NUMERICAL ANALYSIS OF HEAT DISSIPATION OF RECTANGULAR HEAT SINK UNDER NATURAL CONVECTION WITH ORIENTATION EFFECTS

Prashant Rathod[1], Hanoca.P.[2]

[1]*M.Tech Student, Thermal Power Engineering, Rajeev Institute of Technology, Hassan, (India)*

[2]*Assistant Professor, Mechanical Engineering, Rajeev Institute of Technology, Hassan, (India)*

ABSTRACT

Whenever electric current flow through a resistive element say electric chip, heat is generated under natural convection conditions. Electric chips are facing thermal challenges to remove the heat. Effectively that causes rising temperature and failure. For better performance and condition operation additional care must be taken in to account for chip cooling system. One of the effective ways is attaching a heat sink over the chips which removes the heat more effectively and keep the chip at lesser temperature. Heat sink performance will vary with respect to the orientation. Recent study shows that the denser fin arrays or more sensitive to orientations. So in this study similar fin arrays with 0^o,45^o, 90^o, orientation are consider for the numerical simulation under certain condition fins with slots can work better than the plane .different slots shape studied to find the better one of the rectangular fin heat sink.

Keywords: Natural Convection, Cooling, Electronic Component, Numerical Simulation

I. INTRODUCTION

All electronic equipment relies on the flow of and control of electrical current to perform a variety of functions. Whenever electrical current flows through a resistive element heat is generated. Regarding the appropriate operation of the electronics heat dissipation is one of the most critical aspects to be considered when designing an electronic enclosure. Heat generation is an irreversible process and heat must be removed in order to maintain the continuous operation. Pure conduction, natural convection or radiation cool the components to some extend where as today electronic devices need more powerful and complicated systems to cope with heat. Therefore new heat sinks with larger extended surfaces highly conductive materials and more coolant flow are keys to reduce the hot spots. The current study is for optimizing the electronic cooling by using CFD. The objective of the present study is to obtain a consistent set of correlations for all orientations of plate-fin heat sinks. At the end of the study our efforts converge to a single correlation covering a wide range of angles between vertical and horizontal and inclined orientations of the heat sink. In the year 2001, Chang, J e [1] Conducted CFD analysis of a 30-W socketed CPU of a desktop computer with minimum air flow rate and minimum heat sink size. They this using only the fan in the power supplies for all air movement in the chassis. A duct was employed to direct the air flow over the CPU and then to the inlet air vents of the power supply.

They allowed the use of this duct more than 10°C reduction of the CPU case temperature relative to an unducted design. The CFD analysis results were confirmed by experiment and the predicted CPU case temperatures agreed within 62.9°C of the experimental values for the ducted cases. Here they have described the methodology of CFD analysis for the heat sink design and described experimental procedures to validate the predictions. Dhiman [2] worked on the flow and heat transfer characteristics of an isolated square cylinder in cross flow placed symmetrically in a planar slit for a range of conditions. They obtained the heat transfer correlations in the steady flow regime for the constant temperature and constant heat flux boundary conditions on the solid square cylinder in cross flow. In addition, variation of the local Nusselt number on each face of the obstacle and representative isotherm plots are presented to elucidate the role of Prandtl number and blockage ratio on drag coefficient and heat transfer.

II. CLASSIFICATION OF COOLING TECHNIQUES

In general thermal management is categorized into active cooling techniques and passive cooling techniques. Mechanically assisted cooling sub systems provide active cooling. Active cooling technique offer high cooling capacity. They allow temperature control that can cool below ambient temperatures. In most cases active cooling techniques eliminate the use of cooling fans or they require less cooling. Air or liquid jet impingement forced liquid convection, spray cooling thermoelectric coolers and refrigeration systems are the examples of active cooling techniques. The passive cooling sub systems are not assisted by mechanical equipments. The conventional passive cooling techniques include applying effective heat spreaders and heat sinks to the electronic package. For a module with spatial limitation, passive cooling technique is often more practical than active cooling. But it is limited to what it can achieve. Therefore recent technologies include the use of thermal energy storage with phase change materials and integration of the heat pipes to the electronic packages that are commonly used to achieve high cooling capacity.

