Available online at www.sciencedirect.com

SCIENCE DIRECT®

International Journal of Heat and Mass Transfer 49 (2006) 682–694

International Journal of HEAT and MASS TRANSFER

www.elsevier.com/locate/ijhmt

Experimental and numerical investigation of single-phase heat transfer using a hybrid jet-impingement/micro-channel cooling scheme

Myung Ki Sung, Issam Mudawar *

Boiling and Two-Phase Flow Laboratory, School of Mechanical Engineering, Purdue University, 585 Purdue Mall, West Lafayette, IN 47907, USA

Received 29 April 2005; received in revised form 17 August 2005
Available online 18 October 2005

Abstract

Experimental and numerical methods were used to explore the cooling performance of a new hybrid device consisting of a slot jet impinging into a micro-channel, thus capitalizing upon the merits of both cooling configurations. The three-dimensional heat transfer characteristics of this device were analyzed using the standard k–ε turbulent model. Numerical predictions for liquid PF-5052 show excellent agreement with experimental measurements. Vorticity effects are shown to greatly influence cooling performance outside the impingement zone. Higher jet Reynolds numbers yielded stronger attachment to the heated surface and lower surface temperatures. The model was also used to optimize the cooling performance for a water-cooled device. Lower surface temperatures were achieved by decreasing jet width and micro-channel height. These findings are used to recommend a simplified hybrid cooling geometry in pursuit of both lower surface temperatures and smaller temperature gradients across the heated surface.

1. Introduction

Many cooling schemes have been examined in recent years in pursuit of thermal solutions to the problem of heat flux escalation from electronic devices. Of these cooling schemes, jet-impingement and micro-channel flow are considered the two most effective solutions for devices demanding very high-flux removal, such as high-performance microprocessors, laser diode arrays, radars and X-ray anodes [1].

1.1. Jet-impingement cooling

Using a dielectric liquid, jet-impingement produces very large heat transfer coefficients in the impingement zone [2]. The abrupt reduction in cooling effectiveness away from the impingement zone can yield large temperature variations along the surface of the heat-dissipating device [3]. This problem can be circumvented by using an array of jets, creating multiple, closely-spaced impingement zones. Earlier efforts provided valuable insight into the effects of coolant thermophysical properties [4] and interference between jets in multi-jet arrays [5]. In their literature survey, Downs and James [6] discussed how the use of multiple jets to enhance both heat removal and surface temperature uniformity, can be marred by jet interference effects. Interference between circular jets has been shown to reduce heat transfer and produce complex spatial variations in the heat transfer coefficient because of eddy formation and boundary layer separation, as revealed in the flow visualization studies by Goldstein and Timmer [7]. Another drawback to using multiple jets to cool a device surface is flow blockage between jets, especially for jets situated towards the center of the device surface, which complicates flow distribution and exit of the spent fluid [2]. Clearly, better means are needed to capitalize upon the merits of multiple jet impingement while facilitating better flow distribution inside of, and exit from, a cooling module.

Numerical methods have been used extensively to model jet-impingement fluid flow and heat transfer using a variety of turbulence models. Craft et al. [8] tested four different

* Corresponding author. Tel.: +1 765 494 5705; fax: +1 765 494 0539.
E-mail address: mudawar@ecn.purdue.edu (I. Mudawar).

0017-9310/$ - see front matter © 2005 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijheatmasstransfer.2005.08.021

Nomenclature

A_{jet}	area of jet
A_t	area copper block's top test surface
C_μ, C_1, C_2	turbulence model constants
c_p	specific heat at constant pressure
d_h	hydraulic diameter of jet, $2W_{jet}$
G	production of turbulent energy
H	height of unit cell
H_{ch}	height of channel
H_{gap}	height of side channels
H_{jet}	height of jet
H_{th}	height from unit cell bottom boundary to thermocouple holes
H_w	height from unit cell bottom boundary to test surface
k	thermal conductivity; turbulent kinetic energy
l	iteration number
L	length of unit cell
L_{jet}	length of jet
L_{out}	distance between jet and channel outlet
L_1, L_2, L_3	distance between thermocouple holes
$\dot{m}$	mass flux flow rate
n	outer normal coordinate at interface between solid and liquid
N	number of jets in flow distribution plate
Nu	Nusselt number
P	pressure
Pr	Prandtl number
Pr_t	turbulent Prandtl number
P_W	electrical power supplied to copper block
q''	heat flux
q''_{eff}	effective heat flux based on top test surface area of copper block
Re	Reynolds number
T	temperature
T_{in}	fluid inlet temperature
u	velocity component in x direction
U_i	velocity vector
v	velocity component in y direction
$\dot{V}$	volume flow rate
w	velocity component in z direction
w_{in}	fluid inlet velocity
W	width of unit cell
W_{ch}	width of channel
W_{jet}	width of jet
W_w	half-width of wall separating channels
x	Cartesian coordinate
y	Cartesian coordinate
y^+	dimensionless distance normal to wall
z	Cartesian coordinate

Greek symbols

ε	dissipation rate of turbulent kinetic energy
μ	dynamic viscosity
μ_l	eddy viscosity
ρ	density
σ_k, σ_ε	empirical constants in k and ε transport equations
ω	specific dissipation rate of turbulent kinetic energy

Subscripts

av	average
f	fluid
in	inlet
out	outlet
s	solid
t	turbulent
Γ	interface between solid and liquid

turbulence models against experimental jet-impingement data. Park et al. [9] examined confined impinging slot jets with the k–ω turbulence model. Baydar and Ozmen [10] investigated confined high Reynolds number impinging air jets using the standard k–ε turbulence model. They concluded that the k–ε model is capable of accurately predicting the flow characteristics.

1.2. Micro-channel cooling

Micro-channel heat sinks can yield heat removal rates comparable to those of jet-impingement schemes using far smaller coolant flow rates and more compact cooling hardware. Two of their drawbacks are large temperature rise along the direction of fluid flow and relatively high pressure drop [1]. Tuckerman and Pease [11] pioneered the use of micro-channel heat sinks for chip cooling. Their heat sink was fabricated by chemically etching parallel micro-channels into a $1.0 \times 1.0\ \text{cm}^2$ silicon wafer. Their heat sink yielded heat fluxes as high as $790\ \text{W/cm}^2$ using water as working fluid. However, this outstanding performance was realized with enormous penalties in both pressure drop and temperature rise along the direction of fluid flow.

Several numerical studies have been published on the fluid flow and heat transfer characteristics of single-phase micro-channel heat sinks. Weisberg et al. [12] presented a two-dimensional model of micro-channel heat sinks by assuming both hydrodynamically and thermally fully developed flow. Fedorov and Viskanta [13] developed a three-dimensional model to account for development of both the velocity and temperature fields along the flow direction. Qu and Mudawar [14] investigated the transport characteristics of a micro-channel heat sink both experimentally and