RESEARCH ARTICLE OPEN ACCESS

Thermal Design and Analysis of Mechanical Housing Using Ansys

Mr.N.Venkatesh,(M.tech) Mr.M. Kedarnath,M.tech(Ph.D)
*M.Tech (Thermal) Department of mechanical engineering, Godavari institute of engineering& technology (autonomous), Rajahmundry, Andhrapradesh, India (nersuvenkatesh@gmail.com)
**Associate Professor Department of mechanical engineering, Godavari institute of engineering &technology (autonomous), Rajahmundry, Andhrapradesh, India (knm576@gmail.com)
Corresponding author: N.Venkatesh

ABSTRACT
Mechanical housing play critical role in ground electronic devices. It houses Printed Circuit Boards (PCBs) on which high heat dissipating components will be mounted. The complexity associated with this requirement is to dissipate the heat to the ambient through constrained heat path. Though heat can be dissipated to a greater extent through means of forced convection but it demands for power. Hence heat is to be dissipated through free convection in addition to conduction. Thermal design and analysis of mechanical housing in free convection environment is a tedious task in commercial FE software packages. Evolving a theoretical approach for thermal design and analysis of mechanical housing considering two basic heat dissipation mechanisms (Conduction and Free convection) has been taken up in this project. The outcome of the project would be a theoretical method using which the thermal designer can quickly design his mechanical housing. Proposed method also investigates whether enhanced cooling mechanism like fins is needed to be adopted or not and if so it enables the designer to formulate optimum design of fins. This method can be used for any size of the mechanical housing and for any heat dissipation rates and hence it offers a universal solution for thermal design. More over in this project the proposed theoretical method will be validated with thermal analysis of mechanical housing using Finite Element Method (FEM) in ANSYS software.
Key Words: Printed Circuit Boards (PCBs), electronic enclosure, aluminium sheet,

---

Date of Submission: 21-08-2017 Date of acceptance: 09-09-2017

I. INTRODUCTION

Electronic enclosures play vital role in aerospace applications. These enclosures house Printed Circuit Boards (PCBs) having high heat dissipating components mounted on them. The intended electronic enclosure consists of five PCBs on which heats dissipating electronic components are mounted. This enclosure is basically consists of fins provided

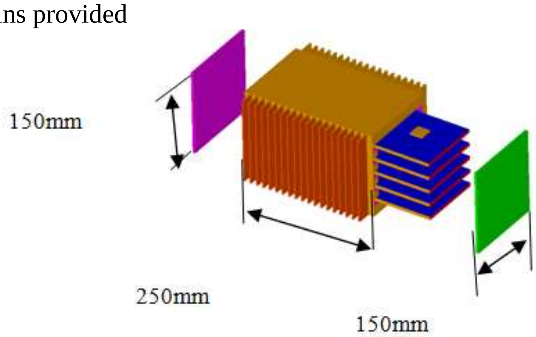


Fig 1 Details of the electronic enclosure.

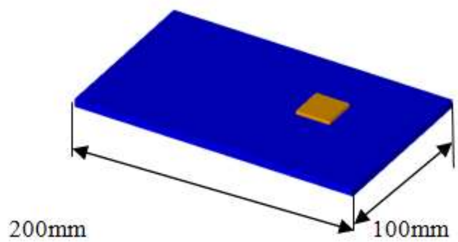


Fig 2 PCB without aluminium sheet.

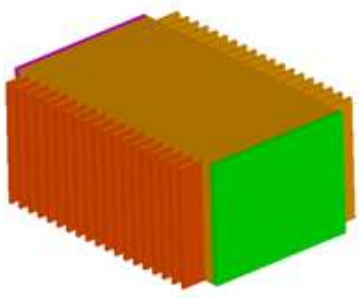


Fig 3 Assembly of the electronic enclosure.

