TEMPERATURE MANAGEMENT FOR PRINTED CIRCUIT BOARDS

FINELINE EXCELLENCE IN PCB

Albert Schweitzer, Fineline
albert.schweitzer@fineline-global.com

November 2021

The development in electronics is characterised by a steady increase in the power density of components. As a result, the problem of thermal load during operation of components and assemblies, due to increasing heat loss, is becoming increasingly important and should be considered with the greatest possible care as early as the planning and design phase. The resulting increase in system reliability is directly related to efficient PCB heat management.

1. General

For many years, developments in electronics have been characterised by a continuous increase in the power density of active components. This development can be shown very well in the following diagrams of the ITRS committee ("International Technology Roadmap for Semiconductors"). The two diagrams below show the development until 2025 with regard to transistor density (Fig. 1) and the ever decreasing structures of semiconductor devices (Fig. 2).

ITRS:
(Intern. Technology Roadmap for Semiconductor):

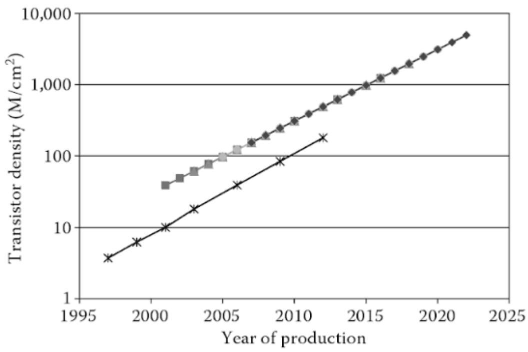


Figure 1: Increasing transistor density

ITRS:
(Intern. Technology Roadmap for Semiconductor):

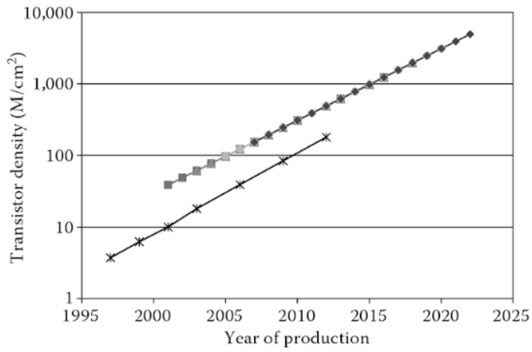


Figure 2: Smaller semiconductor structures

The power consumption of a semiconductor device is proportional to the on-chip clock frequency and the square of the supply voltage. $P \propto f \times V^2$. And although the supply voltages are getting smaller and smaller, their reduction is not sufficient to compensate for the increase in power consumption due to the increase in clock frequency and transistor density.

The trends in semiconductor technology identified by the ITRS Panel show that the heat dissipation of electronic equipment has increased steadily over the last decades and may become a limiting factor in the manufacture of high quality electronic equipment in the foreseeable future. As a consequence, the problem of thermal stress on components and assemblies is becoming more and more important and therefore thermal management in the design of electronic devices is becoming more and more important.

For the above-mentioned reasons, electronic assemblies and devices must be carefully dimensioned not only in electrical but also in thermal terms.

2. Some Basics

A very interesting analogy can be made between thermal quantities of thermodynamics, which we will primarily deal with in the following, and the electrical quantities of electrical engineering, which are generally known to us.

Thermodynamics: Thermal resistance = temperature difference / heat flow
Electrical engineering: Electrical resistance = voltage difference / current

Especially the thermal resistance R_{th} and its calculation should continue to be of interest to us. In analogy to electrical engineering, the same principles for series and parallel connection of thermal resistors apply here.

