International Journal of
Scientific Engineering and Technology Research

www.ijsetr.com

ISSN 2319-8885
Vol.06,Issue.14
April-2017,
Pages:2759-2764

Design and Thermal Analysis of Fins

G. LAKSHMI SRINIVAS[1], PARAYIL HARISH[2], M. SUKENDER REDDY[3], MANI TEJA[4]
[1]Assistant Professor, Dept of Mechanical, Vignan Institute of Technology and Science, Deshmukhi, TS, India,
Email: g.l.srinivas7@gmail.com.
[2]B.Tech Scholar, Dept of Mechanical, Vignan Institute of Technology and Science, Deshmukhi, TS, India.
[3]B.Tech Scholar, Dept of Mechanical, Vignan Institute of Technology and Science, Deshmukhi, TS, India.
[4]B.Tech Scholar, Dept of Mechanical, Vignan Institute of Technology and Science, Deshmukhi, TS, India.

Abstract: Electronic equipment with high power rating generates heat causing temperature rise of the system and if the heat is not dissipated may cause overheating problem leading to failure of system. Design of cooling system with adequate heat dissipation is essential for reliable performance of system. Present project is focused on designing a heat sink for electronic printed circuit board with high wattage. Various methods of heat dissipation have been studied and fins with varying geometry, material and thickness have been analyzed. Steady State thermal analysis determines temperatures and other thermal quantities that do not vary over time. The variation of temperature distribution over time is of interest in many applications such as in cooling. The accurate thermal simulation could permit critical design parameters to be identified for improved life. Optimum design of heat exchanger will be suggested based on the analysis results.

Keywords: Electronic Cooling; Heat Dissipation; Thermal Conductivity; Steady State.

I. INTRODUCTION

Today, almost everybody is familiar with fighter aircraft, battle tanks, warships and submarines. A majority of people have seen them in action, either directly or via television or films. But there is another kind of invisible fight involving the use of radio and radar emissions which is always going on in the atmosphere. This silent battle of beams is commonly called Electronic Warfare. The battlefield of electronic warfare is global and its intensity varies according to different national perceptions of potential threats. In fact, electronic warfare is a catalyst towards the maintenance of regional and global balances which deter the outbreak of armed conflict. Most of the LRU today are of IP55 specification. The IEC's standardized International (or Ingress) Protection Code system is a standard for measuring enclosures capabilities. The IP Code reflects the degree of protection as "IP" followed by two numbers; the first digit shows the extent to which enclosures are protected against particles, and protection to others from enclosed hazards. The second digit indicates the extent of protection against water. We carry IP55 enclosures that demonstrate almost complete protection from particles and a good level of protection against water.

IP 55 Enclosure Characteristics:

- Protection from dirt, dust, oil, and other non-corrosive material
- Complete protection from contact with enclosed equipment
- Protection from water, up to water projected by a nozzle against enclosure from any direction
- Available in aluminum, carbon steel, and stainless steel
- Available in wall-mounted, free standing, trough, and JIC box
- Engraving, silk-screening, or anodizing services available
- Custom with cut-outs, insulation, hinges, latches, or locks.

Advancements in semiconductor technology have led to the significant increase in power densities encountered in microelectronic equipment. As the amount of heat that needs to be dissipated from electronic devices constantly increases, the thermal management becomes a more and more important element of electronic product design. Operating temperature within the limits set by the device design engineers. With the increase in heat dissipation from the electronic devices and the reduction in overall form factors, it became an essential practice to optimize heat sink designs with least trade-offs in material and Manufacturing costs.

