Thermal Design for PCBs of Control Systems to Guided Vehicles

M. Akhila Jyothi,[1] Dr.M.N.Giri Prasad[2], T. Satyanarayana[3]
[1](M.Tcch), ECE, Digital Systems & Computer Electronics, [2]Professor, Department of ECE
[12]JNTUA College of Engineering, Anantapur.
3, Vice President (Systems), Park Controls & Communications (P) LTD, Bangalore.

Abstract- Electronic systems have the unique challenge of managing heat as power dissipation, is a phenomena occurring when current flows through a resistance. Thermal management plays a critical role in determining the functions and performance of electronic systems. As the power requirements of advanced aircrafts steadily move towards higher density and higher heat loads, Requirements of thermal analysis for advanced avionics are becoming more severe and critical to avionics design Without a suitable thermal management design, component temperatures can rise to levels that cause failure. This paper describes some of techniques for controlling the dissipating heat from heat generating electrical components on a Printed Circuit Boards (PCB). For temperature control the approach is to place an arrangement of vias as so called thermal-vias in the PCB underneath the component, the heat from the component will be dissipated through the thermal-vias. Efficient way of placing the copper trace width in order to sustain the nearby by components and traces from thermal rise. Another approach is using the heat sink pad with respect to thermal paste is also an approach which will help for better heat dissipation from case to ambient especially for metal casing components. Different temperature controlling and reliable PCB designs will be discussed. With these analysis results we can foresee any problem arising due to inadequate provision for heat dissipation resulting in poor performance & reliability and ultimately failure of the components and equipment. Key words: Thermal-vias, heat-sink, Juntion temperature, Ambient Temperature, traces .

I. INTRODUCTION

As backbone of electronic devices, rational design of PCBs ensures their high performance. Power electronics devices for future advanced aircrafts require higher performance at the same time more compactness. Though this trends a great increment of the power density per square inch of PCB's, the operating temperature of most of the devices is still settled below 125^0C. Thumb rule says, "The life of an electronic device reduces by half for every 10^0C rise in its operating temperature". It becomes therefore essential to keep the temperature rise of the electronic device/equipment under control for reliable operation for a longer period. Thermal design today is a major limiting factor for performance of avionics equipment. If PCB design partially or even totally fails to meet thermal requirement, electronic devices will definitely suffer from risk of damage or even failure[2]. It's estimated that over half electronic components fail due to high stress which is resulting from thermal environment. Recent years has witnessed wide devices of large-scale and hyper-scale Integrated Circuits (ICs) and surface mount technology and electronic products start embracing development directions towards high reliability and high density[1]. Accordingly, electronic systems are calling for increasingly higher requirement of thermal performance for better reliability of design.

Constantly growing integrity of circuit modules and massive applications of ICs and multi-chip module (MCM) contribute to the improvement of component assembly density which thereafter leads to higher density of heat flow on PCBs. High-quality PCBs not only derive from accurate and rational layout and routing, but also rely on high thermal reliability for secure operation. Therefore, it's of much significance to implement comprehensive thermal dissipation rules and analysis on PCB.

High-quality PCBs not only derive from rational and accurate layout and routing, but also rely on high thermal reliability for secure operation. Therefore, it's of much significance to implement comprehensive and effective thermal dissipation rules and analysis on PCB.

As a fundamental aspect of thermal design, PCB thermal analysis refers to the process of establishing the thermal module of components and set simulation control parameters according to the package type, structure of components and PCB operating environment to estimate values of thermal behavior of PCBs. Based on the result of thermal analysis, thermal problems of PCB can be found out quickly and timely measures can be properly taken and high temperature dense areas can be eliminated, which will determine the path of heat conduction, optimize the positions of key components to fully take advantage of the rate of heat dissipation, increase heat transmission efficiency of heat dissipating ways and determine the space between the components on boards. Ultimately, properly controlling the dissipation of heat in the design will allow to produce a more reliable and military standard PCB design.

II. METHODOLOGY

Airborne electronics are emphasized because they are most challenging to the designer. Constraints in size, weight, volume and operational environment are generally much more restrictive for airborne systems than they are for ground based equipments.

