INTERNATIONAL JOURNAL
FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Thermal Management of an Electronic Board using CFD

Vundrakonda Koushik[1], P. Nithish Reddy[2], B. Shirish[3]

[1]*M tech, Sreenidhi Institute of Science and Technology – Hyderabad*

[2, 3]*Associate Prof., ME Dept, Sreenidhi Institute of Science and Technology – Hyderabad*

Abstract: The rising demand for high performance and multiple functionality in electronic systems and devices continue to be the current great challenges in their thermal management. Thus, the effective removal of heat dissipations and maintaining a safe operating temperature has played an important role in insuring a reliable operation of electronic components. In this paper, a computational domain with thermal components like basechip and transistor placed in its location was considered. A fan was placed near ventilation such that the ambient air was forcely dragged into the enclosure carries the heat from the thermal components. CFD analysis has been conducted to examine the effect of variation in the cooling rate/ heat dissipation inside the components when the fan rotates at 1000 rpm. Investigations were conducted to understand the fluid flow behavior and heat transfer characteristics inside the cabinets. Results were measured at pseudo-steady state i.e., at the end of 1.8 sec, when the heat flux of 2500 w/m^2 input given to the thermal components. The area-weighted average static temperature was found to be around 800 K and 697 K respectively
Keywords: CFD, Forced air cooling, Heat flux, Basechip and Transistor

I. INTRODUCTION

Recent trend towards higher power density and smaller package size in electronic system demands an effective cooling mechanism in order to prevent the electronic components from overheating. Users prefer the computers of higher performance and are therefore more willing to spend reasonably on them. This tendency has motivated the manufactures t employ overdrive technology to improve their products. With this technology, these devices are capable of processing more data within a given period of time and the system performance is therefore regarded as higher. However, this capability is directly related to its heat generation. The larger the amounts of data the system process at a time, the greater the amount of heat it generates. The performance of these devices is directly related to the temperature, therefore it is a crucial issue to maintain the electronics at acceptable temperature levels. Under the condition of multifunction, high clock speed, shrinking package size and higher power dissipations, the heat flux per unit area increased dramatically over the past few years. Besides, the working temperature of the electronic components may exceed the desired temperature level.

For the case of high speed motor, heat transfer analysis and windage losses were determined by using CFD while satisfying at junction temperature [1]. The variation of inlet velocity at inlet to select appropriate fan selection to cool the heatsink was investigated [2]. A numerical analysis was conducted on a novel cooling system for electronics using a combined heat pipe and electronics module [3]. For a case numerical investigation was conducted on mixed convection to determine the heat transfer on IC chips mounted on SMPS board [4]. CFD analysis performed on a novel heat sink and compared with traditional heat sink to determine the heat transfer characteristics in electronic device [5]. By using dielectric fluids for thermal management on electronic component and presented the best available correlations for single phased natural and forced convection including impinging and stream flow are reviewed [6]. In aeronautical industry, electronics cooling is done using CFD analysis in aircraft for positioning the electronic equipment in racks and also associated with ventilation [7]. Heat transfer analysis is also done in telecommunication application for temperature control [8-9]. The application of heat pipes to the electronic equipment which simplifies the entire thermal control unit by reducing weight and pumping equipments [10]. Some investigations are carried for achieving compact, high performance forced liquid cooling of planar integrated circuits and it was found that the convective heat transfer h between the substrate and coolant is the primary impediment for achieving low thermal resistance [11]. Nanofluids are also used as coolant for electronic device which mainly concerns on its thermal conductivity, heat flux and pumping power. It was found that thermal conductivity of water increased by dispersing nanofluid particles such as 14.44% with SiC-water and 9.99% with TiO_2-water at 4% of its volume fraction [12]. In some cases, performance analysis done on two kinds of nanofluids such as Cu-H_2O and CNT- H_2O in a porous medium structured on micro channel heat sinks with two equation model to describe the fluid and heat transfer which

evaluate the thermal resistance performance characteristic [13]. Heat transfer in forced convection through fins was described on a novel structure for cooling silicon chips involving passage of water through fins etched in the back of chips and some formulas for fins and channel dimensions are derived for optimum cooling performance under various conditions [14]. By using CFD Icepack analysis, some thermal designs like appropriate ducting arrangements were made to meet the cooling requirements for chassis and CPU dissipating 313 W and 80 W   and found that 80 W CPU was satisfactorily cooled[15].

