International Journal of Research in Engineering, Science and Management
Volume-1, Issue-10, October-2018
www.ijresm.com | ISSN (Online): 2581-5792

CFD Analysis on Electronic Heat Sink of Al and Cu Metals

M. Praveen Kumar[1], A. C. Uma Maheshwar Rao[2]
[1]Student, Department of Mechanical Engineering, SIET, Hyderabad, India
[2]Assistant Professor, Department of Mechanical Engineering, SIET, Hyderabad, India

***Abstract*—Overheating generated in the electronic equipment is considered as the major problem and causes raise in the temperature resulting in the permanent damage of the electronic components. For this purpose the electronic cooling system is necessary to dissipate generated heat. The present project is focused on designing a heat sink for electronic printed circuit board with high wattage. In this work various arrangement of the fins and the various materials combination has been analyses through ANSYS workbench package. Transient thermal analysis determines temperatures and other thermal quantities that vary over time. The variation of temperature distribution over time is of interest in many applications such as in cooling. The Copper and Copper- Aluminum based composite were analyzed through the package and observed that the Copper- Aluminum based composite resulted in the high dissipation of the heat ensuring the safety of the electronic circuit board**

***Index Terms*—Electronic Cooling, Heat Dissipation, Thermal Conductivity, Transient, Natural Convection, Forced Convection**

I. Introduction

The cooling of the electronic system has turn into a main problem in operating and designing equipment utilized in various of electronic and electrical systems including aerospace. Integrated circuits (ICs) consist of enormous number of resistance, transistors and these ICs generates large amount of heat which reduces the efficiency of the component. Integrated circuits (ICs) needs rapid cooling for consistent running of the electronic component. Furthermore, while the refinement of manufacturing techniques, the sizes of the computer ICs are dramatically reduced. Thus, the heat flux on the chips is keeping climbing in a startling rate. Moreover, the sizes of the electronic devices are reduced to meet the market's needs. As a result, the available space for cooling system is very limited and the convectional cooling techniques cannot satisfy the increasing needs.

There are three major techniques to improve the efficiency of dissipating the heat from the computer components utilizing the air cooling systems: changing the geometry of the conducting material to increase convection surface area, changing the conducting material to increase the surface temperature and achieve higher convection coefficient, and, lastly, increasing the flow rate of the external fluid to reduce the thermal and hydraulic boundary layers.

Several researchers have worked on conjugate heat transfer at electronic systems via CFD. Yu and Webb [1] simulated a complete desktop computer system which uses an 80 W CPU. With the addition of other components (memory, chipset, AGP, PCI cards, floppy drives) a total of 313 W heat is dissipated into the system. They solved the whole domain with a commercially available software, Icepack. To decrease the complexity of their model they modeled CPU heat sink as a volume resistance having the same impedance with the detailed geometry. They improve the cooling of PCI cards with PCI side vents and baffle.

Biswas et al. [2] also used Icepak to study the airflow in a compact electronic enclosure. Their investigated the pressure loss due to the presence of the inlet and the outlet grilles. Linton and Agonafer [3] compared the results of a detailed CFD modeling of a heat sink with experimental data. They found that the coarse model agrees well with the detailed model without losing the characteristics of the heat sink. Sathyamurthy and Runstadler [4] studied planar and staggered heat sink performance with Fluent. They found that the thermal performance of staggered fin configuration is superior over planar fin configuration. Yu et al. [5] combined pin fins with plane fins to obtain a heat sink design called plate-pin fin heat sink. Eveloy et al. [6] used Flotherm as a design tool to predict component temperature on printed circuit boards. Bar-Cohen and his coworkers [7-8] investigated the cooling effect of forced convection heat sinks with various plane fin or pin fin geometries.

In this examination work, Ansys Thermal analysis has been utilized to recognize the thermal dispersal through natural convection for a personal computer and the CPU will scatter 143 W according to the item manual. The heat sink with base plate is composed by the plan imperatives and set to limit the thermal resistance and increment the execution of the perfromance. In this task the design of fins, length of the fins, material selection and determination of the flow stream and heat transfer inside a PC for the powerful conditions. The motivation behind the present work is to decide the heat transfer appropriation in the heat sink varying with the material. In the present analysis Aluminum, Copper and composite (mix of aluminum and copper) are utilized.

