

Thermal Analysis Of Heat Sink For Thyristor Controlled Systems Using 3-D Modeling And Finite Element Analysis.

Authors: Mr.Prasanna Joshi, Mr.N.M. Shelke, Mr.Kiran Tamadaddi.

Affiliation: M/s. INFOTECH PARTNERS, PUNE

Abstract

Electronic systems many a times use heat sinks to dissipate heat generated in the components during circuit operation. Specifically the thyristorized systems require effective heat sinks. The choice and design of heat sinks is goverend by the application and efficiency of heat dissipation.

This paper reports a study carried out using 3-D modeling and finite element analysis for thyristor controlled heat sinks.Using Pro-Engineer as a 3-D modeling software the 3-D model of the heat Sink was created. Using Neutral file format the model was imported into Ansys.

Here the solid model was converted into Finite Element model. Boundary conditions such as maximum temperature of the thyristor, convective heat transfer coefficient, ambient temperature and material properties such as thermal conductivity of the material are used as set of parameters for the program. Heat flux and temperature profiles were used as measure of effeciency.

1. Introduction

Electronic component has made its way into practically every aspect of modern life, from toys and appliances to high-power computers. The reliability of the electronics of a system is a major factor in the overall reliability of the system. The performance of electronics components depends on the passage of electric current to perform their duties. They become potential sites for excessive heating, since the current flow through a resistance is accomplished by heat generation. Continued miniaturization of electronic systems has resulted in a dramatic increase in the amount of heat generated per unit volume, comparable in magnitude to those encountered at nuclear reactors and the surface of the sun. Unless properly designed and controlled, high rates of heat generation result in high operating temperatures for electronic equipment, which jeopardize its safety and reliability.

The failure rate of electronic equipment increases exponentially with temperature. Also, the high thermal stresses in the solder joints of electronic components mounted on circuit boards resulting from temperature variations are major causes of failure. Therefore, thermal control has become increasingly important in the design and operation of electronic equipment.

In this paper, we discuss one of the methods to determine the thermal variations of electronic equipment by means of Finite Element Analysis (FEA).

2. Air cooling: Natural Convection.

Low – power electronic systems are conveniently cooled by *natural convection* and *radiation*. Natural convection cooling is very desirable, since it does not involve any fans that may break down.

Natural convection is based on the *fluid motion* caused by the density differences in a fluid due to temperature difference. A fluid expands when heated and becomes less dense. In

gravitational field, this lighter fluid rises and initiates a motion in the fluid called *natural convection current*. Natural convection cooling is most effective when the path of the fluid is relatively free of obstacles, which tend to slow down the fluid, and is less effective when the fluid passes through narrow passages and over many obstacles.

The magnitude of the natural convection heat transfer between a surface and a fluid is directly related to the *flow rate* of the fluid. The higher the flow rate, the higher the heat transfer rate.

The heat transfer from a surface at temperature T_s to a fluid at temperature T_{fluid} by convection is expressed as,

$$Q_{conv} = h_{conv} A (\delta T) = h_{conv} A \delta (T_s - T_{fluid}) \quad (W)$$

Where, h_{conv} is the heat transfer coefficient and A is the heat transfer surface area. The value of h_{conv} depends on the geometry of the surface and the type of fluid flow, among others things.

3. Cooling load of Electric Equipment

The cooling system of an electronic device must be designed considering the actual *field* conditions. The first step in the selection and design of cooling system is the determination of the heat dissipation, which constitutes the *cooling load*. The easiest way to determine the power dissipation of electronic equipment is to measure the voltage applied V and electric current I at the entrance of the electronic device under full- load conditions and to substitute them into the equation

$$W_e = VI = I^2 R \quad (W)$$

Where, W_e is the electric power consumption of the electronic device, which constitutes the *energy input* to the device.

The first laws of thermodynamics require that in steady operation the energy input into a system be equal to the energy output from the system. Considering that the only form of energy leaving the electronic device is heat generated as the current flows through resistive elements, we conclude that the heat dissipation or cooling load of an electronic device is equal to its power consumption.

$$\text{i.e., } Q = W_e$$

4. Importance of FEA

The *Finite Element method* is a numerical procedure for solving continuum mechanics problem with accuracy acceptable to engineers.