II. METHODOLOGY

A. Problem Description

The problem of this study is to provide the effective cooling in the enclosure for basechip and transistor fins such that performance of electronic chips, reliability and life span of the components increased. Selecting the appropriate cooling technique to dissipate the heat, total power dissipation of recently introduced, new generation microprocessors had been increasing rapidly, pushing desktop system cooling technology close to its limits. Present research focused on heat dissipation inside the enclosure when forced convection takes place due to fan rotation. This study mainly concerns on cooling the overall temperature of the cabinet in which thermal energy is developed from the components like basechip and transistor with fin attachment placed inside an enclosure. Simulations performed in this research were based on use of computational fluid dynamics and results obtained in terms of average static temperature on the components were compared against different cases.

B. Computational Domain

The three dimensional fluid flow and heat transfer in an enclosure having thermal components like basechip and transistor and air is used as working fluid for cooling. The basechip of at X= 0 i.e., at the bottom surface, the basechip and transistor was heated at constant heat flux and the walls with convective heat transfer coefficient.
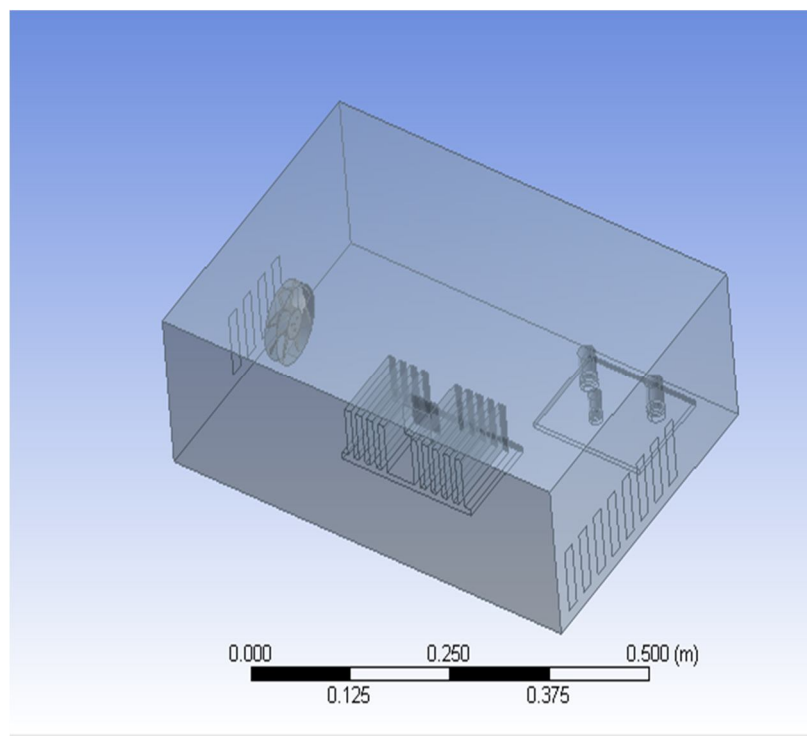

Fig: 1 Geometric model

The following table represents the geometrical parameters of the enclosure and components preset in it. A numerical model was formulated to solve the three dimensional heat transfers inside an enclosure using CFD fluent software package. Using CREO parametric software computational domain has created, and locations of components are placed inside the enclosure according to the dimensions. A fan with casing setup is located near the inlet vents inside the cabinet in such a way the rotating fan intakes the fresh air forcely causes high pressure inside the cabinet. Because the air moves naturally from high pressure to low pressure areas, inside warm air will flows out from the cabinet through exit ventilation.

TABLE 1
Base model specifications

Components	Dimensions (mm)
Enclosure	400*586.75*181.68
Basechip (l*w*h)	195.658*115.569*7.903
Transistor (l*w*h)	160.256*126.356*6.062
Basechip fins (l*h*t)	100.097*59.022*9.139
Distance between fins	7.90
Rotational fan diameter	85.612
Fan shell thickness	17.16
Vent (l*w)	73.652*16.58
Distance between vents	15.78

C. Governing Equations

The governing equations are continuity, momentum and energy equations, which are derived from fundamental principles of heat and fluid flow. The equations are posed to implement SIMPLE (Semi-Implicit Method for Pressure Linked equation) algorithm. Here no-slip assumptions are made.

