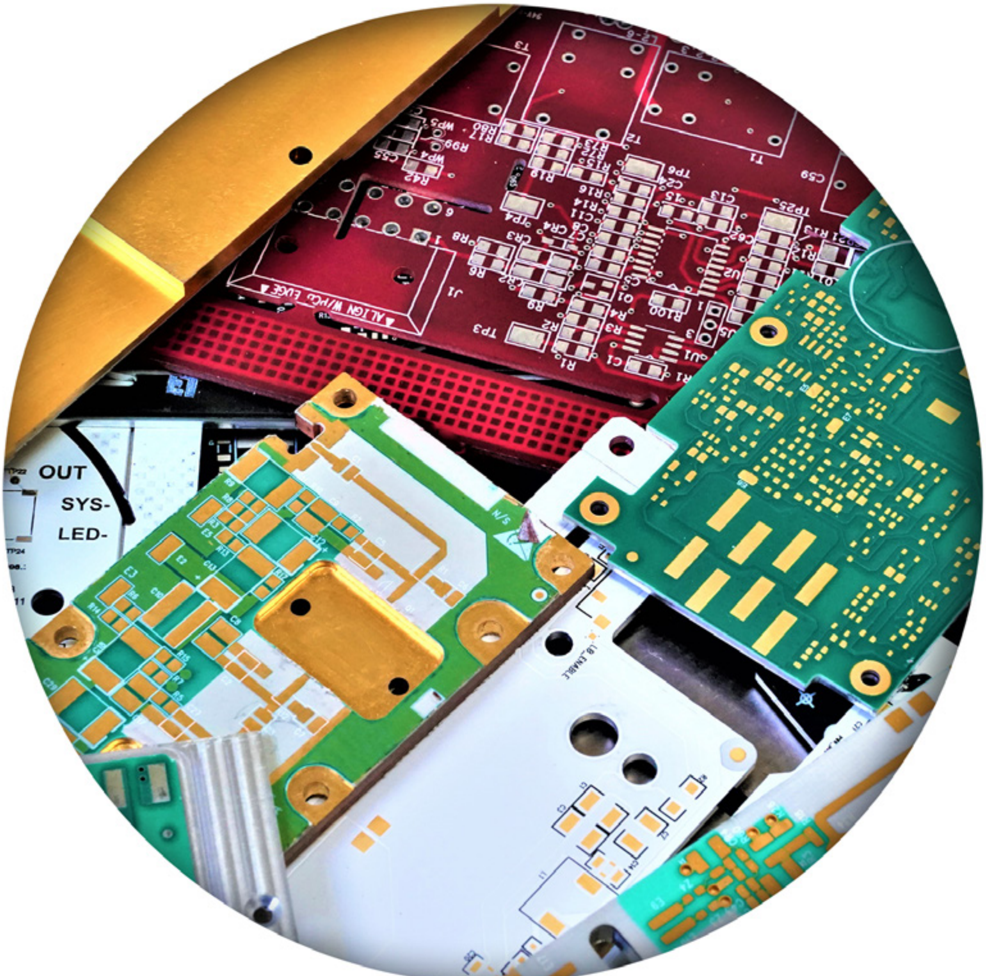
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™

Thermal Management:

A Fabricator's Perspective



Anaya Vardya
American Standard Circuits

I-007e Books

The Printed Circuit Designer's Guide to...™ Thermal Management

A Fabricator's Perspective

Anaya Vardya

American Standard Circuits

BR Publishing, Inc.
dba: I-Connect007
942 Windermere Dr. NW
Salem, OR 97304
U.S.A.

ISBN: 978-1-7342005-5-3

I-007eBooks

Visit I-007eBooks.com for more books in this series.

I-Connect007

GOOD FOR THE INDUSTRY

I-Connect007.com

This book has been reviewed for technical accuracy by the following experts from the PCB industry.

Denis Boulanger
Principal Manufacturing Engineer
Raytheon Technologies

Denis Boulanger has over 35 years' experience in the electronics manufacturing business and is currently a principal manufacturing engineer at Raytheon Technologies. Boulanger was previously a senior manufacturing engineer at Technovative Applications. He spent 25 years in the PCB laminate industry for Rogers, Neltec, Taconic, and Ventec USA in positions from applications engineer to sales and applications engineering manager. Before this, Boulanger held various positions at M/A-Com, from process engineer to manufacturing manager. He holds a master's degree in manufacturing engineering from the University of Massachusetts and bachelor's degrees in both industrial engineering and accounting from the University of Rhode Island.

Dr. Johannes Adam
Consultant, ADAM Research

Dr. Johannes Adam received his doctorate in physics from the University of Heidelberg, Germany, in 1989 with a thesis on the numerical treatment of three-dimensional radiative transfer. He then worked in various software companies, mainly on numerical simulations of electronics cooling and in customer support. In 2009, Dr. Adam founded ADAM Research and now works as a technical consultant for electronics development companies and as a software developer. He is the developer of the thermal simulation program TRM (Thermal Risk Management), a 3D analysis tool targeted at electronics engineers and PCB designers who need to solve complex thermal issues at the PCB level. Dr. Adam's 20 years of experience in this field provide him with deep insight into the physical and electrothermal aspects of PCBs and assemblies. He is a member of the German section of the IPC (FED e.V.) and participates as a speaker in its seminars on thermal topics. Dr. Adam is also a certified interconnect designer (CID).

MEET THE AUTHOR

Anaya Vardya
President and CEO
American Standard Circuits

Anaya Vardya has over 35 years in the electronics manufacturing business and is currently the president and CEO of American Standard Circuits Inc. Anaya was previously COO of Canadian-based Coretec Inc. and senior VP of operations at Merix (both now part of TTM). He was also the corporate development manager at Continental Circuits, and before that, he had a variety of management and non-management positions in IBM Endicott and IBM Austin. Anaya holds a master's degree in chemical engineering from the University of Cincinnati and a bachelor of technology from the Indian Institute of Technology.

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

Contents

Introduction

Thermal management in the printed circuit board (PCB) world is big business! A recent Markets and Markets report projects the thermal management market to reach $16 billion by the year 2024 with an average CAGR of 8% over that period [1]. This is one of the fastest-growing segments of the PCB business and far outpaces the projected growth for the overall industry. While demand was originally driven by high-power telecommunication and mil-aero applications, it has rapidly expanded to include automotive, consumer electronics, and medical sectors. The components used in any electronic assembly generate heat whenever an electrical current flows through them, and the amount of heat depends on the particular attributes of the design (power requirements, design characteristics, transmission speed, etc.).

