International Journal of Thermal Sciences 49 (2010) 721–732

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Optimization of multiple heaters in a vented enclosure – A combined numerical and experimental study

T.V. Radhakrishnan [1], C. Balaji*, S.P. Venkateshan

Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai 600 036, India

ARTICLE INFO

Article history:
Received 14 April 2008
Received in revised form
29 August 2009
Accepted 28 September 2009
Available online 12 November 2009

Keywords:
Experiments
Temperature
Numerical
Optimization
Response surface
Ventilated enclosure
Heaters

ABSTRACT

This paper discusses the results of an experimental and numerical study of fluid flow and heat transfer in an enclosure where multiple heaters are arranged in a staggered fashion. Experiments were carried out for Reynolds numbers, in the range $1800 \leq Re \leq 4500$ and Grashof numbers in the range $2.5 \times 10^4 \leq Gr \leq 3 \times 10^5$. Numerical simulations were carried out for two dimensional, steady, incompressible turbulent flow and the results of the numerical study are compared with the experimental results. The temperature distribution gives an insight into the power management among the heaters, so that the "coolest" heater can be loaded most to maximize the total heat dissipation, for a prescribed temperature excess, for all the heaters. Two methods are used to achieve the target temperature for all heaters, namely (i) trial and error method and (ii) the response surface method. The latter method was adopted, to simultaneously maximize the heat input and minimize the temperature deviation from the target temperature, by employing a composite objective function. The numerically obtained optimal solution was finally verified by carrying out experiments. The method of response surface was found to be effective in optimizing the total heat transfer for a given target temperature.

1. Introduction

Printed circuit boards in electronic equipment, in general have multiple heat generating elements embedded in them. Many such printed circuit boards can be seen arranged in different slots or racks inside the equipment bay of a satellite launch vehicles and multi nodal computing systems. The cabinets are invariably cooled by air flow for low to medium heat flux levels. For carrying out a numerical analysis or experimentation, such systems can be approximated as free standing heat generating elements placed in a ventilated cavity.

A number of studies have been carried out on heat transfer from multiple heat generating elements. Both experimental and numerical investigations on natural, mixed and forced convective heat transfer from multiple heat generating elements have been reported in literature. Lai et al. [1] conducted a numerical study on mixed convection in horizontal porous layers by discrete heat sources kept at isothermal conditions. For Rayleigh numbers in the range of 10–500 and Peclet numbers in the range of 0.1–100, steady state results have been generated. The overall Nusselt numbers increased with the number of heat sources and the Rayleigh number. For Rayleigh number greater than 50, it was observed that the flow is oscillatory and unstable. Choi et al. [2] studied, numerically, heat transfer from an electronic module in an inclined cavity, for natural, mixed and forced convection regimes. The study indicated that the overall Nusselt number strongly depends on the angle of inclination.

An easy-to-use method has been developed by Funk et al. [3] to predict steady state temperatures of PCBs embedded with single or multiple heat sources. The method is based on the Green's function, wherein the heat diffusion equation is solved and is quite fast in determining the temperatures. Heindel et al. [4] studied conjugate heat transfer from multiple flush mounted heaters in a cavity. The studies have been carried out for different combinations of thermal conductivities for the wall and the fluid. For small Rayleigh numbers, the heat transfer is dominated by conduction and as the Rayleigh number increases, convective heat transfer begins to dominate. Hung et al. [5] studied, numerically, laminar flow over multiple heat sources in a horizontal channel. At the entrance to the channel, the flow is split into two, because of which the top flow impinges the heaters only partially. The bottom flow is sucked into the main flow due to pressure difference which enhances the heat transfer rate. The effect of opening at the inlet and the gap between the heat sources on the heat transfer was studied numerically.

Du et al. [6] carried out a numerical study of two dimensional steady mixed convection heat transfer in a vertical channel that had open bottom and top along with protruding discrete heaters

* Corresponding author. Tel.: +91 44 2257 4689; fax: +91 44 2257 0509.
E-mail address: balaji@iitm.ac.in (C. Balaji).
[1] Vikram Sarabhai Space Centre, Indian Space Research Organisation, Trivandrum, India.