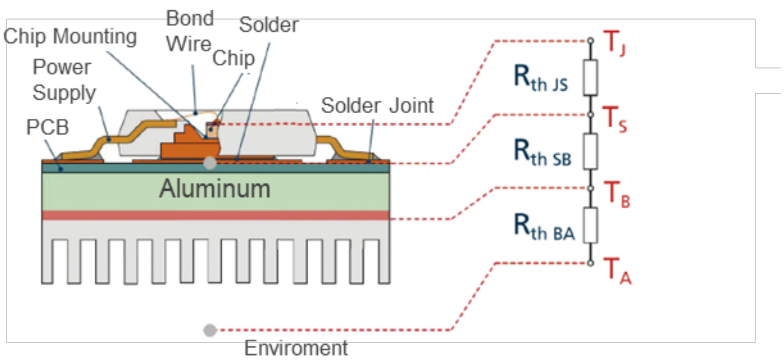


Figure 3: Series connection of thermal resistor
Using the example of an LED module.
Picture: Osram

But first the definition of thermal resistance. The thermal resistance R_{th} is a measure of the temperature difference that occurs in an object when a heat flow passes through it (heat flow = heat per unit of time or heat output). **The thermal resistance thus describes the inability of a material to conduct heat.**

The first law of thermodynamics, which says: "The energy of a closed system is constant", is also important as a basis. Here it means that electrical power is converted into forms of thermal energy. "Energy input - energy output = 0"
The temperature at steady state is the result of the equilibrium of the heating power P [W] and the cooling power in the form of **radiation, convection and conduction.**

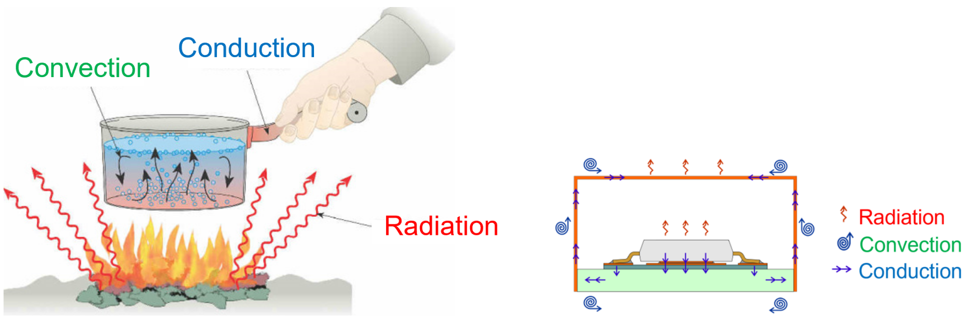


Figure 4: Visual representation of Convection, Conduction and Radiation.
Picture: Pearson Prentice Hall, Inc

The three, by their nature, completely different mechanisms of heat transfer - convection (heat transport), conduction (heat conduction, or molecular heat conduction) and radiation (heat radiation) - have already been mentioned. Although convection is the most effective method of heat reduction, we will only take a closer look at conduction in the following, as it is the determining mechanism for heat transfer in the environment of the PCB.

Conduction also simply called heat conduction, in physics is the heat flow in a solid or a resting fluid as a result of a temperature difference. According to the second law of thermodynamics, heat only flows in the direction of lower temperature. **Heat conduction is a pure energy flow without material transport.**

From the Law of Fourier:

$$\dot{Q} = \lambda\, A \frac{\Delta T}{l}$$

and the formula for heat conduction:

$$\dot{Q} = \frac{\Delta T}{R_{th}}$$

the equation for the thermal resistance can be derived:

$$R_{th\ Conduction} = \frac{1}{\alpha\ \ A}$$

l = Length of the heat transfer path (thickness PCB) [m].
λ = Specific thermal conductivity [W/m K]
A = effective surface area (front plus rear if necessary) [m²].

The **Thermal Conductivity** lambda (λ, k or κ) of a material is determined by the speed at which the heating at one point propagates through the material. **Thermal Conductivity** is therefore the ability of a material to transport thermal energy by thermal conduction.

$$\lambda = \frac{Q * l}{A * t * \Delta T} = \frac{[J] * [m]}{[m^2] * [s] * [K]} = \frac{[W]}{[m] * [K]}$$

The unit of thermal conductivity is given by [J/(m-s-K)], where: [J/s=W] and therefore: **Watt per Meter and Kelvin**.

