Energy Conversion and Management 174 (2018) 218–226

Contents lists available at ScienceDirect

Energy Conversion and Management

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

Minimizing thermal interference effects of multiple heat sources for effective cooling of power conversion electronics

Youngchan Yoon, Seounghwan Hyeon, Dong Rip Kim, Kwan-Soo Lee*

School of Mechanical Engineering, Hanyang University, 222 Wangsimni-ro, Seongdong-gu, Seoul 04763, Republic of Korea

ARTICLE INFO

Keywords:
Electronic cooling
Heat sink
Forced convection
Multiple heat source
Thermal interference

ABSTRACT

The thermal performance of a plate fin heat sink was investigated as a function of the location of multiple heat sources to minimize the thermal interference effect. Numerical analysis was performed under forced-convection conditions to design a cooling system for a high-power heat source. The thermal behaviors of the heat sink and surrounding air were simulated. We also investigated the thermal performances of various base thicknesses under local heat flux conditions to determine the optimal heat sink base thickness using the width of the heat source and conduction coefficient of the heat sink. The optimal location was determined by investigating the effects of the Reynolds number, thermal conductivity of the heat sink, heat transfer rate ratio for multiple heat sources, and width of the heat source. An installation guideline of the plate fin heat sink was prepared to help users avoid the thermal interference effect depending on the width of their heat source. By applying a correlation equation to obtain the optimal location where the maximal temperature of the heat source is minimized, the thermal resistance was decreased by up to 30%.

1. Introduction

In electronic devices, heat is generated during the energy conversion and transfer process. When high-performance electronic devices are integrated in a small location, their heat generation significantly increases. Because the resulting heat adversely affects the reliability of the product, it is necessary to develop a heat sink to effectively dissipate the heat. Most studies on improving the thermal performance of heat sinks focus on the uniform heat flux from the heat sink base. Jang et al. [1] used a pin-fin instead of a conventional plate-fin in a circular heat sink to achieve 30% lighter weight and 20% higher thermal performance. Jeon et al. [2] showed that the thermal performance per unit mass was improved in a plate-fin structure with two fin heights compared with a plate fin with one height. Joo and Kim [3] compared a vertically oriented plate fin type heat sink with a pin-fin type unit, and as a result, the pin-fin type heat sink performed better than the plate-type in terms of unit mass. Kanargi et al. [4] used an oblique pin-fin to achieve the same thermal performance as a straight pin-fin but with reduced fan power of up to 75%. Pakrouh et al. [5] improved the thermal performance of a heat sink by optimizing the design parameters such as fin number, fin thickness, and fin height. Shin et al. [6] optimized the shapes of convergent, uniform, and divergent type heat sinks under corona wind conditions and confirmed their performance in replacing existing CPU cooling devices. Kwon et al. [7] suggested a correlation that could predict the heat transfer coefficient for the radial plate fin heat sink with enhanced performance, based on fin design optimization. Park et al. [8,9] and Li et al. [10] improved thermal performance by installing a chimney around the heat sink to increase the flowrate of cooling air around the fins of the heat sink. They also analyzed the effects of the orientation of the chimney and proposed a correlation equation to predict the thermal performance.

However, in the case of high-power devices such as power electronic equipment, the heat sink is much larger than the heat source to ensure sufficient cooling performance. When the heat is not distributed evenly across the heat sink, the performance of the electronic equipment can suffer [11]. Therefore, it is difficult to apply the previous results because the thermal effects associated with heat distribution must be considered. In this regard, Lee et al. [12] and Muzychka et al. [13] developed a model to calculate the spreading resistance when the heat source is positioned at the center of the heat sink. Yovanovich et al. [14] suggested the spreading resistance model in a semi-infinite heat sink with a strip-shaped heat source. Kim et al. [15] improved the thermal performance of the device by coating the heat sink to reduce the spreading resistance. However, all studies mentioned above considered only the spreading resistance under partial heating conditions for a heat sink with no fin, which limits their wide application.

