International Journal of Thermal Sciences 99 (2016) 161–169

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Optimization of a staggered jet-convex dimple array cooling system

Sun-Min Kim, Arshad Afzal, Kwang-Yong Kim*

Department of Mechanical Engineering, Inha University, 253 Yonghyun-Dong, Nam-Gu, Incheon 402-751, Republic of Korea

A R T I C L E I N F O

Article history:
Received 6 April 2015
Received in revised form
25 August 2015
Accepted 25 August 2015
Available online 23 September 2015

Keywords:
Optimization
Jet-dimpled cooling system
Convex dimples
Thermal resistance
Kriging model
Navier–Stokes equations

A B S T R A C T

Analysis and optimization of a staggered jet-convex dimple array cooling system were performed. A steady incompressible laminar flow and heat transfer in the cooling system were analyzed using three-dimensional Navier–Stokes equations. For the optimization, two design variables, namely the ratio of the height of the dimple to the diameter of the jet and the ratio of the diameter of the dimple to the diameter of the jet, were selected. In addition, thermal resistance was selected as the objective function to evaluate cooling performance. A preliminary parametric study of the objective function was conducted using the design variables, and then, a surrogate modeling of the objective function was performed using the Kriging model. In the optimal design, a lower thermal resistance was achieved with higher pressure drop compared with the reference design. Furthermore, comparative flow analyses for the optimal and reference designs were also performed.

1. Introduction

Recently, impinging jet cooling has been widely used in various industrial engineering fields to achieve high heat transfer rates. Practical applications of impinging jets include cooling of gas turbine blades, deicing of aircrafts, tempering of metal and glass, cooling of microelectronic components, and drying of paper and textiles [1].

The flow structure of the impinging jet is generally divided into three regions, namely the free-jet, stagnation, and wall-jet regions [2]. In the free-jet region, the jet has almost constant velocity along the flow direction before reaching the stagnation region. This free-jet region does not play a significant role in heat transfer compared with the stagnation and wall-jet regions. In the stagnation region, the jet issued from the free-jet region gradually decelerates before impinging on the wall. A high heat transfer rate is generally attainable in this region. After the impingement, the jet is redirected along the target surface in the wall-jet region. A portion of the wall-jet region also significantly contributes to the heat transfer. Therefore, many researchers have focused their studies on the latter two regions, i.e., the stagnation and wall-jet regions.

Hadžiabdić and Hanjalić [3] used large-eddy simulations (LES) to study the vortical and turbulence structures and heat transfer in a normally impinging round jet at the Reynolds number *Re* of 20,000. Jian et al. [4] studied the frequency of a self-excited pulse jet and discussed how the construction of the self-excited nozzle affects the self-excited frequency and amplitude. In addition, Angioletti et al. [5] experimentally and numerically studied an initially laminar and transitional submerged unconfined gaseous jet impinging on a target plane. They tested three turbulence models and presented the velocity maps and local *Nu* distributions.

The heat transfer performance of a laminar annular jet impinging on a surface was compared with that of a standard circular jet having the same mass and momentum effluxes at the nozzle exit by Chattopadhyay [6]. The results of the comparison showed that the heat transfer from the annular jet was about 20% less than that from the circular jet. Bergthorson et al. [7] investigated a round laminar impinging jet and compared the numerical results with experimental data for jet centerline velocities. Furthermore, a combined approach to characterize the flow field and local heat transfer in jet impingement configurations, featuring a mass transfer experiment and a digital visualization technique, was employed by Angioletti et al. [8]. They discussed the entrainment and the related pattern of structure creation by exploiting the PIV (particle image velocimetry) post-processing capability and found an axial velocity pulsation, which weakens the boundary layer stability at stagnation and affects the local heat transfer.

In the last few decades, electronic devices have become progressively smaller. As a heat source becomes smaller, the

* Corresponding author. Tel.: +82 32 872 3096; fax: +82 32 868 1716.
E-mail address: kykim@inha.ac.kr (K.-Y. Kim).

http://dx.doi.org/10.1016/j.ijthermalsci.2015.08.013
1290-0729/

Nomenclature

D	diameter (m)
H	height (m)
Nu	Nusselt number
T	temperature (K)
c_p	specific heat (J/kg K)
p	pressure (Pa)
x, y, z	Cartesian coordinates (m)
Greek symbols	
α	design variable, H_d/D_j
β	design variable, D_d/D_j
ρ	density (kg/m^3)
ν	kinematic viscosity (m^2/s)
μ	dynamic viscosity (Pa s)
τ	shear stress (N/m^2)
Superscripts	
*	normalized value
Subscripts	
f	fluid
j	jet
prj	projected
d	dimple