Thermal Conductivity of Different Substances		
Material		Thermal Conductivity [W/m*K]
Carbon nanotubes	≈	6000
Diamond	≈	2300
Silver	≈	430
Copper	≈	400
Gold	≈	310
Aluminum	≈	240
Nickel	≈	85
Tin	≈	70
Solder (SnCuN)	≈	64
Aluminum Oxid	≈	28
Div. TIM Materials	≈	1 - 9
Polyimide	≈	0.37 - 0.52
Solder Mask	≈	0,25
FR-4	≈	0.2 - 0.9
Air	≈	0.0262

Table 1: Thermal conductivity of different materials.

The higher the value for Lambda λ, the better the thermal conductivity. For values below 0.8 W/m*K, the material in question is considered a thermal insulator. We are therefore dealing with a thermal insulator for the printed circuit board (FR-4 material). These are not exactly the best prerequisites for achieving good thermal conductivity.

3. Cooling concepts for printed circuit boards

There is a wide range of possibilities to realize an efficient heat management for printed circuit boards.

Cooling concepts for printed circuit boards
Thermal Vias
Metal Core printed circuits
Copper Coin / Inlay
Heavy Copper PCBs
Use of Heat Sinks
Use of Heat Pipes
Combination of different methods

Table 2: Heat dissipation concepts for Printed circuit boards. Source: Fineline

The following presentation focuses on the use and calculation of thermal vias and via arrays. Also, the topics metal core PCBs, copper coin / inlay technology and the use of thick copper in PCBs are briefly explained. The topics of heat sinks and heat pipes will not be dealt with further here, as these are "external" cooling methods.

4. Thermal Vias

A thermal via is no different from a standard via in terms of design and appearance. However, the function is different. A standard via has the task of establishing an electrical connection, while the thermal via has the task of establishing a thermal connection.

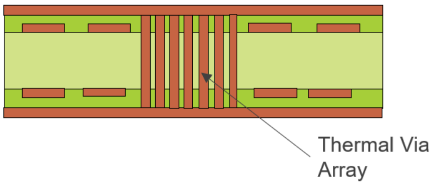
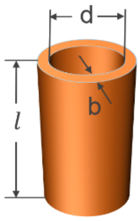


Figure 5: Schematic representation of thermal vias.
Picture: Fineline

First, we want to calculate the thermal resistance of a thermal via. For this purpose, we make the following assumptions:

Copper thickness: b = 25µm,
Diameter: d = 250µm,
Laminate thickness: l = 1,6mm,
λ_{Cu} = 384 W/mK,
A = π (d + b) b; A = 0,021588mm².



We have already learned the formula for the thermal resistance.

$$R_{th} = \frac{l}{\lambda\ A}$$

l = Length of the heat transfer path (thickness PCB) [m].
λ = Specific thermal conductivity [W/m K]
A = effective surface area (front plus rear if necessary) [m²].

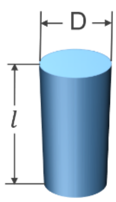
If you now insert the values above into the formula for calculating the thermal resistance, you get the value of 193 K/W for the thermal resistance of a single via.

We note: With a PCB thickness of 1.6mm, with a via diameter of 250µm and a copper thickness of 25µm in the sleeve, a single thermal via has a thermal resistance of 193 K/W.

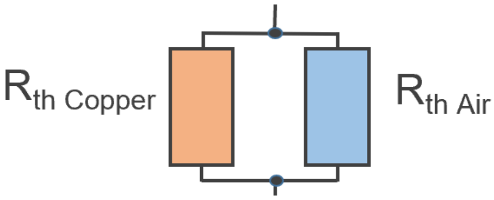
In the calculation carried out so far, it was not taken into account that in addition to the heat flow through the copper, there is also a heat flow through the air in the sleeve. This proportion is negligible, but it is to be calculated here anyway.