* Corresponding author.
E-mail address: ksleehy@hanyang.ac.kr (K.-S. Lee).

https://doi.org/10.1016/j.enconman.2018.08.047
Received 14 June 2018; Received in revised form 3 August 2018; Accepted 12 August 2018

Nomenclature

A	surface area [mm^2]
D_h	hydraulic diameter [mm]
H	height [mm]
h	convective heat transfer coefficient [$W/m^2 \cdot °C$]
k	thermal conductivity [W/m·°C]
L	length [mm]
N	number of fin arrays
P	wetted perimeter [m]
p	pressure [Pa]
$\dot{Q}$	heat transfer rate [W]
R_{th}	thermal resistance [°C/W]
s	space [mm]
T	temperature [°C]
t	thickness [mm]
u	velocity [m/s]

Greek symbols

ρ	density [kg/m^3]
μ	dynamic viscosity [$N \cdot s/m^2$]

Subscripts

a	air
avg	average
b	base
bot	bottom
c	cross cut
ch	channel
$conv$	convective
eff	effective
f	fin
h	heat sink
in	in
$high$	high-power heat source
low	low-power heat source
m	material
p	polyethylene
s	contact area of the heat source
sp	spreading
top	top
u	upstream

To apply these results to practical systems, studies have been conducted on a heat sink with a different type of fin under local heating conditions. Abdoli et al. [16] investigated the thermal performance of pin-fin shapes when the heat source was positioned in the middle of the heat sink. They found that the maximal temperature was lowest for a hydrofoil pin-fin. Lee [17] proposed an arrangement of pin-fins to reduce the maximal temperature when the heat source was installed symmetrically at the front and rear of the heat sink. Turkakar et al. [18] achieved improved thermal performance by making the channel denser in the area where the heat source was installed. Wally et al. [19] added a subchannel to the heat sink channel under nonuniform heat flux conditions. Consequently, the thermal performance decreased by 20%, whereas the pumping power increased by 11%. These studies, however, did not examine the thermal behavior of the heating element to the mounting position.

Cho et al. [20] divided the heat sink into nine regions and installed a heater. The maximal temperature was highest when the heating element was in the front of the heat sink. Emekwuru et al. [21] divided the heat sink into three regions; the best performance was observed when the heat source was located in the middle. Yoon et al. [22] investigated thermal performance as a function of the position of the heat source while moving a strip-shaped heat source continuously from the front to the back side of the heat sink. The best thermal performance was observed when the heat source was located at the back of the center of the heat sink. Although their studies are useful, practical applications still need the investigation of thermal heat sink performance for partial heating problems in terms of locations of multiple heat sources, which is still lacking.

Fig. 1(a) shows the power electronic devices typically used in the industry, which have one or more heating elements with different power levels. In this study, the heating power of multiple heat sources is classified into high power and low power to simulate the actual cooling conditions. The thermal interference effects were analyzed between the multiple heat sources in various operating environments, and a heat source installation guideline was compiled to help avoid further interference. Finally, we suggest the optimal installation locations for high- and low-power heat sources to reduce the maximal temperature of the heat source.

2. Numerical and experimental methods

2.1. Numerical model

As shown in Fig. 1(b), the numerical analysis was performed for cases where high-power heat sources and low-power heat sources were installed as a y-directional, semi-infinite heat sink. The top of the heat sink was a closed system. The assumptions for the numerical analysis were as follows:

(1) The flow is in a steady state.
(2) The air properties are independent of the temperature.
(3) The radiative heat transfer can be ignored.
(4) The top wall of the heat sink is insulated.

Table 1 lists the boundary conditions and governing equations for the numerical analysis. The momentum equation uses a shear stress transport k–ω model for the turbulent flow [22].

2.2. Numerical procedure

The domain of the numerical analysis is presented in Fig. 2.

Commercial extruded heat sinks for cooling electronic devices generally have a fin thickness of approximately 2–3 mm and a base thickness of approximately 10–30% of the total heat sink height. Based on this, we carried out parametric studies of heat sinks.

The heat sink contained fourteen fins, with a fin thickness of 3 mm. The heat sink had a base thickness of 8 mm, a total height of 50 mm, and an overall length of 150 mm. The ratio of each heat source width to length was 0.4. Periodic conditions were applied in the analysis to simulate a semi-infinite heat sink and heating conditions [23].

To investigate the dependence of the number of grids, the hexahedral meshes were varied from 412,000 to 3,325,000. To determine the appropriate air domain, the domain was changed from 0.5 L to 1.5 L toward the direction of the air inlet and outlet. When the maximal temperature deviation of the heat sink was within 0.1 °C, the air domain and reference number of grids were determined to be 1 L and 1,715,000, respectively. ANSYS Fluent release 17.0 was used for performing the numerical analysis. The flow field for the velocity and pressure was calculated using the SIMPLE algorithm. A second-order upwind scheme was used to discretize the convective terms and energy