III. OBJECTIVE AND SCOPE OF THE WORK

1. To increase the convective heat transfer rate from the solid surface to the surrounding thus cooling the system.
2. To compare the total heat transfer rate of different orientation of rectangular fin under natural convection.
3. To compare the total heat transfer rate of rectangular fin to the experimental data.

The objective of the present study is to obtain a consistent set of correlations for all orientations of plate-fin heat sinks including the vertical. At the end of the study our efforts converge to a single correlation covering a wide range of angles between vertical and horizontal and inclined orientations.

IV. CFD ANALYSIS PROCESS

To perform a CFD analysis, the analyst will state the problem and use scientific Knowledge to express it mathematically. Then the CFD software package will embody this knowledge and expresses the stated problem in scientific terms. Finally the computer will perform the calculations dictated by CFD software and the analyst will inspect and interpret their results. In principle, three different major tasks should be done to perform a CFD simulation.

I. Problem identification
   1. Define the modeling goals.
   2. Identify the domain will model.

II. Pre-processing
   1. Create solid model geometry to represent the domain.
   2. Create the mesh (grid).
   3. Set up the physics (physical models, materials &domain properties).
   4. Set up the reference, initial &boundary conditions.
   5. Define solver settings (numerical schemes, convergence controls).

III. Solver execution
   1. Compute and monitor the solution until convergence.

IV. Post-processing
   1. Examine the results with qualitative contour plot or quantitative XY plots.
   2. Consider revision to the model or a grid independent study.

V. CFD ANALYSIS OF THE BASE MODEL

5.1 Specification of the Base Model

The main specification of the heat sink is presented in shown in the Fig. 1. In this aluminium and air parameter are presented. The baseboard of the aluminium size is length and width is 123x157mm. The fin thickness is 2mm. The fin height is 50mm. The aluminium base thickness is 10mm and the fin spacing is 13mm. The fluid width and length size is 246x314mm. 11 aluminium fins are created.

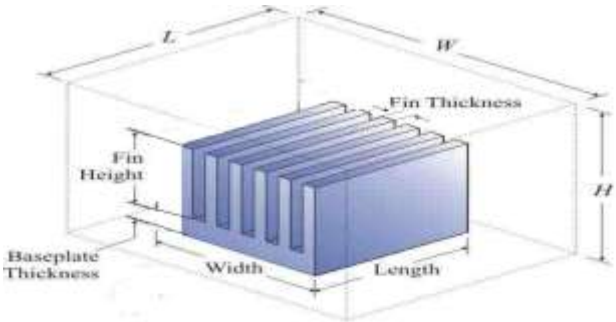


Fig. 1 Symmetry View of the Base Model

Table 1 Properties of Material

Sl. no.	Property	aluminium
1	Thermal conductivity W/Mk	202.4
2	Density [ρ] kg/m^3	2719
3	Specific heat [C_P] J/kg K	871
4	Emissivity	0.6

Table 2 Properties of fluid

Sl. no.	Property	Air
1	Thermal conductivity W/Mk	0.0242
2	Density [ρ] kg/m^3	Incompressible ideal gas
3	Specific heat [C_P] J/kg K	1006.43
4	Viscosity [v] kg/m-s	1.7894e^{-05}

5.2 Mathematical Equations

With the boundary layer approximations the governing equations for convection are as follows.