1) Continuity Equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

2) Momentum Equation (Navier-stokes Equation)

a) X-momentum equation

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

b) Y momentum equation

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

c) Z momentum equation

$$\rho\left(u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right)$$

d) Energy Equation

$$\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right) = \frac{1}{\alpha}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right)$$

D. Computational Grid

First, the fin volumes are discretized, using volume mesh technique. A HEX uniform type of mesh elements used in the grid. Thermal components like basechip and transistor with fins, fine size of elements are used to define the conjugate heat transfer more accurately near the boundary. Such that improving the temperature results along in the z direction comparatively coarse discritisation has been taken because the temperature gradients are small near the walls. The overall mesh of the geometry consists of 1326451 elements of which the skewness of the worst element is 0.900000. After creating mesh, it is necessary to assign the name ports for different parts of the model, such that it was helpful to define boundary conditions. Basechip, transistor, outlets and external wall names are given to different components in the computational domain
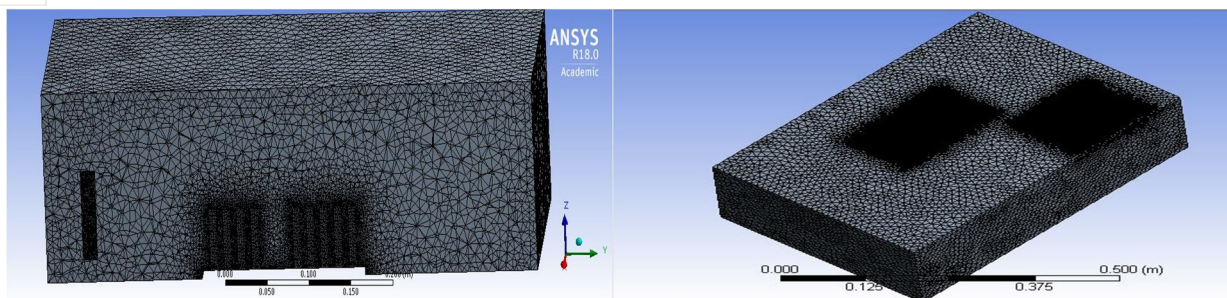
Fig: 2 meshing

E. Boundary Conditions

In CFD analysis, it is necessary to define the boundary conditions to operate the system. Boundary conditions like thermal input like heat flux, temperature and convective heat transfer momentum like velocity and hydraulic diameter like parameters are given as input. In the present work, a turbulent model, of k-omega, SST model was taken to execute the analysis. Thermal boundary conditions like heat flux of 2500 w/m^2 given as input to basechip and transistor and for external wall convective hat transfer coefficient of 25 w/m^2 k is given as input. The ambient air with 300 [K] is passed through the vents while the fan rotates at 1000rpm. Operating conditions like density of air 1.225 kg/m^3 with operating temperature of 288.16 [K] is and also gravity is taken into consideration. The aluminum heat sink is considered for the analysis. It has a thermal conductivity of 205W/m K. Its ease of fabrication and light weight make it excellent material for forming long individual fins.

F. Solution Methodology

FLUENT's segregated solver has taken, which is an implicit solver. The unknown value for a given variable is computed from a set of linear equations, each of which is written for a single cell in the domain. Relaxation factors have direct impact on convergence. Generally default values are used, but if convergence problems occur, then these values are modified. Decreasing these factors gradually helps in convergence.

For the present simulation the finite difference method and the SIMPLE algorithm are applied to solve the governing differential equations. The second order pressure equation, with first order upwind for turbulent kinetic energy is taken. Second order upwind energy equation is taken to solve the transient problem.

