IOP Conference Series: Materials
Science and Engineering

Numerical Study of Conjugate Natural Convection Heat Transfer Using One Phase Liquid Cooling

To cite this article: F A Gdhaidh *et al* 2014 *IOP Conf. Ser.: Mater. Sci. Eng.* **65** 012012

Numerical Study of Conjugate Natural Convection Heat Transfer Using One Phase Liquid Cooling

F A Gdhaidh[1], K Hussain[2] and H S Qi[3]
[1] PhD student in Faculty of Engineering and Informatics at Bradford University, BD7 1DP, UK
[2] Senior Lecturer in Mechanical Engineering and the Head of the School of Engineering at Bradford University, BD7 1DP, UK
[3] Senior Lecturer in Mechanical Engineering at Bradford University, BD7 1DP, UK

E-mail:F.A.Gdhaidh@bradford.ac.uk

Abstract. A numerical study in 3-D is performed using water as a cooling fluid to investigate the one phase natural convection heat transfer within enclosure. A heat source representing a computer CPU mounted on one vertical wall of a rectangular enclosure is simulated while a heat sink is installed on the opposite vertical wall of the enclosure. The air flow inside the computer compartment is created by using an exhaust fan, and the flow is assumed to be turbulent. The applied power considered ranges from $15 - 40\ W$. In order to determine the thermal behaviour of the cooling system, the effect of the heat input and the dimension of the enclosure are investigated. The results illustrate that as the size of the enclosure increase the chip temperature declined. However the drop in the temperature is very small when the width increased more than $50\ mm$. When the enclosure was filled with water the temperature was reduced by 38%. Also the cooling system maintains the maximum chip temperature at 71.5°C when the heat input of 40 W was assumed and this is within the current recommended computer electronic chips temperature of no more than 85°C.

1. Introduction

The natural convection is considered as an effective and most economical cooling strategy for computer electronic devices [1]. However, the conventional method for free convection using air as the medium is not sufficient enough to remove the high heat flux from the modern computer electronic chips [2]. This is due to the low heat transfer rate of air when compared to other high rate fluids.

The results of natural convection inside rectangular enclosures filled with air to cool electronic chips show the limitation of air natural convection cooling [3]. There is a promising alternative way to keep the electronic devices at the safe operation temperature by using liquids. This is justified by the fact that the liquids have relatively high removal of heat when compared to air. In natural convection the fluid motion inside enclosures whether these enclosures were vertical or horizontal occurs due to the temperature gradient change, therefore the isothermal boundary condition is adopted [4-6]. Sathe et al. [7] studied numerically the effect of installed square enclosure filled with dielectric liquid on the chip temperature. They noticed that the highest chip temperature occurred when the upper wall of the enclosure was at fixed temperature and other walls adiabatic. Numerical and experimental studies were carried out by Heindel et al. [8] where two different coolant fluids (water and FC-77) were used within the enclosure to cool heaters where the cold wall is fixed at room temperature. They concluded that as the heat flux increases, the convection coefficients along the heater face and vertical velocity also increases. Further work by Heindel et al. [9] showed that using fins on the heaters enhanced the heat transfer as much as 24 and 15 times for vertical and horizontal cavity orientations respectively when the cold wall kept isothermally at 15℃. In [10, 11] an experimental studies were reported using water as the cooling fluid. Their results indicate that the Nusselt number based on heat source length increases as the heat flux increased. Other investigations on natural convection cooling, using ethylene glycol as fluid to cool heat sources which attached to rectangular enclosures were carried out by Keyhani et al. [12] and Ju and Chen [13]. In both of these studies the cold wall is maintained isothermal at a fixed temperature. They concluded that the enclosure width influences the heat transfer process but this influence is weak when the ratio of enclosure width to heater height was around 4.0.

Desktop computers are complex in terms of heat transfer control and it is impossible to analyse using exact solution methods. Therefore CFD prediction tools are adopted to explore various designs quickly with acceptable accuracy. In the thermal design there are some successes for analysing complex electronics system using CFD. The detailed temperature and flow fields have been simulated through computer chassis with two fans using Flotherm code by Lee and Mahalingam [14].

From the previous studies the far end of the enclosure has been assumed isothermal. However In this study, a desktop computer heat transfer characteristics is examined with the aim to remove excessive heat generated by the electronic chips using one phase water cooling system with adopting a realistic boundary conditions for the cold wall by assuming heat sink with air flow created by exhaust fan.