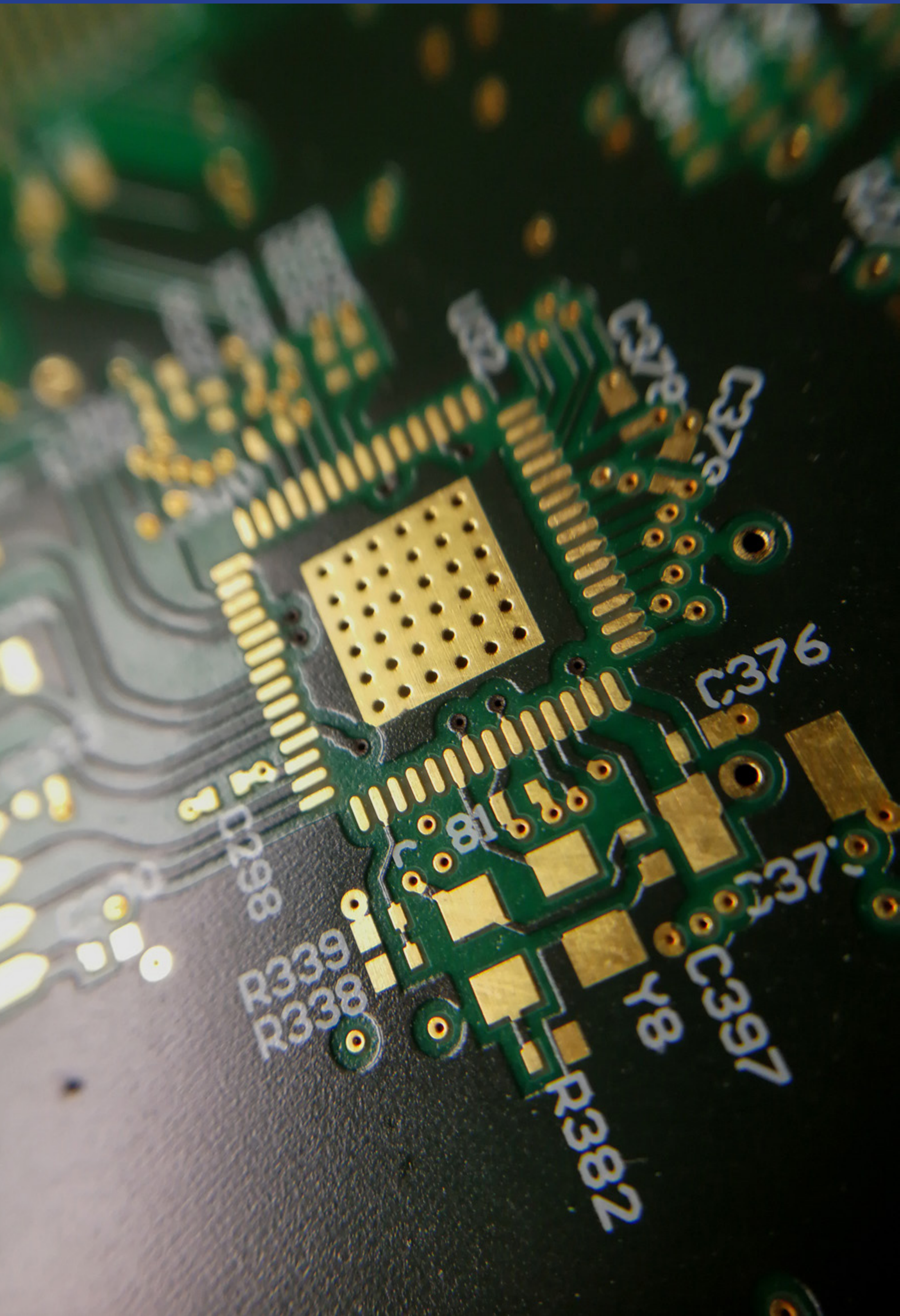
In addition to the heat generated from the electronic components, the resistance of the electrical connections, copper trace configuration, and PCB via structures contribute to the thermal output of the product. While RF/microwave and IMPCB applications hold the lion's share of thermal management challenges, reduced PCB footprints combined with increased component densities can require advanced thermal management solutions on "vanilla" designs.

In our experience working with PCB designers throughout the years, there is a wide range of knowledge on the design side regarding the impact of thermal management design decisions on the PCB manufacturing process, and ultimately, product success. As we strenuously encourage early engagement between the designer and the PCB fabricator in all cases, it is particularly critical when developing an advanced thermal management solution. A disconnect between what the original design manufacturer (ODM) wants in performance, and what the printed circuit fabricator recommends for the application is the biggest reason for an unsuccessful build of a new PCB design. While this book describes a number of techniques, the best solution for an individual design will need to be modeled.

It is important to understand a couple of terms right from the start: thermal conductivity and thermal management. Thermal conductivity is the property of a material to conduct heat, while thermal management is the process of analyzing the system as a whole and effectively dissipating the thermal energy away from the heat source. We have chosen to focus this book on providing designers a thermal management desk reference on the most current thermal management techniques and methods from a PCB fabrication perspective, including a case study on an extreme mixed-technology design that we recently produced. We hope you find value in our efforts.

Reference

1. Markets and Markets, “Markets and Markets report projects the thermal management market to reach $16 billion by the year 2024 with an average CAGR of 8% over that period,” July 2019.


C376
D298
R339
R338
Y8
C397
R382

Chapter 1

Thermal Vias

Heat cannot be efficiently exchanged with stagnant air surrounding a hot device; however, it can be transferred away from the electronic component to the PCB using thermal vias. A thermal via is a good conductor of heat that runs between the top layer and bottom layer of the PCB, dissipating heat through simple conduction. In simple terms, thermal vias are plated holes located under, or electrically connected to, a surface-mounted heat source on a PCB that allows heat transfer through the hole (Figure 1-1).

The efficient vertical heat transfer through thermal vias in the Z-axis is especially important to allow heat distribution over large areas of the PCB in the X-axis. Thermal vias run the design continuum from simple through-hole vias connecting the two external layers to complicated buried and blind microvias in stacked or staggered structures.