Figure 3 Assembly of the electronic enclosure the complexity involved in this application is to dissipate the heat to the ambient through limited heat dissipating mechanism. Though heat can be dissipated to a larger extent using forced convection means but no scope is allowable for forced convection in aerospace applications. Hence heat is to be dissipated through natural convection other than conduction. Many of the failures of electronic components observed in past are due to high temperature. Maximum junction temperature of one such electronic component positioned on a Printed Circuit Board (PCB) of an electronic enclosure resulted to be 112^0 C against the allowable temperature of 90^0 C.

Design of an enhanced heat dissipation mechanism is taken up in this project with a view to reduce the temperature below 90^0 C. As a solution to this problem it is decided to split each PCB into two slices and introduce an aluminium sheet in between to enhance the heat dissipation, as the thermal conductivity of aluminium is more than material of PCB. Thermal design of this enhanced cooling mechanism is done using an analytical method.

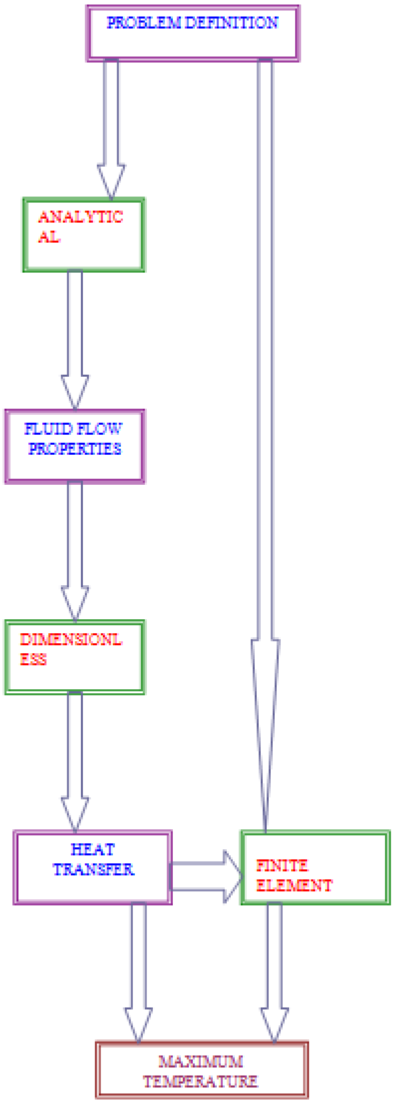
This present work brings out the methodology to obtain the maximum junction temperature of the electronic component. The analysis of this method will be useful to ascertain that the maximum junction temperature is less than the allowable temperature. In later stage the maximum junction temperature of the electronic component and the average wall temperature of the electronic enclosure obtained using analytical method is validated with that of obtained by carrying out Finite Element (FE) Analysis.

As the electronic enclosure is an imported unit theoretical analysis data is not available. The temperature is observed experimentally and found that it is higher than the allowable temperature. As no theoretical analysis data is available in order to estimate the reduction in maximum junction temperature with inclusion of aluminium sheet it is essential to start the estimation procedure from scratch. Hence an analytical method is established to estimate the maximum junction temperature of the electronic component.

C.P. Gupta,et al mentioned results of thermal analysis carried out using FEM for an electronic enclosure in paper titled “System level thermal analysis of OAS cabinet” presented in SDRC user conference, 2000. However they did not establish any analytical formulation in this paper.

In the second stage attention is focused on Finite Element Method (FEM). Suitable elements for carrying out thermal analysis are studied

The basic approach followed in this project is represented in form of flow chart below



As mentioned in the earlier chapter the maximum junction temperature of an electronic component positioned on a Printed Circuit Board (PCB) of an electronic enclosure resulted to be 112^0 C against the allowable temperature of 90^0 C. Design of an enhanced heat dissipation mechanism is taken up in this project with a view to reduce the temperature below 90^0 C. As a solution to this problem it is decided to split each PCB into two slices and introduce an aluminium sheet in between to enhance the heat dissipation, as the thermal conductivity of aluminium is more than material of PCB. Thermal design of this enhanced cooling mechanism is done using an analytical method. This method brings out the methodology to obtain the maximum junction temperature of the electronic component. This outcome of this method will be useful to ascertain that the maximum junction temperature is less than the allowable temperature. This chapter brings out the thermal design details

followed by the analytical method along with the junction temperature calculated using the method.
The two basic mechanisms through which heat transfer takes place in the electronic enclosure are as follows.