corresponding cooling system also needs to be smaller, and therefore, the fluid flows in the cooling systems for microelectronic devices become laminar [9]. Therefore, many researchers have investigated laminar impinging jets for the enhancement of cooling in microelectronic devices. Husain et al. [10] investigated the thermal and hydraulic performances of a microjet array cooling system for the thermal management of a high-power light emitting diode (LED) array. They evaluated the performances of several microjet array configurations and optimized a four-jet array cooling system with two geometric design variables to obtain Pareto-optimal solutions (PODs) for the two objectives, namely the thermal resistance and pressure drop, by analyzing the steady incompressible turbulent flow and conjugate heat transfer in the cooling system. In addition, a silicon-based microjet-impingement heat sink, consisting of up to 16 impinging jets with several parallel and staggered microjet configurations, was investigated by Husain et al. [11] for electronic cooling applications. They reported that the design with a staggered array of 13 impinging jets exhibited the best performance among the various configurations. They also obtained the global Pareto-optimal designs for the overall thermal resistance and pumping power.

In cooling passages, dimples on solid surfaces are usually used to prevent the development of a thermal boundary layer and to increase the production rate of turbulent kinetic energy, thereby enhancing the turbulent heat transfer [12]. Therefore, these dimples are considered as heat transfer enhancers. However, some researchers have reported the dysfunction of dimples under specific conditions. For example, Ekkad and Kontrovitz [13] studied a turbulent jet impinging on a target surface with a dimpled pattern. They reported that dimpled target surfaces—either inline or staggered with respect to the jet location—produced lower heat transfer coefficients than non-dimpled target surfaces at high Reynolds numbers, because the bursting phenomena of the flow inside the dimples at these Reynolds numbers weakened the impingement effect. However, most research works conducted thus far have been focused on the serviceability of the dimpled surface. A numerical study on the flow and heat transfer in a semi-confined axisymmetric laminar air jet impinging on a concave dimpled surface with uniform heat flux was performed by Kanokjaruvijit et al. [14]. They tested various parameters involved in the heat transfer. Moreover, Kim et al. [15] reported a numerical procedure for optimizing a cooling channel with staggered elliptic dimples in order to enhance the turbulent heat transfer and reduce the pressure loss. They employed a hybrid multi-objective evolutionary algorithm (MOEA) to find the optimum designs considering both the heat transfer and friction loss. As introduced above, recently, the design optimization techniques become popular in the designs of thermo-fluid systems [16–20].

Numerical analysis and optimization of staggered dimple channels by using two surrogates, namely the response surface approximation and the Kriging models, were performed by Husain et al. [21]. They obtained an enhanced Pareto-optimal front by performing local resampling of the Pareto-optimal front, which provided relatively more-accurate PODs in the design space of each surrogate model. A further study on the impinging jet-dimple cooling system was conducted recently by Kim and Kim [22]. They numerically evaluated the heat transfer performances of a cooling system by using laminar impinging jets with different combinations of dimple shapes (convex or concave) and relative locations (staggered or inline) of the jets and dimples. They reported that the staggered impinging jet-convex dimple array showed the best heat transfer performance whereas the staggered concave configuration showed the lowest pressure drop. Additionally, they conducted a parametric study with two geometric variables for the staggered impinging jet-convex dimple array.

As mentioned above, Kim and Kim [22] suggested that the staggered impinging jet-convex dimple array is the best impinging jet-dimple configuration. However, the cooling performance of this configuration has not been optimized yet. Thus, the present research was aimed at the optimization of the cooling system with staggered impinging jet-convex dimple arrays using surrogate modeling [23]. The flow and heat transfer analyses were performed using three-dimensional (3D) Navier–Stokes equations. Moreover, the Kriging model [24] was employed to approximate the objective function, i.e., the thermal resistance, to reduce the computation time for the optimization.

2. Numerical analysis

The schematic diagram of the reference geometry of a staggered jet-convex dimple array cooling system is shown in Fig. 1. The diameter of the jet nozzles, D_j, is 2.5 mm, and the distance from the jet exit to the dimpled surface is 22.5 mm. The diameter D_d, height H_d, and pitch of the dimples are 14.07 mm, 4.0 mm, and 20 mm, respectively. The bottom surface is square shaped, with each side having a length of 140 mm, and the number of jets and dimples in the array are 6 × 6 and 5 × 5, respectively.

Using the commercial software ANSYS CFX 15.0 [25], numerical analyses of fluid flow and heat transfer were conducted. The solutions were obtained using the finite volume method to discretize the governing differential equations for the conservations of mass, momentum, and energy, which can be written as follows:

$$\frac{\partial\left(\rho_f v_i\right)}{\partial x_i} = 0 \tag{1}$$