III. RESULTS AND DISCUSSION

As soon as cool air enters into the cabinet, the overall thermal energy decreases. These are the results for transient analysis measured at pseudo steady state ie., at the end of 1.8 sec. In this case the temperatures are found to be high for fraction of seconds which can resists by the alluminium material as the turbulence of air occupies the space in the cabinet rapidly. Figure 3 represents the pattern of air flow inside the cabinet. The outside air was sucked by the rotating fan at 1000 RPM into the cabinet. The center area is cooler than the edges because the flow is impenging the center area first before the flow turns and exit from the chamber. It also shows the recirculation caused by the shape and size of the cabinet as they are frame dependant and also the rotational speed of the fan. Air flow is more rapid around the transistor fins compared to basechip fins. The position of air particle was determined by velocity vector plot in figure 4.
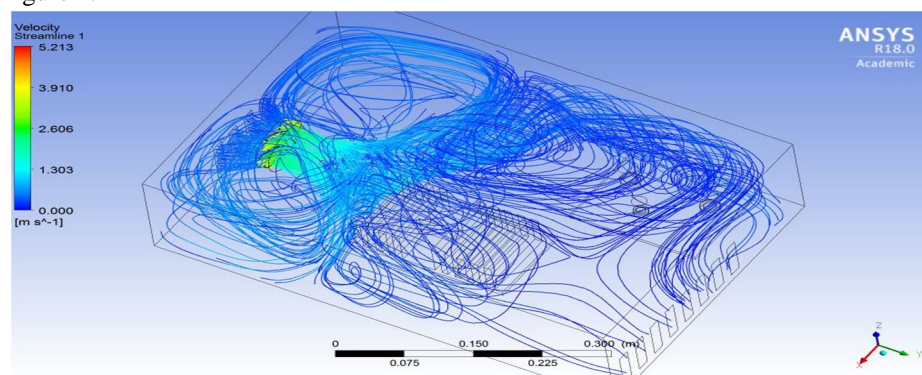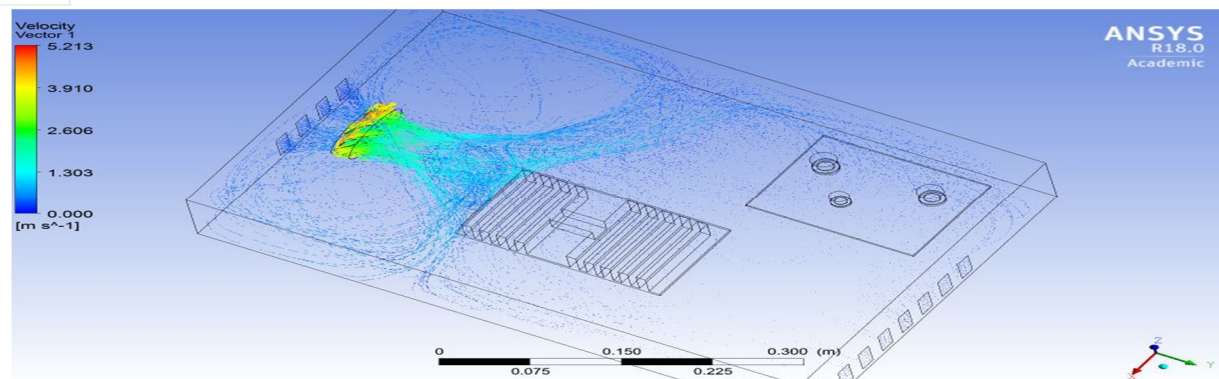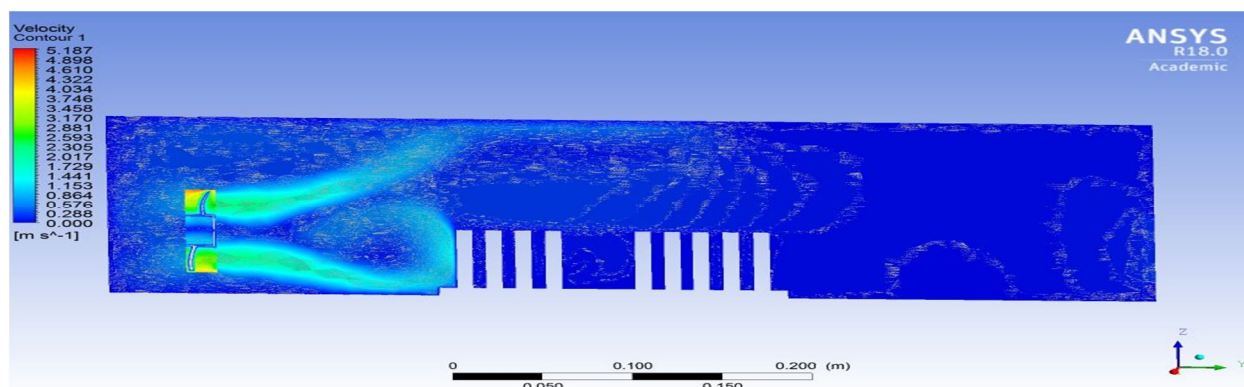