Continuity Equation:

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0 \quad (1)$$

Momentum Equation:

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = g(\rho_\infty - \rho) + \frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right) \quad (2)$$

Energy Equation:

$$\rho c_p\left(u\frac{\partial t}{\partial x} + v\frac{\partial t}{\partial y}\right) = \frac{\partial}{\partial y}\left(k\frac{\partial t}{\partial y}\right) \quad (3)$$

Rayleigh Number:

$$Ra = \frac{g\beta\Delta t L^3}{\alpha\gamma} \quad (4)$$

In determining the heat transfer, aspect ratio (AR) is the most important parameter affecting the heat and fluid flow.

The heat transfer rate inside enclosure is defined as follow:

$$Q = h A_s (T_{wall} - T_f) = m c_p \Delta t \quad (5)$$

Where the convective heat transfer coefficient, h:

$$h = \frac{Nuk}{L} \quad (6)$$

where,

h = Convective heat transfer coefficient (w/m$^{2\circ}$C).

Nu = The Nusselt number.

K = The air thermal conductivity. (w/m°C).

L = The characteristic length of the channel (m).

Prandtl Number:

$$Pr = \frac{v}{\alpha} = \mu c_p k \quad (7)$$

Prandtl number is a dimensionless number used to describe the relative thickness of velocity and thermal boundary layers in natural convection.

5.3 Boundary Condition

The boundary condition is set as pressure inlet is taken atmospheric condition. Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of the specified static pressure is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure will no longer be used and pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior. A set of "backflow" conditions is also specified should the flow reverse direction at the pressure outlet boundary during the solution process. Convergence difficulties will be minimized if you specify realistic values for the backflow quantities. The blue colure indicates the pressure inlet and yellow colure indicates the symmetry wall and red colure indicates the pressure outlet and black colure indicates the heat sink. The boundary condition as shown in Fig. 2.

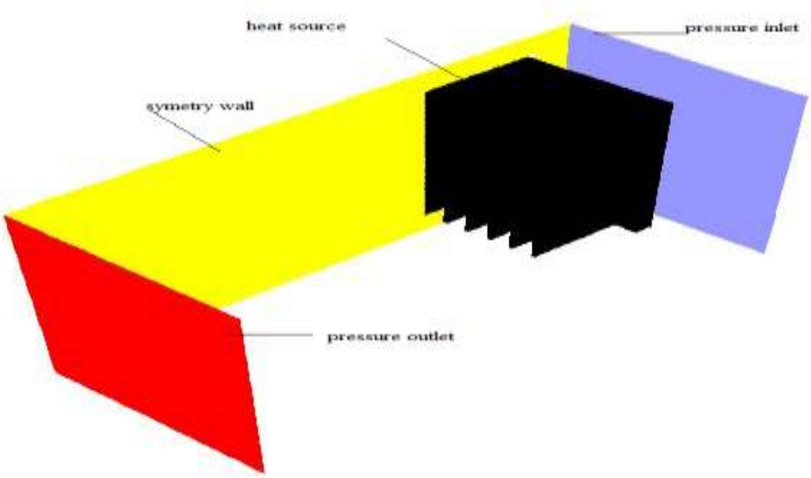


Fig. 2 Boundary Condition

VI. OPTIMIZATION OF THE BASE MODEL

The rectangular fin heat sink placed under an atmospheric condition in a laminar flow different orientation like as horizontal, inclined and vertical to reduce the fin temperature applying the input as 20w.Optimizing the model to find the lesser temperature in different orientation as shown in the Fig. 3.