2. Problem description

A numerical study of coupling between natural convection heat transfer inside enclosure and forced convection within the computer covering case has been investigated. The heat source is mounted on a vertical substrate and the substrate is connected to the enclosure. A heat sink without fan is attached to the other end of the enclosure and the enclosure was filled with water. The air flow inside the computer covering case is created by a fan installed at the end of the computer case. Figure 1 shows a schematic diagram of the three-dimensional system used for the analysis. The heat input chosen was in the range of $15-40W$ with $5W$ increment and the value of power remains constant during the simulation. Table 1 lists all the required dimensions for the simulated model.

Table 1. Physical Model Dimension in mm.

Components	Height	Length	Width
Computer case	352	418	171
Hard drive	23	145	100
Heater	37.5	37.5	5
Enclosure	77	68	50
Substrate	77	68	3
Main board	245	200	1
Power supply	89	138	148

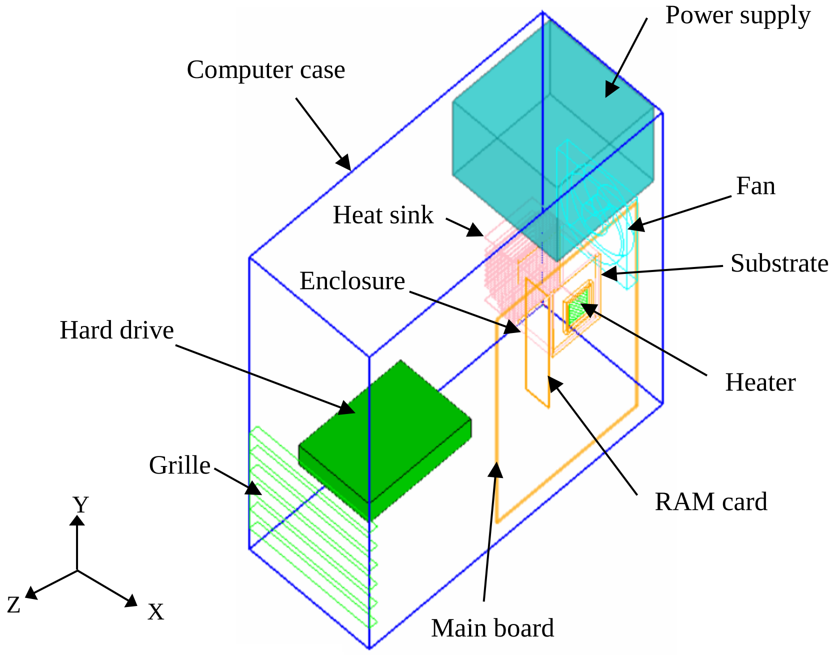


Figure 1. Schematic diagram of the physical model.

The fan is installed at the back of the computer case with 90 mm diameter and a constant flow rate of $0.026\,\mathrm{m}^3/\mathrm{s}$ was assumed.

3. Mathematical model

The mathematical model was constrained by the following assumptions: 3-D steady state heat transfer, contact resistance between the heater/substrate, substrate/enclosure and enclosure/heat sink interfaces equal to $0.18℃/W$. The radiation effects are neglected and an incompressible fluid flow was assumed. The flow through the computer chassis was assumed to be turbulent and hence the $k - \varepsilon$ type model was used.

Applying the above assumptions on the governing differential equations of continuity, momentum and energy, yield the following equations:

Continuity:

$$\nabla . V = 0 \tag{1}$$

Momentum equation:

$$\rho(V.\nabla V) = -\nabla P + \mu\nabla^2 V + F \tag{2}$$

Energy equation:

$$V.\nabla T = \alpha\nabla^2 T + S \tag{3}$$

where ρ is the density, V is the velocity components, μ is the viscosity, P is the pressure, α is the thermal diffusivity, T is the temperature and F and S are the body force and the source term respectively.

In order to complete the mathematical model, the following boundary and initial conditions were used: the flow rate inside the computer case which is created by the fan was assumed fixed. The air inlet temperature was set at 22°C, air physical properties are calculated at the same temperature and the isoflux condition was applied for every test.

The average Nusselt number for substrate/water interface is determined by integrating the local Nusselt number along the substrate surface. Therefore the average Nusselt number ($\overline{\mathrm{N_u}}$) could be written as:

$$\overline{N_u} = \frac{\bar{h}l}{K_w} \tag{4}$$

where $\bar{h}$ is the average heat transfer coefficient, l is the substrate height and K_w is the thermal conductivity of water.