Fig:3 Stream lines

Fig:4 Velocity vector lines

Fig:5 Velocity contour over a plane

Figure 5 shows the velocity contour which determines the air velocity developed inside the cabinet. Air velocity is the function of air density but determing air flow path requires the geometry of the cabinet and fan rotational speed. The air forcely enters into the cabinet and carries the heat from the base plate & transistor fins and leave the cabinet from the vents. However, the velocity of air is not uniform at all points along the cross sectional area of the cabinet. This is because friction between moving air and inside surface of the cabinet and also obstacles present in its path of the air flow.

The temperture distribution for the whole cabinet are shown in the figure 6. It seems that the maximum temperature reached inside the cabinet was 1100.736[K]. The basechip fins are placed perpendicular to fan axis rotation such that red hot spots are formed due to lack of air ciculation in particular areas. Heat dissipation is higher at transistor fins compared to basechip fins. This is due to air flow path inside the cabinet. It was observed that from the streamlines figure air path flow is more rapid around the transistor fins, such that heat dissipation is high. .The area weighted average static temperature at basechip is 803.356 [K] and for transistor is 697.96[K].
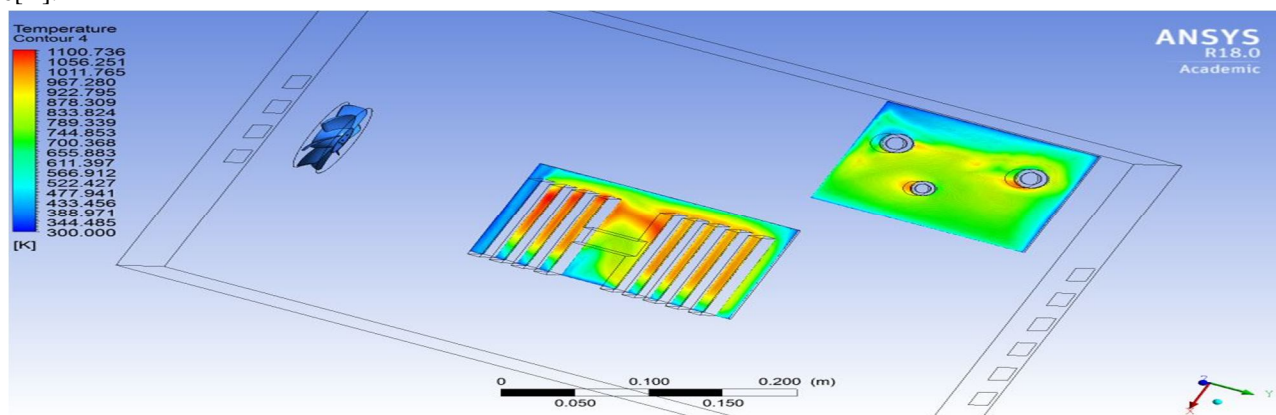

Fig:6 Temperature contour over thermal components

IV. CONCLUSION

CFD analysis were performed, and provided the fundamental understanding of the combined flow and heat dissipation due to forced convection in three dimensional heat transfers inside the enclosure. Since the results are considered at pseudo steady state condition, the cooling rate was improved when it reaches steady state. The cooling rate depends on position of base chip and transistor. Heat dissipation is high at transistor compared to basechip since it placed in centre. This is due to geometry of fins attachment to the basechip and transistor plate. High temperature recorded for fraction of seconds because of lack of air circulation.