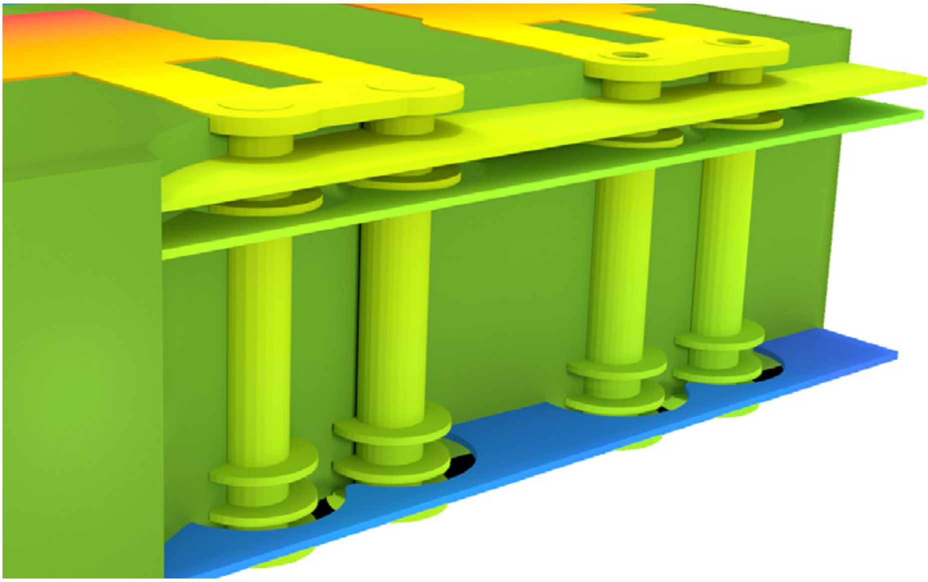


Figure 1-1: Simcenter simulation image courtesy of Siemens Digital Industries Software.

It should be noted that thermal vias, while the least expensive method of thermal management, are not always effective. It depends on the distribution of heat sources, the layout of the planes, and the cooling conditions on the bottom side. Also, if the air on the other side is stagnant, thermal vias are almost only useful with a heat sink on the bottom side.

This chapter will address three common thermal via designs: thermal via arrays, copper planes, and via fill.

Thermal Via Arrays

The premise that the thermal loss of a component is primarily transferred to the base of the package allows for the design integration of a path for heat dissipation into the physical PCB. A very common and cost-effective approach is to place an array of PCB thermal vias directly underneath the component (Figure 1-2). After the component is soldered to the PCB, the base of the component is connected to the thermal vias on the top side of the PCB.

Heat is then dissipated through these thermal vias down to the bottom side of the PCB. The efficiency of heat transfer through thermal vias is

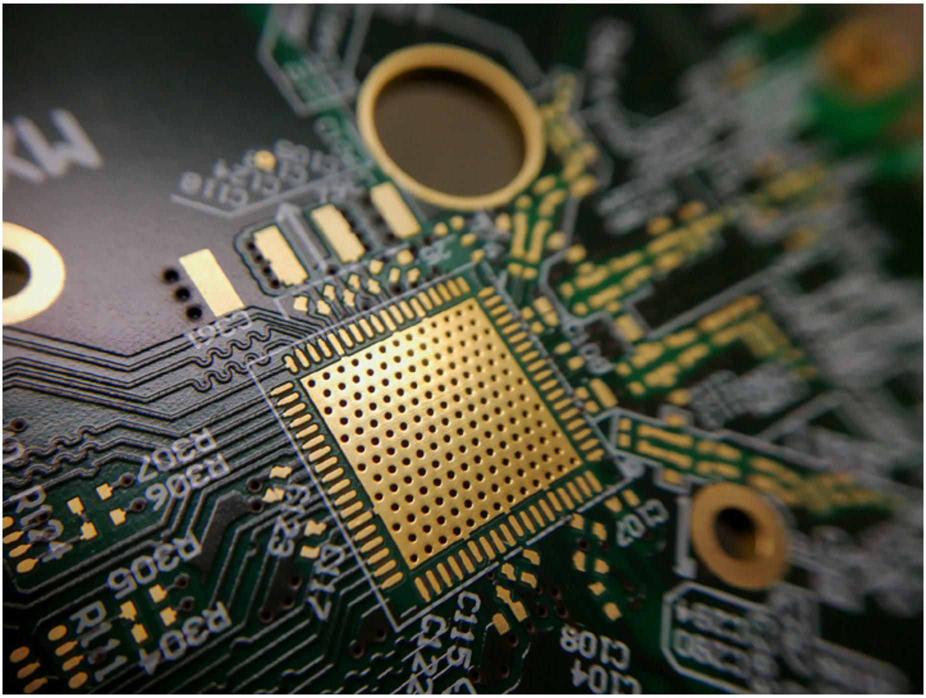


Figure 1-2: A QFN package with PCB thermal vias to assist with heat dissipation.

directly related, but non-proportional, to the amount of copper available to dissipate the heat. That being said, in many cases, the use of standard open vias (unfilled) may not provide the required thermal transfer on their own as the amount of actual copper is limited to the circumferential sides of the via. The most common method to improve this thermal performance is to combine the thermal vias with other thermal management techniques, such as copper planes, via fill, or heat sinks. A thermal simulation should be done.

Design Considerations

A design consideration is that adding unfilled (open) plated through vias to the SMT pad (via-in-pad) creates some challenges because it may result in a solder wicking issue during assembly. This means that solder tends to flow (wick) down into the vias during the assembly reflow process, which may create solder voids on the pad.

If open thermal vias need to be used, there are some things that can be done to minimize this problem: For example, use a small via diameter. The surface tension of solder limits the amount of solder wicking on smaller vias. With 0.3 mm or smaller via diameter, the solder wicking can be reduced. You can also fill the vias with thermally conductive materials. It eliminates the solder wicking but adds cost and an extra manufacturing step (see the via fill section).

Further advice includes the following:

- Eight mils (0.2 mm) is the typical minimum mechanical drilling size. Twelve mils (0.3 mm) is more common and lower cost if the design will permit
- IPC-6012 specifies a minimum 20-µm (0.8-mil) copper plating thickness for a Class 2 PCB, but manufacturers target 25 µm (1 mil). As discussed earlier, the more copper thickness plated in the via, the more heat can be transferred
- PCB thickness may have an impact on the thermal via performance

Copper Planes

There are two basic ways to use copper planes in thermal management; utilizing the existing power and ground planes (passive) and an internal copper core (active). This section will focus on the passive power/ground technique as internal metal cores will be discussed in Chapter 3 on metal-core boards.

Designers should try to incorporate some minimal level of passive heat transfer to dissipate heat into the ground plane. Many heat-generating components include a thermal pad on the bottom of the package to allow heat to be dissipated to a ground plane through plated vias. While this passive method works for some designs, many others require a more active thermal management solution. In either case, utilizing the existing ground planes in a design adds virtually no cost to the PCB.

Via Fill

Filled vias can improve routing density and board assembly and aid with electrical and thermal performance. However, via fill is a significant cost driver requiring specialized equipment and should only be used if absolutely required by the design requirements.

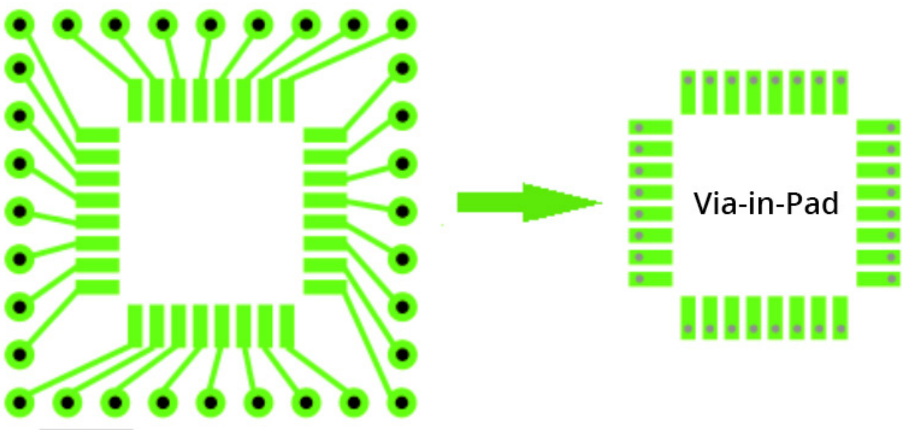


Figure 1-3: Conversion of QFP configuration to via-in-pad.

Advantages of via fill technology include:

- Tighter BGA pitches
- Increased thermal dissipation
- More space on the surface of the PCB for components
- Reduced layer count and/or PCB size
- More design options for improved signal performance
- Via-in-pad technology (Figure 1-3)
  - o Microvia-in-pad
  - o Through-hole via-in-pad

Non-Conductive

The three most commonly used non-conductive products are SAN-EI PHP-900 IR-10F, Taiyo THP-100 DX1, and Peters PP 2795 (Table 1-1). All three products are a thermally cured epoxy formulation. Non-conductive fill has a typical thermal conductivity of 0.25 W/mK. This is the preferred method of filling vias. Most PCB fabricators typically have one that they prefer to use. It is a good idea to work with your PCB fabricator in terms of selection for your design. Most designers leave the choice of the non-conductive supplier to the PCB fabricator.

	SAN-EI PHP-900 IR-10F	Taiyo THP-100 DX1	Peters PP 2795
Solid Content	100%	100%	100%
Tg	160°C	160°C	140°C
Sanding Toughness	Tough	Tough	Easy
Desmear Performance	Good	Good	Dishdown
Hole-Filling Qualities	Good	Good	Inconsistent
Pot Life at Room Temperature	5 Days	10 Days	6 Months

Table 1-1: Non-conductive fill properties.

Conductive Via Fill

The two most commonly used conductive via fill materials are DuPont CB100 and Tatsuta AE3030 (Table 1-2). Both products utilize a silver-coated copper particle formulation, which provides both electrical and thermal conductivity in a cured state. The difference between the two materials is in the overall particulate size and finished coefficient of thermal expansion (CTE); CB100 has a larger particle size and higher CTE. Conductive fill typically has a thermal conductivity ranging from 3.5–10 W/mK. In contrast, electroplated copper has a thermal conductivity of more than 350 W/mK.

	Tatsuta AE3030	DuPont CB100
Filling	Silver-Coated Copper Powder	Silver-Coated Copper and Silver Particles
Copper Particle Size	0.24–0.3 mils	Up to 5 mils
Resin	Epoxy	Epoxy
Percentage Solid Content	100%	92%
Color	Grey	Silver
Viscosity	150 PaS	130 PaS
Density	4.2 g/cm3	5.5 g/cm3
Shelf Life	1.5 months	3 months
Pot Life at Room Temperature	24 hours	24 hours
Storage	Freezer	Freezer
Dry and Curing	60 min. at 170°F 60 min. at 240°F	60 min. and more at 250°F 60 min. and more at 350°F
Volume Resistivity	0.0003 Ohm cm	0.00016 Ohm cm
CTE Below Tg	40	34
ppm/°C Above Tg	86	47
Thermal Conductivity	7.8 W/mK	3.5 W/mK
Tg	171	115

Table 1-2: Conductive fill properties.

Conductive or Non-Conductive?

The primary consideration in this decision is to match the via fill CTE with that of the surrounding laminate material as closely as possible. Why is that important? Because the constant thermal cycles of the PCB in the field can lead to stress fractures and possible electrical opens with mismatched CTEs. You may find that while conductive via fill may offer marginal conductivity in some applications, using non-conductive fill often results in superior thermal and electrical conductivity with minimal cost impact. Increasing the plated copper thickness in the via by even 0.1 mils has a much greater impact on heat transfer than conductive fill. Conductive fill material is also typically significantly more expensive than non-conductive via fill.

Plated Microvias

Today's copper via fill processes involve a two-step plating process using a copper plating bath with highly modified inorganic concentrations and special organic additive packages. These systems were created to effect the preferential plating at the bottom of the hole. These baths have some known limitations and are most well understood when utilized plating the most common small via fill diameters between 0.006–0.010" (6–10 mils). In all cases, the aspect ratio should not exceed 1:1, meaning they should at least be as wide or wider than they are deep. Figure 1-4 shows a cross-section of a mechanically drilled microvia that has been plated shut.

The drill tool design must also be considered; in the case of a 6–10-mil via, which will subsequently be filled with copper plating, this will be either a 165°-point geometry standard drill bit or a conical tool may be used. In either case, a specific plating cycle must be employed to achieve properly filled plated vias.

Another consideration is that of a capture pad or traditional interconnect. In the case of the capture pad, the drilling tool will simply come down and make contact with the termination layer. Where a conical tool is utilized, this contact area may be quite small. In fact, in terms of actual copper contact, it is considerably less than a traditional interconnect.

In a standard multilayer interconnect, the copper thickness of the target layer at lamination plays a factor in how much copper surface area is contacted by the plating in the hole. Doubling the inner layer thickness will double the amount of contact area. This has less if any impact on the contact area of a contact pad using either controlled depth methods.

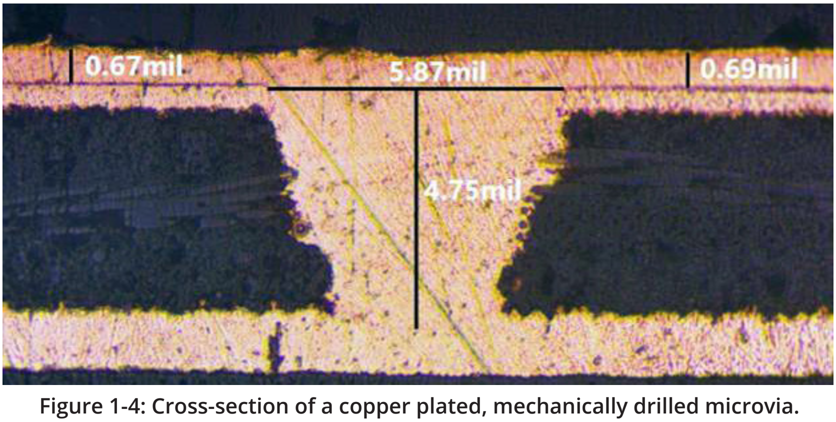
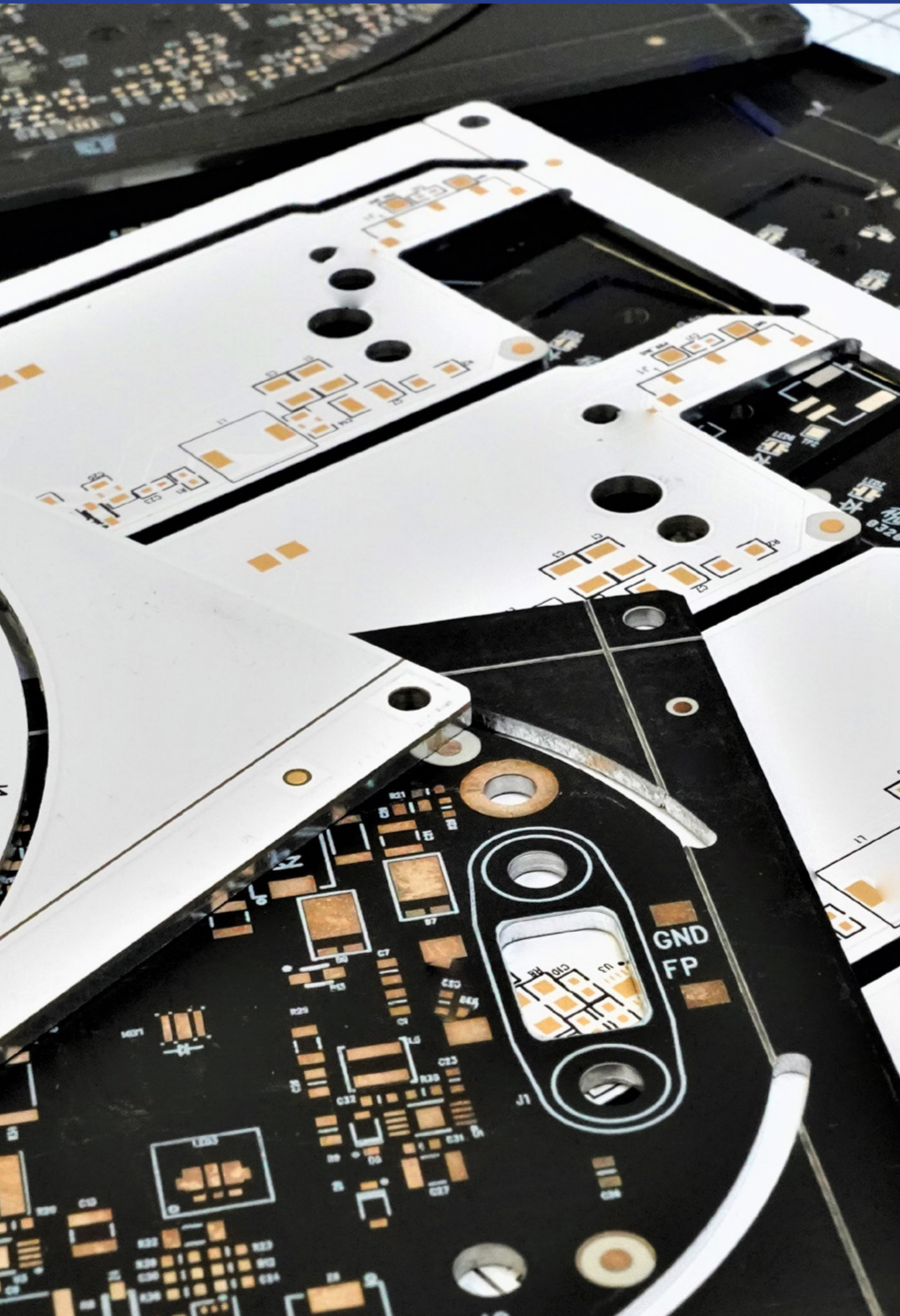


Figure 1-4: Cross-section of a copper plated, mechanically drilled microvia.


GND
FP
J1

Chapter 2

IMPCBs or MCPCBs

Insulated metal PCBs (IMPCB) or metal-clad PCBs (MCPCB) are a thermal management design that utilizes a layer of solid metal to dissipate the heat generated by the various components on the PCBs. When metal is attached to a PCB, the bonding material can either be thermally conductive but electrically isolative (IMPCBs or MCPCBs), or in the case of RF/microwave circuits, the bonding material may be both electrically and thermally conductive. The reason that RF designers usually have the bonding material thermally and electrically conductive is that they are using this not only as a heat sink but also as part of the ground layer. The design considerations are quite different for these different applications.

This chapter will focus on the IMPCB design considerations, and Chapter 4 will focus on RF thermal management. We will focus on things designers should be discussing with their PCB supplier to ensure manufacturability and a successful product launch. Since the choices, options, and decisions can be extremely complicated, it is critical to engage early and collaborate with the PCB fabricator about the specific design to ensure the most cost-effective solution.

Some of the more common applications of IMPCBs include:

- **Power Conversion:** Thermal-clad offers a variety of thermal performances, is compatible with mechanical fasteners, and is highly reliable
- **LEDs:** Using thermal-clad PCBs ensures the lowest possible operating temperatures and maximum brightness, color, and life
- **Photovoltaic Energy:** Renewable energy to power telecommunications, military camps, residential and commercial structures, and battery charging stations
- **Motor Drives:** Thermal-clad dielectric choices provide the electrical isolation needed to meet operating parameters and safety agency test requirements

- **Solid-State Relays:** Thermal-clad offers a very thermally efficient and mechanically robust substrate

- **Automotive:** The automotive industry uses thermal-clad boards, as they need long term reliability under high operating temperatures coupled with their requirement of effective space utilization

IMPCB Benefits

- Excellent surface mount cooling
- High electrical isolation, insulation, and thermal dissipation
- Low cost
- Robust thermal performance
- Thermal conductivity of the dielectric in the range of 0.6–8 W/mK
- Manufacturability (integrates with standard PCB processing)

Thermal Properties Explained

A thorough understanding of a number of different thermal properties is needed to be able to design the appropriate IMPCB solution to a thermal condition, including thermal conductivity, thermal impedance, and thermal resistance.

Thermal Conductivity

- Measurement of the ability of a substance to conduct heat (W/mK)

- A material property, meaning that it does not change when the dimensions of the material change, as long as it is made up uniforml. For example, the thermal conductivity of 1 cm^3 of gold is exactly the same as the thermal conductivity of a 100 m^3 of gold

- Generally obtained in the industry using one of two tests: The D-5470 test, or the E-1461 standard ASTM tests

- The D-5470 test measures the thermal impedance in Kcm^2/W of the sample and determines the thermal conductivity through the following relation:

*Thermal conductivity = Thermal diffusivity * Specific heat capacity * Density*

Thermal Impedance

- This is the opposite of thermal conductivity. It is a measurement of the ability of a material to oppose the flow of heat, so from a PCB point of view, we want this value to be low. The lower the thermal impedance, the quicker heat flows through the PCB and to the heat sink where it is dissipated

- Its value depends on the thermal conductivity of the material and its thickness; in other words, this is not a material property, but is an object property, as changing the thickness of the material will change this value. However, changing the area of the material will not change this value (as long as the thickness stays constant)

- For example, the thermal impedance of a sheet of laminate is the same as the thermal impedance of a cut piece of the laminate, say 1 cm^2 of it. Whereas the thermal impedance of a sheet of gold of 1-mm thickness is different from the thermal impedance of a sheet of 2-mm thickness
- This is generally obtained using the D-5470 test mentioned above and relates to the thermal conductivity via the following relation:

Thermal impedance = Thickness / Thermal conductivity

Thermal Resistance

- Thermal resistance (measured in K/W) is basically the same as the thermal impedance. The difference is that it takes into account the area of the sample as well as the thickness and conductivity
- Changing either the thickness, or the area of the material, will change the associated value of the thermal resistance as follows:

*Thermal resistance = Thickness / (Thermal conductivity * Sample area)*

Single-Sided IMPCBs

In its simplest form, an IMPCB is a piece of copper foil that is bonded to a thermally conductive dielectric and a metal substrate (Figure 2-1). Typically, a PCB supplier can buy the copper foil laminated to the base metal from several different laminate manufacturers. A laminate selector guide is provided in Appendix B.

Figure 2-1: Single-sided IMPCB.

Some of the key design factors to consider include the following.

Copper Thickness

Typically ranging from 1–6 ounces with 1 and 2 ounces being the most commonly used. The thicker the copper, the more expensive the cost of the PCB.

Thermally Conductive Prepreg

This is one of the most important elements of this construction and what typically differentiates the various suppliers. This is the substance that both electrically isolates the copper circuitry from the main metal and helps with the rapid transfer of heat between the two. It ensures that heat generated by the components is dispersed to the base metal (heat sink) as quickly as possible. The prepreg is typically an organic resin with ceramic fillers to increase thermal conductivity. The filler type, size, shape, and percentage are some of the factors that determine the thermal conductivity performance. The usual ceramic fillers are Al_2O_3, AlN, BN, etc.

The performance of the various prepregs is measured by the thermal conductivity (W/mK) and thermal impedance (Km2/W). The higher the thermal conductivity, the better the heat transfer, and the lower the thermal impedance, the better the heat transfer. However, it is also important to understand that the better the heat transfer associated with the prepreg, the greater the cost. Therefore, it is critical not to over-design. To put this in perspective, the thermal conductivity of FR-4 is approximately 0.2–0.4 W/mK, whereas the thermally conductive prepregs that are available on the market today range from 1–7 W/mK. Apart from thermal conductivity, the thickness of the dielectric can be critical. Typically, the thickness of the dielectric ranges between 2–6 mils, with 3-mil dielectric as the most common.

Base Metal

Aluminum is the most common base metal used. The two most common types are 5052H32 and 6061T6. 5052H32 is typically less expensive and a lot more available than 6061T6. The thickness of the aluminum typically ranges between 40–120 mils, but 40 and 60 mils are the most common thicknesses available.

Metal Base Material	Thermal Conductivity (W/mK)	Thermal Expansion (ppm/K)	Comments
Aluminum 5052 H32	138	25	Al-Mg-Cr alloy: Best for bending and mechanical forming, most popular choice, low-cost.
Aluminum 6061 T6	167	25	Al-Mg-Si-Cu alloy: Best for CNC machining and V-cut scoring, medium-cost.
Copper C110	386	17	Pure Cu: Low CTE, high thermal conductivity, high-cost.

Table 2-1: Properties of various base metals.

There are also cases where copper is used as a base metal. Copper is sometimes used for better thermal conductivity, mechanical strength, and CTE match to thicker copper foils. In most applications, the thermal advantages of the copper base plate are insignificant because the thermal resistance of the base metal is small relative to the thermal resistance of the dielectric layers and the components. This is a significantly more costly solution, as well as significantly heavier. A brief comparison of the various base metals is illustrated in Table 2-1.

Maximum Operating Temperature

Work with your PCB fabricator and raw material supplier to ensure that the MOT you require is being met by the material selected.

Breakdown Voltage

Ensure that you understand the voltage at which the material dielectric will breakdown and short the circuit. As a general rule, the thinner the dielectric, the lower this value will be.

Panel Utilization

The IMPCB laminate materials are significantly more expensive than FR-4 materials. As a ballpark, a 0.062” IMPCB material may be three times more expensive than an 0.062” FR-4 material. It is, therefore, extremely important to understand how the board/array designs utilize the production panel. This is another area where early engagement with the PCB supplier is important. The most popular size for a working panel on these materials tends to be 18” x 24.” As many PCBs are processed as arrays, it is critical to ensure that array designs are such that panel utilization is maximized. Many large PCBs may be processed without rails through the assembly operation due to the rigidity of the material. This can vastly help improve panel utilization.

Machining/Fabrication

Scoring is the most common process used for square or rectangular shapes. The advantage of scoring is that it assists in maximizing material utilization since zero spacing is needed between parts to score them. In contrast, routing is the most expensive process since it is slower and requires spacing between parts and will likely reduce material utilization. Make sure that the PCB fabricator has a scoring system that is specifically designed for scoring aluminum. The scoring machine should be equipped with a lubrication system. It is recommended to use diamond-coated scoring blades and router bits when dealing with aluminum base metal.

Solder Mask

There are many single-sided IMPCB designs that are used for LED lighting applications. A majority of these applications require white solder mask. Thus, it is important to address this as all white solder masks are not made equal. A lot of LED customers are looking for consistency in the color of their white solder mask. The marketplace today has several different solder masks that are marketed as LED solder masks.

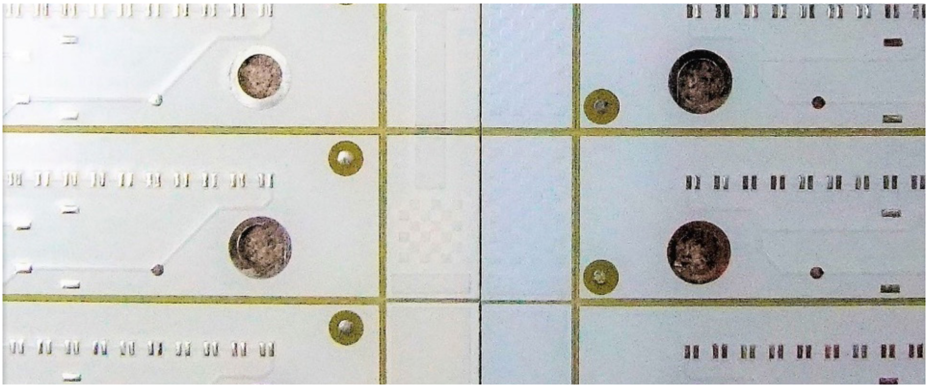


Figure 2-2: Different colors on two different types of solder mask.

The issue is that they visually look different when you put them side by side. Some solder masks have a "bluish" hue to them, and others have a "yellowish" hue (Figure 2-2). Also, the colors look different with one coat vs. two coats of solder mask, so this is another decision that will need to be made. Another consideration is that there can be an interaction between the surface finish, the solder mask, and subsequent heat processing steps in the assembly process. Some solder masks tend to change colors more than others with additional heat.

Figure 2-3: Solder mask "browning" with multiple reflows.

Boards with lead-free hot air solder leveling (HASL) tend to become "yellower" the more heat they are subjected to. We have stopped reworking boards through lead-free HASL (only one pass is allowed). Figure 2-3 illustrates the same solder mask after lead-free HASL vs. a board that has been through two assembly reflow cycles.

Boards with ENIG after the solder mask process may get "pink" with subsequent reflow. The reason for this is that "dirty" rinses in the ENIG process with gold residue form a complex with the titanium pigment in the solder mask which then turns the solder mask purple in the high-temperature assembly process. Thus, it is important for the PCB supplier to manage the rinses on the ENIG bath very carefully (Figure 2-4).

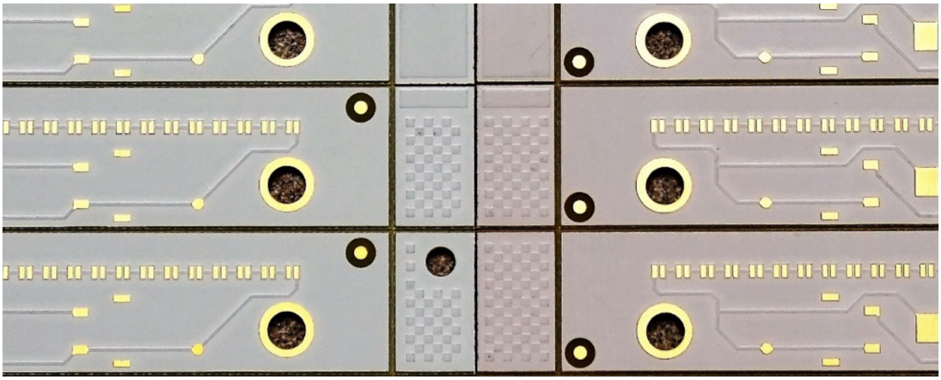


Figure 2-4: "Pinking" solder mask on an ENIG board with multiple reflow cycles.

Manufacturing Process for Single-Sided IMPCB

The manufacturing process for single-sided IMPCBs is illustrated in Figure 2-5.

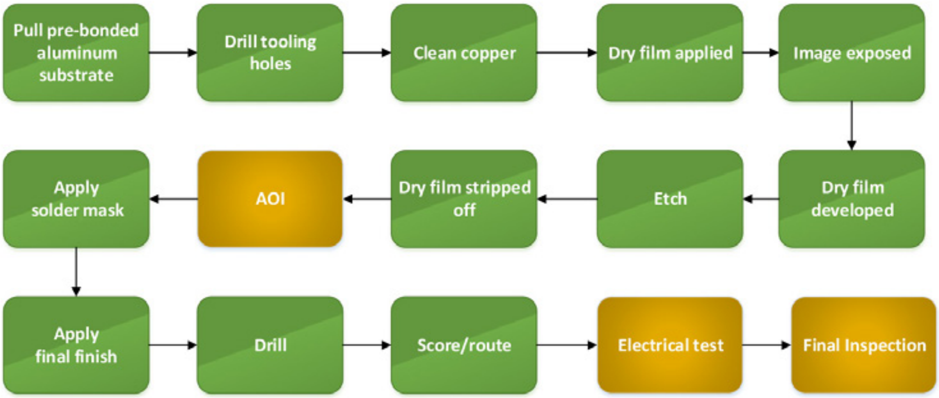


Figure 2-5: Single-sided IMPCB process flowchart.