4. Solution procedure

In present study the commercial (ANSYS –CFD) software which has heat transfer capability was used to examine the system. This software is based on a Finite-Volume scheme to convert the governing equations to algebraic equations that can be solved numerically. The "SIMPLE" algorithm (semi-implicit method for pressure linked equations) is used to handle the coupling between pressure and velocity. To avoid the divergence in the iteration process, Under-relaxation techniques are used to slow down the changes between iterations.

Although there are different regions (solids and fluids), and these have different thermal properties and equations, however, the numerical solutions within the computational domain for continuity, momentum and energy equations are obtained simultaneously in all regions.

The effects of the elements number on the results have been examined. The final optimised mesh data points of 311,204 were used, and this gave stable solution.

5. Results and discussion

A numerical study was carried out to examine the thermal behaviour of the proposed system. Results and analysis of the flow and the thermal field are presented in this section. During the design of the cooling system, a number of limitations/constrains were considered, such as the available space and the maximum allowed operation temperature of the system. These constraints are applied to obtain the optimum width of the enclosure when the applied heat input was set at $40\ W$.

The second test was to examine the effect of using different heat input level (Q). Heat sources in the range of $15 - 40\ W$ with $5\ W$ increment were used. According to deSorgo [15] when two surfaces in contact, less than 1% of the surfaces is in touch because all the solid surface have certain roughness regardless of how smooth they are. Therefore, a thermal resistance value of $0.18\ ℃/W$ between any two solid surfaces in contact was assumed.

5.1 Enclosure width effects

Figure 2 illustrates the maximum heater temperature for different enclosure widths. A range of $0 - 60\ mm$ with $10\ mm$ increments was used, and the heat input was fixed at $40\ W$. In the case when the enclosure width is $0\ mm$, the heater temperature surface was high and reaches a temperature of 115.4°C.

An enclosure of 20 mm wide reduced the temperature sharply to 76.4°C. A further reduction can be achieved by increasing the width of the enclosure. A 60 mm enclosure reduces the temperature to 70.6°C. . The change in the temperature is due to the increase in the heat transfer area of the enclosure wall surfaces and also the effect of the water thermal properties. Moreover the water flow inside the enclosure helps to cool regions closer to the hot wall.

The results show that only 1% change in temperature between the 60 and $50\ mm$ enclosure widths. Hence a $50\ mm$ enclosure was used as the optimum dimension for our enclosure to carry further analysis. Moreover a $50\ mm$ enclosure filled with water reduced the initial temperature without enclosure by 38%. Note that the volume of water inside the enclosure can be obtained using the dimension of enclosure and this result in a total volume of water of 0.207 litre.

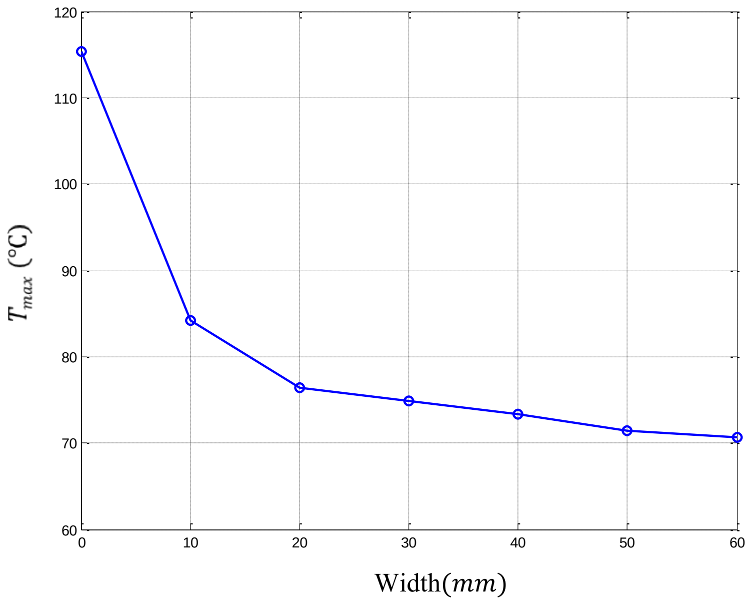


Figure 2. The enclosure width effects on the maximum heater temperature.

5.2 *Heat input effects*

In this investigation the heat input applied to the heater varies between $15 - 40W$. The heater material used corresponds to extruded aluminium with thermal conductivity of ($k_h = 205\ W/mK$), producing constant heat input. The substrate material chosen was copper with thermal conductivity ($k_s = 387\ W/mK$). Further, the fluid Prandtl number was assumed to be 5 corresponding to water. The fluids and the material properties are assumed constant. The effects of different values of heat energies on the maximum system temperature and average Nusselt number are presented.