REFERENCE

[1] KR Anderson, J Lin, C McNamara, V Magri (2015)"CFD Study of Forced Air Cooling and Windage Losses in a High Speed Electric Motor", Journal of Electronics Cooling and Thermal Control, vol: 5, pp:27-44

[2] S Jeevaraj (2015) "Heat Transfer Analysis of Heat Sink by Computational Fluid Dynamics", International Journal on Emerging Technologies (Special Issue on NCRIET-2015), vol: 6(2), pp: 24-28

[3] B Saengchandr and NV Afzulpurkar (2009) "A Novel Approach for Cooling Electronics Using a Combined Heat Pipe and Thermoelectric Module", American J. of Engineering and Applied Sciences, vol: 2(4), pp: 603-610

[4] VK Mathew and TK Hotta (2018) "Numerical investigation on optical arrangement of IC chips on a SMPS board cooled under mixed convection", Thermal Science and Engineering Progress, vol: 7, pp: 221–229

[5] BR Alvarado, P Li, H Liu and A Hernandez Guerrero (2010) "CFD Analysis of Flow and Heat Transfer in a Novel Heat Sink for Electronic Device", ASME International Mechanical Engineering Congress & Exposition, pp: 1-8

[6] A Bar-Cohen (1993) "Thermal Management of Electronic Components using Dielectric Liquids", JSME International Journal, vol: 36, no: 1, pp: 1-25

[7] F Stancato, LC dos Santos and M Pustelnik (2017) "Electronic Package Cooling Analysis in an Aircraft using CFD", SAE Technical Paper

[8] A Ghiraldi (1988) "Passive Conditioning Systems (PCS) for Temperature Control in Telecommunications Equipment Enclosure", International Telecommunication Energy Conference, pp: 368-373

[9] R Boukhanouf and A Haddad (2009) "A CFD Analysis of an Electronics Cooling Enclosure for Application in Telecommunication Systems" Applied Thermal Engineering, vol: 30(16), pp: 2426-2434

[10] CJ Feldmanis (1972) "Application of heat pipes to electronic equipment cooling", Thermophysics conference, pp: 1-13

[11] DB Tuckerman and RFW Pease (1981) "High-Performance Heat Sinking for VLSI", IEEE Electron Device Letters, VOL: 2, NO: 5, pp: 126-129

[12] A Ijam and R Saidur (2012) "Nanofluid as a Coolant for Electronic Device (Cooling of Electronic Device)", Applied Thermal Engineering, vol: 32, pp: 76-82

[13] T Tsai and R Chein (2007) "Performance Analysis of Nanofluid- Cooled Micro Channel Heat Sinks", International Journal of Heat and Fluid Flow, vol: 28, pp: 1013-1026

[14] RW Keyes (1984) "Heat Transfer in Forced Convection through Fins", IEEE Transactions on Electron Devices, VOL: 31, NO: 9, pp: 1218-1221

[15] CW Yu and RL Webb (2001) "Thermal Design of a Desktop Computer System Using CFD Analysis", Seventeenth IEEE SEMI-THERM Symposium, pp: 18-26

[16] NA Roberts and DG Walker (2010) "Convective Performance of Nanofluids in Commercial Electronics Cooling Systems", Applied Thermal Engineering, vol: 30, pp: 2499-2504

[17] A Sakanova, C Keian and J Zhao (2015) "Performance Improvements of Micro Channel Heat Sink using Wavy channel and Nanofluids", International Journal of Heat and Mass Transfer, vol: 89, pp: 59–74

[18] K NavinRaja, HA Mohammed and CW Lim (2013) "Numerical Investigation of Trapezoidal Grooved Micro channel Heat Sink using Nanofluids", Thermochimica Acta, vol: 573, pp: 39–56

[19] G Colangelo, E Favale, M Milanese, A de Risi and D.Laforgia (2017) "Cooling of Electronic Devices: Nanofluids Contribution", Applied Thermal Engineering, vol: 127, pp: 421–435

[20] M Behnia, W Nayakam and J Wang (1998) "CFD Simulations of Heat Transfer from a Heated Module in an Air Stream: Comparison with Experiments and a Parametric Study", Intersociety Conference on Thermal Phenomena, pp: 143-151

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089   (24*7 Support on Whatsapp)