International Journal of Heat and Mass Transfer 113 (2017) 295–309

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Effects of channel shape on the cooling performance of hybrid micro-channel and slot-jet module

Yanjun Zhang, Shuangfeng Wang *, Puxian Ding

Key Laboratory of Enhanced Heat Transfer and Energy Conservation of the Ministry of Education, School of Chemistry and Chemical Engineering, South China University of Technology, Guangzhou 510640, Guangdong, China

ARTICLE INFO

Article history:
Received 26 February 2017
Received in revised form 17 May 2017
Accepted 24 May 2017
Available online xxxx

Keywords:
Hybrid micro-channel
Slot-jet module
Cross-section shape
Cooling performance
Parameters optimization

ABSTRACT

This paper investigates the effect of channel shape on the micro-channel and slot-jet module with the realizable k-ε turbulent model. Cooling performance of three channels with a same cross-section area but different shapes (rectangular, trapezoid and circular) are comparatively discussed. The hybrid module with circular channel has the maximum pressure drop at the same flow rate. While the hybrid module with trapezoid channel achieves the best cooling performance. Its superiority in the cooling performance enlarges with the heat flux rising and the pump power increasing, as compared with the other two hybrid modules. The local thermal resistance in the trapezoid channel exhibits peak-shape distribution, which is very different from the other two shapes channel. In addition, the cooling performance of the trapezoid channel module can be further improved by the optimization of the three geometric parameters (channel height, channel bottom width, and channel corner angle). When the optimal value for the three parameters is respectively adopted, the temperature on the bottom surface of the module can be reduced by 12.22%, 14.85% and 7.15% as compared with the worst design, and the temperature difference can also be reduced by 63.60%, 74.86% and 57.16%. What's more, the influence level of these parameters is also compared. Before the reference value, the channel height has the greatest influence on the module bottom surface temperature, whereas after the reference value, the corner angle becomes the largest influence factor. As for the temperature difference on the module bottom surface, it is just opposite.

1. Introduction

During the past few decades, a rapid advance has been made in the packed electronic devices, solar photovoltaic concentrators, laser diode arrays and other high heat production fields, where a more efficient thermal management system is urgently needed. Micro-channel flow and jet impingement are two traditional strategies employed to dissipate the high heat flux [1]. While, deficiencies still remain in them [2]. High pressure drop and poor temperature uniformity exist in the micro-channel chips. As for the jet impingement, there is a big temperature difference on the cooled surface and an abrupt reduction of the heat transfer coefficient away from the impingement region [3]. Arrays of the jets can be used to decrease the temperature difference, but local heat transfer coefficient will be reduced by the interaction between adjacent jets. Hybrid micro-channel and slot jet is one of the most efficient cooling module, which have the advantage of the micro-channel flow and jet impingement simultaneously. It not only provides a very high heat removal capability, but also maintains a high degree of temperature uniformity on the cooling surface [4]. Lelea [5] numerically investigated the micro-tube heat sink with tangential impingement jet and variable fluid properties. He found that the jet impingement heat sink reached both lower peak temperature and lower temperature difference compared with classical micro-tube heat sink. The fluid viscosity was also found has a great influence on the temperature and velocity field of the heat sink. Barrau et al. [2,6] experimentally and numerically studied on a new hybrid jet impingement/micro-channel cooling scheme. It was shown that the hybrid scheme exhibited excellent cooling performance. Yang et al. [7–9] experimentally and numerically studied the unsteady heat transfer and fluid flow in cylindrical channel with a single row of 10 aligned impinging jets. They found that the flow unsteadiness increased as cross-flows accumulated within the impingement channel. Kayansayan and Kucuka [10] investigated the impingement cooling of a semi-cylindrical concave channel. Heat transfer rate at the impingement surface of the concave channel was found higher than that of the flat channel, because of the effect of the channel curvature. Liu and Feng [11] numerically researched the effect of jet nozzle position on the impinge-

* Corresponding author.
E-mail address: sfwang@scut.edu.cn (S. Wang).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.05.092
0017-9310/