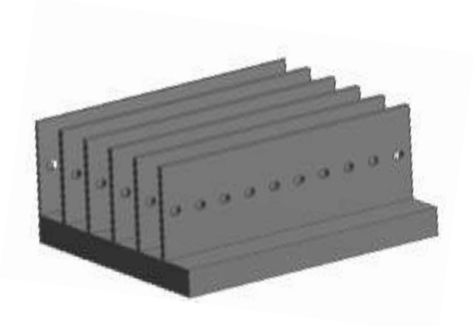


Fig. (a) With circular slot

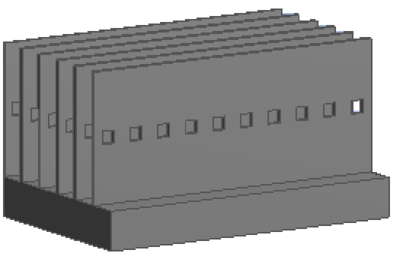


Fig. (b) With square slot

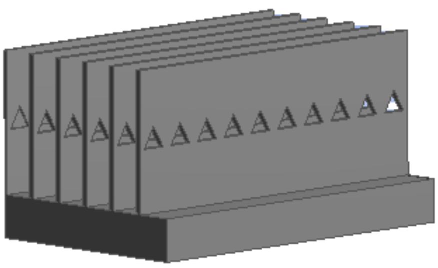


Fig. (c) With triangle slot

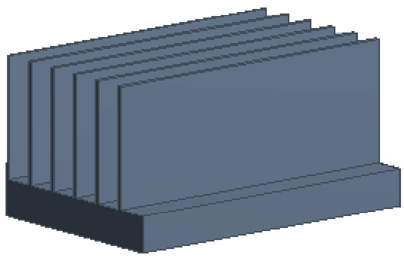


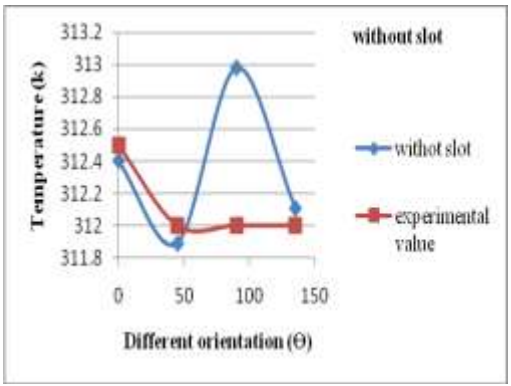
Fig. (d) Without slot

Fig. 3 Optimization of the Different Model

Table 3 Optimization Model Dimensions

Geometry	dimensions	Horizontal dimensions	Vertical dimensions	Offset dimensions
Circular slot	5mm	6.15	25	12.5
Square slot	5mm	6.15	25	12.5
Triangle slot	5mm	6.15	25	12.5

VII. RESULT AND DISCUSSION



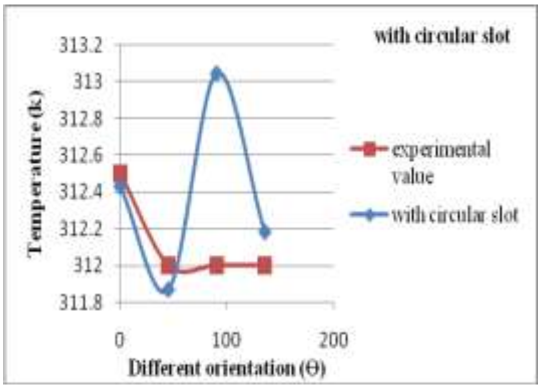
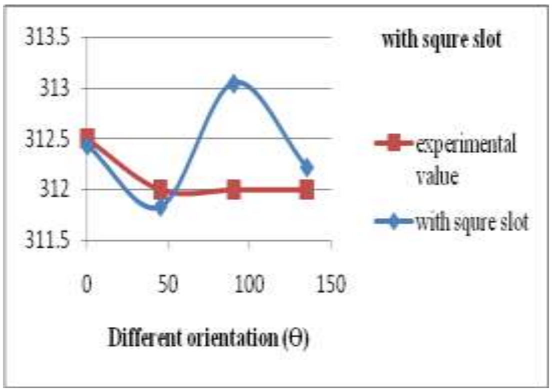


Fig. (a) **Fig. (b)**



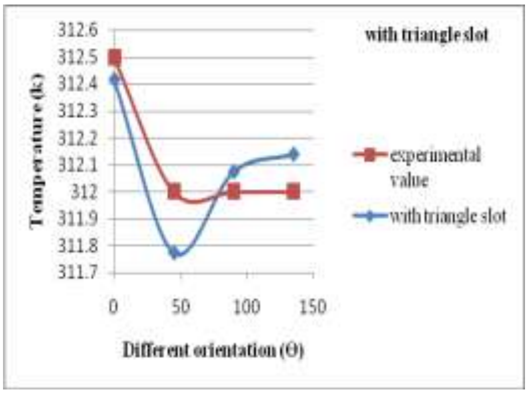


Fig. (c) **Fig. (d)**

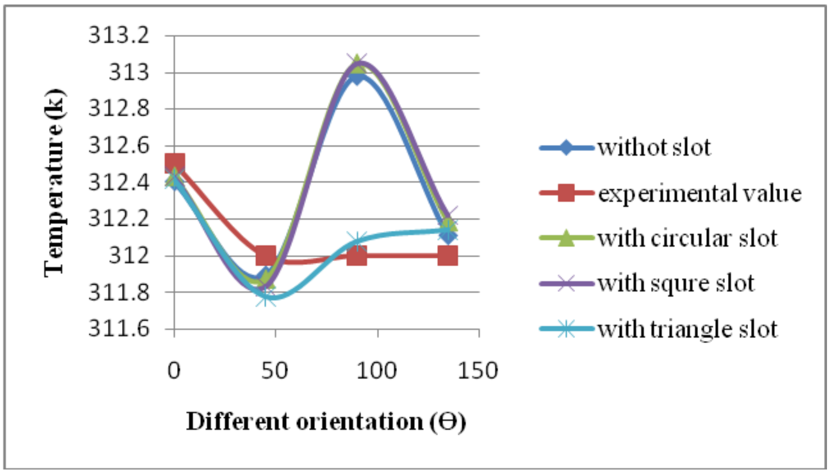


Fig. (e)

Fig. 4 Excess Temperature of Rectangular Fin in Different Orientation of the Model

Table 4 Numerical Analysis of the Temperature Result

Geometry	Number of fins	Heat input (w)	Lowest maximum temperature experimental value(k)	Lowest maximum temperature numerical model value(k)
Without slot	11	20 w	312.54 k	311.8913 k
With circular slot	11	20 w	312.51 k	311.8679 k
With square slot	11	20 w	312 k	311.8277 k
With triangle slot	11	20 w	312 k	311. 7735 k

The orientation of rectangular fin heat sink with different slot studied compare with experimental value there is accept triangle slot all the slot rising temperature obtained. But in the triangle slot maintained less temperature for the chip then the remaining slot as shown in Fig. 4.

7.1 Flow Field Analysis

The heat dissipation performance of a rectangular heat sink is directly related to the flow patterns induced by thermal buoyancy. The flow field using 11 fin heat sink under 20 w load the flow cross section using y-z direction at different orientation slot finally the triangular slot maintaining lesser temperature in the remaining slot as shown in Fig. 5 at different contour temperature.

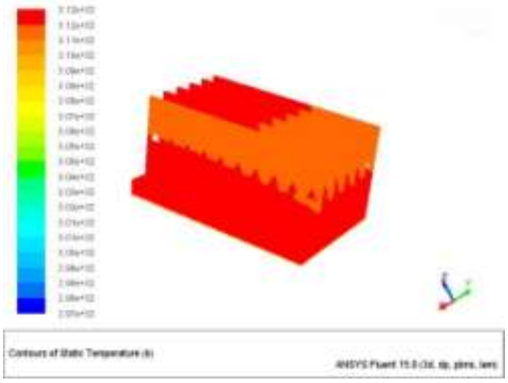


Fig. (a) With Slot Triangle Inclined

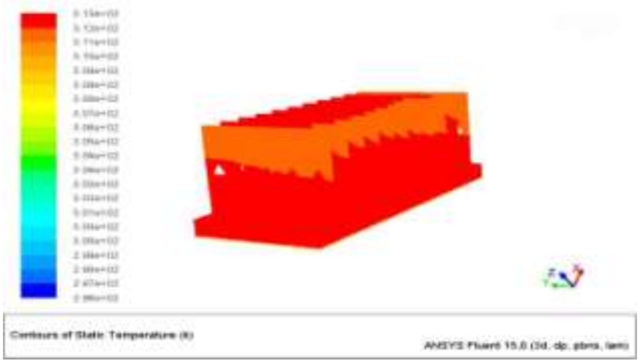


Fig. (b) With Slot Triangle135^{o}

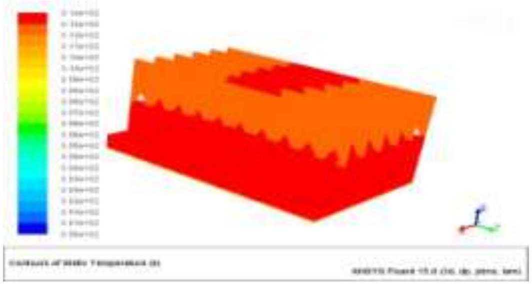


Fig. (c) With Slot Triangle Horizontal

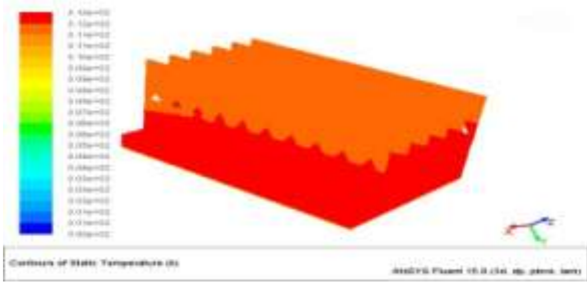


Fig. (d) With Slot Vertical

Fig. 5 Different Contour Temperature of Triangle Slot

VIII. CONCLUSION

Fins with and without slot are studied at different orientation to find the rising temperature of the chip. From the simulation result fin with slot is working similar and even better under some orientations, when compare with plane fin. Different slot shape study shows that the fin wth triangle slot is maintaining less rising temperatures for the chip then the remaining slots.

8.1 Future Work

Present study shows that rectangular slot is working better than the remaining slots. In future study the slots at different location can be studies for the furthers operation.

IX. ACKNOWLEDGMENTS

Apart from the efforts of me, the success of this work depends largely on the encouragement and guidelines of many. I take this opportunity to express my gratitude to the people who have been instrumental in the successful completion of this work. I express immense gratitude to my guide **Mr. Hanoca P,** Department of Mechanical Engg., Rajeev Institute of Technology, Hassan for having extended his priceless technical expertise. I also thank him for his patient thoughtful guidance and also for his most generous support and encouragement towards the development of the work. I express my sincere thanks to **Dr. D A Dhananjaya**, Vice Principal & Professor & Head, Department of Mechanical Engg., Rajeev Institute of Technology, Hassan for his valuable encouragement and support in carrying out the work by providing all the facilities

Nomenclature

A	Surface area, m^2
H	Fin height, mm
h	Convective heat transfer coefficient, w m^{-2} k^{-1}
K	Thermal conductivity, w m^{-2} k^{-1}
L	Fin length, mm
W	Width of the fin array, mm
t_b	Base thickness
t_f	Fin thickness
p	Pressure, pa
Q	Rate of heat transfer, w
s	Fin spacing, mm
ΔT	Temperature difference, k
T	Temperature, k

Greek Symbols

α	Thermal diffusivity
β	Coefficient of thermal expansion of air, k^{-1}
Cp	Heat capacity, J kg^{-1} k^{-1}
ε	Emissivity, dimensionless
θ	Angle defined
ϑ	Kinematic viscosity, m^2 s^{-1}
μ	Dynamic viscosity, pa s
ρ	Density, kg $^{m-3}$
φ	Finning factor, dimensionless
σ	Stefan Boltzmann constant, 5.67x 10^{-8} $wm^{-2}k^{-4}$

Subscripts

CFD	Computational fluid dynamics
CPU	Central processing unit

REFERENCES

[1] Abdullatif Ben-Nakhi, Ali J. Chamkha, Conjugate natural convection in a square enclosure with inclined thin fin of arbitrary length, International Journal of Thermal Sciences 46. Pp. 467–478, 2007.