The purpose of FEA is to model the behavior of structure under the system of loads not just its geometry. The extent of influence of geometrical details on the behavior varies from case to case. Classical methods describe the problem with partial differential equations but yield no answers because the geometry and loading are complicated. In practice most problems are too complicated for a closed form mathematical solution. A numerical solution is required and the most versatile method that provides it is the finite element method.

5. i) How ANSYS Treats Thermal Modeling

The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. The finite element solution performed with ANSYS calculates nodal temperatures. These nodal temperatures are used to obtain other thermal quantities such as heat flux, thermal gradient and heat flow rate. The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation

5. ii) Tasks in a Thermal Analysis

The procedure for doing a thermal analysis involves three main tasks:

1. Build the model.
2. Apply loads and obtain the solution.
3. Review the results.

6. i) Aim of Analysis: - To produce a heat sink with maximum heat flow rate with optimum weight.

The heat sink is as shown in fig.1.



Fig.1

6.ii) Assumptions

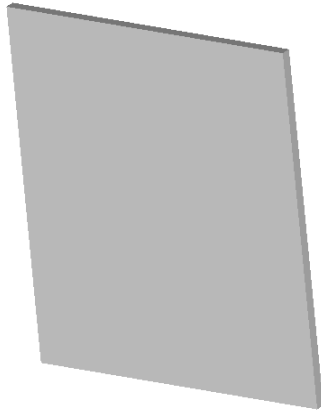
- Steady state operating conditions exist.
- Source temperature is at 60°C, ambient air temperature is 30°C
- Air cooling: Natural convection.
- Average surface temperature exposed to air is 50 °C.
- Plate is horizontally mounted and effect of radiation is not considered.
- Thermal insulation is provided on the inner sides of the heat sink.

7. Procedure

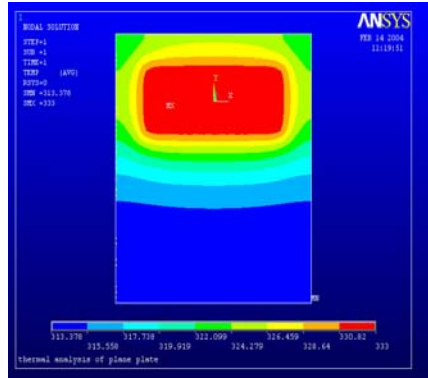
As the heat sink is *insulated* from inside, the actual area that takes part in heat dissipation was only the *fin surface area*. Hence we idealize the heat sink as a *flat plate with fins*. We had done *five* numbers of iterations.

Case I) Flat Plate (35mmx40mm)

The *flat plate of size 35x40* was created in *Pro-Engineer* software and it was imported in the *ANSYS* environment using *IGES (Initial Graphics Exchange Format)*. Element used is *SOLID87*. The element has one degree of freedom, temperature, at each node. The element is applicable to a three-dimensional, steady-state or transient thermal analysis. Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system.



Flat plate 35mmx40mm

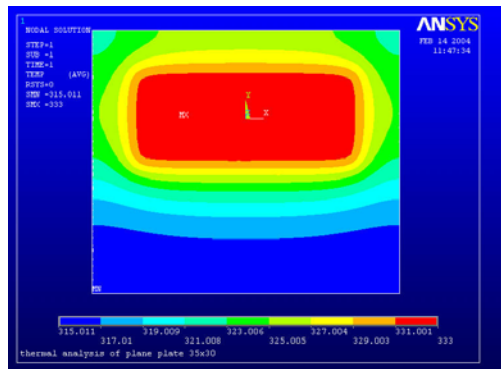


Temperature Distribution

Using a 2mm element size the solid model was converted into *finite element model* which consists of only nodes and elements. At the source, a temperature of 60°C is applied as Degree of Freedom. The top side of the plate which is subjected to convection is having a convective coefficient of 6.35 W/m²K, and is exposed to air at temperature of 30°C.

From the temperature distribution plot we know that bottom portion of the flat plate is not subjected to much variations of temperature and it results in increase in weight. Hence we have cut that area and the new size would be 35mm x 30mm.