Nomenclature

u_i	fluid velocity,
ρ	coolant density
p	coolant pressure
μ	dynamic viscosity of the coolant
k_f	thermal conductivity of the coolant
c_p	specific heat of the coolant
μ_t	turbulent viscosity
σ_k	the turbulent Prandtl number for k
k_s	solid thermal conductivity
T_f	coolant temperature
T_s	wall temperature
T_b	average temperature of the hybrid module bottom surface
ΔT	temperature difference on the module bottom surface
Q	flow rate of the water
P	pump power
u, v, w	velocity component in the direction of coordinate axis
V_n	velocity component perpendicular to the solid-fluid interface
L_{jet}	length of the jet slot
W_{jet}	width of the jet slot
H_{ch}	channel height
W_b	channel bottom width
A_{jet}	cross-sectional area of the slot-jet
α	channel corner angle
β, w	non-dimensional geometric parameters
T_{opt}, ΔT_{opt}	temperature and temperature difference at optimal point
h	local heat transfer coefficient
R_{con}	convection thermal resistance
q	heat flux

ment cooling of gas turbine blade leading edge. They concluded that the side entry jet was desirable to improve the cooling performance of the impingement. Barik et al. [12] explored the heat transfer enhancement using different surface protrusions in the rectangular channel. The heat transfer enhancement rate with triangular protrusions was found to be higher compared to rectangular and trapezoidal protrusions. Husain et al. [13] numerically explored a novel hybrid design with pillars inserted into the rectangular micro-channel. They found pillars in the channel would contribute to a heat transfer rate enhancement. Besides, a kind of manifold micro-channel heat sink was also investigated having favorable capacity to improve the temperature uniformity of the cooled object [14,15]. Sung and Mudawar [4,16–19] experimentally and numerically conducted a series of exploration on the cooling performance of hybrid micro-channel and slot jet module. The rectangular shape micro-channel was used in the hybrid module and the standard k-ε model was employed to analyze the fluid flow and heat transfer characteristics of their hybrid scheme. Sung and Mudawar also analyzed the effects of jet pattern on single-phase [20] and two-phase [21] cooling performance of hybrid micro-channel. It was shown that the decreasing-jet-size pattern achieved the lowest bottom wall temperatures but the equal-jet-size pattern provided the smallest gradients in bottom wall temperature in the condition of single phase. But the pressure drop in the two-phase region was highest for equal-jet-size pattern followed by the decreasing-jet-size and increasing-jet-size patterns, respectively.

However, there are few studies which sufficiently compare the cooling performance of the hybrid micro-channel and slot jet modules with different channel shapes. In this work, a three-dimensional numerical model based on the realizable k-ε turbulent model is developed to research the fluid flow and the heat transfer characteristics of the hybrid module. The pressure drops and the cooling performances of three kinds of hybrid module with different cross-section shape of channels (rectangular, trapezoid and circular) are comparatively analyzed. Subsequently, three geometric parameters of trapezoid channel including channel height, channel bottom width, and channel corner angle are analyzed to search for the optimal design of the hybrid module.

2. Geometry of the hybrid micro-channel and slot-jet modules with different channels

The schematics of the hybrid micro-channel and slot-jet modules with different channels are depicted in Fig. 1. The unit cells consisting of one slot jet and single micro-channel for each kind of modules are illustrated in Fig. 2. They have an equal channel cross-section area, a same channel length, and a same jet orifice area as well. The fluid from the slot jet strikes at the heated bottom surface and runs rapidly outwards. The geometric parameters of each unit cell are listed in Table 1. It can also be noticed that the micro-channel is thermally and hydro-dynamically symmetrical with respect to the boundary conditions. Thus, only a quarter of unit cell is analyzed, as shown in Fig. 3(1).

3. Mathematical model

3.1. Governing equations

Numerical computations were accomplished using the Fluent 6.3 CFD commercial software. The realizable k-ε turbulent model [22], which has been proven to be accurate enough in predicting turbulent flow including strong streamline curvature and vortices, was employed to predict the fluid flow and heat transfer characteristics of the present hybrid scheme. The following conditions were assumed for solving the governing equations: (1) steady state, (2) single phase and turbulent flow.

The governing equations are written in Cartesian tensor notation as follows.

For the fluid region:

Continuity equation:

$$\frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}$$

where u_i is the coolant velocity.

Momentum equation:

$$u_j \frac{\partial \rho u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left((\mu + \mu_t)\frac{\partial u_i}{\partial x_j}\right) \tag{2}$$

where ρ is the coolant density, p is the coolant pressure, μ is the dynamic viscosity of the coolant, and μ_t is the turbulent viscosity.

Energy equation:

$$\rho c_p u_j \frac{\partial T_f}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(k_f + \frac{c_p \mu_t}{Pr_t}\right)\frac{\partial T_f}{\partial x_j}\right) \tag{3}$$

where k_f is the thermal conductivity of the coolant, c_p is the specific heat of the coolant, T_f is the coolant temperature, and $Pr_t = 0.85$ is the turbulent Prandtl number.

The transport equations of the realizable k-ε turbulent model: