P.Chennakesavarao Int. Journal of Engineering Research and Applications www.ijera.com
ISSN : 2248-9622, Vol. 4, Issue 10(Part - 5), October 2014, pp.83-94

RESEARCH ARTICLE OPEN ACCESS

A Thesis on Design Optimization of Heat Sink in Power Electronics

P.Chennakesavarao[1] P.Srihari[2]
M.E (Thermal Engg.) student, Aditya Institute of Technology and Management, Tekkali.
Associate.professor, Aditya Institute of Technology and Management, Tekkali.

ABSTRACT

The heat sinks are used in electronic systems to remove heat from the chip and effectively transfer it to the ambient. The heat sink geometry is designed by the mechanical engineers with the primary aim of reducing the thermal resistance of the heat sink for better cooling in the electronic systems. Due to the proximity of the heat sink with the ICs, the RF fields created by RF currents in the ICs/PCBs gets coupled to heat sinks. Hence, the coupled RF current can cause radiated emission. This radiated noise from the device can couple and disturb the functioning of the nearby electronic systems. Also this radiated emission from the device poses a problem to the system compliance with respect to EMI/EMC regulations. The international EMI/EMC standards require the radiated emission from the electronic devices to be kept below the specified limits.

As a result the design of Heat Sink is very important factor for the efficient operation of the electronic equipment. In this project design optimization of a Heat sink in a Power amplifier is performed to reduce the weight and size .Power amplifier is electronic equipment mounted in an army vehicle. The power modules inside the amplifier generates a heat of 1440 Watts and a temperature of 140 0c.Two Heat sinks are used to dissipate the heat generated inside the equipment and maintain a temperature of less than 850c. The existing heat sink which is being used is weighing around 10.3kgs and height of 51mm; as a result the unit is very robust.

The objective of my project is To design & optimize the heat sink to reduce the weight and size. The optimized heat sink should also dissipate heat generated by power modules and maintain a temperature of less than 850c inside. To achieve the design a steady state thermal analysis will be performed on the heat sink and plot the Temperature distribution on the fins. Based on the above analysis results we will increase/decrease the number of fins, thickness of fins, and height of fins to reduce the weight of the heat sink. We will perform CFD analysis of the power amplifier by mounting the optimized heat sink and plot temperature, pressure and velocity distribution in the power amplifier enclosure. Efforts are made to optimize temperature, pressure and velocity distribution in the power amplifier enclosure by reorienting the power modules in the enclosure.

UNIGRAPHICS software is used for 3D modeling SOLID WORKS FLOW SIMULATION software is used for thermal and CFD analysis.

Key words: RF currents: Radio Frequency currents EMI/EMC: Electromagnetic interference/ Electromagnetic compatibility

I. INTRODUCTION

1.1 INTRODUCTION TO HEAT SINK

The heat sink is a very important component in cooling design. It increases the component surface area significantly while usually increasing the heat transfer coefficient as well. Thus, the total resistance from the component junction to the surroundings is reduced significantly, which in turn reduces the junction temperature within a device. As a result, obtaining correct performance characteristics for heat sinks is extremely important in cooling design solution.

1.1.1 HEAT SINK TYPES

Heat sinks can be classified in terms of manufacturing methods and their final form shapes. The most common types of air-cooled heat sinks include:

Stampings: Copper or aluminium sheet metals are stamped into desired shapes. they are used in traditional air cooling of electronic components and offer a low cost solution to low density thermal problems. They are suitable for high volume production, because advanced tooling with high speed stamping would lower costs. Additional labor-saving options, such as taps, clips, and interface materials, can be factory applied to help to reduce the board assembly costs.

Extrusion: These allow the formation of elaborate two-dimensional shapes capable of dissipating large heat loads. They may be cut, machined, and options added. A cross-cutting will produce omni-directional, rectangular pin fin heat sinks, and incorporating serrated fins improves the performance by approximately 10 to 20%, but with a slower extrusion rate. Extrusion limits, such as the fin height-to-gap fin thickness, usually dictate the flexibility in design options. Typical fin height-to-gap aspect ratio of up to 6 and a minimum fin thickness of 1.3mm, are attainable with a standard extrusion. A 10 to 1 aspect ratio and a fin thickness of 0.8″can be achieved with special die design features. However, as the aspect ratio increases, the extrusion tolerance is compromised.

Bonded/Fabricated Fins: Most air cooled heat sinks are convection limited, and the overall thermal performance of an air cooled heat sink can often be improved significantly if more surface area can be exposed to the air stream. These high performance heat sinks utilize thermally conductive aluminium-filled epoxy to bond planar fins onto a grooved extrusion base plate. This process allows for a much greater fin height-to-gap aspect ratio of 20 to40, greatly increasing the cooling capacity without increasing volume requirements.

Castings: Sand, lost core and die casting processes are available with or without vacuum assistance, in aluminium or copper/bronze. This technology is used in high density pin fin heat sinks which provide maximum performance when using impingement cooling.

Folded Fins: Corrugated sheet metal in either aluminium or copper increases surface area and, hence, the volumetric performance. The heat sink is then attached to either a base plate or directly to the heating surface via epoxying or brazing. It is not suitable for high profile heat sinks on account of the availability and fin efficiency. Hence, it allows high performance heat sinks to be fabricated for applications. The objectives of these experiments are to evaluate the performance of heat sinks, determine the effect of flow velocity, and to analyze results by comparing them.

1.2 THEORY:

A heat sink works by conducting heat through its base and fins and then converting it to the surroundings. The equation for thermal resistance of a heat sink is shown in Equation (1)

$$R_{HS} = \frac{T_{base} - T_{air}}{Power} \quad \text{... ... (1)}$$

The thermal resistance can be found from experimental testing, simulation, or manufacturer's data. In this lab, you will be using Equation (1) to find heat sink resistance and comparing results to published correlations.

The heat sink resistance includes both the conduction and convection resistance, although the convection resistance typically dominates. Equation (2) is a somewhat simplified equation for the heat sink resistance. This equation would be completely accurate if conduction and convection were in parallel. However, this is not completely the case for heat sinks, so this equation incorporates some inaccuracies.

$$R_{HS} = \frac{1}{1/R_{\text{conduction}} + 1/R_{\text{convection}}} \quad \text{... ... (2)}$$

Conduction resistance can be lowered by changing the geometry and choosing a material with a higher thermal conductivity. Convection resistance can be lowered by increasing the heat sink surface area or the heat transfer coefficient. The heat transfer coefficient will change with both geometry and flow velocity for a given fluid type.

More commonly, heat sink resistance is found using the equation for fin resistance given below.

$$R_{fin} = \frac{1}{h\eta_f A_{fin}} \quad \text{... ... (3)}$$

Here ηf, the fin efficiency, takes into account the conduction resistance and is equal to the actual fin heat transfer divided by the fin heat transfer if it were to have been all at the heat sink base temperature. The heat sink resistance becomes

$$R_{HS} = \frac{1}{h\eta_O A_{tot}} \quad \text{... ... (4)}$$

where

$$\eta o=1-NAfin(1-\eta f)/Atot \quad \text{... ... (5)}$$

Here ηo is the overall surface efficiency, N the number of fins, and Atot the total heat sink surface area.

1.3 HEAT TRANSFER PRINCIPLE

A heat sink transfers thermal energy from a higher temperature to a lower temperature fluid medium. The fluid medium is frequently air, but can also be water or in the case of heat exchangers, refrigerants and oil. If the fluid medium is water, the 'heat sink' is frequently called a cold plate. In thermodynamics a heat sink is a heat reservoir that can absorb an arbitrary amount of heat without significantly changing temperature. Practical heat sinks for electronic devices must have a temperature higher than the surroundings to transfer heat by convection, radiation, and conduction.

To understand the principle of a heat sink, consider Fourier's law of heat conduction. Joseph Fourier was a French mathematician who made important contributions to the analytical treatment of heat conduction. Fourier's law of heat conduction, simplified to a one-dimensional form in the x-direction, shows that when there is a temperature gradient in a body, heat will be transferred from the higher temperature region to the lower temperature region. The rate at which heat is transferred by conduction, q_k, is proportional to the product of the temperature gradient and the cross-sectional area through which heat is transferred.

$$q_k = -kA\frac{dT}{dx} \qquad \ldots\ldots (6)$$

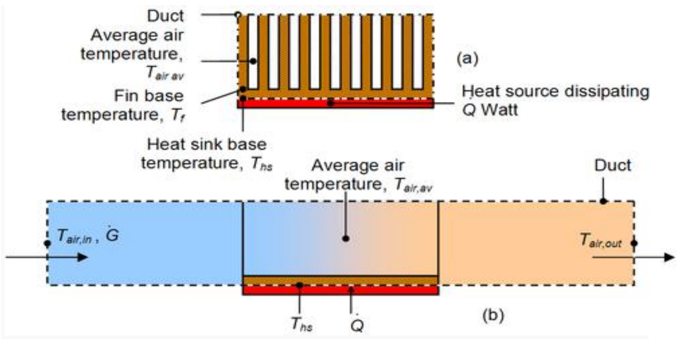


Figure 1.1: Sketch of a heat sink in a duct used to calculate the governing equations from conservation of energy and Newton's law of cooling

Consider a heat sink in a duct, where air flows through the duct, as shown in Figure 1.1. It is assumed that the heat sink base is higher in temperature than the air. Applying the conservation of energy, for steady-state conditions, and Newton's law of cooling to the temperature nodes shown in Figure 1.1 gives the following set of equations.

$$\dot{Q} = \dot{m}\, c_{p,in}\left(T_{air,out} - T_{air,in}\right) \qquad \ldots\ldots (7)$$

$$\dot{Q} = \frac{T_{hs} - T_{air,av}}{R_{hs}} \qquad \ldots\ldots (8)$$

Where

$$T_{air,av} = \frac{T_{air,in} + T_{air,out}}{2} \qquad \ldots\ldots (9)$$

Using the mean air temperature is an assumption that is valid for relatively short heat sinks. When compact heat exchangers are calculated, the logarithmic mean air temperature is used. $\dot{m}$ is the air mass flow rate in kg/s.

The above equations show that

- When the air flow through the heat sink decreases, this results in an increase in the average air temperature. This in turn increases the heat sink base temperature. And additionally, the thermal resistance of the heat sink will also increase. The net result is a higher heat sink base temperature.
- The increase in heat sink thermal resistance with decrease in flow rate will be shown in later in this article.
- The inlet air temperature relates strongly with the heat sink base temperature. For example, if there is recirculation of air in a product, the inlet air temperature is not the ambient air temperature. The inlet air temperature of the heat sink is therefore higher, which also results in a higher heat sink base temperature.
- If there is no air flow around the heat sink, energy cannot be transferred.
- A heat sink is not a device with the "magical ability to absorb heat like a sponge and send it off to a parallel universe".

Natural convection requires free flow of air over the heat sink. If fins are not aligned vertically, or if pins are too close together to allow sufficient air flow between them, the efficiency of the heat sink will decline.

II.OBJECTIVE AND METHODOLOGY

2.1 PROBLEM DEFINITION:

The heat sinks are used in electronic systems to remove heat from the chip and effectively transfer it to the ambient. The heat sink geometry is designed by the mechanical engineers with the primary aim of reducing the thermal resistance of the heat sink for better cooling in the electronic systems.

In the present project Design optimization of a Heat sink in a Power amplifier is performed to reduce the weight and size. Power amplifier is electronic equipment mounted in an army vehicle. The power modules inside the amplifier generates a heat of 1440 Watts and a temperature of 160 0c.Two Heat sinks are used to dissipate the heat generated inside the equipment and maintain a temperature of less than 850c. The existing heat sink which is being used is weighing around 10.3kgs and height of 53mm, as a result the unit is very robust.

The objective of my project is

- To design & optimize the heat sink to reduce the weight and size.
- The optimized heat sink should also dissipate heat generated by power modules and maintain a temperature of less than 850c inside.
- To achieve the design a steady state thermal analysis will be performed on the heat sink and plot the Temperature distribution on the fins.
- Based on the above analysis results we will increase/decrease the number of fins, thickness of fins, and height of fins to reduce the weight of the heat sink.
- We will perform CFD analysis of the power amplifier by mounting the optimized heat sink and plot temperature, pressure and velocity distribution in the power amplifier enclosure.
- Efforts are made to optimize temperature, pressure and velocity distribution in the power amplifier enclosure by reorienting the power modules in the enclosure
- UNIGRAPHICS software is used for 3D modeling
- SOLID WORKS FLOW SIMULATION software is used for thermal and CFD analysis.

2.2 METHODOLOGY:

- A 3D model of the existing heat sink was created using UNIGRAPHICS NX.
- The 3D model is converted into parasolid file and imported into SOLID WORKS FLOW SIMULATION.
- Steady state thermal analysis was carried out on the existing heat sink. The heat generation of individual heat modules is applied on the heat sink. The CFM of the fan is applied as convection for the heat sink.
- Based on the above thermal results, the existing heat sink is optimized for size and weight.
- The Optimized heat sink is mounted in the power amplifier enclosure and CFD analysis is carried out to plot the temperature, pressure and velocity distribution in the power amplifier enclosure.
- Based on the above CFD analysis, the power modules are reoriented from zero convection areas to the convection areas to reduce the temperature.

III. 3D MODELING OF POWER AMPLIFIER

3.1 INTRODUCTION TO UNIGRAPHICS

The power amplifier assembly is an electronic enclosure used in army vehicles to amplify the signals. Some of the important and functional parts of the power amplifier include heat sinks, power modules and fans. The 3D model of the power amplifier assembly is created using UNIGRAPHICS NX software from the 2d drawings. UNIGRAPHICS NX is the world’s leading 3D product development solution. This software enables designers and engineers to bring better products to the market faster. It takes care of the entire product definition to serviceability. NX delivers measurable value to manufacturing companies of all sizes and in all industries. NX is used in a vast range of industries from manufacturing of rockets to computer peripherals. With more than 1 lakh seats installed in worldwide many cad users are exposed to NX and enjoy using NX for its power and capability.

3.2 CREATING A SOLID MODEL

Modeling provides the design engineer with intuitive and comfortable modeling techniques such as sketching, feature based modeling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When you use a sketch, a rough idea of the part becomes represented and constrained, based on the fit and function requirements of your design. In this way, your design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic requirements are not lost when the design is edited.

The strategy you use to create and edit your model to form the desired object depends on the form and complexity of the object. You will likely use several different methods during a work session. The next several figures illustrate one example of the design process, starting with a sketch and ending with a finished model. First, you can create a sketch "outline" of curves. Then you can sweep or rotate these curves to create a complex portion of your design.

3.3 ASSEMBLY CONCEPTS

Components

Assembly part files point to geometry and features in the subordinate parts rather than creating duplicate copies of those objects at each level in the assembly. This technique not only minimizes the size of assembly parts files, but also provides high levels of associativity. For example, modifying the geometry of one component causes all assemblies that use that component in the session to automatically reflect that change. Some properties, such as translucency and partial shading (on the Edit Object Display dialog), can be changed directly on a selected component. Other properties are changed on selected solids or geometry within a component. Within an assembly, a particular part may be used in many places. Each usage is referred to as a component and the file containing the actual geometry for the component is called the component part.

Top-down or Bottom-up Modeling

You are not limited to any one particular approach to building the assembly. You can create individual models in isolation, then later add them to assemblies (bottom-up), or you can create them directly at the assembly level (top-down). For example, you can initially work in a top-down fashion, then switch back and forth between bottom-up and top-down modeling.

Multiple Loaded Parts

Many parts can be simultaneously loaded at any given time. These parts may have been loaded explicitly (such as with the Assembly Navigator's Open options), or implicitly as a result of being used by some other loaded assembly. Loaded parts do not have to belong to the same assembly. The part currently displayed in the graphics window is called the displayed part. You can make edits in parallel to several parts by switching the displayed part back and forth among those parts. The following figure shows two different assembly parts (MOUNT_ASSY.PRT and MOUNT2_ASSY.PRT) which both use many of the same components. The difference in the two is that due to a design change, assembly MOUNT2_ASSY.PRT uses components BODY2 and BUSHING2, which differ slightly from those used by MOUNT_ASSY.PRT (BODY and BUSHING). The remaining components are used by both assemblies.

The following scenario illustrates how component parts used by multiple assemblies are loaded:

- Before you open a file, there is no displayed part or any loaded parts.

- When you open MOUNT_ASSY.PRT, component parts BOLT, BUSHING, BODY, NUT, PIN, and YOKE are also loaded. MOUNT_ASSY.PRT becomes the displayed part and the work part. If you then open MOUNT2_ASSY.PRT, only the component parts not also used by the previously opened assembly (BODY2 and BUSHING2) are loaded. MOUNT2_ASSY.PRT then becomes the displayed part and work part.
- You could also open up a third assembly part that does not share any common components with the previously opened files. The new assembly part and all component parts it uses are then loaded, and the new assembly part becomes the displayed part and work part

Design in Context

When the displayed part is an assembly, it is possible to change the work part to any of the components within that assembly (except for unloaded parts and parts of different units). Geometry features, and components can then be added to or edited within the work part. Geometry outside of the work part can be referenced in many modeling operations. For example, control points on geometry outside of the work part can be used to position a feature within the work part. When an object is designed in context, it is added to the reference set used to represent the work part.

Associativity Maintained

Geometric changes made at any level within an assembly result in the update of associated data at all other levels of affected assemblies. An edit to an individual piece part causes all assembly drawings that use that part to be updated appropriately. Conversely, an edit made to a component in the context of an assembly results in the update of drawings and other associated objects (such as tool paths) within the component part. See the next two figures for examples of top-down and bottom-up updates.

Mating Conditions

Mating conditions let you position components in an assembly. This mating is accomplished by specifying constraint relationships between two components in the assembly. For example, you can specify that a cylindrical face on one component is to be coaxial with a conical face on another component. You can use combinations of different constraints to completely specify a component's position in the assembly. The system considers one of the components as fixed in a constant location, then calculates a position for the other component which satisfies the specified constraints. The relationship between the two components is associative. If you move the fixed component's location, the component that is mated to it also moves when you update. For example, if you mate a bolt to a hole, if the hole is moved, the bolt moves with it.

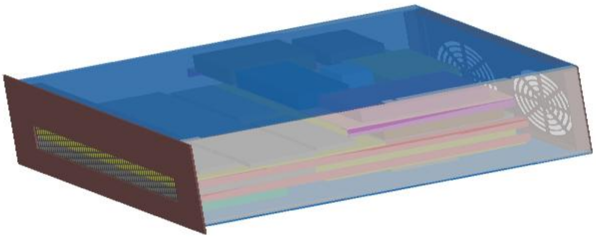


Fig.3.1 shows 3D model of the Power amplifier Assembly structure with heat sinks and power modules.

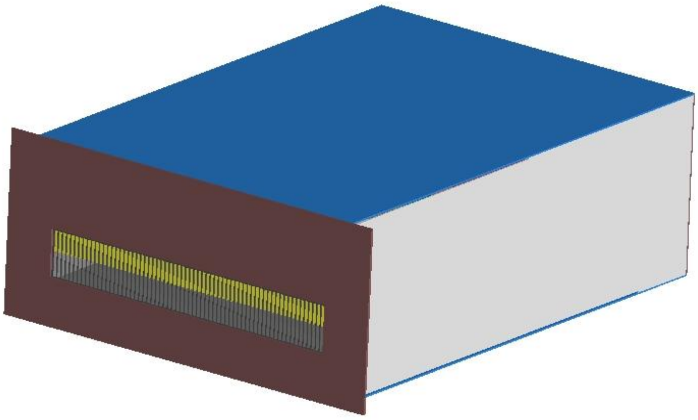


Fig.3.2 shows front view of 3d model of Power amplifier
Assembly
structure

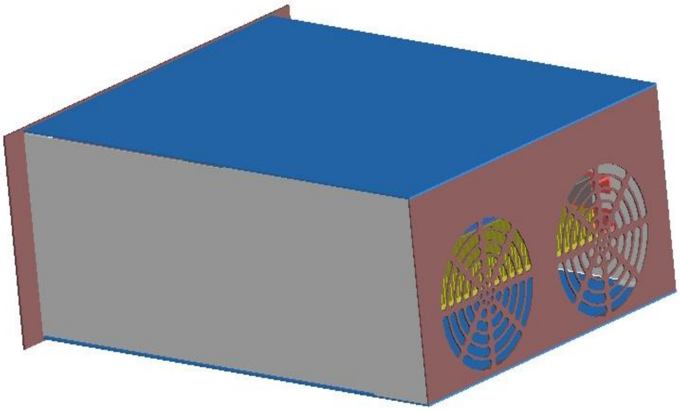


Fig.3.3 shows rear view of Power amplifier assembly

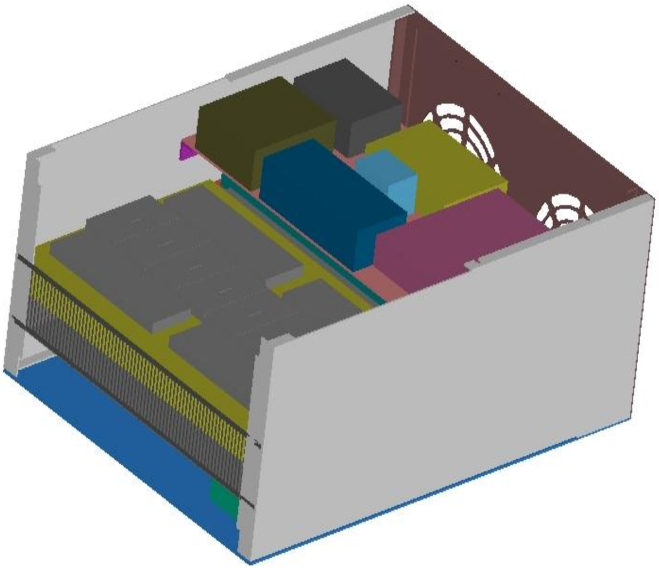


Fig.3.4 shows power amplifier Assembly structure with power modules

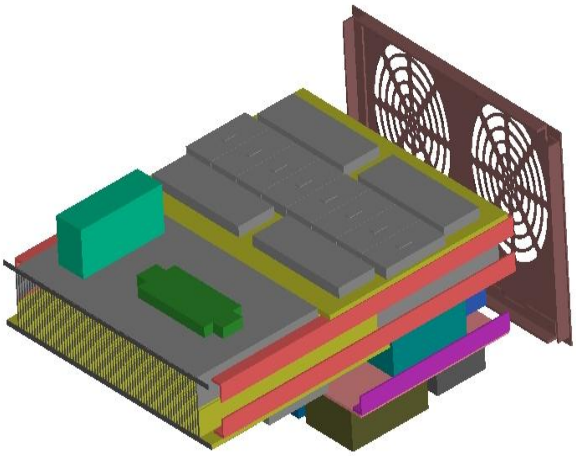


Fig.3.5 shows power modules and heat sinks in power amplifier assembly

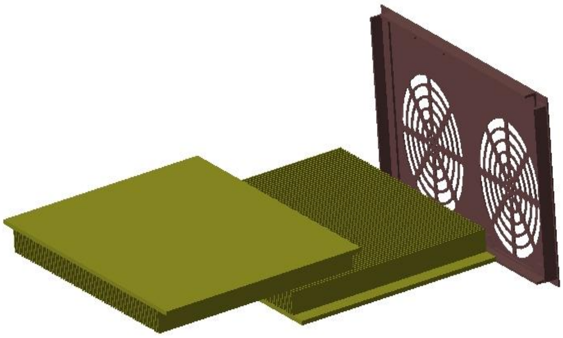


Fig.3.6 shows the heat sinks used in power amplifier assembly enclosure

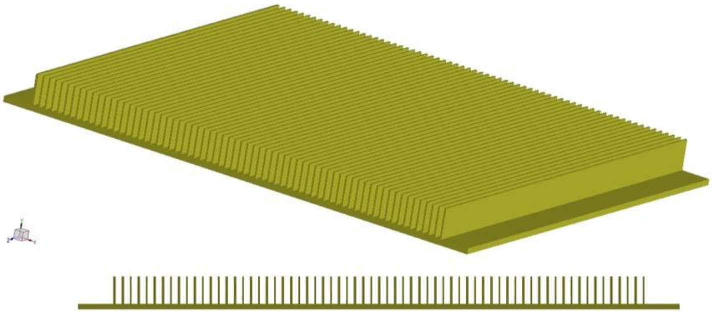


Fig.3.7 shows the detailed view of heat sinks in power amplifier used for analysis

IV.THERMAL ANALYSIS OF HEAT SINK

4.1 INTRODUCTION

SOLIDWORKS FLOWSIMULATION is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

SOLIDWORKS FLOWSIMULATION is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. SOLIDWORKS FLOW SIMULATION provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of SOLIDWORKS FLOW SIMULATION also provides a means to ensure that users are able to see the effect of a design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc.

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are:

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions).

4.2 THERMAL ANALYSIS

SOLIDWORKS FLOWSIMULATION supports two types of thermal analysis:

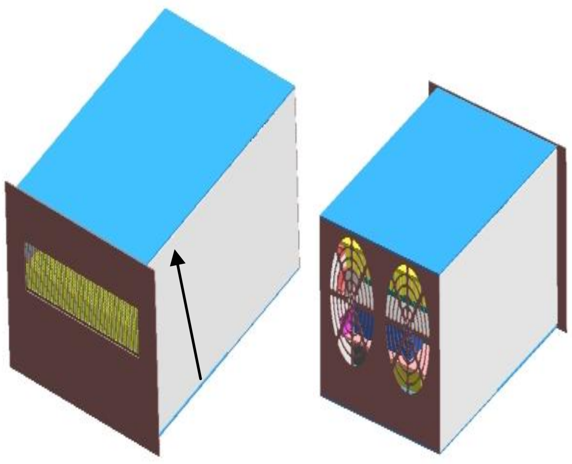
1. A steady-state thermal analysis determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.
2. A transient thermal analysis determines the temperature distribution and other thermal quantities under conditions that vary over a period of time

We can use steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects also makes the analysis nonlinear. Transient calculations are time dependent and SOLIDWORKS FLOWSIMULATION can both solve distributions as well as create video for time incremental displays of models.

3D model of the amplifier developed using UNIGRAPHICS:



Duct created for passage of air to

2 fans of CFM

Fig.4.1 The above figure shows the Enclosure of the power amplifier

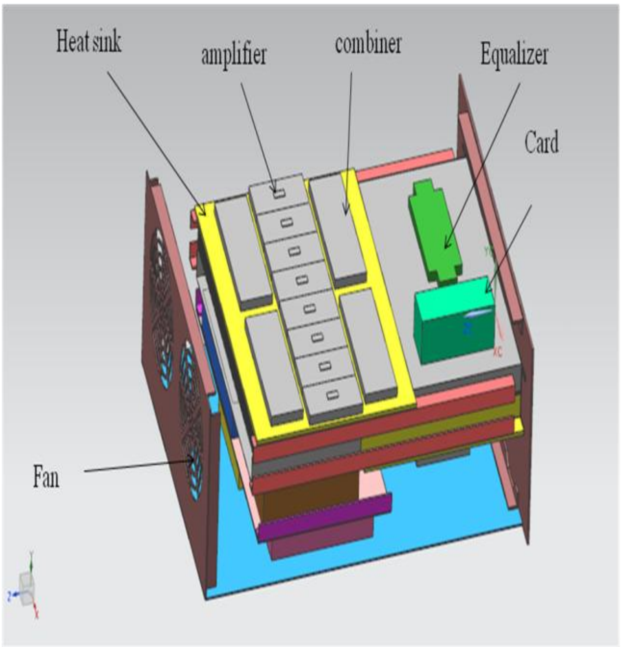


Fig.4.2 The above figure shows the components inside the Enclosure of the power amplifier

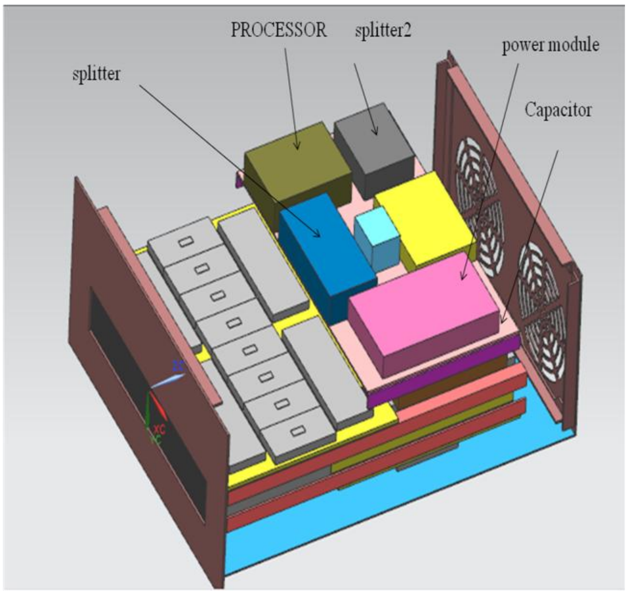


Fig. 4.3 The above figure shows the heat modules inside the Enclosure of the power amplifier

The original heat sink shown above is weighing around 10.3 Kgs as the result the power amplifier unit becomes robust. The objective of my project is to reduce the weight of the heat sink and at the same time the heat generated inside the amplifier must be less than 85 degrees.
Steady state thermal analysis is carried out using SOLIDWORKS FLOWSIMULATION software to find the temperature distribution on the original heat sink.

4.3 THERMAL ANALYSIS OF ORIGINAL HEAT SINK :

The specifications of the original heat sink are as follows:

Weight of the heat sink=10.3 Kgs
Fin height = 46 mm
Base plate Thickness = 5 mm
Number of fins = 51
Thickness of fin = 3mm

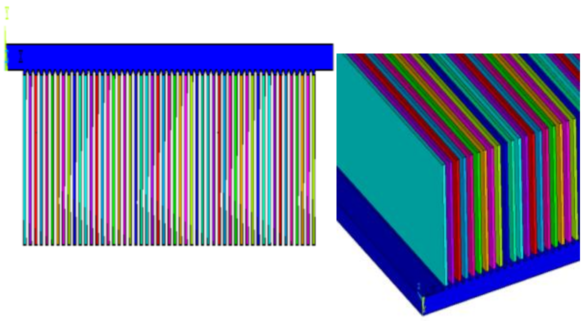


Fig. 4.4 The above figure shows the Details of fins in the heatsink

Material Properties:

The heat sink is made up of Aluminium 6061 Alloy material. Forced convection is applied by using 2 fans of 300 CFM each. The analysis is carried out at worst atmospheric temperature of 55 degrees.

Thermal conductivity of Aluminium = 167 W/mk
Film coefficient (air) = 26.4 w/m2k (Assumed 2 fans with 300 CFM each)
Bulk temperature = 328 K (55 c)
The below formulae is used to calculate film coefficient of air.

$$h_{plate} = \frac{k}{L} Nu_L$$

$$\mathrm{Re} = \frac{\rho V L}{\mu} = \frac{V L}{\nu}$$

$$Nu_L = 0.664 \cdot (Pr)^{1/3} \cdot \sqrt{Re_L}$$

where:

- V is the mean velocity of the object relative to the fluid (SI units: m/s)
- L is a characteristic linear dimension, (travelled length of the fluid; hydraulic diameter when dealing with river systems) (m)
- μ is the dynamic viscosity of the fluid (Pa·s or N·s/m² or kg/(m·s))
- ν is the kinematic viscosity ($\nu = \mu / \rho$) (m²/s)
- ρ is the density of the fluid (kg/m³)

Boundary Conditions:

Heat Generation values from the heat sources are appled as volume sources on the heat modules and convection film coefficient is applied on the fin surfcaes of heat sink.The below table lists the heat generated by individual module.The below figures shows the heat generations applied and the convection applied on the heat sink.

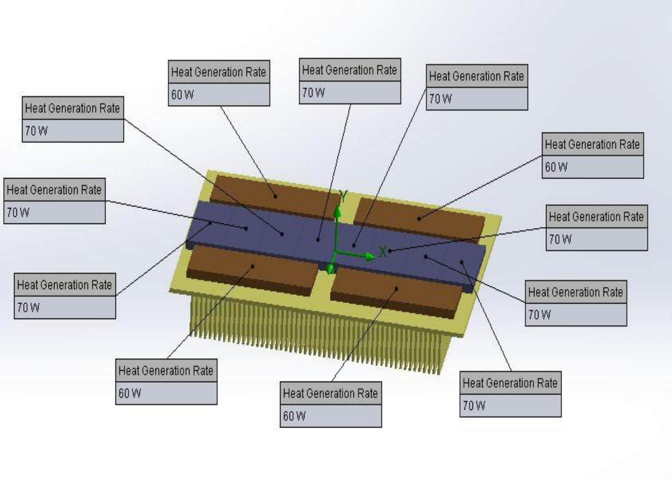


Fig.4.5 shows the heat generation values applied on modules of the heat sink.

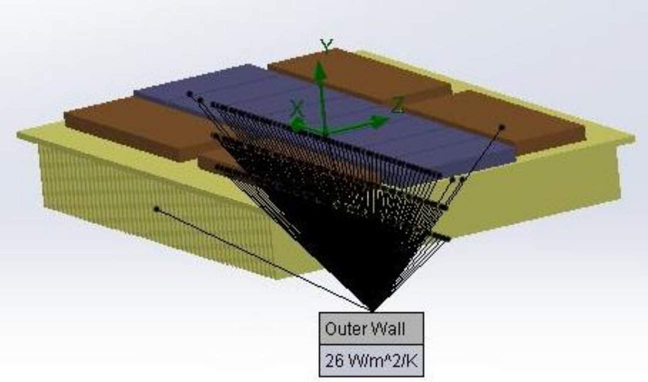


Fig.4.6 shows the convection applied on the fin surfaces of heat sink

V. Results:

From the thermal analysis the maximum temperature observed on the existing heat sink is 356 k (83 degrees)

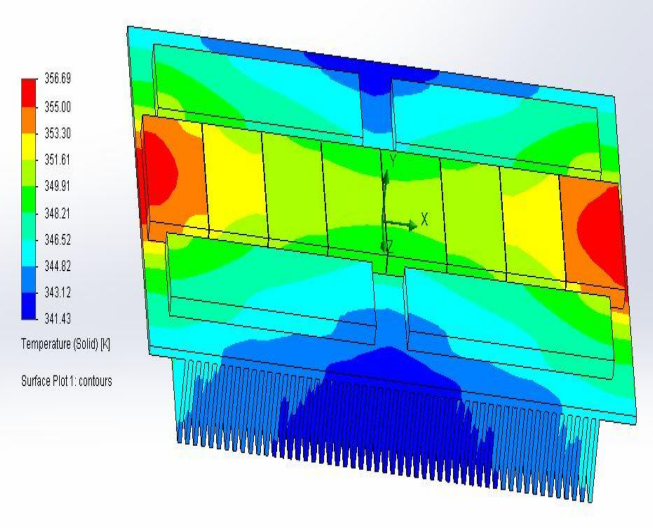


Fig. 5.1 Shows Temperature distribution from Thermal analysis for existing heat sink

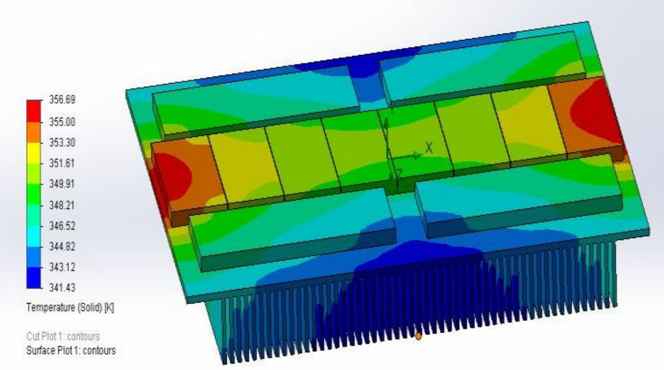


Fig.5.2 shows the temperature distribution for existing heat sink

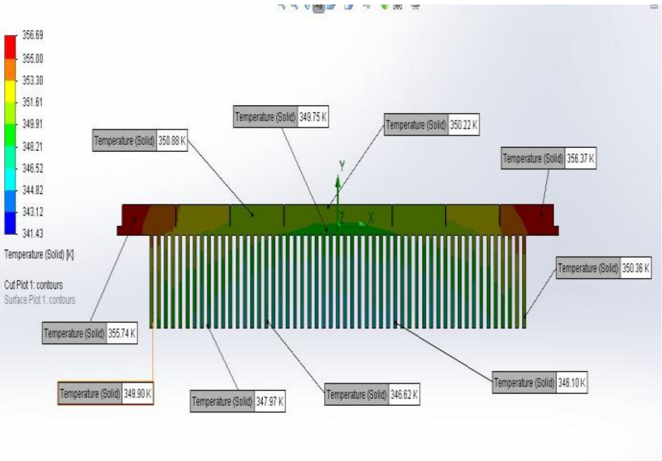


Fig.5.3 shows the temperature distribution along the cross section of the existing heat sink

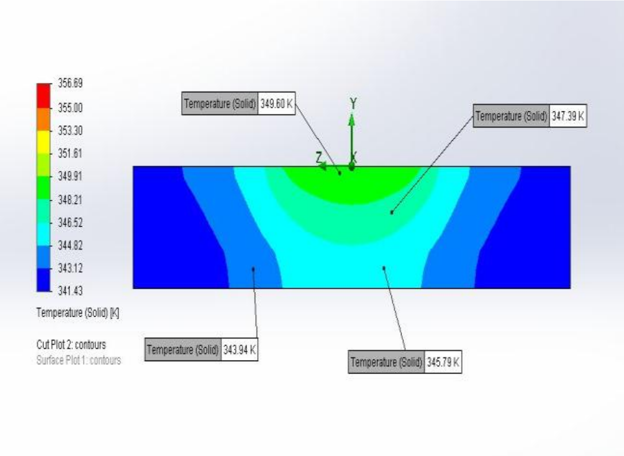


Fig.5.4 shows the temperature distribution along the cross section of the existing heat sink

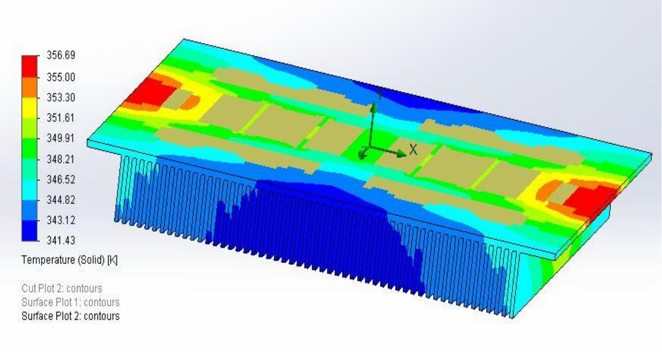


Fig.5.5 shows the temperature distribution on the tip of the fin for existing heat sink

From The above results it is observed that:

- Maximum temperature observed at heat generation patches is 356 K (83 0C)
- Max temp observed at tip of fin is 341 K (68 0C)

From the results it is observed that the though the maximum temperature is within the design limit, the temperature has not reached the tip of the fin. So the fin height can be reduced to reduce the weight. The below table shows the summary of specifications and temperature obtained on the original heat sink.

S.No	**Description**	**Original**
1	Weight (Kgs)	10.3
2	Base plate thickness (mm)	5
3	Fin Height (mm)	46
4	Number of Fins	51
5	Fin thickness (mm)	3
6	Temperature observed (K)	356

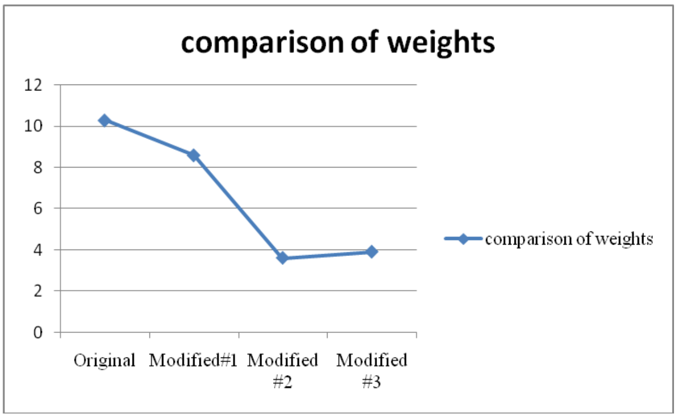
Table: Summary of results for original heat sink

VI. RESULTS AND DISCUSSIONS

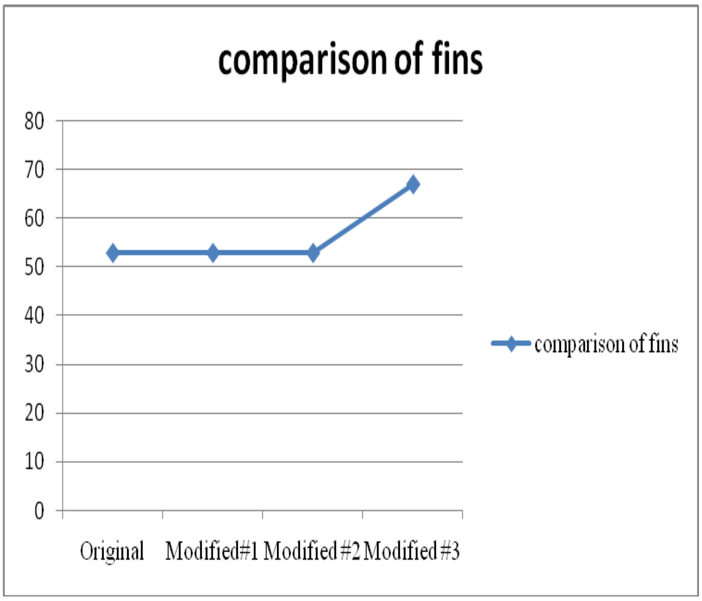
In the present project Design optimization of a Heat sink in a Power amplifier is performed to reduce the weight and size. Power amplifier is electronic equipment mounted in an army vehicle. Two Heat sinks are used to dissipate the heat generated inside the equipment and maintain a temperature of less than 850c. The existing heat sink which is being used is weighing around 10.3kgs and height of 53mm, as a result the unit is very robust. In order to reduce weight of the heat sink and maintain temperature below 850c,three modified heat sinks are designed and thermal analysis is carried out. As CFD analysis is very expensive in terms of time and resources, initially thermal analysis is performed on the heat sink. After concluding the optimised heat sink, a CFD analysis is done to find the temperature, pressure and velocity of fluid inside the enclosure. The comparison table of results and specifications of original and modified heat sinks are tabulated below. The comparison graphs are also plotted below.

		Heat Sink			
S.No	Description	Original	Modified#1	Modified #2	Modified #3
1	Weight (Kgs)	10.3	8.6	3.6	3.9
2	Base plate thickness (mm)	5	5	5	5
3	Fin Height (mm)	46	46	24	24
4	Number of Fins	51	51	51	67
5	Fin thickness (mm)	3	3	3	2
6	Temperature observed (K)	356	392	364	350

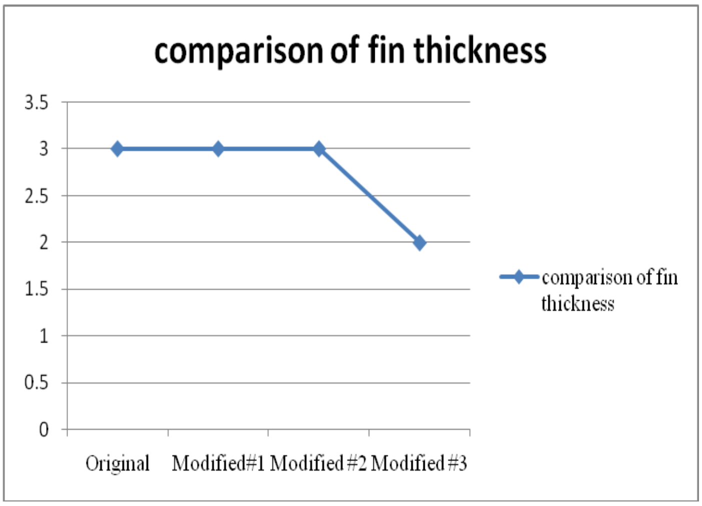
Table 6.1: Comparison table of original and modified heat sinks



