Thermal Design and Analysis of Telecom Board

#1Rahul R. Sarode, #2Prof. A. M. Elgandelwar

#1Mechanical Engineering Department, MIT Pune, Pune University, Pune, India
#2Mechanical Engineering Department, MIT Pune, Pune University, Pune, India

Abstract— Thermal transport from device's core, to the ambient environment is highly three dimensional. Heat generated at the die is removed through several paths and so the total resistance inside the device package is divided into two part between case and junction and Printed Circuit Board (PCB) and junction. These resistance value are under control of device manufacturer. Additional layers of resistivity are introduced between the device case and environment. This is where thermal solution resistance comes into the picture. That resistance consists of thermal interface material and thermal resistance of heat sink itself. For these complete modelling of the PCB, thermal coupling between the devices, and from PCB to PCB must be accounted for. For these analysis of telecom board, ANSYS ICEPAK Computational Fluid Dynamics (CFD) is used.

Keywords: PCB, Junction, Thermal Resistance, CFD

I. INTRODUCTION

High reliability and availability of electronic equipment belong to the major requirements for almost all applications. This is especially true for telecom systems which often are used under extreme ambient conditions. Simultaneously the power density keeps on increasing up to several kilowatts inside a rack system and this electrical power is mainly converted into heat. Because reliability and failure rate of components are directly related to temperatures, the key is the ability to control the cooling mechanisms at any point and in any level of detail inside the system.

Thermal management and full-level thermal optimisation procedures can only be successful if they have access to all cooling-related data such as air velocity distribution, pressure field, local heat transfer effects and more[1]. One of the most powerful tools which are suitable for this task is CFD, Computational Fluid Dynamics. Ideally Thermal Simulation accompanies the whole development process to ensure that the building blocks e.g. power supplies - as well as the whole system meet the required cooling target.

Increasing in the use of telecommunication system power-flow-regulating devices in the form of powerelectronics are also required in a wide range. Power electronic components contains of solid-state switches and passive components such as capacitors and inductors. Solid-state switches are mainly metal oxide semiconductor field effect transistors (MOSFETs) and insulated gate bipolar transistors (IGBTs). Solid-state switches having types metal oxide semiconductor field effect transistors(MOSFETs) and insulated gate bipolar transistors (IGBTs). Heat is generated in these switches because of switching and conduction losses. Heat fluxes can range up to 200 W/cm^2 to 300 W/cm^2 depending on the device geometry [2].With this power density being transformed to heat in the package, which is why leads to the urgent need to efficiently dissipates the generated heat.

For thermal design, commercial software of computational fluid dynamics (CFD) provides a widely used method in solving the coupled fluid dynamics/heat transfer problem encountered in electronic system and products design. Currently, most of the thermalmanagement solutions used in telecommunication systemis passive cooling and active cooling method such as conduction and natural convection and forced convection. The design use conduction to transport heat from electronic components to the surface of the electronic enclosure and conduction through bond wires and leads to the printed circuit board which provides the primary means for chip-to-chip communication with interconnection PCBs where heat removal typically occurs [3].Inside the board the heat is conducted in thickness and in-plane directions. The board construction (layer structure and materials) has a strong influence on the heat spreading inside the board.Using PCBs with high conductivity power and ground planes that include insulated metal substrates and/or embedded heat pipes will provide improved thermal performance. At the board surface the heat is transferred to the environment by means of natural convection and radiation. Thus PCB becomes one of main heat dissipating paths.

In this paper work minimizing the temperature rise of the circuit board and components will be commonly desired. 3D model of telecommunication system is prepared and analysis done in ANSYSICEPAK. Results of analysis and experimental results are compared. Fig.1 shows 3D model of telecommunication system built in ANSYSICEPAK.