- Conduction
- Free convection

The elements of electronic enclosure from which heat transfer takes place are listed in Table 1

Sl. No.	Element of enclosure	Heat transfer
1.	Component to PCB	Conduction
2.	PCB to enclosure	
3.	Front, Rear, Side, Top and bottom walls to ambient	Convection
4.	Fins to ambient	

Table 1 Elements of heat transfer.

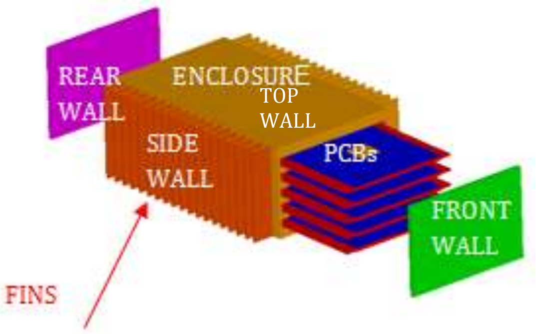


Fig 4 Elements of encloses.

The higher temperature observed in earlier design of the electronic enclosure is due to the reason that the thermal conductivity of material of PCB i.e. epoxy (0.346 W/m K) is much lesser. This caused the temperature of enclosure to exceed the allowable temperature in spite of fins provided to the electronic enclosure. In order to reduce the temperature heat dissipation path shall be enhanced. There are different ways to accomplish this viz.

- Increasing the number of copper layers in PCB
- Providing heat sinks to high heat dissipating components

First option is not feasible as the total electronic enclosure along with PCBs is an imported unit. And second option is also not feasible because heat dissipation by all components is identical and providing heat sink to one of component will not be helpful. Rather heat sink should be providing to whole of the PCB. Hence it is decided to sandwich a sheet with high thermal conductivity by slicing each PCB into two halves along thickness. In order to meet this requirement though high thermally conductive material like gold, silver and copper are not preferred due to high cost involvement.

As a cost effective solution aluminium is chosen, as its thermal conductivity is moderately high (200 W/m K). The intended aluminium sheet will be bonded to PCBs. A thickness of 3.125 mm is chosen for aluminium sheet. The basis for selection of thickness of aluminum sheet is lesser the thickness lesser will be the thermal conduction resistance and more will be the heat dissipation. Even then opting for thickness lesser than 3.125 mm makes bonding process complicated. Hence thickness of 3.125 mm is chosen which optimum thickness for bonding process is. Each aluminium sheet sandwiched between two halves of PCBs is here after will be referred as module. The PCBs with out and without aluminium sheet are shown in Figure 5 and 6 respectively.

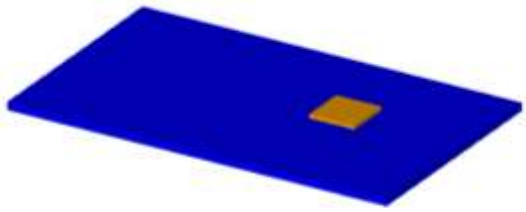


Fig 5 PCB without aluminium sheet.