II. THEORETICAL BACKGROUND

The primary function of thermal analysis is to predict the temperatures of components and parts within a product. By visualizing heat fluxes, thermal bottlenecks, and missed shortcut opportunities, thermal analysis seeks to eliminate any detected thermal compliance issues. For this purpose, CFD simulation has been used to design efficient heat removal paths for electronics devices at package, board, and system levels. There are three main stages of Computational Analysis, they are:

1. Pre processing
2. Solving
3. Post processing

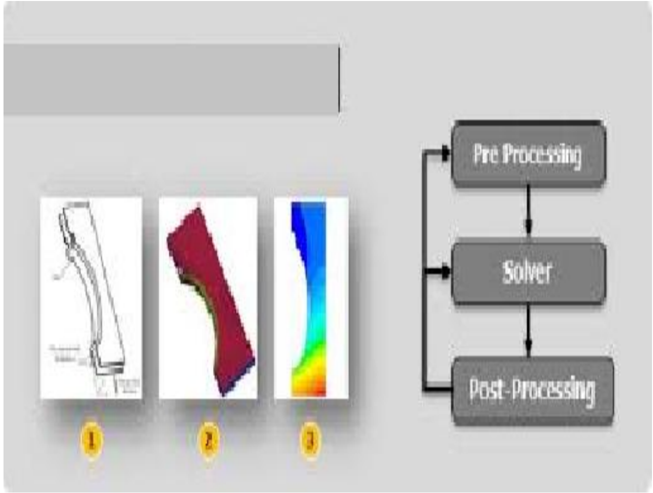

Fig 1 Steps in Computational Analysis.

A. Pre Processing: Finite element analysis is comprised of pre-processing, solution and post-processing phases. The goals of pre-processing are to develop an appropriate finite element mesh, assign suitable material properties, and apply boundary conditions in the form of restraints and loads.

The finite element mesh subdivides the geometry into elements, upon which are found nodes. The nodes, which are really just point locations in space, are generally located at the element corners and perhaps near each mid side. For a two-dimensional (2D) analysis, or a three-dimensional (3D) thin shell analysis, the elements are essentially 2D, but may be "warped" slightly to conform to a 3D surface. An example is the thin shell linear quadrilateral; thin shell implies essentially classical shell theory, linear defines the interpolation of mathematical quantities across the element, and quadrilateral describes the geometry. Developing the mesh is usually the most time-consuming task in FEA. In the past, node locations were keyed in manually to approximate the geometry. The more modern approach is to develop the mesh directly on the CAD geometry, which will be (1) wireframe, with points and curves representing edges, (2) surfaced, with surfaces defining boundaries, or (3) solid, defining where the material is. Solid geometry is preferred, but often a surfacing package can create a complex blend that a solids package will not handle. As far as geometric detail, an underlying rule of FEA is to "model what is there", and yet simplifying assumptions simply must be applied to avoid huge models. Analyst experience is of the essence. The geometry is meshed with a mapping algorithm or an automatic free-meshing algorithm. The first maps a rectangular grid onto a geometric region, which must therefore have the correct number of sides. Mapped meshes can use the accurate and cheap solid linear brick 3D element, but can be very time-consuming, if not impossible, to apply to complex geometries. Free-meshing automatically subdivides meshing regions into elements, with the advantages of fast meshing, easy mesh-size transitioning (for a denser mesh in regions of large gradient), and adaptive capabilities. Disadvantages include generation of huge models, generation of distorted elements, and, in 3D, and the use of the rather expensive solid parabolic tetrahedral element. It is always important to check elemental distortion prior to solution. Material properties required vary with the type of solution. A linear statics analysis, for example, will require an elastic modulus, Poisson's ratio and perhaps a density for each material. Thermal properties are required for a thermal analysis. Examples of restraints are declaring a nodal translation or temperature. Loads include forces, pressures and heat flux. It is preferable to apply boundary conditions to the CAD geometry, with the FEA package transferring them to the underlying model, to allow for simpler application of adaptive and optimization algorithms. It is worth noting that the largest error in the entire process is often in the boundary conditions. Running multiple cases as a sensitivity analysis may be requiring.

