Thermal Management of Electronic Devices used in Automotive Safety – A DoE approach

Vinod Kumar, Vinay Somashekhar and Srivathsa Jagalur
Autoliv India Private Limited, Bangalore, India

***Abstract**: Electronic devices used in automotive safety has a trend to reduce size in packaging and increase performance of the product, both of which contribute to exponential increase in power consumption of the system consequently increasing the heat generation. Designers have very less choice on PCB board components but have flexibility in designing the enclosures over the PCBs. This calls the need for an optimal enclosure design to maximize the heat dissipation and to reduce the temperature well below the allowable operational limit. Majority of heat generated by these devices get dissipated by conduction and natural convection modes. Heat generated by the chip on PCB due to power consumption is transferred to the cover and housing via heat sink and thermal pads respectively by conduction. The surface film coefficients estimated from the bench tests were used to simulate convection heat transfer from housing and cover to ambient using Abaqus.*

To arrive at an optimal house and cover design, a DoE approach is used with geometrical, material and positioning as parameters. Efficient cover and housing design ensuring better functionality with maximum heat dissipation capability in terms of conduction and convection is achieved with minimum number of physical tests by DoE approach of thermal analysis using Abaqus.

***Keywords**: Thermal management, Steady State Heat Transfer, Automotive safety, DoE*

1. Introduction

Electronic equipment 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. Electronic components depend on the passage of electric current to perform their duties, and they become potential sites for excessive heating, since the current flow through a resistance is accompanied 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 jeopardizes 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 the past, thermal analysis of electronic systems relied upon a combination of the empirical data and analytical techniques which were based on regular geometries and simple boundary conditions. For the analysis of more complex systems, electro-thermal analogue representations of the heat transfer, based on thermal resistance network methods were employed. Numerical schemes were also employed for the analysis of conductive heat transfer problems, but these depended on the prescription of the illusive convective heat transfer coefficient on the convective surfaces. Realistically the conjugate problem can only be accurately represented by Computational Fluid Dynamic (CFD) analysis that simultaneously solves the appropriate governing equations for the solid and fluid domains, and couples both at the solid-fluid interface. However to date, the predictive accuracy of CFD codes dedicated to the thermal analysis of electronic equipment has not been comprehensively validated. Consequently, CFD based thermal designs still require lot of experimental verification during the design phase, thereby diminishing the potential of this design method to reduce product development cycle times.

With ever-reducing product development cycle times preventing extensive prototyping, increasing reliance is being placed on numerical predictive techniques, a need therefore exists to comprehensively evaluate the predictive accuracy of these codes, so as to anticipate what extent of concurrent experimentation would be required during the design cycle. Although a thermal management system may make use of all modes of heat transfer to maintain temperatures within their appropriate limits, the simplest form is by conduction and convection heat transfer.

In this work authors wants to demonstrate the thermal management using steady state heat transfer analysis involving conduction and convection modes in a simplified manner. For thermal management FE simulation challenges lies in chip modelling, PCB modelling, design of enclosures, thermal boundary conditions and positioning of these electronic devices.

With increase in occupant safety consciousness in an automotive, the electronic control of Air bags, breaking and seatbelt pre-tensioners have become standard equipment in almost all the vehicles. The Figure 1 shows a typical safety control systems where such electronic circuitry is used and the structure of one such circuit board shown in Figure 2.