According to the military standards, for the reliability of the design for military environments, there are some standards for the boards which they have to be under go through. Tests are performed for the purpose of demonstrating the quality or reliability of devices subjected to the specified conditions over an extended time period. With this reliability tests the board level functionality we can come to know that whether the design and the quality is suitable for the defense applications or not reliable to that conditions. For material wise, design wise, failure rates are to be identified with the tests.

According to MIL-STD-833, the most reliable environmental considerations that contribute to the thermal management challenges include Humidity Test, Thermal Shock, and Variable Vibration Test, Temperature Cycling Test, Burn-In Test[3].

Temperature Cycling Test: According to MIL-STD-801G, this test is to come to know that whether the material can withstand sudden changes in the temperature of the surrounding atmosphere without experiencing any physical damage or deterioration of the board performance. The device shall be thermally stabilized at the minimum specified test temperature with no power applied to the device. The specified power supply voltage and the specified input logic patterns using VLW and VHW input voltage levels shall then be applied to the device under test and the outputs shall be monitored.

Burn-In Test : This test is performed to eliminate marginal semiconductor devices or those with defects resulting from manufacturing aberrations that are evidenced as time and stress dependent failures. Without the burn-in, these defective devices would be expected to result in early lifetime failures under normal use conditions. It is the intent of this test to operate the semiconductor device at specified conditions to reveal electrical failure modes that are time and stress dependent. All devices shall be operated at the maximum rated power related to the test temperature for 160 hours minimum at the specified test conditions[8].

Thermal Shock Test: The purpose of this test is to determine the resistance of the part to sudden exposure to extreme changes in temperature and the effect of alternate exposures to these extremes. From ambient temperature to sudden high temperature and from high temperature to low temperature. This sudden chance can make the components to work in not as expected. So according to component temperature readings we can come to know the stability of that particular component. Sudden temperature changes can permanently or temporarily affect the performance of the boards.

For any component, according to its functionality and requirement electrical specifications will be mentioned in the device specifications document. According to this electrical specification we can come to know the minimum and maximum functional voltage and current ratings of that component. Thermal specifications are also be defined. So with respect to that, thermal characteristics of the device are being considered. With maximum and minimum operating temperature range will be specified. The main purpose of the thermal specifications is to determine the thermal characteristics of microelectronic devices. This includes junction temperature, thermal resistance, case and mounting temperature and thermal response time of the microelectronic devices.

Thermal design is based on the basic theory of heat transfer. Where there's temperature difference, there's heat transfer from high temperature zone to low temperature zone. Heat transfer can be achieved through heat conduction, heat convection and heat radiation.

The temperature rise is proportional to the square of the conductor resistance and rate of current flow, which eventually increase the DC loss. Temperature rise means how much hotter the trace will get with respect to the current flow in it. We have to decide how much temperature rise the board can handle based on the operating environment.

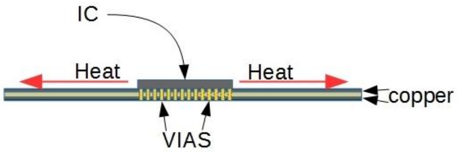

Fig.1: Heat Flow in an IC

Power dissipation is an important issue in present-day design of the PCB. Power dissipation will result in the temperature changes and difference and pose a thermal problem to a chip. In addition to the issue of reliability, excess heat will also negatively affect safety and mainly the electrical performance. The working temperature of an IC should therefore be kept below the maximum allowable limit of the worst case for better case. The ever-shrinking chip size may cause the heat to concentrate within a small area and leads to the high power density. Removing the heat effectively becomes the critical issue to be resolved.

$$P = I \times I \times R = I^2 \times R$$

$$P = V/R \times V = V^2 / R$$

Power dissipation isn't the power that may be drawn by the chip and put into a load, but it is the power wasted by the chip when it drives a load.

$$Pdstatic = Vcc * Icc$$

Where Vcc and Icc are the supply voltage and currents found in the datasheet. Then for voltage regulators since there is not a source voltage.

Power dissipation = Vin * Iout

When Voltage is applied and the switch is enabled, Load current Iout is been drawn. Since current is drawn by the load, the chip will become hot due to the power dissipation. The junction temperature T_j will differ from the ambient temperature or case temperature. In cases the junction temperature T_j can be related to power dissipated in the chip.

Ambient temperature is a term which refers to the temperature in a room, or the temperature which surrounds a chip. For electronic components, power dissipation along with ambient temperature in nearby components and the components' own power dissipation represent the main source of temperature extremes. The highest operating temperature that a semiconductor in an electronic device can withstand is called its junction temperature. The operating junction temperature range specifies how hot the "junction" which is the active part of the integrated circuit can be allowed to get before it goes into the thermal shutdown. When current is drawn by the load, the chip will become hot due to power dissipation.

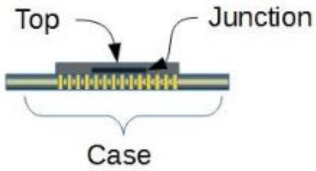

Fig.2: Temperature flow defining for an IC

Thermal resistance is a measurement and a heat property of a temperature difference by which an object or material resists a heat flow. It is the ability of a material to resist the flow of heat. It is the quantity of heat that passes in the unit time. Thermal resistance is the reciprocal of thermal conductance, i.e., lowering its value will raise the heat conduction and vice versa.

Heat transfer occurs at a lower rate in materials of low thermal conductivity than in materials of high thermal conductivity. Correspondingly, materials of high thermal conductivity are widely used in heat sink applications and materials of low thermal.

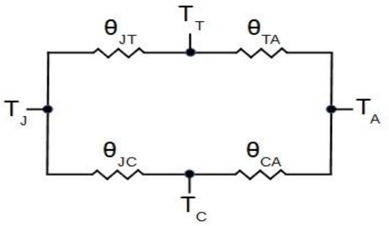

Fig.3: Thermal Flow from Junction to Ambient

As presented in the figure.3. T_J is defined as the temperature of the junction (the internal working portion of the component), T_T is the temperature of the "top" of the package (typically the plastic enclosure of the component), T_C is the temperature of the "case" (this is the temperature of the highly thermally conductive pads of the component and the attached PCB) and T_A is ambient environment's temperature. The goal of the electronics designer is to produce the lowest thermal resistance possible between the junction and the ambient environment. With the exception of θ_{CA}, the thermal resistances of the system (θ_{JT}, θ_{TA} and θ_{JC}) are defined by the properties of the component and can be pulled from the data sheet for particular required component. As such, the primary challenge for the designer is the reduction of the thermal resistance of the component's case to the ambient environment by reducing the resistance. How well we are able to lower this thermal resistance (θ_{CA}) will largely define the temperature differential (or lack thereof) that will develop between the ambient environment and the junction temperature of the component. Generally, θ_{JA} figures typical of op amps and other small devices are on the order of 90-100°C/W for a plastic 8-pin DIP package, as well as the better SOIC packages.

Case temperature T_C in ^{0}C, is the temperature at a specified accessible reference point on the package in which the microelectronic chip is mounted.

In ICs, one temperature reference point is always the device junction, taken to mean the hottest spot inside the chip operating within a given package. The other relevant reference point will be either T_C, the case of the device, or T_A, that of the surrounding air. This then leads in turn to the above mentioned individual thermal resistances, θ_{JC} and θ_{JA}.

$$\theta_{JA} = \theta_{JC} + \theta_{CA}$$

$$T_J = T_A + (Pd \times \theta_{JA})$$

$$\theta_{JA}= T_J - T_A/(Pd)$$

When a device operates in a system under the steady state condition, the maximum power dissipation is determined by the maximum junction temperature rating, the ambient temperature, and junction-to ambient thermal resistance.

$$Pd_{MAX} = (T_{J\,MAX} - T_A) / R\theta_{JA}.$$

In general, a device with a thermal resistance θ equal to 100°C/W will exhibit a temperature differential of 100°C for a power dissipation of 1 W, as measured between two reference points. Note that this is a linear relationship, so 1 W of dissipation in this part will produce a 100°C differential (and so on, for other powers). For the AD8017AR example, θ is about 95°C/W, so 1.3 W of dissipation produces about a 124°C junction-to-ambient temperature differential. It is of course this rise in temperature that is used to predict the internal temperature, in order to judge the thermal reliability of a design. With the ambient at 25°C, this allows an internal junction temperature of about 150°C. In practice most ambient temperatures are above 25°C, so less power can then be handled.

For any power dissipation P (in watts), one can calculate the effective temperature differential (ΔT) in °C as:

$$\Delta T = Pd \times \theta$$

where θ is the total applicable thermal resistance.

The other path for thermal conduction is the top case of the component. As the plastic packaging of most power components do not provide a good thermal path to the ambient environment the efficiency of thermal dissipation of the design is more heavily dependent on the design's ability to dissipate the thermal energy to the surrounding environment through its case.

Temperature raise (^{0}c) = Thermal resistance * power dissipation

Temperature margin (^{0}c) = Tmax - maximum junction temperature

Thermal vias:

The PCB by its nature is not a good thermal conductor. It is made of substrate materials that are insulating electrical interconnections between components. The thermal conductivity of a typical substrate material is about $\lambda \sim 0.2$ W/mK. However, copper, the material of the conductive traces of a PCB, has a high thermal conductivity of $\lambda \sim 390$ W/mK. So for better heat flow control the approach is to place an arrangement of vias as so called thermal vias in the PCB underneath the component on pad [4]. . The thermal conductivity of thermal vias is higher. The base of the component is connected to the thermal vias on the top side of the PCB. The heat flux is transferred through these vias down to the bottom side of the PCB and then can be coupled into the heat sink. For heat spreading the thermal vias are can be connected to power or ground planes of the PCB.

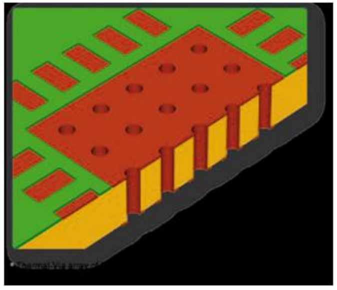

Fig.4: Cross Section of Thermal vias across the layers of the PCB

The heat flux flows mainly only through the very small cross sectional area of the copper plating at the hole wall of the vias. An array of thermal vias of the particular diameter drill can control most of the heat flow accordingly depending on the current flow through that component. This leads to a significant reduction of the temperature of the component that could increase the life time and reliability of the component and the whole system. To illustrate the performance of thermal via arrays we can calculate with respect to the thermal conductivity and the thermal resistance. The effective thermal conductivity λ_{eff} and the thermal resistance, R_{th} of a thermal via array can be calculated with the following equations:

$$\lambda_{\mathrm{eff}} = \frac{1}{\mathrm{A}}(\lambda_{\mathrm{hole}}\ .\ \mathrm{A}_{\mathrm{hole}} + \lambda_{\mathrm{via}}\ .\ \mathrm{A}_{\mathrm{via}})$$

$$R_{th} = \frac{d}{\lambda_{eff}\ .\ A}$$

Where

λ_{eff} is the effective thermal conductivity,

R_{th} is the thermal resistance,

A is the surface area of the thermal array or the thermal pad,

d is the thickness of the PCB.

The centre of the vias can be filled to enhance the thermal conductivity by replacing the air with some material of better thermal conductivity. They can be filled with conductive Silver paste which increases the thermal conductivity slightly. That small gain in thermal conductivity could also be achieved when a few microns of additional copper are plated into the vias. A filling of the vias with some kind of solid metal such as solder or copper results in a much better thermal conductivity if a void-free filling can be achieved. In general the inside of the vias is filled with solder. Terminology of via is: The "outside diameter of the via" is the total diameter including the rim and the filler. The "thickness of the plating near the outside diameter of the via" is the thickness of the plating on only one side. The relation is, diameter of the filler + (2x thickness of the plating) = total outside diameter of the thermal via[7].

It is well known that the thermal resistance of the package, especially, θ_{JA}, is highly dependent on the PCB in which the parts are mounted for thermal testing. The effect of the PCB is more critical when the package has extremely low θ_{JC} (or thermal resistance between junction-to-case) because the thermal resistance between case to PCB, and PCB to ambient air, becomes more dominant than that between die to package case. The remaining conduction paths extend from the device into the rest of the layers (both copper and dielectric) through the thermal vias underneath the device. These conduction paths, if optimized through proper design of the thermal via array, are the most efficient paths in the structure for removing heat from the device.

Generally, vias should be placed near to the respective IC pad, incase if it is placed far and if the current carrying is high then it may affect the nearby signals or sometimes it may end up burning of that trace. So it is better to use less than 20 mils of distance from the IC pad to the vias[6].

For less than 1 ampere of current carrying device, it is better suggested to use 12 mil of drill and 24 mil of pad which can maximum bare up to 0.8 amperes. For more than 4 ampere of current carrying device, 24 mils drill and 40 mil pad is been suggested .incase of 3 amperes current carry, we can use two to three 12mil drill/24 mil pad thermal vias for better heat flow.

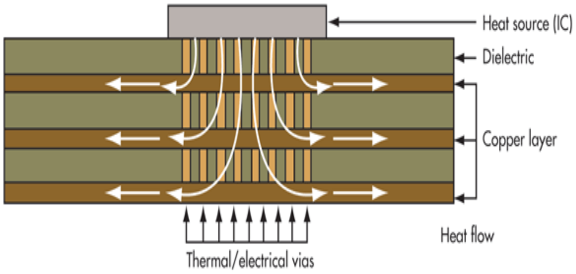

Fig.5: IC thermal conduction can be achieved through the use of thermal vias and copper planes

Heat sink: The case temperature is defined as the hottest temperature on the top of the device. In most instances, this is at the center of the top surface or lid of the device. The case temperature measurement can be performed with (in order of accuracy) an IR camera, a thermocouple. The impact of using a heat sink pad is to measure the package top surface can be very substantial, reducing the heat flow between the ambient and actual surface temperature by 50% or more.

The heat energy generated by the test die is allowed to flow normally along preferential thermal conduction paths. The quantity of heat flowing from the die to the top of the package is actually unknown in the measurement, but is assumed to be the total power of the device. The amount of energy flowing from the die to the top of the package during test is similar to the partitioning of the energy flow in an application environment.

If the heat is not dissipated efficiently, the resulting high temperatures not only alter the output signals but also damage components . so the heat Sink to Air Thermal Resistance is to be known as This is necessary if a heat sink is added to component with respect to the factor Where θja is the thermal resistance between a heat sink and the air when the heat sink is applied to a component. This value is a function of air velocity, usually provided by the manufacturer of the heat sink.

Sometimes, rather than specifying maximum operating temperature, a maximum case temperature will be given. Running the device at the maximum case temperature (without a heat sink) results in the die running at the recommended operating junction temperature written as T_C. T_C is normally measured at the center of the package top-side surface. When a high-efficiency heat sink is applied to the top surface of a device for which $R\theta_{JC}$ is small compared to $R\theta_{JA}$:

$$T_J = T_A + \{ [R\theta_{JA} + R\theta_{JC} + R\theta_{(CS)}] \} * \text{power dissipation}$$

The package thermal performance $R\theta_{JA}$ is reported to be the sum of two resistances: $R\theta_{JC}$ and $R\theta_{CA}$. $R\theta_{CA}$ stands for the case-to-ambient thermal resistance.

$$R\theta_{JA} = R\theta_{JC} + R\theta_{CA}$$

Here, $R\theta_{(CS)}$ is the case-to-heat sink thermal resistance of the thermal interface material . This equation is the most accurate for packages where $R\theta_{JC}$ is small compared to $R\theta_{JA}$, meaning that most of the heat can be dissipated through the top surface of the package when a sufficiently efficient heat sink is applied. So for the metal casing components which has more heat dissipating factor it is better to use with a thermal paste compound with a heat sink pad for controlling the temperature from case to ambient mostly. During the reliability level testing's, The rise in junction temperature during the on period shall be verified by means of measuring junction temperature using the change in body diode voltage drop or calculated by applying the following equation.

$\Delta T_J = P_T R\theta_{JC} (1 - Exp - t/Tp)$, where $P_T = V_{DS} I_D$

TP = thermal time constant of device package, and the heat sink used.

t = heating time, $R\theta_{JC}$ s= thermal resistance junction to case, for the period of heating time specified, of the device and any necessary heat sink used.

This test is intended to allow the case temperature to rise and fall appreciably as the junction is heated and cooled; thus, it is not appropriate to use a large heat sink or a high power short pulse.