The maximum system temperature. In this section the maximum temperature in the system which occurs in the heater is studied. As mentioned earlier, the recommended electronic chip temperature should not exceed 85°C in most electronic devices [16]. However, the ideal maximum working temperature is in range of $65 - 70$°C. Figure 3 illustrates the maximum temperature in the system used in this study for different values of heat input.

Figure 3 illustrates that the maximum temperature of the cooling system increased as the heat input increased. The curve is almost linear and that is because the temperature is a function of heat input when the other conditions are fixed.

Figure 4 shows the temperature distribution for two values of heat input 15 and 40W. The maximum temperature increased from 40.02°C at 15W to reach 71.48°C at 40W. The contour lines of temperature are nearly vertical in the fluid region because the heat transfer in the enclosure is controlled by conduction due to the slow water motion within the enclosure. Further, the maximum temperature is located at the centre of the heater as shown. For clarity the temperature in the enclosure, substrate and heater are also illustrated in Figure 4.

Figure 5 shows the substrate/water interface temperature for the full range of the applied heat input. The heat input flows from the heater and the substrate towards the water. The flat part of the curve denotes zero temperature gradient where the temperature is maximum. This region corresponds to the size of the heater. The zero gradient region is due to the higher heater and substrate thermal

IOP Conf. Series: Materials Science and Engineering **65** (2014) 012012 doi:10.1088/1757-899X/65/1/012012

conductivity. The positive gradient region of the curve represents the increase in the thermal boundary layer while the negative gradient represents the decrease in the thermal boundary layer.

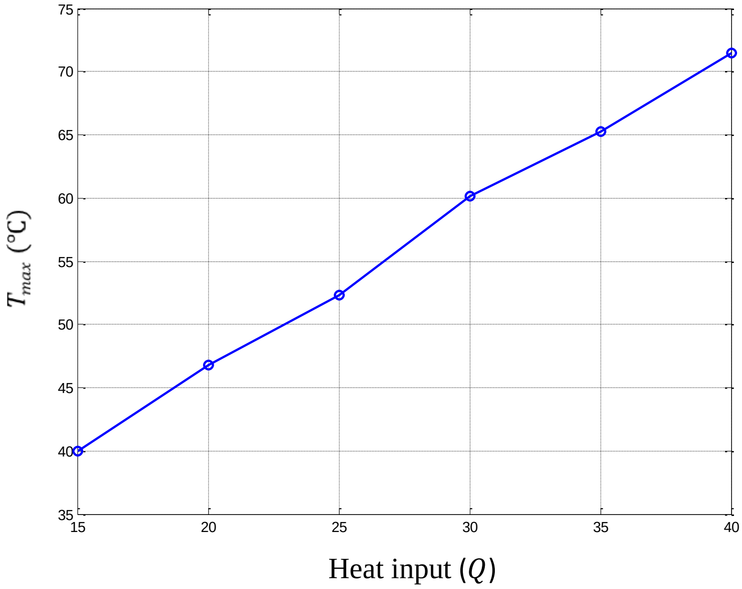
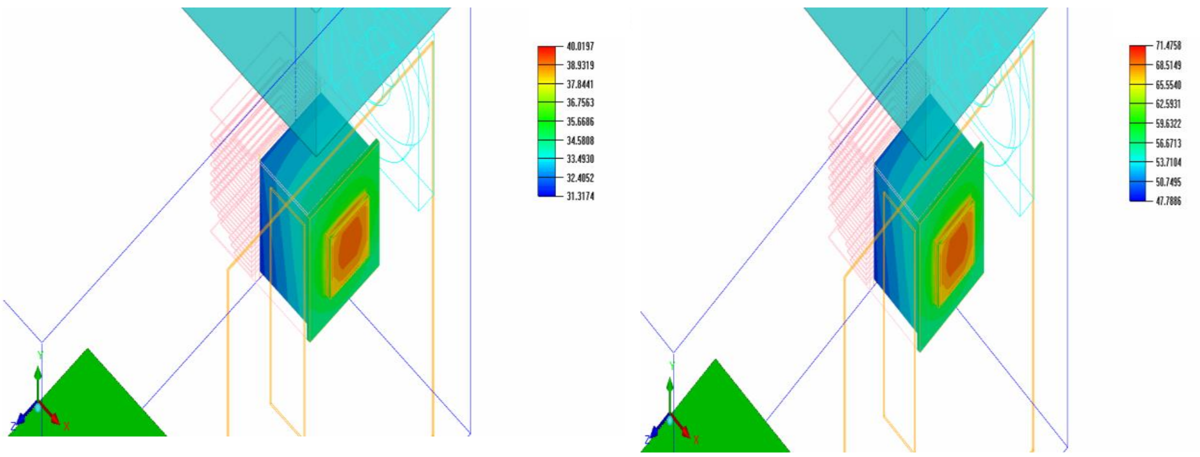