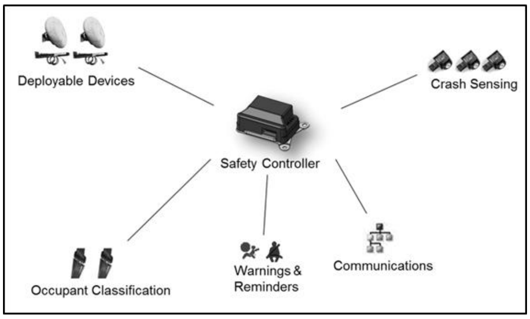


Figure 1. Safety Controller

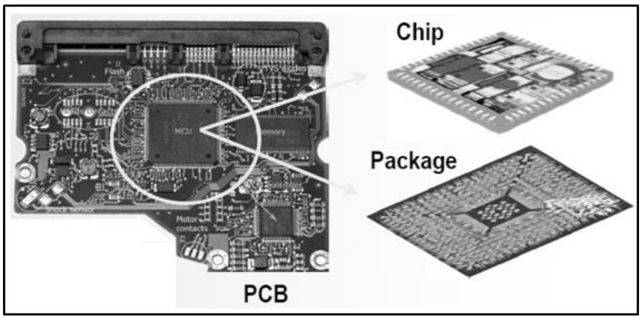


Figure 2. Circuit Board

To arrive at an optimal house and cover design, a DoE approach is used with geometrical, material and positioning as parameters. Efficient cover and housing design ensuring better functionality with maximum heat dissipation capability in terms of conduction and convection is achieved with minimum number of physical tests by DoE approach of thermal analysis using Abaqus.

2. Numerical Model

In this study, the prediction of a component's thermal interaction is investigated on the PCB. The current design is referred base model. Component is centrally located on the PCB. The PCB is housed inside enclosures. The base model of the enclosure consists four components outer fin, House, Inner heat sink and Case which is made of aluminum material. The Figure 3 shows the base model of the Safety Control Device.

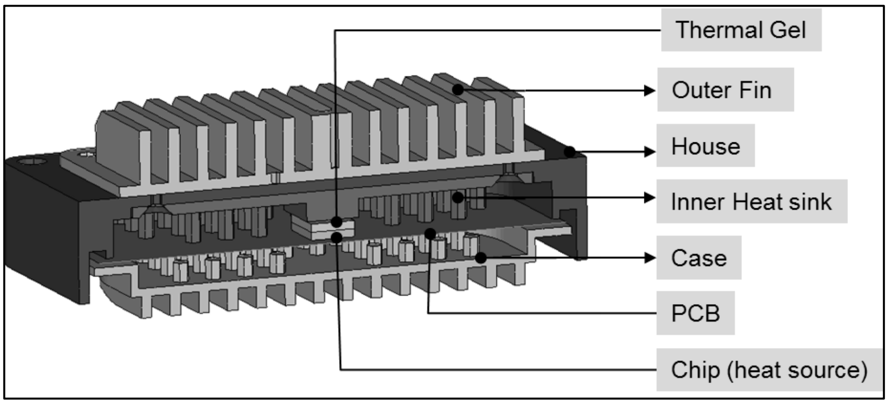


Figure 3. Safety Controller Base Model

2.1 Problem Description

The heat generated by electronic circuit in Safety Control Device must be dissipated effectively to improve reliability and to keep the operating temperature of PCB component well below the desired limit. Here chip (resistor) is the heat generating component with operational limit of -35°C to 150°C. To serve this purpose of heat dissipation, aluminum fins are mounted at different locations in order to minimize the temperature of PCB at Chip PCB interface.
Heat generated by the chip is transferred to thermal gel which are cured in place to give a permanent and durable thermal interface. In the uncured state, these materials are soft enough to assemble components under low force and then cure to a harder state. Thermal gel carries the heat to Inner heat sink, House and Outer fin in top side. Heat flow path for current design is as shown in the Figure 4.