B. Solving: While the pre-processing and post-processing phases of the finite element method are interactive and time-consuming for the analyst, the solution is often a batch process, and is demanding of computer resource. The governing equations are assembled into matrix form and are solved numerically. The assembly process depends not only on the type of analysis (e.g. static or dynamic), but also on the model's element types and properties, material properties and boundary conditions. The first result of a solver ship always contains the shifts of the individual nodes. In subsequent steps, distortions, stresses and knot forces can then be calculated. In addition to mechanical variables, thermal, electrical or magnetic variables can also be calculated if a thermal or electromagnetic solver is used accordingly. At the end of a solving run, the results are passed on to the postprocessor of the FE program

C. Post Processing: After a finite element model has been prepared and checked, boundary conditions have been applied, and the model has been solved, it is time to investigate the results of the analysis. This activity is known as the post-processing phase of the finite element method. The magnitude of principal stresses or of a scalar failure stress such as the Von Mises stress may be displayed on the model as colored bands. When this type of display is treated as a 3D object subjected to light sources, the resulting image is known as a shaded image stress plot. Displacement magnitude may also be displayed by colored bands, but this can lead to misinterpretation as a stress plot**.**

III. DESIGN OF FINS

Dimensions of Heat Source:

Heat Source Length = 0.323m
Heat Source width = 0.008m
Heat Source thickness = 0.120 m
Heat Sink width = 0.323 m
Heat Sink thickness = 0.064 m

Heat Source area A_S = $0.002584 m^2$
Heat Sink area A_b = $0.020672 m^2$

Thermal Input parameters:

Convective Heat Transfer coefficient, h = $5 W/m^2 * K$
External Resistance, R_0 = 1.25
Thermal Conductivity, k = 176 W/m-K
Time variation = Steady State
Convection = Natural
Power Input (RFSM) = 27 W
Assignment of heat source = Volumetric
Ambient condition = 55°C

Design of Fin Calculation:

$a = (A_s / \pi)^{0.5}$ = 0.02867 m
$b = (A_b / \pi)^{0.5}$ = 0.08111 m
$\epsilon = a/b$ (dimensionless heat source radius) = 0.3525
$\tau = t/b$ (dimensionless plate thickness) = 0.3081
Biot Number= $1/ (\pi * k * b * R_0)$ = 0.017
(Effective Biot Number)
Biot Number = $(h*b)/k$ = 0.0023
$\lambda_c = \pi + (1/ (\pi^{0.5} * \epsilon))$ = 4.737

Analytical solutions for spreading resistance for the configuration have recently been developed by the present authors, and have been submitted for publication elsewhere. These solutions are in the form of an infinite series with special functions, and require computation of a few hundred terms. Using the analytical solutions, the present authors have developed approximate co relation, and they are presented here:

$\Phi_c = (\tanh (\lambda_c * \tau) + (\lambda c /Bi))/ (1+ (\lambda_c /Bi)*\tanh (\lambda_c * \tau))$
= 1.113
Dimensionless resistance, Ψavg =
$(\epsilon * \tau / (\pi)^{0.5}) + 1/2*(1- \epsilon)^{3/2} * \Phi_c$ = 0.350
$\Psi max = (\epsilon * \tau / (\pi)) + 1/ (\pi)^{0.5} * (1- \epsilon) * \Phi_c$ = 0.467
$R_{spread\ average} = \Psi avg / (\pi)^{0.5} * k * a$ = 0.039
$R_{spread\ max} = \Psi max / (\pi)^{0.5} * k * a$ = 0.052
$R_{Component} = t/(k*A_b)$ = 0.032
$R_{cond} = t/ (k * A_b)$ = 0.006
Total Power, Q = 36Watts
$T_{fin\ base}$ = 70^oC
$T_{ambiance}$ = 55^oC
$\Delta T = T_{fin\ base} - T_{ambiance}$ = 15^oC
Fins Area Required, A_s = $0.6162 m^2$
Temperature at fin base, T1 = 70^oC
Room Temperature, T2 = 55^oC

Once the Fin Area required is known the fin is designed according to the considered space constraints. To find out the fin dimensions, five parameters are required namely Length, Height, Thickness, Spacing, Number of fins and Height of Fin. Out of these Any three values have to be assumed i.e. have to be taken from a data book. To arrive at the best Fin design three cases are considered where the three fixed parameters are varied and the best is selected using trial and error method.

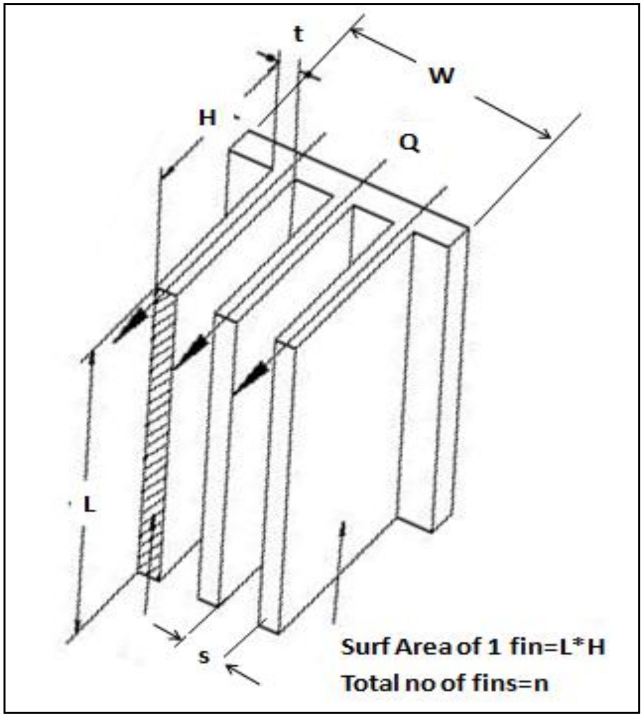

Fig 2. Fin Dimensional Specification

Case 1:
Fin Length, L = 370 mm
Fin Width, W = 64 mm
Fin thickness, t = 2 mm
Gap/Spacing between 2 fin = 4 mm
Number of fins, N = (W-t)/ (t+s) = 10.33333
Number of fins round off to higher, N = 10
Fin Height, H = 78.888mm

Case 2:
Fin Length, L = 150 mm
Fin Width, W = 220 mm
Fin thickness, t = 8 mm
Gap/Spacing between 2 fin = 10 mm
Number of fins, N = (W-t)/ (t+s) = 11.77777
Number of fins round off to higher, N = 12
Fin Height, H = 102.5mm

Case 3:
Fin Length, L = 150 mm
Fin Width, W = 220 mm
Fin thickness, t = 2 mm
Gap/Spacing between 2 fin = 8 mm
Number of fins, N = (W-t)/ (t+s) = 21.88888
Number of fins round off to higher, N = 22
Fin Height, H = 54.584m

IV. ANALYTICAL MAXIMUM TEMPERATURE OF THE SWITCH BOARD

The Formula used to find out T_{max} is

$$T_j = T_A + R_{\emptyset jA} \times P_D$$

Where

T_j = Junction temperature

T_A = Ambient temperature

$R_{\emptyset jA}$= Thermal Resistance Junction to ambient

P_D= Power Dissipated

Here T_A= 55, $R_{\emptyset jA}$= 0.85, P_D= 27 Watts

T_j = 77.95°C

The $R_{\emptyset jA}$ values are taken from the JS5555 Standards.

Similarly the analytical T_{max} for the three cases of Switch Board Matrix with Fins is calculated and the results are shown below:

Table 1. Analytical Tmax Comparison

Case No.	Case 1	Case 2	Case 3
T_{max}(°C)	62.02	62.18	64.008

V. THERMAL ANALYSIS OF THE SWITCH BOARD FOR MAXIMUM TEMPERATURE

Once the Fin dimensions are known considering the given input parameters and thermal considerations, the fin is designed in CAD software and then necessary thermal analysis is done to verify whether the obtained design serves the purpose.