1290-0729/$ – see front matter
doi:10.1016/j.ijthermalsci.2009.09.012

Nomenclature

A	area m^2
C_p	specific heat at constant pressure, J/kg K
C_v	specific at constant volume, J/kg K
E	total energy, J
Gr	Grashof number, $\frac{g\beta\Delta TL^3}{\nu^2}$
g	gravitational acceleration, 9.81 m/s^2
G_k	generation of turbulent kinetic energy due to mean velocity gradient, J
G_b	generation of turbulent kinetic energy due to buoyancy, J
h	heat transfer coefficient W/m^{2}K
I	turbulence intensity, m^2/s^2
K	turbulent kinetic energy m^2/s^2
k	thermal conductivity W/m-K
L	height of the heater, m
Nu	Nusselt number, $\frac{hL}{k}$
P	pressure Pa
Pr	Prandtl number $\frac{\mu C_p}{k}$
p	penalty parameter
Q	heat input, W
q_v	volumetric heat generation rate, W/m^3
R	universal gas-law constant, 8.314 $\times$ 10^3 J/kmolK
Ra	Rayleigh Number, Gr Pr
Re	Reynolds number, $\frac{uL}{\nu}$
Ri	Richardson number, $\frac{\mathrm{Gr}}{\mathrm{Re}^2}$
S	dimensionless length, X/W
S_k	user defined source term in RNG turbulence equation, W/m^3
S_ε	user defined source term in RNG turbulence equation, W/m^3
T	temperature, K
T_{avg}	average temperature of the heater, K
T_∞	ambient temperature, K
T_w	wall temperature, K
ΔT_{ref}	reference temperature difference, $\frac{q_v L^2}{k_s}$, K
u	horizontal component of the velocity, m/s
u_∞	inlet velocity, m/s
v	vertical component of the velocity, m/s
W	width of the chamber, mm
X	location of the heater in the horizontal direction from left wall, mm
x	horizontal distance, m
y	vertical distance, m

Greek symbols

α	thermal diffusivity, m^2/s or inverse effective Prandtl number
β	coefficient of thermal expansion, 1/K
δ_{ij}	Kronecker delta
δ	perturbation parameter
ε	emissivity of the surface or dissipation rate of turbulent kinetic energy, kg/m^{2}s^2
Φ	dimensionless average temperature, $\frac{T_{avg}-T_\infty}{\Delta T_{ref}}$
μ	viscosity, Ns/m^2
ν	kinematic viscosity, m^2/s
ρ	density of air, kg/m^3
σ	Stefan Boltzmann constant, 5.67x 10^{-8} W/m^{2}K^4
τ	shear stress, N/m^2

Subscripts

avg	average
eff	effective
∞	inlet and ambient
f	fluid
i	heater number
n	experiment number
s	solid
v	volumetric

installed on one side. Studies were carried out for $0 \leq \mathrm{Ra} \leq 10^7$, $0 \leq \mathrm{Re} \leq 200$ and $1 \leq A \leq 6$, where A is the aspect ratio. It was found that the entrance lengths exert a negative effect on the cooling of the components in the natural and mixed convection regimes, at low Reynolds numbers. At high Reynolds numbers, the effect of the entrance length was found to be negligible. Joseph et al. [7] also carried out experimental investigations on natural convection from embedded heat sources placed in a vertical channel. Deng et al. [8] carried out a numerical study of natural convection heat transfer in a horizontal enclosure with discrete heat sources. Heat sources of different types, orientation and size have been considered. In this study, a combined temperature scale method and a unified heat transfer characteristic analysis for convenient representation of heat transfer due to discrete heat sources were proposed.

Keyhani et al. [9] carried out an experimental study of heat transfer from discrete heater elements in a vertical cavity. In this study, unheated and heated elements of equal dimensions were mounted on one of the walls. The opposite wall was kept at a constant temperature. The heat transfer data and the flow visualization photographs indicated that stratification is the primary factor influencing the temperature of the heated sections. Sultan [10] conducted experiments to study forced convection heat transfer from multiple protruding heat sources in a horizontal channel of small aspect ratio with passive cooling. Perforated holes were arranged at the base of channel in a staggered manner in two rows between the heat sources. Due to an increase of the temperature between the heaters, the outside air is drawn naturally through the perforated holes. The effect of the size of holes on the heat transfer for a range of Reynolds numbers was studied. It was found that the heat transfer coefficient was enhanced for all values of the hole/open area ratio. However, holes with $\beta = 0.0409$ gave the best thermal performance for $376 < \mathrm{Re} < 6170$, where β is the ratio of total hole area to one side area of the heater.

Rodgers et al. [11] carried out a numerical study of forced convection from printed circuit boards with heat generating elements and verified its accuracy based on experiments. The heat sources used were SO16, TSOP 48 and PQFP 208 [12] packages. The objective of the study was to create bench mark test data for numerical modeling. Baskaya et al. [13] investigated, experimentally, heat transfer from an array of heat sources in a rectangular channel, in the mixed convection regime. The configuration was typically an electronic package. It was observed that when the Reynolds number decreased and/or Grashof number increased, an enhancement in heat transfer was obtained through buoyancy driven secondary flow. The study gave guidelines for the placement of electronic packages subjected to flow in a rectangular channel.

The above discussion focused on numerical and experimental studies on heat transfer and flow characteristics for multiple heat generating elements. Optimization methods and studies applicable to multiple heat generating elements, most of which are numerical, are discussed next.

Perez et al. [14] used a quadratic response surface approximation for engineering design problems. A methodology to reduce the size