© 2015, IERJ All Rights Reserved

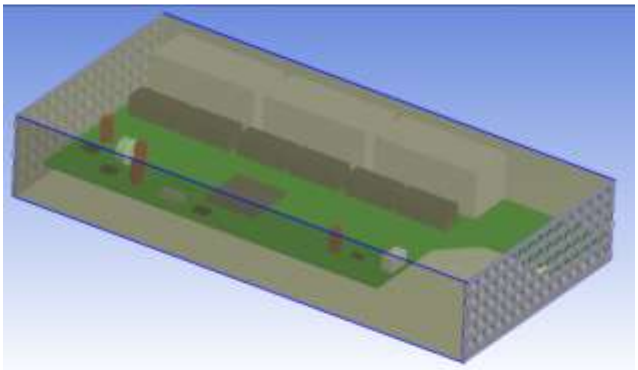

Fig.1 3D model of telecommunication system built in ANSYS ICEPAK

II. EXPERIMENTAL SET UP

Experimental set up consists of wind tunnel, temperature sensors and display. There were two temperature sensors used on the IC package surface and one used to measure board temperature. These all three readings were taken to the display where observer can read it. Fig.2 shows the experimental set up layout.

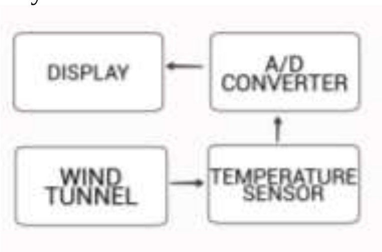

Fig.2 Experimental set up layout

III. MESH GENERATION

The present study is based on application of computational fluid dynamics (CFD) to the telecommunication board cooling. Modeling of fluid flow requires the specification of the geometry through a computational grid. The geometry chosen for simulation properly represent the physics of problem.

3.1Modelling

3D geometry of telecommunication system is drawn in ANSYSICEPAK. Mesher-HD type is used for meshing. ANSYSICEPAK is a popular proprietary software package used for CAD and mesh generation. Model already drawn into other CAD packages can be imported in to ICEPAK for meshing purpose. Fig.3 shows 3D model meshed using Mesher-HD type of meshing.

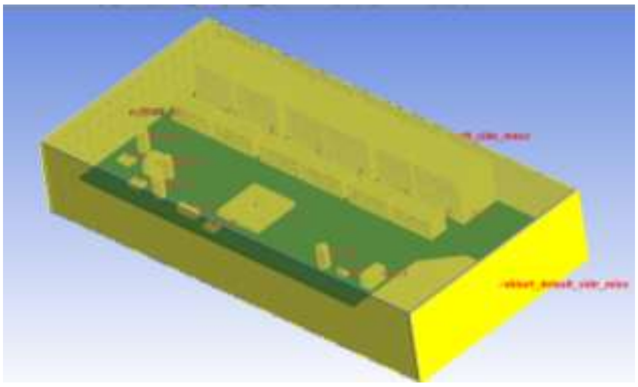

Fig.3 Telecom board meshing

IV. MODEL SIMULATION

Mesh file created by ICEPAK is imported into ANSYS Fluent. Further this is solved by a natural convection model.

4.1 **Governing Equations**

© 2015, IERJ All Rights Reserved

For all flows, FLUENT solves numerically the Conservation Equations for Mass, Momentum and additional Transport equations (if the flow is turbulent) for a fluid in a given flow geometry. The transport equations have different forms depending type of model used. In this work for telecommunication cabinet flow is set as a natural convection flow. And all the governing equations are solve according to the given setting.

For flows general equation for conservation of mass or continuity equation can be written as follows:

$$\frac{\partial \rho}{\partial t} + div\,(\rho u) = 0 \quad (1)$$

Navier-Stokes Equations

$$\frac{\partial(\rho u)}{\partial t} + div\,(\rho u u) = -\frac{\partial p}{\partial x} + div\,(\mu\, grad\, u) + \left[-\frac{\partial(\rho \overline{u'^2})}{\partial x} - \frac{\partial(\rho \overline{u'v'})}{\partial y} - \frac{\partial(\rho \overline{u'w'})}{\partial z}\right] + S_{\ddagger x} \qquad (2)$$

$$\frac{\partial(\rho v)}{\partial t} + div\,(\rho v u) = -\frac{\partial p}{\partial y} + div\,(\mu\, grad\, v) + \left[-\frac{\partial(\rho \overline{u'v'})}{\partial x} - \frac{\partial(\rho \overline{v'^2})}{\partial y} - \frac{\partial(\rho \overline{v'w'})}{\partial z}\right] + S_{\ddagger y} \qquad (3)$$

$$\frac{\partial(\rho w)}{\partial t} + div\,(\rho w u) = -\frac{\partial p}{\partial z} + div\,(\mu\, grad\, w) + \left[-\frac{\partial(\rho \overline{u'w'})}{\partial x} - \frac{\partial(\rho \overline{v'w'})}{\partial y} - \frac{\partial(\rho \overline{w'^2})}{\partial z}\right] + S_{\ddagger z} \qquad (4)$$