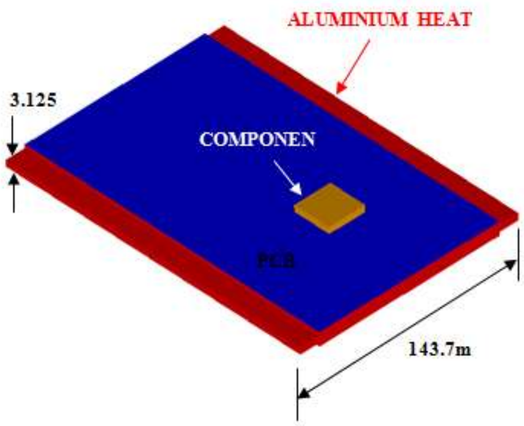


Fig 6 PCB with aluminium sheet.

As the electronic enclosure is an imported unit theoretical analysis data is not available. The temperature is observed experimentally and found that it is higher than the allowable temperature. As no theoretical analysis data is available in order to estimate the reduction in maximum junction temperature with inclusion of aluminium sheet it is essential to start the estimation procedure from scratch. Hence an analytical method is established to estimate the maximum junction temperature of the electronic component.

- The maximum junction temperature of electronic component should not exceed 90^0 C.

1.1 DESIGN INPUTS

- Heat load of each PCB is 20 W (Total heat load is 20 x 5 = 100 W

1.2 ANALYTICAL METHOD

Analytical method comprises of following three steps

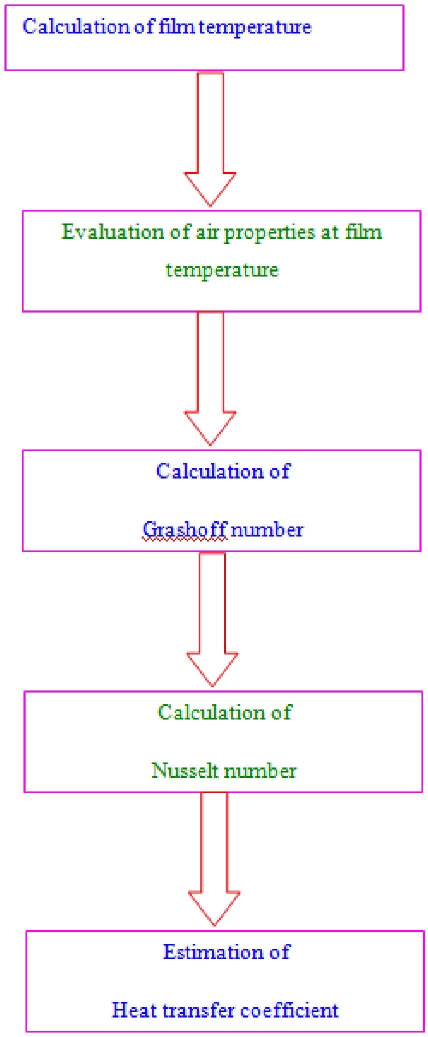
- Estimation of average wall temperature of enclosure
- Module temperature
- Maximum junction temperature of electronic component.

The proposed analytical method demands for assumption of average temperature of the enclosure. Taking assumed temperature of the enclosure as starting point the total heat load will be calculated and verified with actual total heat load of 100 W. If it results to be 100 W the assumed wall temperature of the enclosure will be the actual average temperature of the enclosure otherwise new value of the enclosure is to be assumed and the analytical method should be reiterated. This process shall continue till the heat load of enclosure results to be 100 W with the assumed enclosure temperature.

- The wall temperature of enclosure is assumed to be 63.25^0 C.
- An ambient temperature of 25^0 C is considered.

1.3 Estimation of Heat Transfer Coefficient – Front & Rear, Top Wall, Bottom Wall, Finned walls, Fin efficiency

As the front and rear walls are identical heat transfer coefficients will be same. Estimation procedure for heat transfer coefficient is represented in form of flow chart below.