WALL WITHOUT FINS

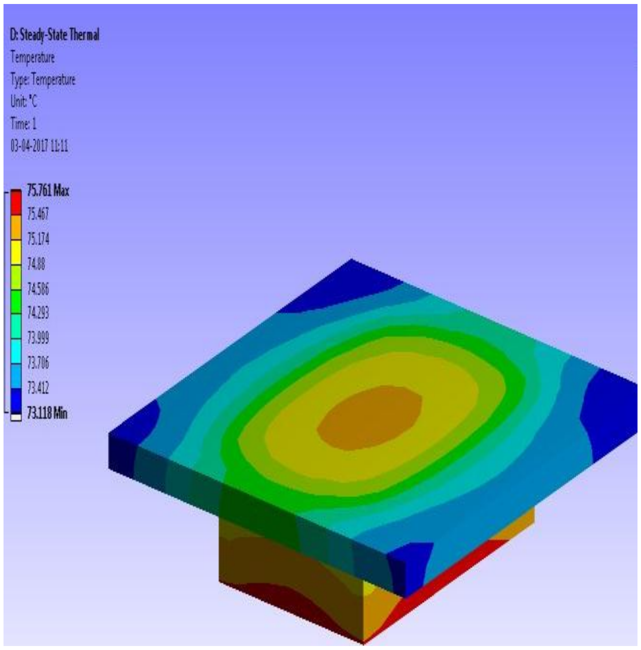

Fig 3. Switch Board Matrix and the wall without Fins

In this case only the Switch Board Matrix and the Enclosing wall have been considered. A heat of about 27 Watts is been generated in the Switch board Matrix and that information is fed into ANSYS. Once the analysis is completed it can be seen that the temperature of the SBM varies between 73.1°C to 76.8 °C, thereby producing a maximum of 7°C more than the specified limit (70°C).

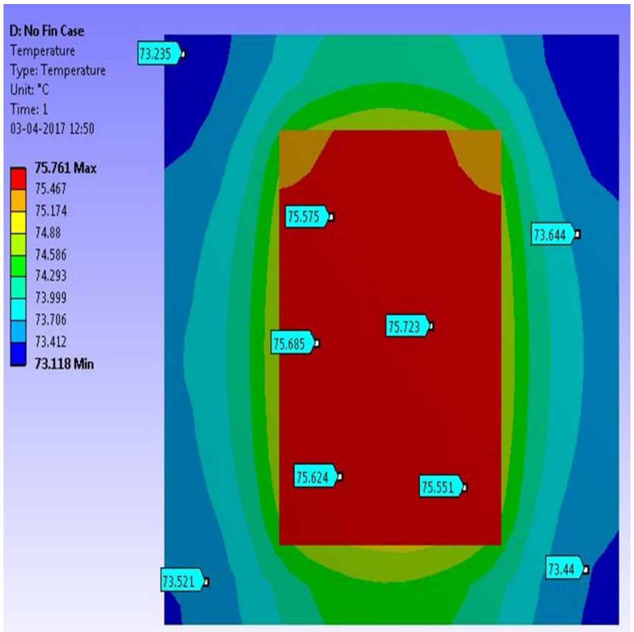

Fig 4. Switch Board Matrix and the wall without Fins Tmax Location

The above analysis shows the maximum temperature location on the Switch Board Matrix. As mentioned the temperature has to be below 70°C which isn't the case here.

RECTANGULAR FINS:

Using Analytical formulae three different dimensions of rectangular fins have been designed. The four dimensional parameters namely Length, Width, Thickness and Spacing have been varied and the Maximum temperatures of the Switch Board Matrix have been compared.