Figure 3. The variation of maximum temperature with different values of heat flux.

The increase in the thermal boundary layer is due to the reduction in the convection coefficient and the increase in the local bulk fluid temperature. While for the negative temperature gradient the thermal boundary layer start to decrease. The effects of the enclosure walls thickness are clearly shown at the substrate interior corners having steeper gradients. This is again due to the high thermal conductivity for both enclosure and substrate.



(a) (b)

Figure 4. Temperature distribution within enclosure, substrate and heater for (a) 15W and (b) 40W.

The different curves denote the change of temperature for various heat input levels. It is clearly shown that as the heat input increases the temperature level also increases.

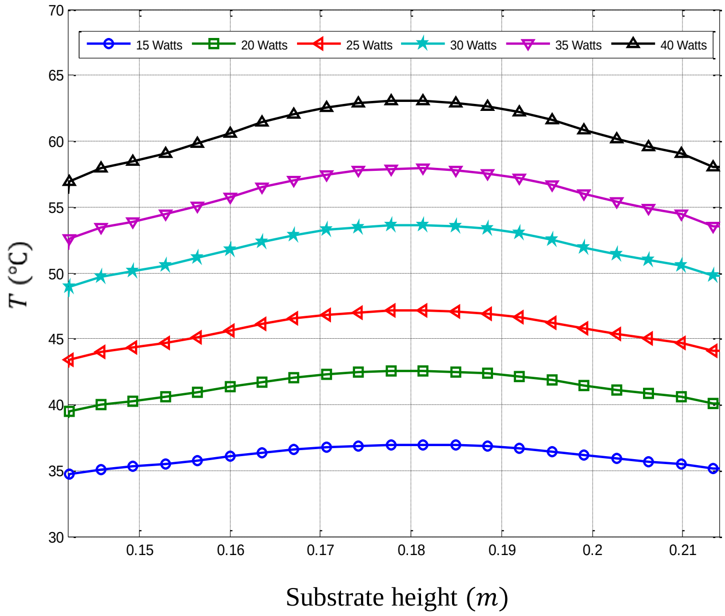


Figure 5. The local substrate/water interface temperature with different applied heat input.

5.2.1 Average Nusselt number. The Nusselt number illustrates the ratio between the convection and conduction heat transfer, and hence a larger Nusselt number corresponds to a more active convection. In this study the average Nusselt number is calculated along substrate/water interface.

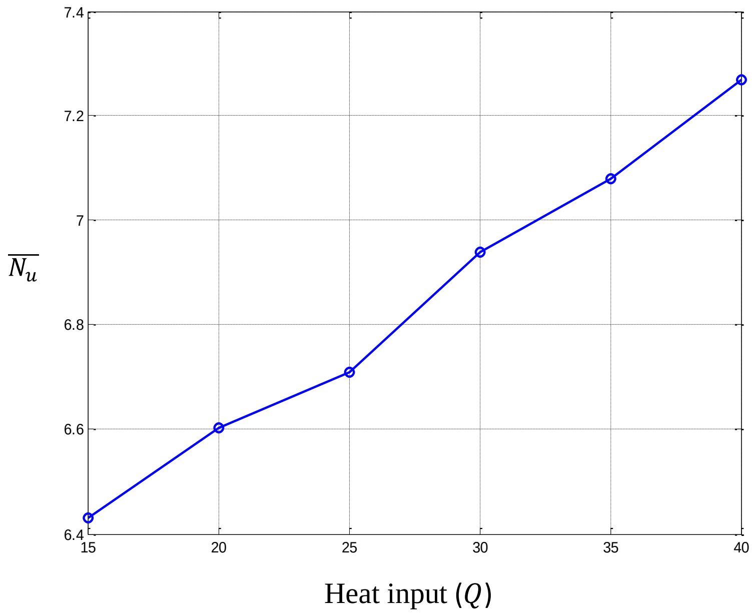


Figure 6. Average Nusselt number with different values of heat input

Figure 6 illustrates the variation of mean Nusselt number for different values of heat input. It is evident from the figure that, the average Nusselt number increases as the heat input increases due to the increase in buoyancy driven flow inside the enclosure. As a result the average heat transfer coefficient increases and the thermal boundary layer decreases.