1.4 Summary of heat transfer coefficients

Heat transfer coefficients obtained for all walls of the enclosure are summarized in Table 2

Sl. No.	Enclosure wall	Heat transfer coefficient
1.	Front wall	5.94 W/m^2 k
2.	Rear wall	5.94 W/m^2 k
3.	Top wall	5.34 W/m^2 k
4.	Bottom wall	2.53 W/m^2 k
5.	Side wall	5.65 W/m^2 k
6.	Fins	5.65 W/m^2 k

Table 2 Heat transfer coefficients.

1.5 Overall convection heat transfer

Number of fins, n = 19

A_s: Area of sides excluding fins = 2(0.25 – n x t_f) 0.15 = 0.057 m^2

$$Q_{total} = h A \Delta$$

A_t: Area of top wall = Area of bottom wall = 0.25 x 0.15 = 0.0375 m^2
A_{fr}: Area of front and rear walls = 2 x 0.15 x 0.15 = 0.045 m^2
A_f: Surface area of fins = 2 x n x L_c x p_f = 0.309 m^2

$$Q_{total} = [h_s A_s + h_t A_t + h_b A_b + h_{fr} A_{fr} + \eta_f h_f A_f](T_{chas} - T_a)$$

In convection environment total heat transfer can be expressed as follows
From aboveQ$_{total}$ = 100.42 W

As the estimated total heat transfer matched with actual heat load, assumed enclosure wall temperature is the actual wall temperature.
Hence wall temperature of enclosure = 63.25^0 C
Note: As shown in the above calculation always it is not necessary that in the first iteration itself the estimated total heat transfer matches with actual heat load. In this case initially enclosure temperature is assumed to be 50^0 C. Then the
iteration process was repeated until at the instance of wall temperature of enclosure = 63.25^0 C estimated total heat transfer matched with actual heat load. To avoid confusion calculation for iteration with this assumed temperature is only presented above

1.6 Estimation of Average Temperature of Module
Estimation of average temperature of module will be done by adding following three components to the average wall temperature of the enclosure.

- Local enclosure temperature rise
- Module to chassis interface temperature rise
- Module temperature rise

1.7 Estimation of local enclosure temperature rise
w: Width of enclosure rail (Module length) = 200 mm = 0.2 m
t: Thickness of enclosure rail = 12.5 mm = 0.0125 m
l: Length of enclosure rail = 12.5 mm = 0.0125 m
Local enclosure temperature rise = q_{module} x θ = 20 x 0.0125 = 0.25^0 C

1.8 Estimation of module to chassis interface temperature rise
A_c: Contact area = 2 x w x t = 2 x 0.2 x 0.0125 = 0.005 m^2
Thermal contact resistivity depends upon contact force, material and plating properties and use of thermal enhancement materials. Actual values are generally obtained from testing.
For the present case
Thermal contact resistivity = R_c = 0.00032 ^{0}Cm^2/W

$$\theta = \frac{R_c}{A_c} = \frac{0.00032}{0.005} = 0.0645 \ {}^0C/W$$

Module to enclosure interface temperature rise = q_{module} x θ = 20x0.0645=1.29^0 C

1.9 Estimation of Module temperature rise
P: Module power = 20 W
W: Width of aluminium sheet = 0.2 m
t: Thickness of aluminium sheet = 0.003125 m
l: Length of aluminium sheet = 0.14375 m
K_{al} = 200 W/mK
Module temperature rise can be expressed as

$$\Delta T_{ag} = \frac{Pl}{12K_{Al}wt} = 1.916 \ {}^0C$$

From all of above average module temperature can be expressed as follows
Average module temperature = Enclosure temperature + Local enclosure Temperature rise + Module to enclosure interface temperature rise + Module temperature rise
= 63.25 + 0.25 + 1.29 + 1.916 = 66.7^0 C
Average module temperature = 66.7^0 C
Estimation of maximum junction temperature of component will be done from following three thermal resistances.