Case 1

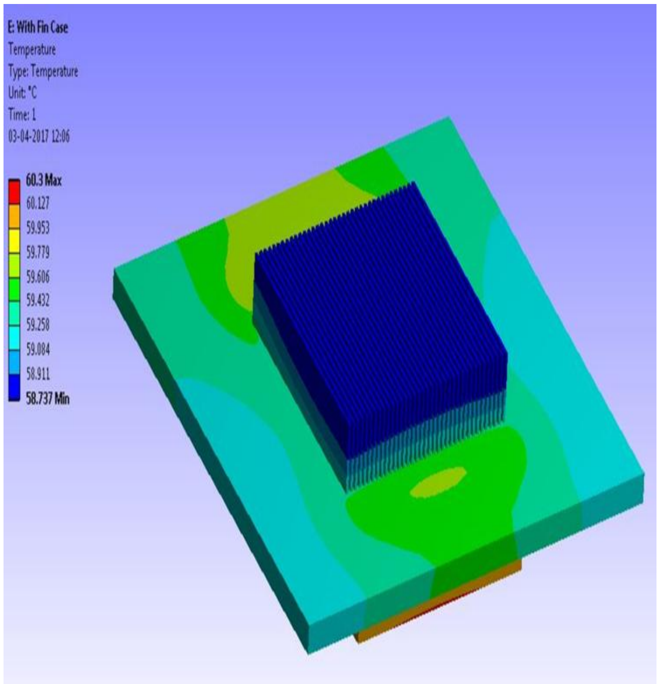

Fig 5. Rectangular Fin Case 1.

The fin dimensions which have been obtained in the **Case 1** are used in designing the fin in the CAD software and analysis is performed in ANSYS considering the a heat source of 27 Watts with the Fins attached to the Switch Board Matrix and the enclosure. The temperature varies between 58.737°C and 60.3°C which is well below the specified limit of 70°C

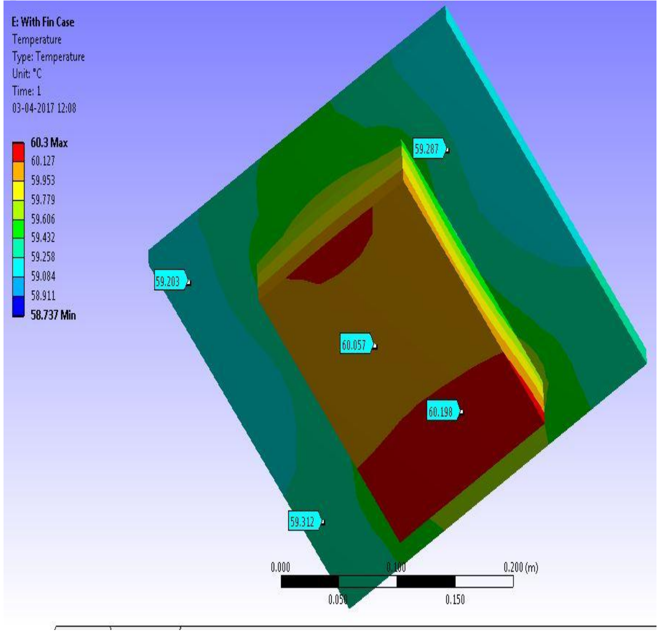

Fig 6. Rectangular Fin Case 1 Tmax location.

The above analysis shows the maximum temperature location on the Switch Board Matrix. As mentioned the temperature has to be below 70°C and in this case the condition is satisfied.

Case 2:

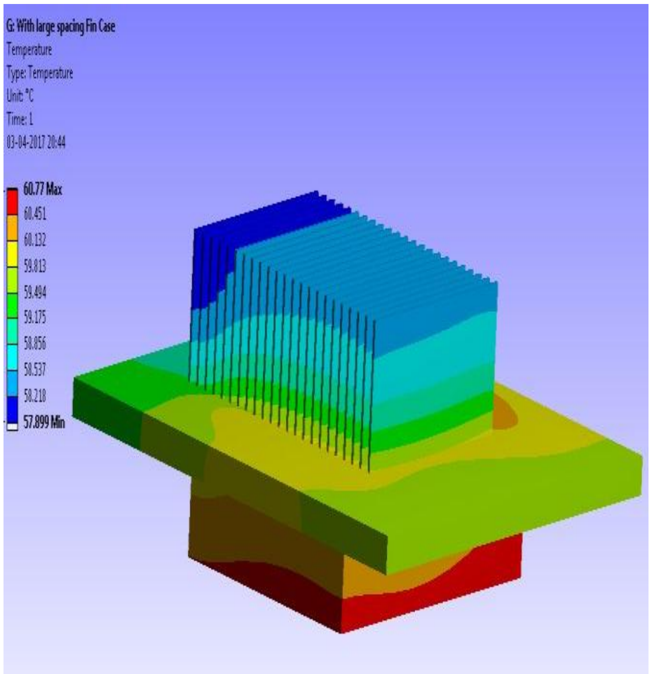

Fig 7. Rectangular Fin Case 2 Tmax

The fin dimensions which have been obtained in the **Case 2** are used in designing the fin in the CAD software and analysis is performed in ANSYS considering the a heat source of 27 Watts with the Fins attached to the Switch Board Matrix and the enclosure. The temperature varies between 57.899°C and 60.77°C which is well below the specified limit of 70°C

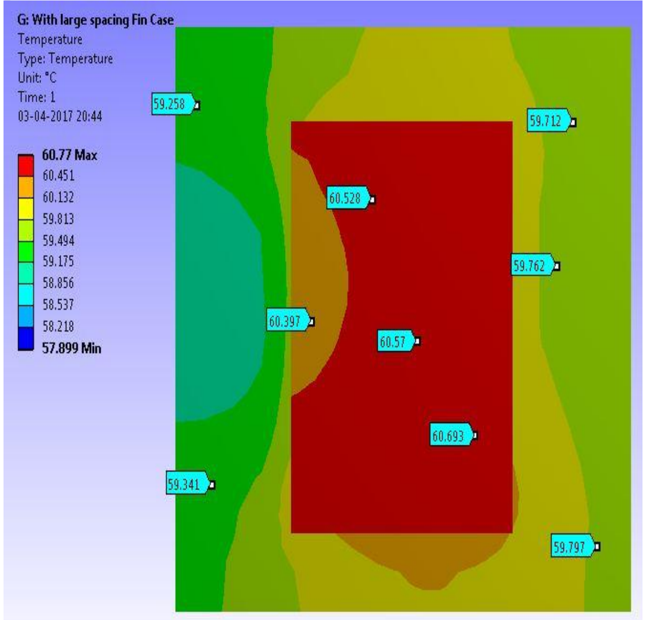

Fig 8. Rectangular Fin Case 2 Tmax

The above analysis shows the maximum temperature location on the Switch Board Matrix. As mentioned the temperature has to be below 70°C and in this case the condition is satisfied.

Case 3:

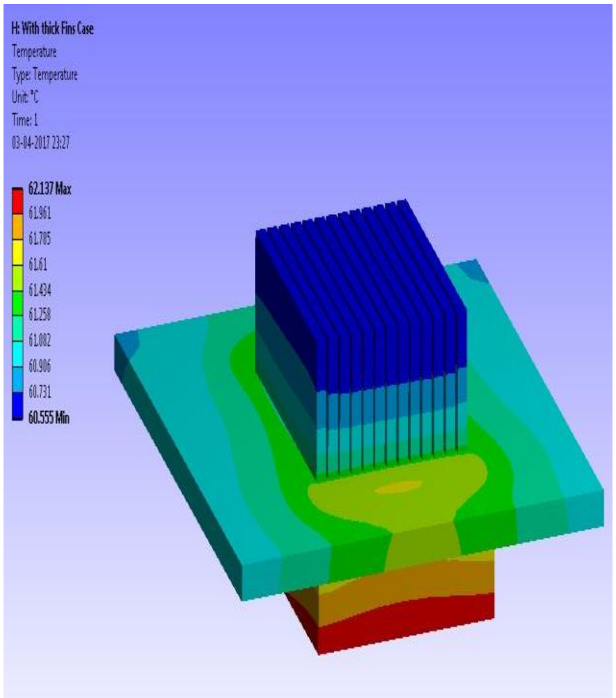

Fig 9. Rectangular Fin Case 3

The fin dimensions which have been obtained in the **Case 3**are used in designing the fin in the CAD software and analysis is performed in ANSYS considering the a heat source of 27 Watts with the Fins attached to the Switch Board Matrix and the enclosure. The temperature varies between 60.555^{o}C and 62.137^{o}C which is well below the specified limit of 70^{o}C