[2] B.E. Launder, B.I. Sharma, Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc, Letters in Heat and Mass Transfer 1.pp.131–138, 1974.

[3] B. Yazicioglu, H. Yuncu, Optimum fin spacing of rectangular fins on a vertical base in free convection heat transfer, Heat Mass Transfer 44.pp. 11–21,2007.

[4] B. Yazicioglu, Performance of rectangular fins on a vertical base in free convection heat transfer, MS thesis, Middle East Technical University, Ankara, 2005.

[5] D.C. Wilcox, Turbulence Modeling for CFD in DCW Industries, La Canada, CA, 2000, p. 314.

[6] D. Kim et al, Thermal optimization of micro channel heat sink with pin fin structures, Proceedings of the IMECE'03, IMECE2003-42180, 2003.

[7] Deepak Gupta, Momin Nausheen, A.D. Dhale , CFD Analysis & Simulation of Pin Fin for Optimum Cooling of Motherboard, IJEDR,Volume 2, Issue 2,2014 2321-9939.

[8] F. Harahap, H. Lesmana, Measurements of heat dissipation from miniaturized vertical rectangular fin arrays under dominant natural convection conditions, Heat Mass Transfer 42.pp. 1025–1036, 2006.

[9] G. Mittelman, A. Dayan, A. Dado-Turjeman, A. Ullmann, Laminar free convection underneath a downward facing inclined hot fin array, Int. J. Heat Mass Transfer 50. pp. 2582–2589, 2007.

[10] H. Jonsson and B. Moshfegh, Modeling of the thermal and hydraulic performance of plate fin, strip fin, and pin fin heat sinks-influence of flow bypass, IEEE Trans. Compon. Packag. Technol., 24. Pp. 117-123, 2000.

[11] Kercher, D. S, Cooling Devices for Thermal Management of Electronics, IEEE Transactions on Components and Packaging Technologies, 26. pp. 359-366, 2003.

[12] Sanjeev D. Suryawanshi, Narayan K. Sane Natural Convection Heat Transfer From Horizontal Rectangular Inverted Notched Fin Arrays Journal of Heat Transfer AUGUST 2009, Vol. 131.

[13] Yongping Chen, Chengbin Zhang, Mingheng Shi, Jiafeng Wu Three-dimensional numerical simulation of heat and fluid flow in noncircular micro channel heat sinks International Communications in Heat and Mass Transfer 36. pp. 917–920, 2009.

[14] S.A. Nada Natural convection heat transfer in horizontal and vertical closed narrow enclosures with heated rectangular finned base plate International Journal of Heat and Mass Transfer 50. pp. 667–679, 2007.

[15] M. Dogan, M. Sivrioglu Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel International Journal of Heat and Mass Transfer 53. pp. 2149–2158, 2010.

[16] Yue-Tzu Yang, Huan-Sen Peng Numerical study of pin-fin heat sink with uniform fin height design International Journal of Heat and Mass Transfer 51.

[17] Ren-Tsung Huang, Wen-Junn Sheu, Chi-Chuan Wang Orientation effect on natural convective performance of square pin fin heat sinks International Journal of Heat and Mass Transfer 51. pp. 2368–2376, 2008.

[18] R.Mohan and Dr.P.Govindarajan. Thermal analysis of CPU with composite pin fin heat sinks, International Journal of Engineering Science and Technology, Vol. 2(9), 2010, 4051-4062.