To calculate the thermal resistance of the air column within the via, we use the following assumptions:

Diameter: d = 250µm
Laminate thickness: l = 1,6mm
λ_{Luft} = 0.026 W/mK
A = π (D/2) ²; A = 0,049mm²



As expected, after inserting the values into the formula, we get a very high value, namely 1244712 K/W. Following the analogy between electrical engineering and thermodynamics, we are dealing here with a parallel connection of thermal resistors.



$$R_{th\,Total} = \frac{R_{th\,Copper}\; x\; R_{th\,Air}}{R_{h\,Copper} + R_{th\,Air}}$$

The calculation, according to the calculation that seems very familiar to us, gives the value and thus the thermal resistance of 192.98 K/W. We can therefore neglect the part that the air column contributes to the thermal resistance. In all further considerations it is no longer taken into account.

Now we want to have a look at several other variants of thermal vias:

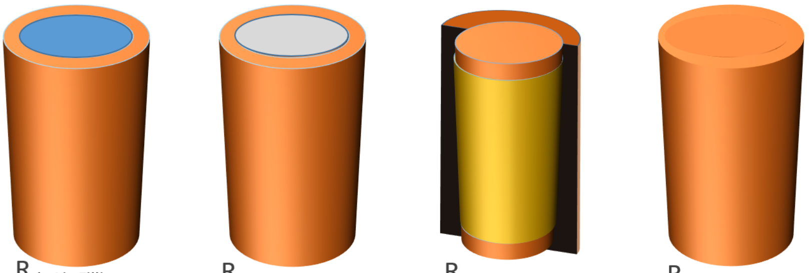


Figure 6: Thermal resistance with different via fillings.
Picture: Fineline

Next, by varying various parameters, we want to develop a feeling for how we can influence the thermal resistance of a via.

Calculation of the thermal resistance of a via:

with b = 25µm; λ = 384W/mK;

d:	A:	Rth: (l = 1,6mm)	Rth: (l = 1,5mm)	Rth: (l = 1,0mm)	Rth: (l = 0,5mm)
150	0,0137375	303,31	284,35	189,57	94,78
200	0,0176625	235,90	221,16	147,44	73,72
250	0,0215875	193,01	180,95	120,63	60,32
300	0,0255125	163,32	153,11	102,07	51,04
350	0,0294375	141,54	132,70	88,46	44,23
400	0,0333625	124,89	117,09	78,06	39,03
450	0,0372875	111,74	104,76	69,84	34,92
500	0,0412125	101,10	94,78	63,19	31,59
600	0,0490625	84,93	79,62	53,08	26,54
700	0,0569125	73,21	68,64	45,76	22,88
800	0,0647625	64,34	60,32	40,21	20,11
900	0,0726125	57,38	53,80	35,86	17,93
1000	0,0804625	51,78	48,55	32,36	16,18

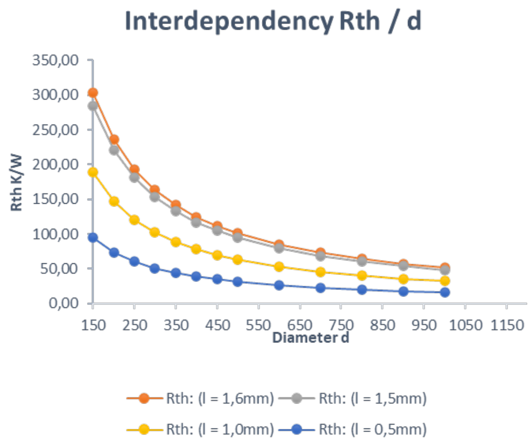
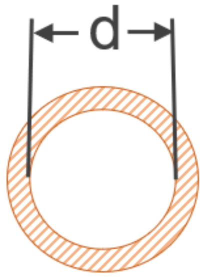


Table 3: Variation of different parameters in the environment of a via.
Source: Fineline