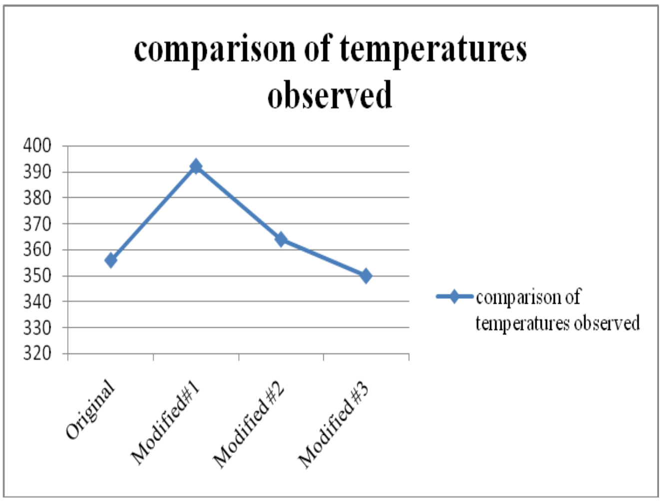
Graph 1: Comparison weights for original and modified heat sinks



Graph 2: Comparison of number of fins for original and modified heat sinks



Graph 3: Comparison of fin thickness for original and modified heat sinks



Graph 4: Comparison of temperatures observed for original and modified heat sinks

The following observations are made from the CFD analysis of the enclosure:

1. Sufficient amount of fluid flow is observed in the fin region.
2. The maximum temperature observed is 349 k (76 0C) in the enclosure.
3. New heat sink design helped in reducing temperatures.
4. Max pressure difference observed is 618 pa between inlet and outlet.
5. Max temperature of the fluid observed ~ 67 C at fins region.

VII. CONCLUSIONS AND FUTURE SCOPE

In this project the Heat sink of the power amplifier has been optimized for more heat transfer, less weight and less pressure drop using CFD analysis. The heat sink is mounted in the power amplifier on which different electronic components are mounted which dissipates heat.

- Steady state thermal analysis has been done for 4 models of heat sink.(1 original heat sink and 3 modified heat sink models)
- Max temperature on the original heat sink is 83 0c and weight of the of the heat sink is 10.3 Kgs.
- Max temperature on the modified model#1 heat sink is 119 0c and weight of the of the heat sink is 8.6 Kgs i.e. a mass reduction of 16.5%.
- Max temperature on the modified model#2 heat sink is 91 0c and weight of the of the heat sink is 3.6 Kgs i.e. a mass reduction of 65%.
- Max temperature on the modified model#3 heat sink is 77 0c and weight of the of the heat sink is 3.9 Kgs i.e. a mass reduction of 62%.

The CFD analysis of the power amplifier has been carried out by mounting the optimized heat sink model#3.From the CFD analysis it is observed that Max pressure difference observed is 618 Pa between inlet and outlet. The maximum temperature on the heat sink observed is 76o C.The weight of the heat sink is also reduced to 3.9 Kgs. From the analysis it is concluded that the power amplifier with modified model#3 heat sink is the best design in terms of weight and heat transfer.

Future Scope: As the power amplifier is used in defense equipment it is subjected to structural loads. Structural analysis has to be carried out to check the integrity of the power amplifier for structural loads.

REFERENCES

[1.] Takeshi Hirasawa, Kenya Kawabata and Masaru Oomi “Evolution of the Heatsink Technology” Furukawa Review, No. 27 2005.

[2.] Nikhil Seshasayee “Understanding Thermal Dissipation and Design of a Heatsink”. Texas Instruments Incorporated SLVA462–May 2011.

[3.] Stephen D. Turner, and Ahmed M. Zaghlol, had presented a paper on “Thermal Design Techniques improve Solid State Power Amplifier Performance”.

[4.] Mahendra Wankhede, Vivek Khaire, Dr. Avijit Goswami and Prof. S. D. Mahajan, “evaluation of cooling solutions for outdoor electronics”.

[5.] Hussam Jouhara and Brian P. Axcell “Modelling and Simulation Techniques for Forced Convection Heat Transfer in Heat Sinks with Rectangular Fins”.

[6.] Behnia, M., Copeland, D., and Soodphadakee, D., 1998, “A Comparison of Heat Sink Geometries for Laminar Forced Convection,” Proceedings of The Sixth Intersociety Conference on Thermal and Thermo mechanical Phenomena in Electronic Systems, Seattle, Washington, USA, May 27–30, pp. 310–315.

[7.] Majid Bahrami, Golnoosh Mostafavi, Mehram Ahmad., “Effects of geometrical parameters on natural convective heat transfer from vertically mounted rectangular interrupted fins” jul 8-12, 2012, Rio Grande, Puerto Rico. ASME 2012.

[8.] Charles D. Jones and Lester F. Smith “OPTIMAL ARRANGEMENT OF RECTANGULAR FINS ON HORIZONTAL SURFACE FOR FREE CONVECTION HEAT TRANSFER”. Journal of heat transfer volume 92 Online August 11, 2010.

[9.] Schmidt R., Chu R., Kamath V., Ellsworth M., Iyengar M., Porter D., Lehman B., "Maintaining Datacomm rack Inlet Temperatures with water Cooled heat Exchangers", Proceedings of the Pacific Rim/ASME International Electronic Packaging Technical Conference (INTERpack), Paper number IPACK2005-73468, San Francisco, California, July 2005.

[10.] Shin, D., 1998, "Thermal Design and Evaluation Methods for Heat Sink", Tech. Report. 7th International FLOTHERM User Conference Verona, Italy - T226 .

[11.] Shah, R.K., and Bhatti, M.S., 1987, "Laminar Convective Heat Transfer in Ducts", in Kakaç, S., Shah, R.K., and Aung, W., Handbook of Single-Phase Convective Heat Transfer", John Wiley & Sons, p. 3.42.

[12.] M. E. Steinke and S. G. Kandlikar, "Single-phase heat transfer enhancement techniques in micro channel 17–19, 2004, pp. 141–148and mini channel flows," in Proc. 2nd Int. Conf. Micro channels Mini channels, Rochester, NY, Jun.