- Thermal resistance to aluminium sheet
- Thermal contact resistance between component body and module
- Junction to case thermal resistance

2.0 Estimation of Thermal resistance to aluminium sheet

t_b: Thickness of bond = 0.00025 m
t_{pwb}: Thickness of bond & PCB = 0.0015625 m
Ac: Area of the component = 0.02375 x 0.02375 = 0.00056 m^2
K_{bond} = $K_{silicnerubber}$ = Thermal conductivity of bond = 0.224 W/mC = K_b
K_{PCB} = $K_{glassepoxy}$ = Thermal conductivity of PCB = 0.346 W/mC = K_{PCB}
Using above thermal resistance to aluminium sheet can be expressed as follows

$$\theta = \frac{t_b}{K_b \ x \ A_c} + \frac{t_{PCB}}{K_{PCB} \ x \ A_c} = 10.05 \ {}^0C/W$$

2.1 Estimation of Thermal contact resistance between component body and module
Thermal contact resistance between component body and the module is determined by dividing thermal contact resistivity parameters by the area of contact in order to determine the thermal resistance.
R_c = 0.000645 ^{0}Cm^2/W

$$\theta = \frac{R_c}{A_c} = \frac{0.000645}{0.00056} = 1.15 \ {}^0C/W$$

2.2 Junction to case thermal resistance

For the present case Junction to case thermal resistance is considered as

$\theta_{jc} = 5\ ^0C/W$

Heat load for the component for which maximum junction temperature is to be estimated is taken as 1 W.

Using above maximum junction temperature of the component can be expressed as

Maximum junction temperature = Average module temperature + $[\theta_{jc}+\theta_l+\theta_c]$ x 1 W

= 66.7 + [5 + 10.05 + 1.15] x 1 = 83^0 C

Maximum junction temperature = 83^0 C

2.3 OBSERVATIONS

- The maximum junction temperature of the component is 83^0 C, which is less than the allowable temperature of 90^0 C.
- By providing aluminium sheet to the PCBs temperature reduction from 112^0 C to 83^0 C is achieved.

2.4 FE modeling

Geometry of the electronic enclosure is built up in 3-D CAD software. All the subsystems like 5 PCBs and components are considered for modeling. Geometric model of the electronic enclosure in exploded view is shown in Figure 7

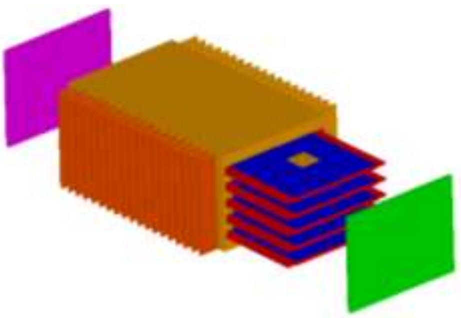


Fig 7 Geometric model of the electronic enclosure**.**

2.5 Physical properties

The wall thicknesses of walls of electronic enclosure, PCBs, fins and aluminium sheet given in Table 3 are defined as physical properties to the FE model

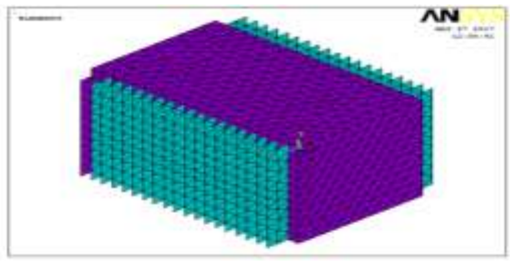


Figure 8 FE model of the electronic enclosure.

Sl. No.	Part	Thickness
1.	Walls of electronic enclosure	3 mm
2.	PCbs	1.5625 mm
3.	Aluminum sheet	3.125 mm
4.	Fins	3.125 mm

Table 3 Physical properties.

2.6 Material properties

For carrying out steady state thermal analysis only thermal conductivity is required as material property. Thermal conductivity values of materials of all subsystems are given in Table 4

Sl. No.	Sub system	Material	Thermal Conductivity
1.	Wallsof enclosure	Aluminium	200 W/mK
2.	Sheet		
3.	Fins		

Table 4 Material properties.