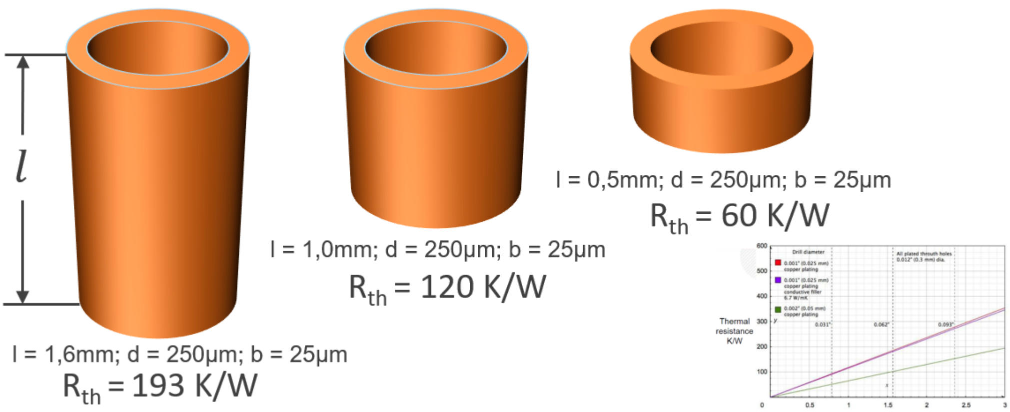


Figure 7: Thermal resistance at different board thicknesses.
Picture: Fineline

Now we want to calculate the thermal resistance of a via array.

The formula for this is very simple. You only have to divide the calculated value of a thermal via by the number of vias in the via array.

$$R_{th\ \text{Via-Array}} = R_{th\ \text{Single-Via}} / \text{Number of vias in array!}$$

In the previous calculation, the FR-4 material between the vias in the via array was not yet taken into account. The contribution of the FR-4 material to the thermal resistance is very small, since we are dealing with a thermal insulator. If this FR-4 material is taken into account, the thermal resistance decreases in the range of 2-3 K/W per via.

A very common problem is mentioned here. Open thermal vias under the thermal pad of a component can cause the so-called solder wick problem. The solder tends to flow into the vias during the reflow process and creates voids in the vias. The problem can be solved to a large extent by choosing thermal vias with a small diameter (finish diameter). A recommendation can be found in the following figure 8.

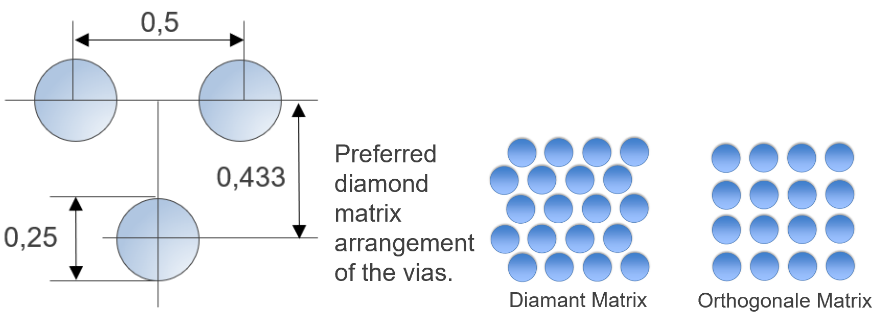


Figure 8: Fineline recommendation for diameter and pitch of thermal vias.
Picture: Fineline

Of course, the number of vias in the via array also plays an important role. Figure 9 shows the original footprint of the thermal pad of the GaN Systems Inc. component GS6650xP (Layout #1). In various layouts (Layout #2 - Layout #8) an ever-larger thermal pad with an increasing number of thermal vias was created to find an optimum for the number of thermal vias.