Copper Trace:

Generally, the power rating of any electrical component does not tell us how much power it will dissipate, but simply how much power it may dissipate without sustaining damage. If the actual amount of dissipated power exceeds a component's power rating, that component will increase temperature to the point of damage. A PCB has a number of components mounted on it which can be active or passive, high power or low power, but all of them produce heat during operation. Copper traces are utilized to connect one element node to another node. The shape of these traces determine one very important aspect of a PCB. The main parameters that define the behavior of a trace are its characteristic impedance, propagation delay and losses which impacts more on high-speed PCB functionality analysis and design . Their characteristics determine how the connected I/O buffers interact with one another. However, the right impedance and other trace characteristics characteristics are essential to generating accurate results.

During operation, heat is transferred from and to the PCB track by conduction through component leads, between components through air by convection or simply by radiation without any medium. During operation, the components mounted on the board take electrical power as input and a part of it is converted into heat as wasted energy. The heat generated in a component during operation gets transferred by conduction, convection and radiation; conduction of heat occurs through the component lead terminals and to and from copper tracks in the PCB. Convection of heat occurs through the surrounding air. Heat transfer by radiation occurs between components in the board. The net result of the heat transfer phenomena is temperature rise of the components on the board, which affects the functioning of the components and in extreme cases cause failure due to thermal stress.

Generally according to standards, a 40 mil trace is equal to 1mm trace width which can carry up to 1 ampere of current for signal carrying, the trace width will be 6 mil or 10 mil is been recommended. In case of power supply (Vcc) and ground tracks, the trace width will be double to the signal carrying track with more copper pouring is required.

Consider for an IC LMZ10503, which is a switcher regulator which can give variable output voltage which takes 5volts as a direct input supply and output voltage it will provide is 3.3V and output current is 3ma. So according to this electrical characteristics it take 9.9 watts i.e., W= 3.3 /3 = 9.9W. It means almost 10 ampere of current it will take. So 400 mil trace width is been suggested.

This benefit is using a larger copper pattern than electrically required to sink the thermal energy away from the component. For the surface mount component this would be the power dissipation in the component divided by the interface area between component and board. The plot below shows the thermal resistance vs. copper thickness from 1 Oz through 8 Oz for 25mm square copper area. Notice that the as the thickness of the copper increases so does the performance.

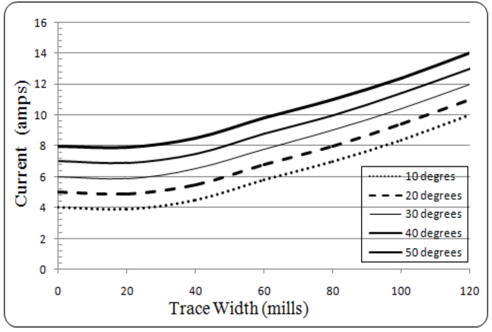

Fig.6: Current Carrying Capacity with respect to Trace Width

From the graph it is observed that thermal optimization during PCB development for a one sided PCB can be directly estimated from the graph. Increasing copper thickness and copper area will reduce the thermal resistance of the board and will reduce the operating temperature of the component. If the trace width from the transmitter part to the receiver are not designed properly according to their electrical characteristics specifications with required impedance then they may lead to the signal distortion. The Unequal impedance of the source output, and receiver or load causes impedance mismatch. This mismatch means the transmitted signal is not fully absorbed within the receiver and the excess energy is reflected back to the transmitter. This process continues back and forth until all the energy is absorbed. The amount of reflection actually depends on the characteristic impedance of the transmission line which is the track i.e..,. If the characteristic impedance of the trace matches the source or load impedance then the signal will pass without any disturbance to the receiver chip. In order to avoid signal distortion at the source and destination, the PCB trace impedance must be matched to source and load impedance at the source and destination ends. This presents a considerable challenge that requires careful PCB design to mitigate the effects of signal degradation caused by impedance discontinuity. The design considerations for setting the value of the impedance for the conductor is typically the output impedance of the transmitter and the input impedance of the receiver.The acceptable range (tolerance) for the impedance will need to be determined and taken into account during the design phase as well as when specifying the PCB parameters. The impedance requirements will have to be specified based on the layer and the trace widths.