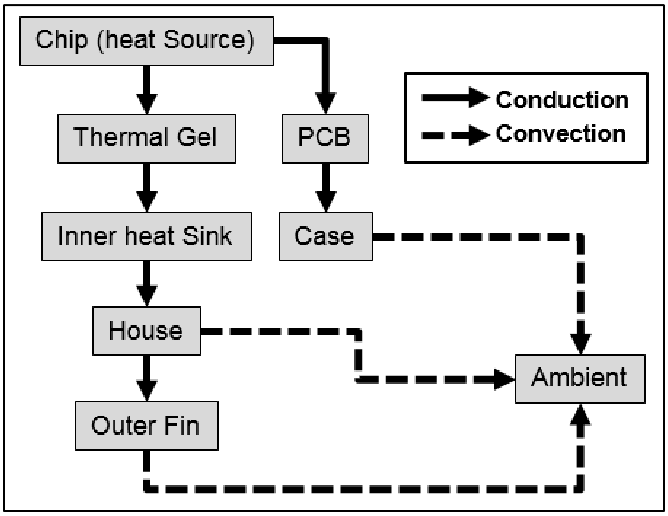


Figure 4. Heat Flow Path

2.2 Objective and Performance Parameter

Based on these house topologies, a methodical approach is employed to permit both the modeling methodology applied and solver capability to be carefully evaluated. Different configuration of the housing and case, poses the greatest challenge for numerical prediction due to it generating a high degree of component thermal interaction. Measured *component junction temperature* is used as the primary benchmark criterion, while component and PCB surface temperature measurements serve to validate the component-PCB modeling methodology. All component dimensions and constituent material thermo-physical properties correspond to nominal vendor specifications. Objective of this work is to reduce this *component junction temperature* by finding an optimal house and cover design.

2.3 Modeling Approach

Keeping in mind the physical structure of Component shown in Figure 5, where the core resisting layer which is sandwiched between a plastic or ceramic layer coating. The Component is modelled as multi-layered with DC3D8 (Continuum Thermal Bricks) elements in which the heat is generated at mid layer of resistor and is conducted further. The Heat generation is modeled using surface heat flux.

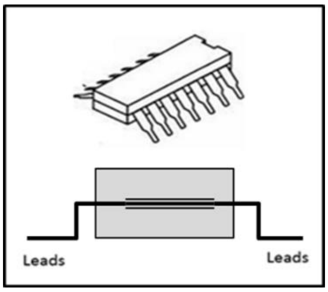


Figure 5. Physical Structure of a Component

PCB is modelled as general purpose shell elements (DS3 or DS4) with thermal degrees of freedom and orthotropic material property with higher thermal conductivity along in-plane directions and least thermal conductivity along thickness direction. Thermal gel and other components of the assembly is modelled with solid elements (DC3D8) with appropriate connections assumed.

Conduction heat transfer happens due to contact between two components. Transfer of heat depends on the surface finish of surfaces in contact. In simulation it is assumed to have ideal contact between two components with no gaps in between and the two components are tied to each other. Convective heat transfer coefficient for natural convection is assumed 5-10 W/m^2K

The thermal material properties of the components used are as listed in the Table 1.

Components	**Material**	**Conductivity (W/mK)**
House, Case, Inner Heat sink, Outer fin	Aluminum	237
PCB	Substrate (Ortho)	K11=32.7, K22 =32.7, K33=0.28
IC Component	Composite	0.16
Thermal Gel	Thermal Gel	6

Table 1. Material Properties

3. Experimentation

A prototype of current baseline model was tested and temperatures on PCB, Case, Outer Fin and house is measured using a thermocouple. The Figure 6 shows the measured temperatures on different components of the assembly. Conditions assumed for the testing: Room temp = 25°C, Voltage =9 V, Current = 1.8 A, Safety Control Device position – vertical

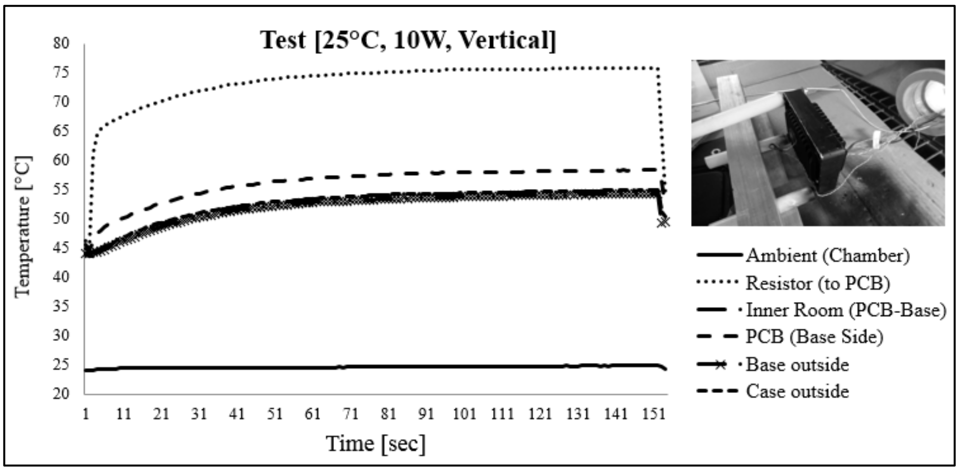


Figure 6. Physical Test Results

4. Numerical Model Validation

The present simulation model is correlated with a bench test by tuning parameters like Convective film coefficient and Convection surface.