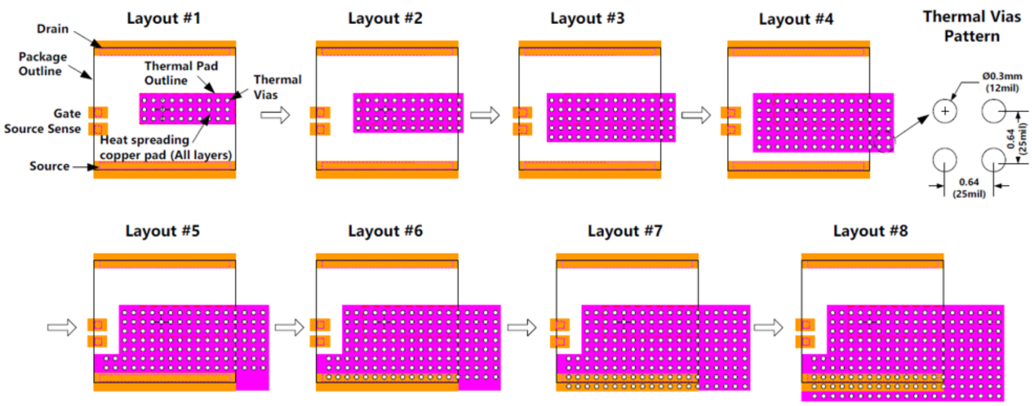


Figure 9: Thermo-Pad Layout #1- Layout #8 to optimize the number of thermal Vias in a via array. Picture: GaN Systems Inc.

By means of the following graph (Fig. 10) the optimum for the number of vias can be seen very clearly. This is layout #5 with a total of 123 thermal vias. Here a pitch of 0.65mm and a diameter of 0.3mm was defined. The arrangement of the vias could be further optimized by a "diamond matrix".

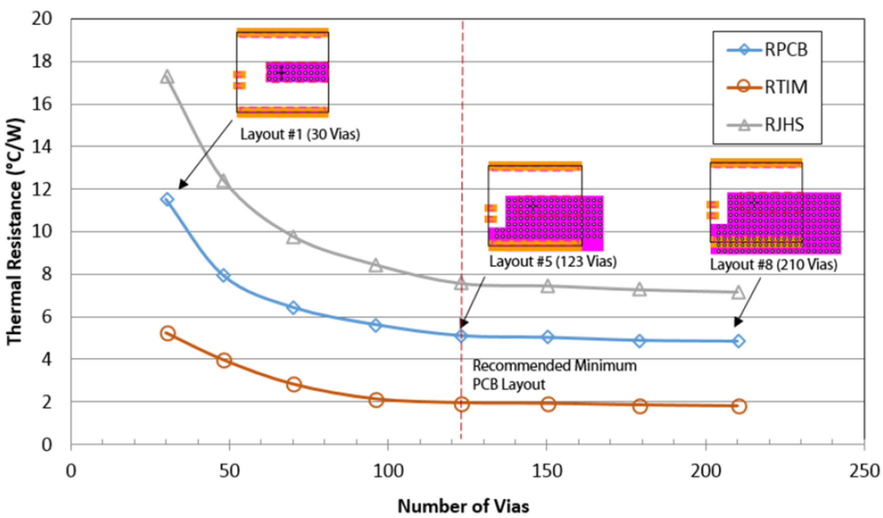


Figure 10: Optimum determination for the maximum number of thermal vias in a via array. The optimum is achieved with layout #5 with a total of 123 thermal vias. Picture: GaN Systems Inc.

5. Heat spread through copper inner layers

In all PCB layouts, the thermal resistance decreases proportionally to the number of PCB layers. The underlying effect is the so-called thermal spread. Figure 11 shows the principle of the heat spread.

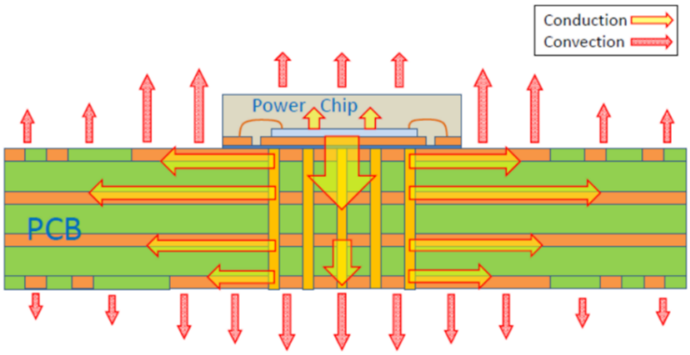


Figure 11: Principle representation of heat spread.
Picture: ON Semiconductor

The thermal resistance of a 2-layer board can be reduced by 30% by adding another 6 layers (8-layer ML board). Very good information on this topic can be found in the IPC-2152, where the positioning of the inner layers in relation to the heat source and the layers in relation to each other is discussed in detail.