III. RESULT & DISCUSSION

When heat is generated in a component as a result of its working, it is conducted through the leads of the devices to the copper tracks in the PCB. The heat then flows to other components through lead terminals soldered to the copper tracks from a hotter to a cooler component. So, adequate trace width is necessary to ensure the desired amount of current can be transported without overheating and damaging your board. So, it seems obvious with DC current that, to route power to a chip or connector use a thicker trace. Higher current flow increases the temperature of the copper traces therefore, temperature rise is a design parameter for how much added heat you would like to design for. Theoretically, PCB current carrying capacity is determined by cross-sectional area of trace and temperature rise.

If the trace has a resistance of 0.328 Ohms, when 1.12 Amps flows through the trace there is 0.409 watts of power generated in the trace. This causes the trace to heat up. As the trace heats up the energy conducts away from the trace into the circuit board. The heat spreads out into the circuit board heating it up. The effect of the trace will impact more on the nearby components functionality because PCB track width is based on current flowing through the track. As a result, as soon as power starts or order modification is implemented on trace, it's possible to cause a super large transient surge or even burning down of a trace between pads.

The integration of thermal vias for heat dissipation into the structure of printed circuit boards is a reliable technique which provides a highly efficient way to dissipate heat from electronic components with respect to a multi layer board especially for the power switching modules. So using of the thermal vias technique is considered to be the reliable technique for the temperature controlling of the device. The principle of integrating thermal vias into the PCB construction is also suitable for bare die attachment, providing a much lower thermal resistance in the thermal pathway. The lower number of thermal interfaces in the thermal pathway leads to a reduced overall thermal resistance and to a much higher efficiency in the heat dissipation of the chip. This is a significant reduction of the temperature of the component that could increase the life time and reliability of the component and the whole system.

IV. CONCLUSION

This paper discusses what needs to be considered to make a thermal design guideline of PCB more practical and reliable for today's requirement with high component density and high current density. The advantage of thermal via arrays has been demonstrated and the factors about the trace current carrying capacity significantly for a multi layer board are identified. And also the effects of heat sink pad for controlling more heat generation towards case to ambient temperature are been discussed. How to formulate a new PCB trace thermal design guideline with consideration of these factors are introduced in details. The temperature controlling method, which is suitable to control the maximum temperature around components which are having more heat, heat dissipation factor and reliable to military standards are discussed.

V. REFERENCES

[1]. T.L. Wu, F. Buesink, and F. Canavero, "Overview of Signal Integrity and EMC Design Technologies on PCB: Fundamentals and Latest Progress" in *Electromag. Compat., IEEE Trans. on,* vol 55, pp. 624-638, Aug , 2013.

[2]. V.Lakshminarayanan et.al "Thermal effects in a Printed Circuit Board due to heat emission from mounted components" IMAPS Nordic Conference on Microelectronics Packaging (NordPac) 2017.

[3]. Dr.John Humphries: "Thermal management of electronic devices", Engineer IT Magazine, EE Publishers, June 3,2010, website : http://www.ee.co.za.

[4]. Yi Wang et.al "Thermal Design Guideline of PCB traces under DC and AC Current", 978-1-4244-2893-9/09/ IEEE page-1240-1246, 2009.

[5]. Nadezhda Kafadarova,Anna Andonova : "PCB thermal design improvement through thermal vias" , Proc. 8th International WSEAS Conference, 2009.

[6]. S. Velusamy, W. Huang, J. Lach, M. Stan, and K. Skadron, "Monitoring temperature in FPGA based SoCs," in *ICCD '05*. Washington, DC, USA: IEEE Computer Society, 2005, pp. 634–640.

[7]. Dr.N.V.Srinivasulu : "Thermal Analysis of Printed Circuit Board (PCB) of Avionics equipment ", IJARCET,Volume1,issue 6, 2012.

[8]. Environmental Test Methods For Semiconductor Devices , Department of Defense Test Method Standard PART 1: Test Methods 1000 Through 1999.

[9]. Thermal Design basics : MT-093 Thermal design analysis , Analog Devices.

[10]. Allen M.Lush : Modeling Heat conduction in printed circuit boards using finite element analysis, Electronics cooling magazine, May 1,2004.

[11]. Darvin Edwards : PCB design and its impact on device reliability,Electronic Design Magazine,June 5,2012.