4.2 Buoyancy-Driven Flows and Natural Convection

The importance of buoyancy forces in a mixed convection flow can be measured by the ratio of the Grashofand Reynolds numbers:

$$\frac{Gr}{Re^2} = \frac{g\beta\Delta T L}{\vartheta^2} \qquad [5]$$

$$Ra = \frac{g\beta\Delta T L^3 \rho}{\mu\alpha} \qquad [6]$$

$$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p \qquad [7]$$

$$\alpha = \frac{K}{\rho c_p} \qquad [8]$$

Rayleigh numbers less than 10^8 indicate a buoyancy-induced laminar flow, with transition to turbulence occurring over the range of $10^8 < Ra < 10^{10}$.

1) 4.2.1 The Boussinesq Model

By default, ANSYS Icepak uses the Boussinesq model for natural-convection flows. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation:

$$(\rho - \rho_0) g \approx -\rho_0 \beta (T - T_0) \qquad [9]$$

Where, ρ_0 is the (constant) density of the flow, T_0 is the operating temperature, and β is the thermal expansion coefficient. Above equation is obtain by using the Boussinesq approximation

$$\rho = \rho_0 (1 - \beta \Delta T) \qquad [10]$$

to eliminate ρ from the buoyancy term. This approximation is accurate as long as changes in actual density are small.

© 2015, IERJ All Rights Reserved

2) 4.2.2Incompressible Ideal Gas Law

In ANSYS Icepak, if you choose to define the density using the ideal gas law, ANSYS Icepak will compute the density as

$$\rho = \frac{P_{op}}{\frac{R}{M}T} \qquad [11]$$

Where,*R* is the universal gas constant and P_{op} is defined as the**operating pressure** in the**advanced problem setup**panel. In this form, the density depends only on the operating pressure and not on the local relative pressure field, local temperature field, or molecular weight. When the Boussinesq approximation is not used, the operating density, ρ_0, appears in the body-force term in the momentum equations as$(\rho-\rho_0)g$. This form of the body-force term follows from the redefinition of pressure in ANSYS ICEPAK as

$$P'_s = P_s - \rho_0 g x \qquad [12]$$

The hydrostatic balance in a fluid at rest is then

$$P'_s = 0 \qquad [13]$$

The definition of the operating density is thus important in all buoyancy-driven flows. Fig.4 shows overview of the solution method to solve conservation equation in ANSYS ICEPAK

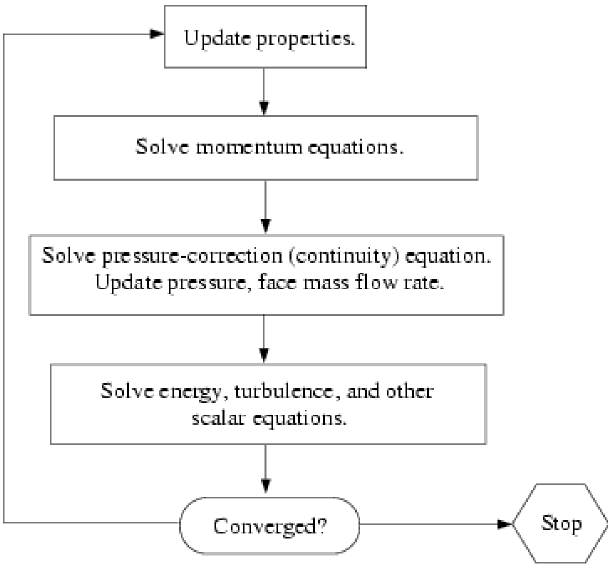

Fig.4 Overview of the Solution Method [4]

In this work solution is done with boundary conditions of 0.3 m/s velocity at the two ends of cabinet and pressure is set to the atmospheric pressure. Comparison of numerical results and experimental results are done in next section.