6. Influence of the copper thickness of the inner layers on the thermal resistance

An optimum can also be found for the copper thickness with regard to thermal resistance. See Figure 12.

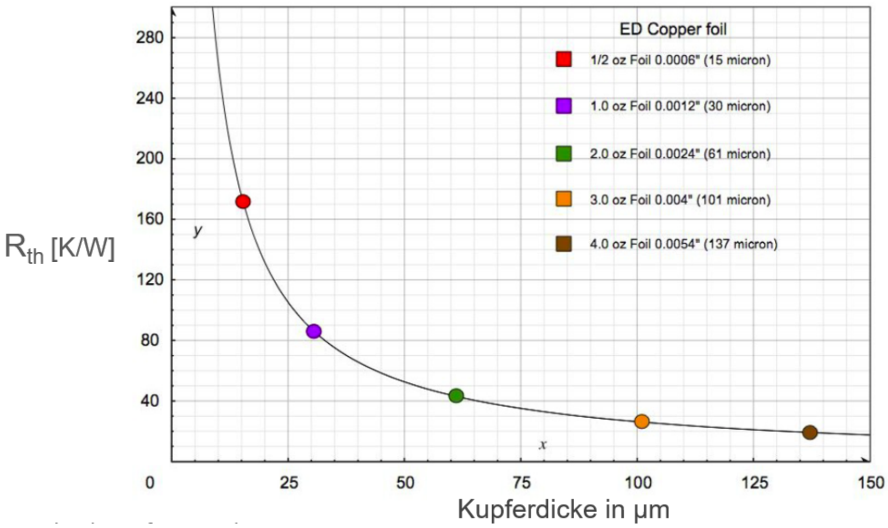


Figure 12: The thermal resistance as a function of copper thickness.
Picture: TTM

As can easily be seen, an optimum is achieved with a copper thickness of 70µm. The price for the increase to 105µm copper thickness, which shows very little effect, is about as expensive as the increase by two additional copper layers. Increasing the number of layers is, as shown above, but much more effective in terms of thermal management.

6. Metal-Core Printed Circuit Boards. MCPCB

MCPCB is a generic term for all types of printed circuit boards that have a metal layer or a metal core. Also, IMS printed circuit boards (Insulated Metal Substrate) belong to this category. If you can manage with only one layer in the layout of your circuit, as is often the case with PCBs for LED applications, MCPCBs are an effective tool to keep the thermal resistance very low.

Compared to FR-4 materials, the thermal conductivity of aluminum is reduced by factor 1000 higher, as can be seen in Table 1.

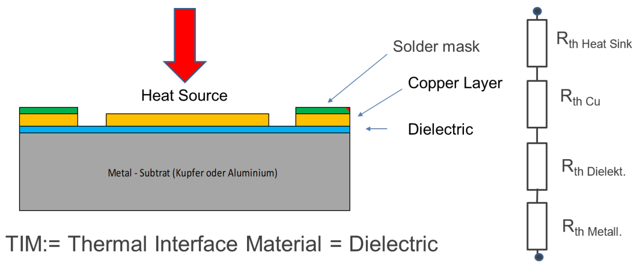


Figure 13: Basic MCPCB structure.
Picture: Fineline

Special attention should be paid to the dielectric (insulation layer between copper and metal). Very good thermal conductivity, low thermal resistance and high dielectric strength are required here. To further reduce the thermal resistance of the TIM material, filler materials within the dielectric, such as aluminum nitrides (117W/mK) or beryllium oxides (251W/mK), are often used.