International Journal of Research in Engineering, Science and Management
Volume-1, Issue-10, October-2018
www.ijresm.com | ISSN (Online): 2581-5792

II. Computational Thermal Analysis

Pre-processing is the first step to achieve modeling goals and computational grid is created. In the second step numerical models and boundary conditions are set to start up the solver. Solver is terminated, the results are examined which is the post-processing part.

A. *Design of a Heat Sink*

The Fig. 1 shows the schematic representation of the arrangement for the experimentation. Since the test setup is an open domain, the atmospheric temperature is the temperature of the air blown to the heat sink. The atmospheric air is passed around the heat sink which is heated then is exhausted by blower and also the pressure drop has been noted. The heat transfer can be calculated by eq. (1).

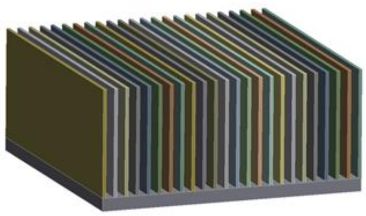

Fig. 1. The schematic representation of the arrangement for the CAD model

$$Q = h * s * (Tp - Ta) \tag{1}$$

Q =heat transferred (J/s = W); h = heat transfer coefficient (W/m^2 K); S = transfer surface(m^2), Tp = Plate temperature(K), Ta = Air temperature(K).

For convection we use the convection heat transfer coefficient hc, W/(m^2 K). A different approach is to define h through the Nusselt number Nu, which is the ratio between the convective and the conductive heat transfer eq. (2):

$$Nu = \frac{Convective\ heat\ transfer}{conductive\ heat\ treansfer} = \frac{(hc*L)}{k} \tag{2}$$

Where Nu = Nusselt number, h_c = convective heat transfer coefficient, k = thermal conductivity(W/mK), L = characteristic length(m). The convection heat transfer coefficient is then defined as following: $hc = \frac{(Nu*k)}{l}$.

The Nusselt number depends on the geometrical shape of the heat sink and on the air flow. For natural convection on flat isothermal plate the formula of Na is given in table 1. Where Ra is the Rayleigh number defined in terms of Prandtl number (Pr) and Grashof number (Gr). If Ra < 106 the heat flow is laminar, while if Ra > 106 the flow is turbulent.

$$Ra = Gr * Pr \tag{3}$$

The Grash of number, Gr is defined as following:

$$Gr = \frac{g*L3*\beta*(Tp-Ta)}{l} \tag{4}$$

TABLE I
Nusselts Correlation for Type of Fins and Flow

	Vertical fins		Horizontal fins
Laminar flow	$Nu = 0.59 * Ra^{0.25}$	Upward laminar flow	$Nu = 0.54 * Ra^{0.25}$
Turbulent flow	$Nu = 0.14 * Ra^{0.33}$	Downward laminar flow	$Nu = 0.27 * Ra^{0.25}$
		Turbulent flow	$Nu = 0.14 * Ra^{0.33}$

Where: g = acceleration of gravity = 9.81(m/s^2), L = 0.130, longer side of the fin (m), β = air thermal expansion coefficient. For gases, is the reciprocal of the temperature in Kelvin: $\beta = 1/Ta$; β = 0.033

Where Tp = 75^{o}C, Plate temperature (°C), Ta = 30^{o}C, Air temperature (°C), η = air kinematic viscosity, 1.6^{-5} at 30 °C.

$$G_r = \frac{9.81*0.130*0.130*0.130*0.033*(75-30)}{1.6^{-5^2}}$$

$G_r = 7.2*10^{-7}$

For plate temperature, Tp, set an expected value. Finally, the Prandtl number, Pr is defined as:

$$Pr = \frac{(\mu*Cp)}{k} \tag{5}$$

Where, μ = air dynamic viscosity, is 1.86^5 at 30 °C, c_p = air specific heat = 1005 J/(Kg*K) for dry air- k = air thermal conductivity = 0.026 W/(m*K) at 30 °C

$$Pr = \frac{1.86^{-5}\ X\ 1005}{0.026} = 0.7189 \tag{6}$$

Convective heat transfer coefficient:

$$hc = \frac{(Nu*k)}{L} \tag{7}$$

$$h_c = \frac{57.59*0.026}{0.130}$$

$h_c = 10.01$ (For vertical walls) and $h_c = 9.16$ (For horizontal walls)

B. *Design of Fins*

Given:
Heat load: 143 W
Base size = 130 X 130 sq.mm
Assumptions:
The ambient temperature of air is T_A = 30^{o}C
Surface Temperature T_F = 75^{o}C
Thickness of Fin = 1.5 mm
Heat transfer coefficient = 10.01 W/m^{2}K
Average temperature (or) film temperature =
$T_F = (T_s + T_A)/2$ (8)
= (75+30)/2 = 52.5 ^{o}C

The properties of dry air at 30^o C from “Heat and mass transfer data book” by “C P Kothandaram” are
Density (ρ) = 1.1652852 kg/m^3
Thermal conductivity (k) = 0.0257 w/mk
Coefficient of viscosity = 18.7 X 10^{-6} m^2/s
Prandtl number (pr) = 0.7189
$\beta = 2/(T_s + T_A) = 0.019047619$ (9)

Design constraint

The surface temperature of components should not exceed 85°C

C. Natural Convection Calculation

Fin spacing: To find optimum fin spacing, it is first necessary to find out Rayleigh number (R_a) ;

It can be calculated using the formula:

$$R_a = \frac{g \times \beta \times (T_s - T_A) \times L^3 \times P_r}{\vartheta^2} \quad (10)$$

Where,

g = Acceleration due to gravity, 9.8 m/s^2

L = 0.130 m, Base length or fin length

$$R_a = \frac{9.18\ X\ 0.019047619\ X\ (45) X\ 0.130 X 0.130 X 0.130 X\ 0.7189}{0.0000186 X 0.0000186}$$

R_a = 51882150.28

$$\text{Optimum Fin Spacing} = S_{opt} = 2.714 \times L/{R_a}^{0.25} \quad (11)$$

S_{opt} = 2.714 x 0.130/51882150.28$^{0.25}$

S_{opt} = 0.004157179 m.

Number of fins (n):

The number of fins is given by

$$N = \frac{Totalbasewidth}{(finthickness\ + S_{opt})} \quad (12)$$

$N = \frac{130}{(1.5+4.1)} = 23$

Fin height:

The heat dissipation through Natural convection is given as

$$Q = h \times A \times (T_s - T_A) \quad (13)$$

Q = Heat dissipated = 143 W

143 = 10.01 X A X (75-30)

A = 0.317 m^2

But A= Total area = Total Fin area + Base area

0.317 = (2 X N X L X H) + (L X W)

= (2 X 23 X 0.130 X H) + (0.130 X 0.130)

H = 50mm.

D. ***Modelling & Finite Element Analysis***

Thermal Analysis of heat sink is performed in Ansys Workbench. Ansys workbench is used for pre and post processing whereas Ansys Mechanical Solver is used for solving. Two Cases of heat sinks have been analysed by varying the material and keeping the geometry intact. i.e. no. of fins and fin spacing.

Case 1: Copper,

Case 2: Aluminium-Copper Composite

1) Geometry

3D CAD models for copper, aluminium and Composite heat sinks are developed in Solidworks software as per the dimensions obtained from the calculations.

2) Analysis Setup

Step 1: Geometry import and Simplification

Geometry in the neutral file format is exported from solidworks software and imported into ansys Design Modeller. Required geometric operations have been carried out to simplify the geometry for improved meshing.

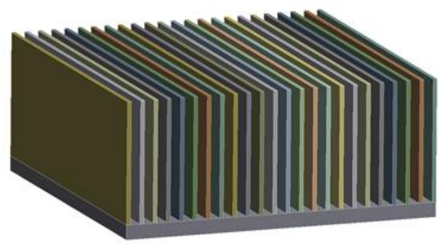

Fig. 2. 3D CAD model of heat sink

Step 2:

The geometry file is now updated in the Mesh model of the Ansys. The suitable materials are assigned to the models and mesh is generated.

Fig. 3. Mesh model

The Boundary conditions calculated are applied on the geometric entities.

