IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
ISSN: 2278-1684 Volume 4, Issue 5 (Nov. - Dec. 2012), PP 41-46
www.iosrjournals.org

Thermal Design and Analysis of Electronic Enclosure of Ground Radar

Anjaiah Madarapu [1] Murali Mohan Raju.T [2] M.Manzoor Husain [3]
[1,2]Associate Professor, Mechanical Engineering Department, School of Engineering & Technology, Guru Nanak Institutions Technical Campus, Hyderabad, A.P., INDIA.
[3]Professor, Mechanical Engineering Department, JNT University, Hyderabad, A.P., INDIA

Abstract: *When an electronic component is powered heat will be developed which will in turn increases the component temperature. For safe functioning of these components manufacturer specifies allowable temperature. If the component temperature exceeds these limits component fails. To get rid of this situation heat dissipation mechanism must be effective. Thermal design of an enclosure for PCBs of a ground RADAR is taken up in this work. The primary objective is to enhance the heat dissipation rate with the proposed enclosure in forced convection environment. Selection of fan and estimation of cooling rate is also be done. The complexity associated with this requirement is to estimate the heat transfer coefficient for the proposed enclosure in forced convection environment. This is due to the reason that the standard literature limits its scope to presentation of empirical correlations for heat transfer coefficients for standard geometries like plate, pipe, etc. Where as in the current requirement heat transfer coefficient is to be estimated for the enclosure with complex geometry. As part of the work an analytical method is established to estimate the heat transfer coefficient and the same is validated with commercial CFD software package. After validation process maximum component temperature is calculated in order to ascertain the cooling effectiveness of the proposed thermal design by ensuring that maximum temperature is within the prescribed limits. Apart from meeting the primary objective i.e. providing suitable heat dissipation mechanism and ensuring component temperature within limits another intended outcome of the work is an analytical method to calculate heat transfer coefficient for electronic enclosures quickly without depending on commercial CFD software.*

I. Introduction

The effect of the heat developed is raising the temperature of the component mounted on PCB. Each component will have allowable temperature specified by the manufacturer crossing which will result into either malfunctioning of the component or permanent failure of the component. As the total reliability of the electronic enclosure is highly dictated by the component, a cooling mechanism is essentially required for an electronic enclosure. Compared to other modes of heat transfer, forced convection means will provide faster cooling rates. This work deals with designing a cooling scheme for an electronic enclosure of a RADAR in forced convection environment. In the whole exercise estimation of heat transfer coefficient is a tedious task as standard literature mentions about standard geometries like plate, cylinder etc but not about this kind complex geometries like enclosures with PCBs. An analytical method is established during this work in which an expression for heat transfer coefficient is derived. This expression will be validated with that of evaluated using commercial CFD software package. The outcome of this work would be an analytical method which any thermal designer can make use of as a hand calculator to quickly arrive at the cooling scheme with out depending on CFD software. The solid model of the electronic enclosure in assembled configuration is shown in Fig. 1.

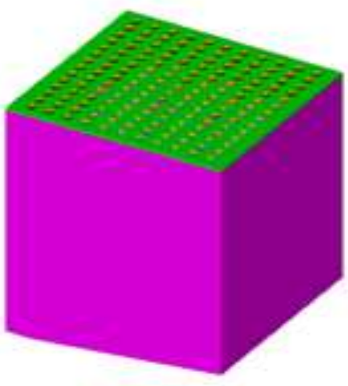

Fig. 1.Solid model of the electronic enclosure

The solid model of the electronic enclosure in exploded view is shown in Fig. 2.

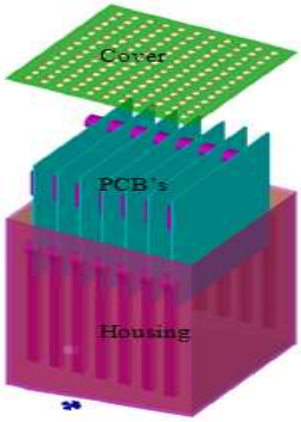

Fig. 2. Exploded view of the electronic enclosure

II. Design Inputs

- Number of PCBs = 7
- Heat load is 280 W (7 PCBs 40 x 7 = 280 W)
- Air inlet temperature = 40^0 C

III. Design Constraint

- Maximum junction temperature of electronic component should not exceed 100^0 C.
- Maximum allowable temperature raise from air inlet to component hot spot temperature = 100 – 40 = 60^0 C.

IV. Thermal Design Of Housing Of The Electronic Enclosure

As mentioned earlier the maximum junction temperature of an electronic component positioned on a Printed Circuit Board (PCB) of the electronic enclosure should be limited to 100^0 C. Design of an enclosure is taken up in this work with a view to keep the temperature below 100^0 C. As a part of the design process selection of fan is also accomplished. Thermal design of this enclosure is done using an analytical method. This method brings out the methodology to obtain the maximum junction temperature of the electronic component.
The cross section of the housing from air inlet point towards the last PCB (In air flow path) plays vital role in forced convection heat transfer. This cross section is nothing but the cross section of air duct. Two possible cross sections of air duct are shown in Fig. 3.