2.7 Boundary conditions

- Heat load of 20 W is applied on each PCB.
- The convection heat transfer coefficient calculated for all the walls along with fins using the analytical method are applied to the FE model.
- An ambient temperature of 25^0 C is applied.

2.8 Thermal Analysis

The FE model is solved for temperature distribution. Maximum wall temperature is observed to be 82.4^0 C. The temperature distribution plot is shown in Fig 9

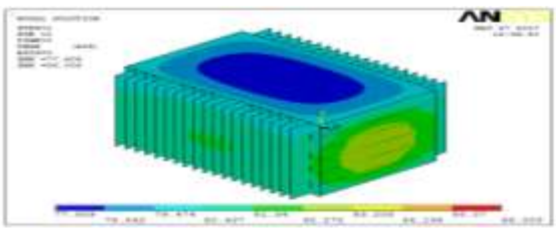


Fig 9 Temperature distribution of electronic enclosure.

In order to show that maximum temperature is developed on component on PCB the above temperature distribution plot discarding front wall is shown in Figure 10

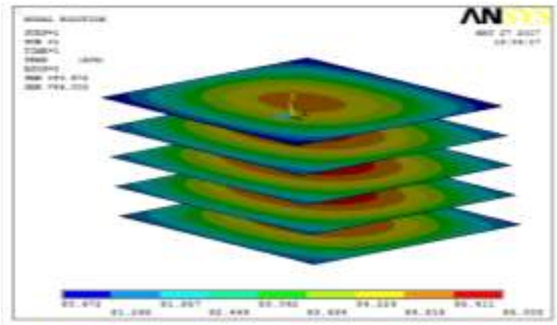


Fig 10 Temperature distributions of PCBs

2.9 Observations

The maximum junction temperature of component on PCB is observed to be 86^0 C.

3.0 RESULTS AND CONCLUSIONS

The results obtained using analytical method is compared with that of FE method in Table 4

Sl. No.	Temperature	Analytical method	FEM
1.	Maximum junction temperatureofcomponent	**83^0 C**	**86^0 C**

Table 4 result and conclusion

- The temperature estimated using analytical method is in good agreement with that of obtained from FEM analysis.

3.1 Cost analysis

Free convection		Forced convection	
Material	Cost	material	cost
Aluminum plate(without fins) PCB s are inserted b/w aluminum plate	500	Blower fan	3800
Aluminum plate(with fins)	350		
Total cost	850	Total cost	3800

Table 5 cost analysis.

- Without aluminum plate temperature in housing=112^0C
- With aluminum plate temperature in housing =83^0C (by analytical method)
- Using FEM analysis reducing temperature =86^0C

- The degree of closeness indicates the confidence of the analytical method.
- With the proposed aluminium sheet remarkable temperature reduction i.e. from 112^0 C to 86^0 C is achieved.
- There by the design solution recommended in this project work the primary goal is reached.
- Additional advantage gained is during the process of estimating temperature an analytical method is established which is validated with FE method.
- Hence any thermal designer can adopt this analytical method as a hand calculator and quickly arrive at the solution.
- Moreover the proposed solution (Aluminum sheet) can be adopted for any kind of heat dissipation requirements at low cost.
- The outcome of the project is a low cost thermal solution for achieve temperatures with a view to protect electronic components from harsh thermal environments.

BIOGRAPHIES

International Journal of Engineering Research and Applications (IJERA) is **UGC approved** Journal with Sl. No. 4525, Journal no. 47088. Indexed in Cross Ref, Index Copernicus (ICV 80.82), NASA, Ads, Researcher Id Thomson Reuters, DOAJ.

N.Venkatesh. "Thermal Design and Analysis of Mechanical Housing Using Ansys ." International Journal of Engineering Research and Applications (IJERA) , vol. 7, no. 9, 2017, pp. 77-83.