Usually the MCPCB materials are more expensive than FR-4 materials. Before use, the use of e.g. Thermo-Vias in combination with FR-4 base material should be tested.

7. Copper Coin / Inlay Technology

The Coin / Inlay technology was developed to improve the heat dissipation of high-performance components. It is perhaps the most cost-intensive, but also the most effective and direct method of conducting heat away from a specific location around the PCB. There is no difference between the terms "copper co-in" and "copper inlay"; they describe the identical technology. The round shape, hence, the name Coin, offers advantages in terms of mechanical tolerances in the manufacturing process.

As shown in Figure 14, a copper block, e.g. in a round, rectangular or other shape, is implemented in the circuit board either with or without connection of the inner layers. The outer layers of the circuit boards are always contacted as well.

There are different ways of integrating the inlay into the PCB. The most cost-effective variant is the press-fit method, in which only the outer layers of the PCB are contacted.

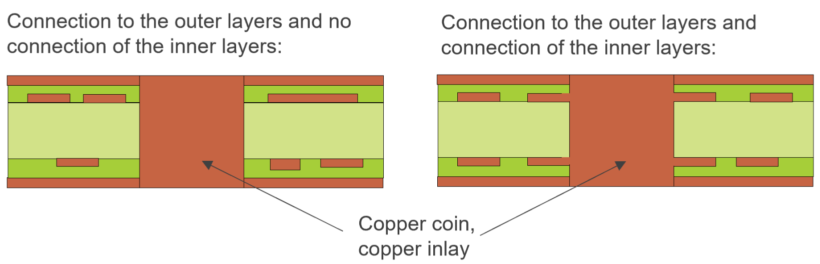


Figure 14: Basic structure of a copper coin / inlay printed circuit board.
Picture: Fineline

Compared to a Thermo-Via-Array, a copper coin has about 10 times lower thermal resistance as shown in Figure 15.

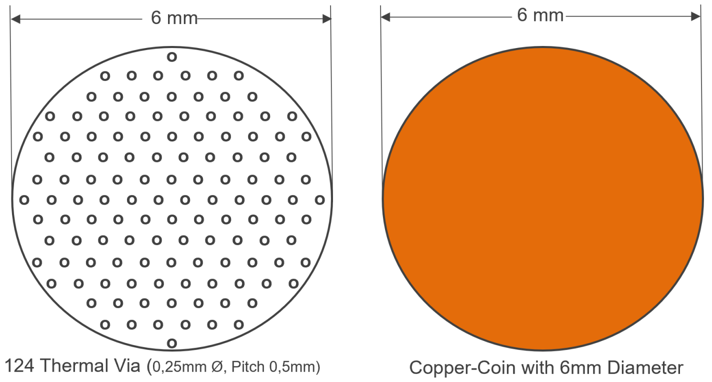


Figure 15: Comparison of a thermal via array with a copper coin / inlay.
Picture: Fineline

8. Thick Copper Circuit Boards

The printed circuit board industry speaks of thick copper in copper thicknesses of 70 µm - 400 µm. Thick copper technology is particularly important for Current carrying capacity of conductor tracks. To reduce the thermal resistance, a copper thickness of 70µm is sufficient for economic reasons, as shown in Figure 12.

9. Closing Remarks

The importance of thermal management in printed circuit boards should be clear to every designer and layout artist, because the system reliability of any electronics and electronic assembly is directly dependent on efficient thermal management at the board level. It is highly recommended to consider thermal management at the beginning of the development cycle to achieve cost and time targets and the desired reliability.

10. References

[1] GaN Application Note, "PCB Thermal Design Guide for GaN Enhancement Mode Power Transistors, (2016) GaN Systems Inc."
[2] Dr. Johannes Adam, "Die Leiterplatte als Kühlkörper"
[3] John H. Lienhard IV/V, "A Heat Transfer Textbook"
[4] Younes Shabany, "Heat Transfer, Thermal Management of Electronics"
[5] John H. Lau, "Thermal Stress and Strain in Microelectronics Packaging"