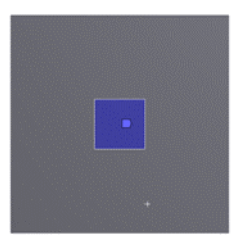

Fig. 4 (a). Heat input 143W

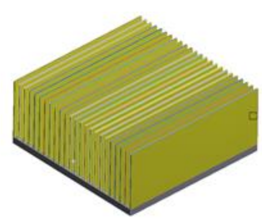

Fig. 4 (b). Convective heat transfer coefficient on vertical surfaces 10.01 W/m^{2}C

International Journal of Research in Engineering, Science and Management
Volume-1, Issue-10, October-2018
www.ijresm.com | ISSN (Online): 2581-5792

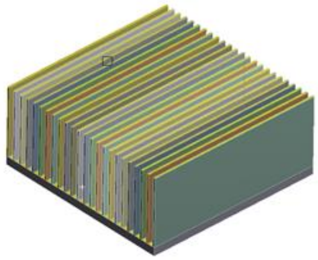

Fig. 4 (c) Convective heat transfer coefficient on horizontal surfaces 9.16 W/m^{2}C
Fig. 4. Mesh control model

The above mentioned boundary conditions are same for copper materials i.e. case1, but whereas for the composite model (aluminium and copper) the meshing and geometry differs. The 0.37mm copper sheet is covered on the 0.75mm aluminium fins and separate mesh sizing is adopted for the proper element quality.

Geometry:

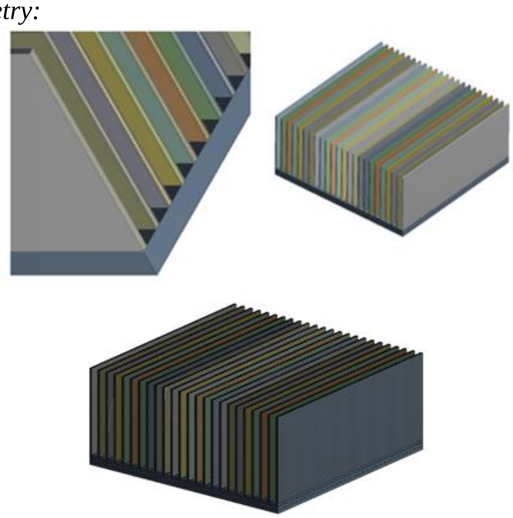

Fig. 5. Geometry and mesh model of composite fins

III. Result

The temperature plots have been observed on both fins and base plate after completion of analysis with 143W on processor along with the natural convection parameters on the fins.

A. Aluminium

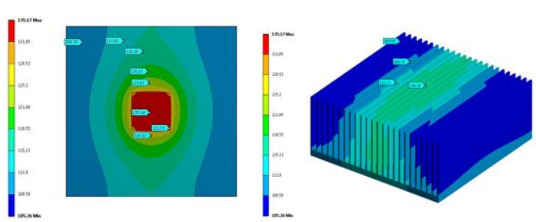

Fig. 6. Temperature on base plate and fins of Aluminum model ^{o}C (Case 1)

The Fig. 6, shows the temperature distribution on the base plate of heat sink with the aluminum as base metal. The maximum temperature of 135.14^{o}C is found on the core and the minimum temperature is observed as 109.78^{o}C on the base plate. The above figure shows the temperature distribution on the fins, maximum temperature 118^{o}C and minimum temperature is at the 115.64^{o}C.

B. Copper

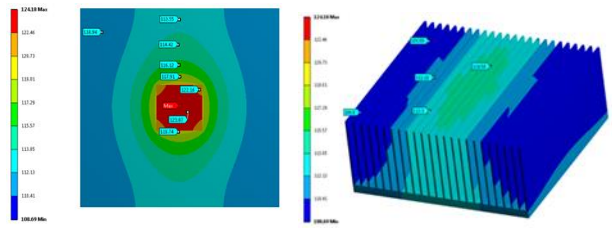

Fig. 7. Temperature distribution on the base plate and fins of Copper (Case 2)

The Fig. 7 shows the temperature distribution on the base plate of heat sink with the copper as base metal. The maximum temperature of 123.87^{o}C is found on the core and the minimum temperature is observed as 110.94^{o}C on the base plate. The temperature distribution on the fins, maximum temperature 114^{o}C is found on the central fins the temperature is decreasing towards the end and minimum temperature is at the 108.8^{o}C.