6. Conclusion

Steady state 3-D natural conjugate convection analysis in a rectangular enclosure and forced convection within a computer covering case has been conducted numerically. One can conclude from the results of this research the following:

- The size of the enclosure has an effect on the value of the maximum temperature. However, it was also found that the temperature variation remains constant beyond certain enclosure size.
- The temperature at substrate/water interface in the place corresponding to the heater remains constant due to the high thermal conductivity of the heater and the substrate.
- The values of mean Nusselt number rises as the heat input increase from $15W$ to $40W$. That is due to the thermal boundary layer being thin and also the average convection coefficient increased.
- It was found that the addition of an enclosure with water had a greater effect on the value of the maximum temperature when compared to the case without enclosure. A reduction of 38% of maximum temperature was reached when the enclosure with water was added.
- The introduction of the water system reduced the temperature to 71.5℃ which is much below the recommended manufacture temperature for the desktop computers.

7. References

[1] Ridouane EH, Campo A and Chang JY 2005 Natural convection patterns in right-angled triangular cavities with heated vertical sides and cooled hypotenuses *J. Heat Transfer*.**127**1181–1186.

[2] Bar-Cohen A1991Thermal management of electronic components with dielectric liquids*Proceedings of the ASME/JSME Thermal Engineering Joint conference*.**2**xv-xxxix.

[3]KelleherMD, Hnock R H and Yang K T1987 Laminar natural convection in a rectangular enclosure due to a heated protrusion on one vertical wall, Part 1: Experimental investigation *Proceedings of the Second ASME/JSME Thermal Engineering Joint Conference*. **2**169-177.

[4] Hasnaoui M, Bilgen E and Vasseur P1992 Natural convection heat transfer in rectangular cavities heated from below*J. Thermophysical Heat Transfer*.**6** 255-264.

[5] Valencia A and Frederick R1989 Heat Transfer in square cavities with partially active vertical walls*Int. J. Heat and Mass Transfer*.**32**1567-1574.

[6] Selamet E, Arpaci V S and Borgnakke C1992 Simulation of laminar buoyancy driven flows in an enclosure*Numerical Heat Transfer*.**22**. 401-420.

[7] Sathe S B and Joshi Y1991 Natural convection arising from a heat generating substrate-mounted protrusion in a liquid-filled two-dimensional enclosure*Int. J. Heat Mass Transfer*. 342149-2163.

[8] Heindel T. J, Ramadhyani S and Incropera F P 1995Conjugate natural convection from an array of discrete heat sources: part 1 two- and three-dimensional model validation*Int. J. Heat and Fluid Flow*.**16** 501-510,

[9] Heindel T J, Incropera F P and RamadhyaniS1996 Enhancement of natural convection heat transfer from an array of discrete heat sources*Int. J Heat and Mass Transfer*.**39**479- 490.

[10] Bhowmik H, Tso C P, Tou K W and Tan F L2005 Convection heat transfer from discrete heat sources in a liquid cooled rectangular channel*Applied thermal engineering*.**25** 2532-2542.

[11] Bhowmik H and Tou K W2005 Experimental study of transient natural convection heat transfer from simulated electronic chips*Experimental Thermal and Fluid Science*.**29** 485- 492.

[12] Keyhani M, Chen L and Pitts D R1991 The aspect ratio effect on natural convection in an enclosure with discrete heat sources*J. Heat Transfer*.**113**883– 891.

[13] Ju Y Land Chen Z Q1996 Numerical simulation of natural convection in an enclosure with

discrete protruding heaters*Numer. Heat Transfer*.**30** 207–218.
[14] Linton R and Agonafer D1995 Coarse and Detailed CFD Modeling of a Finned Heat Sink*IEEE transactions on COMFQNENTS, packaging and manufacturing technology- part A*.**18**517-520.
[15] Miksa deSorgo 1996 Thermal Interface Materials *Electronics Cooling Magazine* September, 1, No 3.
[16] RamaswamyC, Joshi Y, Nakayama W and Johnson W2002 High-speed visualization of boiling from an enhanced structure*Int. J. Heat and Mass Transfer*.**45**4761- 4771.