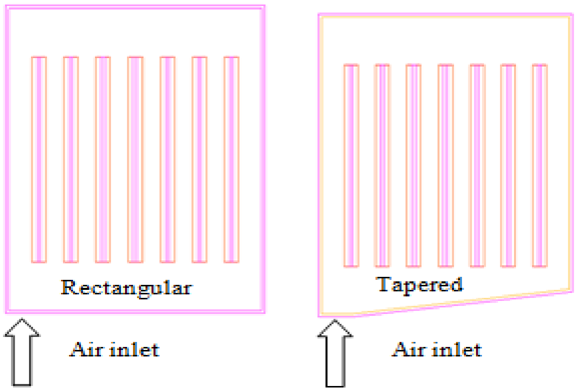

Fig.3. Two cross sections of air duct

Generally in confined ducts (Originated due to obstructions by PCBs) same air inlet velocity will not be maintained as airflow progresses. Due to this maximum benefit cannot be gained from the specified mass flow rate of the cooling air. Especially this problem arises in rectangular ducts due to which the cross section of the duct will be tapered from inlet towards outlet so that loss of air velocity can be minimized due to convergent nozzle effect. Hence housing with tapered cross section at the bottom (Where the fan will be mounted) is preferred for the proposed design.

The dimensions of the housing are worked out based on the PCB dimensions however keeping the dimensional constraints imposed by the user.
The dimensional model of the housing is shown in Fig. 4.

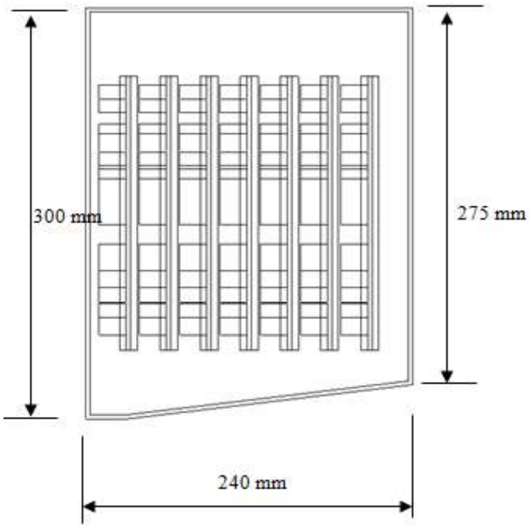

Fig. 4. Dimensional model of the housing

V. Analytical Method

Analytical method comprises estimation of two major contributors for maximum allowable temperature raise from air inlet to component hot spot temperature i.e. 60^0 C.

- ΔT_A due to heat input from fan and electronics to cooling air
- ΔT_B due to thermal resistance across the convection film from cooling air to component surface

Axial flow fan is selected as it suits well with present requirement.
Past experience with air-cooled electronic systems has shown that satisfactory thermal performance can be

$$\text{Weight velocity flow 'G' through the duct}$$

$$\text{formed by PCB, } G = \frac{W}{A\,n}$$

$$W : \text{Flow rate} = 10\text{ g/sec} = 0.01\text{ Kg/sec}$$

$$n : \text{Number of ducts} = 7$$

$$A : \text{Area} = 0.2x0.0025 = 0.0005\text{m}^2$$

$$G = \frac{0.01}{0.0005x7} = 2.857\,\frac{\text{Kg}}{\text{secm}^2}$$

Obtained if the cooling air exit temperature from the electronic housing does not exceed 71^0 C [1].
Hence the allowable cooling air temperature rise, $\Delta T_A = 71\text{-}40 = 31^0$ C.
From the heat capacity equation

$$Q = m\, c_p\, \Delta T \qquad (1)$$

Where Q: Total Heat load = 280 +25 = 305 W
m: Required cooling air flow rate
C_p: Specific heat = 1008 J/KgK
ΔT: Temperature difference = 31^0 C

From the above

$$m = \frac{Q}{C_p \Delta T} = \frac{305}{1005x31} = 0.097\text{ Kg/sec}$$

$$m = 0.097\text{ Kg/sec} = 9.7\text{ g/sec } \cong 10\text{ g/sec}$$

The temperature rise can be determined from the convection relation

$$J = 0.0094$$

$$h_C = 0.0094x1005x2.857[0.6969]^{\frac{-2}{3}}$$

$$h_C = 34.33\,\frac{W}{m^2K}$$

$$Q = h_c.A.\Delta T_B \qquad (2)$$

From which

$$\Delta T_B = \frac{Q}{h_c.A}$$

Where Q: Heat Load of PCB

A: Surface area

h_c: Heat transfer coefficient

$$Q = h_c.A.\Delta T_B$$

$$\Delta T_B = \frac{Q}{h_c.A}$$

$$Q = \text{Heat load of each PCB} = 40\text{ W}$$

$$h_c = 34.33\,\frac{W}{m^2K}$$

$$A = 1.3x0.2x0.225 = 0.0585\,m^2$$

$$\Delta T_B = \frac{40}{34.33x0.0585} = 19.91^0C$$

Heat transfer coefficient has to be determined from the geometry of the air duct i.e. formed by the air space between the PCBs using following expression [2].