V. RESULTS & DISCUSSION

Fig.5 shows the predicted temperature field distribution of the components on PCB. The simulation predicted maximum temperature occurs in the IC package with a value of about 154.35°C, the temperature rise of this IC is highest on the board as it consumes the most power. This temperature compares to an experimentally determined 151.6°C, giving an error of 1.81%. Also at another point on IC package temperature predicted is 96.35 °C and experimental temperature reading by temperature sensor is 94.2°C with an error of 2.23%.

© 2015, IERJ All Rights Reserved

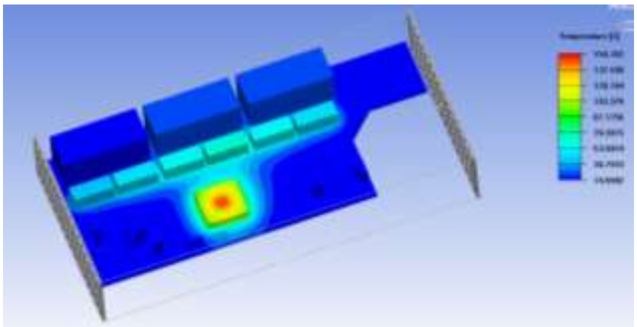

Fig.5 Predicted temperature field distribution of the components on PCB

Table 1shows the comparison between simulated temperature and the average experimental data.From this results we can say that results predicted by ANSYS ICEPAK and experimental results are match within considerable error.

Table 1 Comparison of experimental and numerical value

Sr. No.	Temperature Point	Experimental Value(°C)	CFD Value
1	P1	151.6	154.35
2	P2	94.2	96.35

Fig. 6 shows velocity distribution in cabinet. In which we can describe air stream from two side gets mixed up and move against the gravity towards the max Y side of the cabinet performing cooling operation.

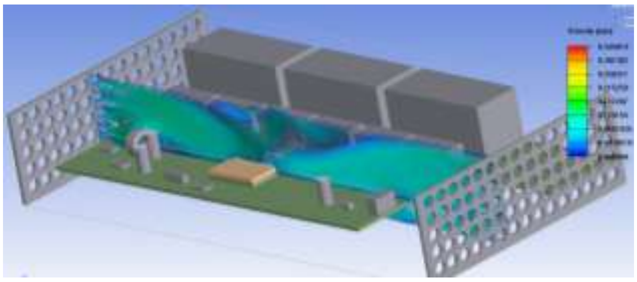

Fig. 6 Velocity distribution in cabinet

REFERENCES

1. Anton Breier, Manfred Schlenk (2002), Cooling of Telecom Systems – Thermal Optimization Using CFD Methods, IEEE, PP. 486-490.
2. D. B. Baer (2001), emerging cooling requirements &systems in telecommunications spaces, IEEE, PP. 95-100
Roger R. Schmidt (2002),Computer and Telecommunications Equipment Room Cooling: a Review of Literature, IEEE, PP. 753-766 ANSYS 2016 help document
3. Noh, Hong-Koo, KyuSop Song and SooKeun Chun (2003),The Cooling Characteristics on the Air Supply and Return Flow System in the Telecommunication Cabinet Room. IEEE, PP. 777-784
4. Patrick A. Gillan (2004),Fresh Air - Natural Asset, IEEE, PP .470-477
5. ZhipengDuan and Yuri *S.* Muzychka, Impingement Air Cooled Plate Fin Heat Sinks Part I - Pressure Drop Model, IEEE, PP. 429-435
6. ZhipengDuan and Yuri *S.* Muzychka, Impingement Air Cooled Plate Fin Heat Sinks Part II - Thermal Resistance Model, IEEE, PP. 436-443
7. A. G. AgwuNnanna , A. Haji-Sheikh, D. Agonafer, Effects of Variable Heat Transfer Coefficient, Fin Geometry, and Curvature on the ThermalPerformance of Extended Surfaces,IEEE, PP. 292-297
8. MadhusudanIyengar and Avram Bar-Cohen,Design for Manufacturability of SISE Parallel Plate Forced Convection Heat Sinks,IEEE,PP.141-148
9. Hyunjong Kim, KyoungJoon Kim, YeonwonLee,IEEE,Thermal Performance of Smart Heat Sinks for Cooling High Power LED Modules, PP. 962-967

© 2015, IERJ All Rights Reserved