4.1 Simulation at Room Temperature

The Figure 7 shows the thermal profile and maximum measured temperature on different components of the system in room temperature when it reached steady state. In Room temperature case initial temperature of the assembly is considered to be at 25°C

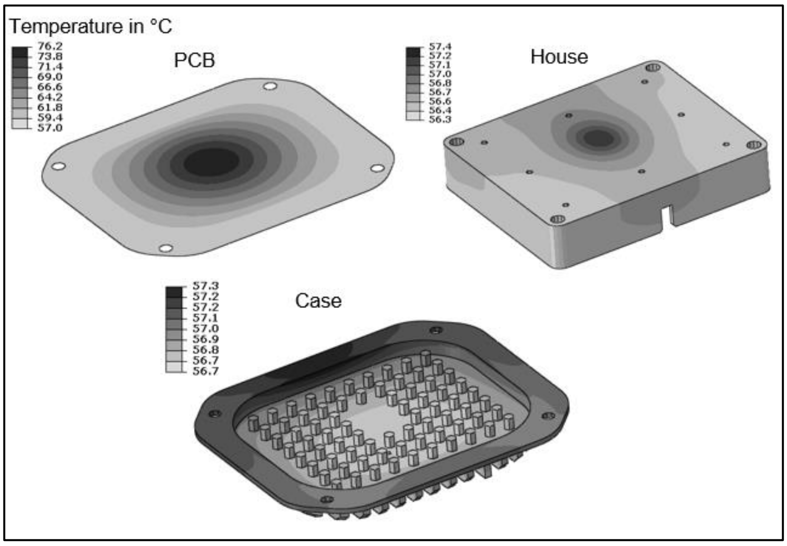


Figure 7. Temperature plot for Room Temperature case

The Table 2 shows the comparison of measured temperatures on different components in simulation and physical test. The simulation results correlated well with the physical test results. Thus providing a benchmark model to go for further experiments with numerical model.

Component	Test (max °C)	Simulation (max °C)
House	54	57
Case	55	57
PCB (Junction Temperature)	76	76

Table 2. Results for Room Temperature case

4.2 Simulation at High Temperature

The benchmark model is simulated in an ambience of higher temperature to see the effect on the temperature rise at the junction. The Figure 8 shows the thermal profile and maximum measured temperature on different components of the system at high ambient temperature when it reached steady state. In high temperature case initial temperature of the assembly is considered to be at 85°C