Component hot spot temperature

$$T_s = T_{\text{in air}} + \Delta T_A + \Delta T_B$$

$$= 40 + 31 + 19.91 = 90.91^0\,C$$

$$h_C = J.C_p.G[P_r]^{\frac{-2}{3}} \qquad (3)$$

Where

J: Colburn factor

C_p: Specific heat of fluid

G: Weight velocity flow through duct

μ: Viscosity of the fluid

K: Thermal conductivity of the fluid

Colburn factor 'J' can be expressed in terms of Reynold's number (R_e) as follows.

$$J = \frac{6}{R_e^{0.98}} \qquad (4)$$

Reynold's number for the air duct formed by the PCBs can be expressed as follows.

Fluid mean temperature, T_f

$$= \frac{71+40}{2} = 55.5^0 C$$

$$\text{At } T_f = 55.5^0 C, \mu_{air} = 19.66x10^{-6} \frac{Kg}{m\,sec}$$

$$R_e = \frac{2.857 \text{ x } 0.005}{19.66x10^{-6}} = 726$$

$$R_e = \frac{G\,D}{\mu} \quad (5)$$

$$D : \text{Hydraulics diameter} = \frac{4x\text{Area}}{\text{Perimeter}}$$

a : Height of PCB = 200 mm = 0.2 m

d : Gap between PCBs = 2.5 mm = 0.0025 m

$$D = \frac{2x0.2x0.0025}{0.2+0.0025} = 0.005$$

As component hot spot surface temperature < 1000 C and hence design is satisfactory.

VI. Computational Fluid Dynamics (Cfd) Analysis

Geometry of the electronic enclosure is built up in 3-D CAD software [3]. All the subsystems like 5 PCBs and components are considered for modeling. Geometry model is converted into CFD model by discretizing the electronic enclosure, PCBs using linear three nodded triangular shell elements. Where as air surrounding the PCBs with in the enclosure are discretized using 4 nodded solid linear tetrahedron elements. CFD model of the electronic enclosure is shown in Fig. 5.

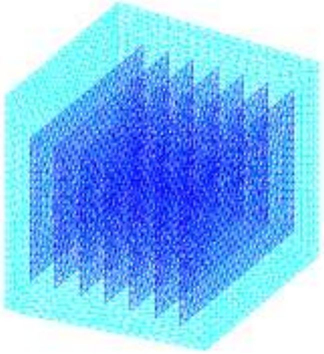

Fig. 5. CFD model of the electronic enclosure

For carrying out steady state CFD analysis thermal conductivity is required as material property. Rest of other material properties will be in built in the software. Thermal conductivity values of materials of all subsystems are given in Table 1.

Sl. No.	**Sub system**	**Material**	**Thermal Conductivity**
1.	Walls	Aluminum	200 W/mK
2.	PCB	Epoxy	18 W/m K
3.	Air		0.0263 W/m K

Table 1 Material properties

- Heat load of 40 W is applied on each PCB.
- An ambient temperature of 45^0 C is applied.
- An inlet fan is created in the FE mode with mass flow rate of 10 gm/Sec.
- Vent (Opening for air exit to ambient) is defined on perforated holes provided on the cover.
- Flow surfaces are defined on the front and rear faces of PCBs.

The CFD model is solved for temperature distribution. In order to show that maximum temperature is developed on component on PCB the temperature distribution plot-discarding housing is shown in Fig. 6.

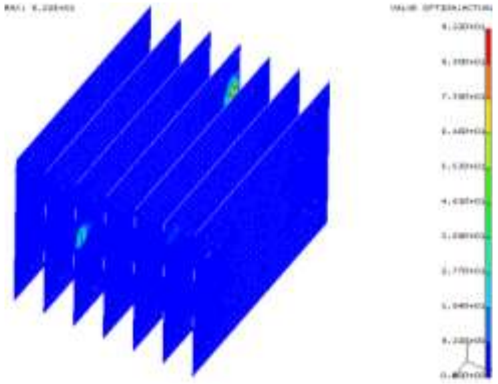

Fig. 6. Temperature distribution of PCBs

The maximum junction temperature of component on PCB is observed to be 92.2^0 C.

VII. Results And Discussion

The maximum component temperature estimated using analytical method is compared with that of obtained from CFD in Table 2.

	Analytical method	CFD
Maximum component temperature on PCB	90.9^0 C	92.2^0 C

Table 2 Comparison of results

VIII. Conclusion

Maximum junction temperature estimated using analytical method is matching well with that of obtained using CFD. The degree of closeness established indicates the confidence of the analytical method. Based on this validation study this analytical method can be comfortably used as a digital calculator to quickly accomplish the thermal design task of electronic enclosures.

References

[1] Colburn, A. P., “A method for correlating forced convection heat transfer data”, Trans. A. I. Ch. E. 29,174.

[2] Whitaker S., “Forced convection heat transfer correlations for flow in pipes, flat plates, single cylinders, spheres and for flow in packed beds”, AI Ch. E. No. 2, 361.

[3] ESC manual.