Case II) Flat Plate (35mmx30mm)

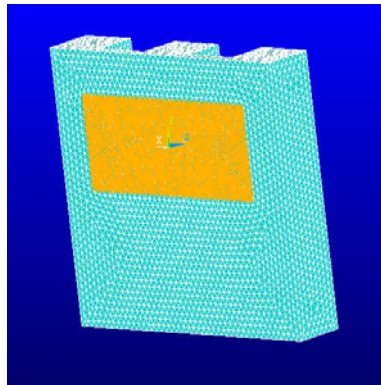


Temperature Distribution

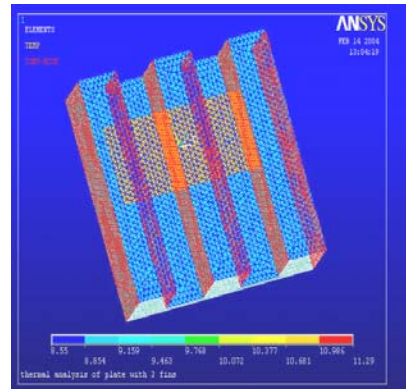
The same analysis was carried and the results were plotted. By comparing the results of the flat plate with 35mmx40mm size and the plate with 35mmx30mm size, the increase in heat flux rate is very small. Hence the size of this plate becomes a bench mark for future analysis. The new size of plate is 35mmx30mm.

Case III) Flat Plate (35mm x30mm) with 3 fins of height 5mm

Keeping the same boundary conditions the plate was checked for the effect of three fins.



Mesh model with temperature as DOF



Mesh model with convection as load

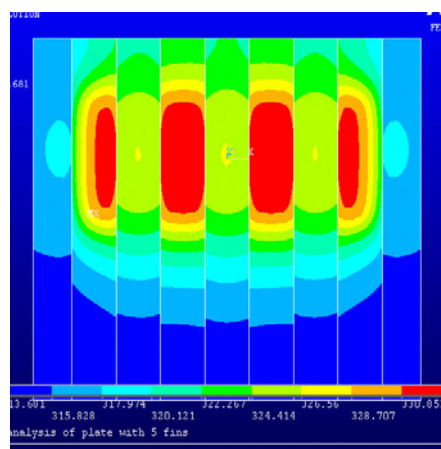
By comparing the results of the flat plate and the plate with fins, there was 38% increase in heat flux rate due to increase in surface area. Now further, fin height was reduced from 5mm to 3mm.

Case IV) Flat Plate (35mm x30mm) with 3 fins of height 3mm

With the decrease in the height of the fins, there was slight decrease in heat flux rate. Now the numbers of fins were increased to five and again analyzed for the same boundary conditions.

Case V) Flat Plate (35x30) with 5 fins of height 3mm

With the increase in number of fins to five, there is 11% increase in heat flux. Also the reduction in height of fins helps in carrying the heat easily, thus the heat flux increases.



Temperature distribution of plate with five fins

8. Results

Analysis Results

<i>Case no.</i>	<i>Plate configuration in mm</i>	<i>Mass of plate in gms.</i>	<i>Average Heat flux in W/m²</i>	<i>Area in mm²</i>	<i>Convective coefficient in W/m²K</i>
I	Plane plate 35x40	3.906	411.500	1400	6.350(horizontal face)
II	Plane plate 35x30	2.900	413.500	1050	6.584(horizontal face)
III	Plane plate (35x30) with 3 fins of height 5mm	11.718	569.500	1950	8.550 (horizontal face) 11.290 (vertical face)
IV	Plane plate (35x30) with 3 fins of height 3mm	8.200	548.250	1590	8.550 (horizontal face) 12.830 (vertical face)
V	Plane plate (35x30) with 5 fins of height 3mm	7.700	606.500	1980	9.630 (horizontal face) 12.830 (vertical face)

9. Conclusion

From the above analysis results of the heat sink using Ansys, we conclude that the *Case V* results are satisfactory. In this case, with increase in surface area & decrease in mass, we achieved more heat flux than case III & IV. Thus our aim of analysis is served. The important aspects of these analyses are the idealization had to match that of actual conditions. Application of boundary conditions, flow of heat convection play very important role. Results from the software are just a guideline to the given task, but ultimately it is the user who has to judge the answer and decide the percentage-range he is satisfied with. The results affect the type of analysis.

10. References:

1. Yunus Cengel. "Heat Transfer a practical approach". Tata McGraw-Hill Edition
2. Frank P. Incropera, David P. Dewitt. "Heat and Mass transfer".John Wiley & Sons.
3. www.ansys.com