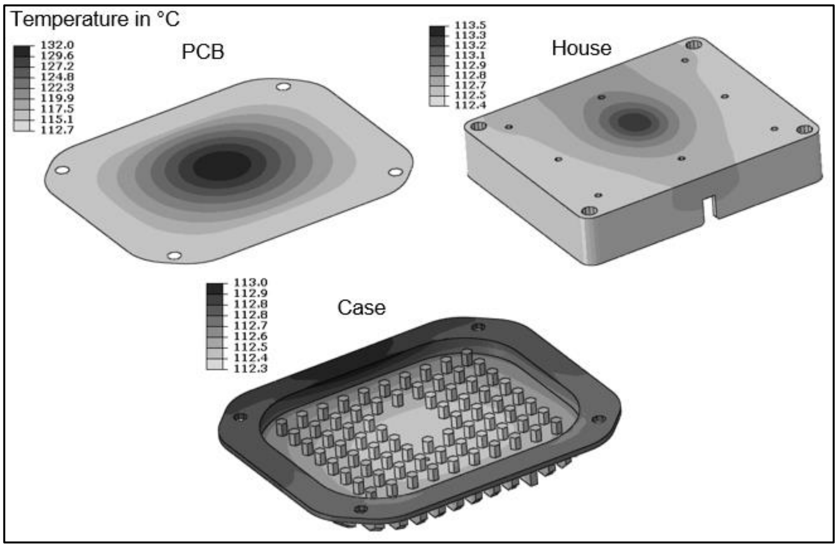


Figure 8. Temperature plot for High Temperature case

4.3 Limitations of Base model

Having multiple components for conduction of heat increases the mass of the enclosure assembly, and makes the design a complex assembly. Increase in number of components also increases the effort of material handling and lead time in assembly process. When two or more mating components are assembled by means of screws, the ability of transfer of heat over their contacting surfaces determined by certain parameters of contacting solids such as surface roughness, flatness deviation, gap thickness etc. which is difficult to avoid while manufacturing. And is also complicated to model such losses in simulation.

This calls for need of reduction of number of components in enclosure design with optimal conductive material and surface area for natural convection through DoE approach.
With the current validated Simulation model, Design of Experiments were performed in order the make the Baseline design more robust. The important criteria considered are reduction of number of components, reduction of Assembly weight and increase of Surface area in order to achieve minimum possible temperature on PCB and maximum heat dissipation.

5. DoE results

The list of design iterations performed are as tabulated in the Table 3.
The Table 3 shows the brief structure of modified design highlighting key parameters like Number of components, final mass of enclosure assembly and *component junction temperature* on PCB at chip PCB interface. Design no. 4 gives an optimal solution with lower junction temperature and with minimum number of components.

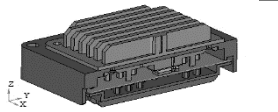
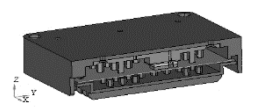
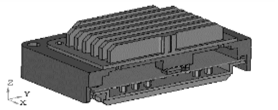
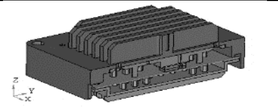
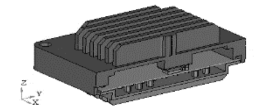
Design No.	Model Assy.	No. of Enclosure Comps	Assy. mass [gms]	Component Junction Temp. [°C] with ambient at 25°C	Component Junction Temp. [°C] with ambient at 85°C
Base Model		4	285	76.17	132.02
1		3	210	94.66	156.62
2		3	254	76.16	138.17
3		3	265	76.38	138.38
4		2	220	67.7	129.75

Table 3. DoE Results

6. Conclusion

With Abaqus authors were able to build a Simulation model to replicate the actual physical model, which was further used to modify the design and arrive at final design with lesser components, better functionality and with minimum design cost. Further the modified model can be analyzed for CFD to see the effects of forced convection and Radiation when number of components on the PCB is increased.

References

1. Abaqus User's Manual, Version 6.13-1, Dassault Systémes Simulia Corp., Providence, RI.
2. Giuseppe Petrone & Giuliano Cammarata – Numerical Modelling for Thermal Design of Electronic Equipments
3. Hiroyuki Komaki & et al., Development of the electronic "Safing" system for airbag ECUs
4. Mangesh D Shende (1) & Dr. Ashish Mahalle (2) - Cooling Of Electronic Equipments with Heat Sink: A Review of Literature
5. McGraw Hill, Heat and Mass Transfer: Fundamentals and Applications - Chapter 15-6, 15-7, Cooling Of Electronic Equipment
6. Wikipedia - http://en.wikipedia.org/wiki/Thermal_management_of_electronic_devices_and_systems
7. Z. J. Zuo, L. R. Hoover & A. L. Phillips – An integrated thermal architecture for thermal management of high power electronics

Acknowledgments

The authors gratefully acknowledge Autoliv Electronics France and Autoliv Electronics Germany for their technical support. The base model was built and tested at Autoliv Electronics Germany. Authors would like to express their gratitude towards Autoliv India, who funded this work and helped them to use the computational facilities to undertake the numerical analyses.