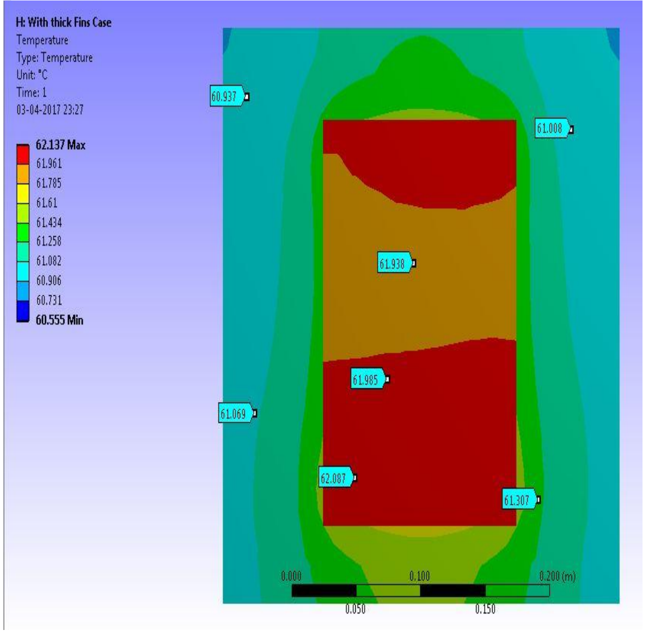

Fig 10. Rectangular Fin Case 3 Tmax

The above analysis shows the maximum temperature location on the Switch Board Matrix. As mentioned the temperature has to be below 70^{o}C and in this case the condition is satisfied.

VI. COMPARISON OF ACHIEVED RESULTS

After performing thermal analysis on the obtained designs it is seen that the design of fin in **Case 1** maintains the temperature of the Switch Board Matrix below or at 60.3^{o}C which is well below the limit.

Table 2. Comparisons between Analytical and Computational Design

Condition	With out Fins	Fins Case 1	Fins Case 2	Fins Case 3
T_{max}(^{o}C) Analytical	77.9	62.0	62.2	64.0
T_{max}(^{o}C) Thermal Analysis	75.7	60.3	60.8	62.1

Since there is negligible discrepancy between the analytical and computational results it can be said that the obtained results are optimum and reliable.

VII. CONCLUSION

Design of fins for the required problem statement i.e. 'for a Switch Board Matrix in a Replacement Line Unit which emits 27 Watts such that the temperature of the Matrix is maintained below 70^{0}C is done. Three cases of variation in fin geometry are considered and the fin design which results in lowest maximum temperature of the Switch Board Matrix is found out. Comparison between the Analytical and Computational results is done and it is found out that the difference between them is less.

VIII. REFERENCES

[1] Lee, S., Optimum De sign and Selection of Heat Sinks, Proceedings, 11th IEEE Semi-Therm Symposium, San Jose, Cal., USA, 1995, pp. 48-54.

[2] Kordian T., (1998), Hot air rises and heat sinks – Everything you know about cooling electronics is wrong, ASME Press, New York, USA.

[3] Azar, K., McLeod, R. S., Caron, R. E., Nar row Channel Heat Sink for Cooling of High Powered Electronic Components, Proceed ings, 8th Annual IEEE SemiTherm Symposium, Aus tin, Tex., USA, 1992, pp. 12-19.

[4] Finite Element Analysis and Experimental Study of Convective Heat Transfer Augmentation from Horizontal Rectangular Fin by Triangular Perforations By Kumbhar D.G. ,Dr. N K Sane, Chavan S.T:[International Conference on Advances in Mechanical Engineering, SV National Institute of Technology, Gujarat, India, August 3-5, 2009.