C. Composite Metal (Copper + Aluminum)

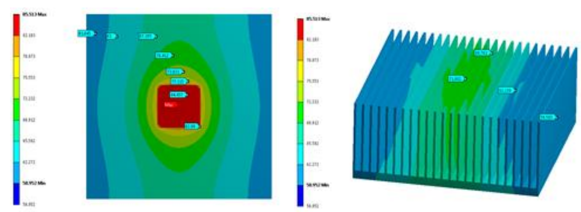

Fig. 8. Temperature distribution on base plate and fins of composite material (Case 3)

The Fig. 8, shows the temperature distribution on the base plate of heat sink with the copper as base metal. The maximum temperature of 84.457^{o}C is found on the core and the minimum temperature is observed as 61.645^{o}C on the base plate. The temperature distribution on the fins, maximum temperature 71.002^{o}C is found on the central fins and minimum temperature is at the 59.593^{o}C.

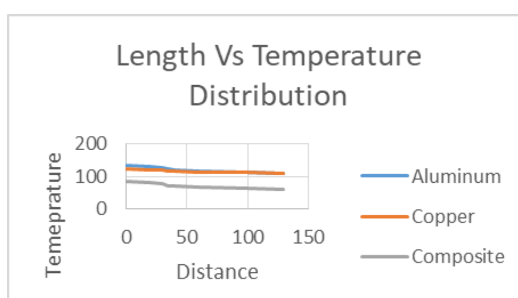

Fig. 9. Graph of Temperature vs. Distance

The above graph describes the temperature distribution for three materials with the same heat source. The temperature distribution on the copper + aluminum composite bonded metal is very far less than compared to that of other two metals and it is meeting the design conditions.

TABLE I
TEMPERATURE VALUES FOR THREE DIFFERENT MODELS

Material	Temperature		Design Temperature	Deviation
	Maximum	Minimum		
Aluminum	135.17	105.26	85	58%
Copper	124.18	108.69	85	46%
Composite (Aluminum + Copper)	85.5	58.95	85	0

IV. CONCLUSION

In the current project the thermal analysis of the fins is carried out by using the Ansys workbench thermal module. AMD Athlon – II processor is selected as the interest of analysis and extended surface geometry parameters (Fins: no. of fins, height, thickness and gap) are calculated. The analysis is under the effect of natural convection and the medium heat transfer coefficient is found. Three materials are considered i.e. Aluminum, Copper and composite (Aluminum and copper fitted together).

The aim of the project is to restrict the temperature not more than 85ºC on the CPU (prescribed by the manufacturer) when is at the peak performance. In case of the aluminum metal the maximum temperature is 135.17ºC which is 58% more than the desired limit. The copper material got heated up to 124.18ºC, 46% more than normal, but in the case of the composite i.e. aluminum and copper binded material it is almost equal to desired temperature 85.5ºC. The temperature vs. distance graph clearly depicts the distribution of the temperature contours on the fins from the center of the core to other end is very much less for composite fins compared to the conventional metals.

As the range of the temperature is in the control in the case of the composite material, it can be selected as the best in the case of natural convection mode where the forced convection is not possible (due to minimum space).

REFERENCES

[1] R. L. Webb, "Next generation devices for electronic cooling with heat rejection to air," *Trans. ASME, J. Heat Transfer*, vol. 127, pp. 2–10, 2005.
[2] C. W. Yu and R. L. Webb, "Thermal design of a desktop computer system using CFD analysis," in *Proc. 17th IEEE Semi-Therm Symp*., 2001, pp. 18–26.
[3] R. Biswas, R. B. Agarwal, A. Goswami, and V. Mansingh, "Evaluation of airflow prediction methods in compact electronic enclosures," in *Proc. 15th IEEE Semi-Therm Symp*., 1999, pp. 48–53.
[4] R. L. Linton and D. Agonefer, "Coarse and detailed CFD modelling of a finned heat sink," *IEEE Trans. Compon., Packag., Manuf. Technol.- Part A*, vol. 18, no. 3, pp. 517–520, Sep. 1995.
[5] P. Sathyamurthy, P. W. Runstadler, and S. Lee, "Numerical and experimental evaluation of planar and staggered heat sinks," in *Proc. IEEE Inter Soc. Conf. Thermal Phenomena*, 1996, pp. 132–139.
[6] X. Yu, J. Feng, Q. Feng, and Q.Wang, "Development of a plate-pin fin heat sink and its performance comparisons with a plate fin heat sink," *Appl. Thermal Eng*., vol. 25, pp. 173–182, 2005.
[7] V. Eveloy, P. Rodgers, and M. S. J. Hashmi, "Numerical prediction of electronic component heat transfer: An industry perspective," in *Proc. 19th IEEE Semi-Therm Symp*., 2003, pp. 14–26.