Pedestal Technology

For LED applications with a demanding heat transfer profile, integrating pedestal technology may be the optimal solution. The pedestal construction on metal-core PCBs allows the designer to get a direct path between the LED (heat generator) and the metal core of the PCB. A typical LED is soldered to a copper pad and the heat must transfer through the PCB dielectric material before it gets to the metal core. Another option is to use thermal vias to get a more direct transfer to the metal core. Both of these designs are suited for lower-heat applications but may not be optimum for high-heat generators. A pedestal is simply a buildup of copper plating from the metal core up to the surface of the PCB. Thermal conductivity of up to 400 W/mK can be achieved using pedestal technology if the metal core is copper. The cross-section view in Figure 2-6 visually highlights the "pedestal" effect of this construction.

Figure 2-6: LED pedestal construction.

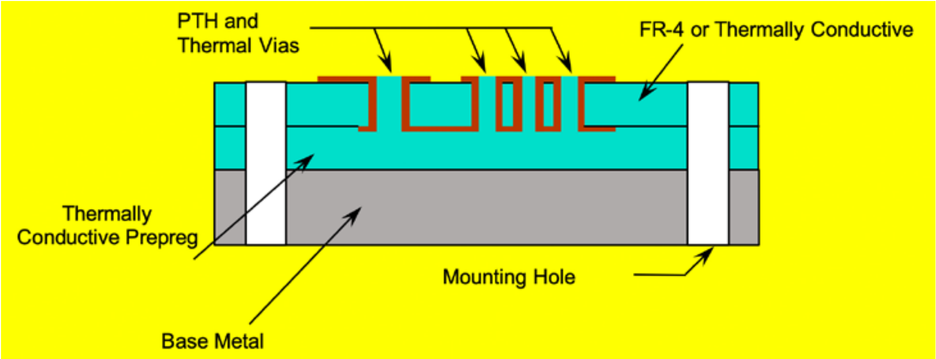


Figure 2-7: Double-sided IMPCB.

Double-Sided/Multilayer IMPCB

The PCB supplier manufactures a double-sided or multilayer IMPCB and then bonds it, utilizing a thermally conductive prepreg to metal (Figure 2-7). The bonding process is done in the same multilayer press that is used to manufacture a multilayer PCB. A lot of the design factor considerations discussed in the single-sided IMPCB section apply to double-sided as well. Some additional considerations to think about include the following.

Copper Weights on All Layers

The thicker the copper, the more expensive, and remember that the outer two layers will get additional copper since the vias will need to be plated. Lines and spaces should follow the design guidelines of the PCB fabricator based on the copper weights of each of the layers.

Double-Sided/Multilayer Construction

It is important to decide whether you can use FR-4 for your multilayer construction or if you require thermally conductive prepregs and cores. If you need thermally conductive cores and prepregs, there are a number of options available, but core thicknesses are limited, so it is best to work with the PCB fabricator or laminate supplier on constructions that would make sense. The prepregs tend to be low flow, so it is important to work with a PCB fabricator that understands the press cycles and flow dynamics of the specialty prepreg that need to be used.

Thermally Conductive Prepreg

Choose the prepreg to bond the PCB (double-sided or multilayer) to the metal based on thermal conductivity required and thickness of the copper circuitry. From a PCB manufacturing perspective, several different factors need to be taken into account for in the process of bonding the PCB to base metal:

- Ensure that you don't have delamination between the PCB and the metal. There are design factors that can impact this and process conditions in the lamination process
- Have a method to control the flow of the prepreg through the plated through-holes (PTHs) to the top side, and then have a method to remove any flow that ended up on the top surface of the PCB
- There are a number of mismatched CTEs in this package. It is important to balance the copper in the construction as much as possible from a PCB perspective and have a press cycle that minimizes warpage

Base Metal

Aluminum is the most common; however, there are many applications that will also use copper as the base metal. In general, for this kind of construction, if aluminum is the metal of choice, I recommend using the 6061T6 alloy.

Manufacturing Process for Double-Sided/Multilayer IMPCB

The manufacturing processes for double-sided and multilayer IMPCB are illustrated in Figures 2-8 and 2-9. The processes are similar to double-sided and multilayer PCBs described in detail in our book *Fundamentals of Printed Circuit Board Technologies* (more books listed in our further reading section).

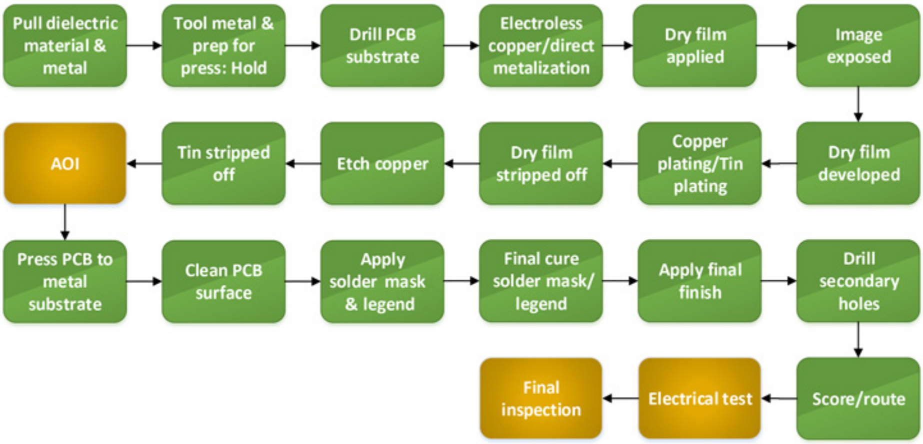


Figure 2-8: Double-sided IMPCB process flowchart.

There are two major differences. First, when one utilizes the thermally conductive prepregs, the press cycles need to be well-defined due to the low-flow nature of the prepreg. Second, there is an additional press cycle since after the board is completed, one need to bond the PCB to the aluminum. The aluminum prep before bonding and the press cycle associated with the bonding are critical.

IMPCB Testing Methods

This is a bit of a caveat emptor (buyer beware) caution regarding the testing of IMPCBs, particularly when the PCB fabricator is in China or otherwise outside of the U.S. There are a few factors to be considered:

- Datasheets provide several different ways that laminate suppliers can test these materials for thermal conductivity, and to date, there are no IPC standards for this. A thorough understanding of the test methods utilized is required since all materials advertised as 2 W/mK may not result in similar performance (Appendix A describes the various test methods)
- Some laminate suppliers 100% HiPot test their materials, others will test them if you request it at an extra cost, and many don't have the ability to test their material
- There are many suppliers that can supply single-sided IMPCB materials, but the supply base is quite limited for multilayer cores with high thermally conductive prepreg and thermally conductive prepreg that can be used to bond the multilayer PCBs to the metal

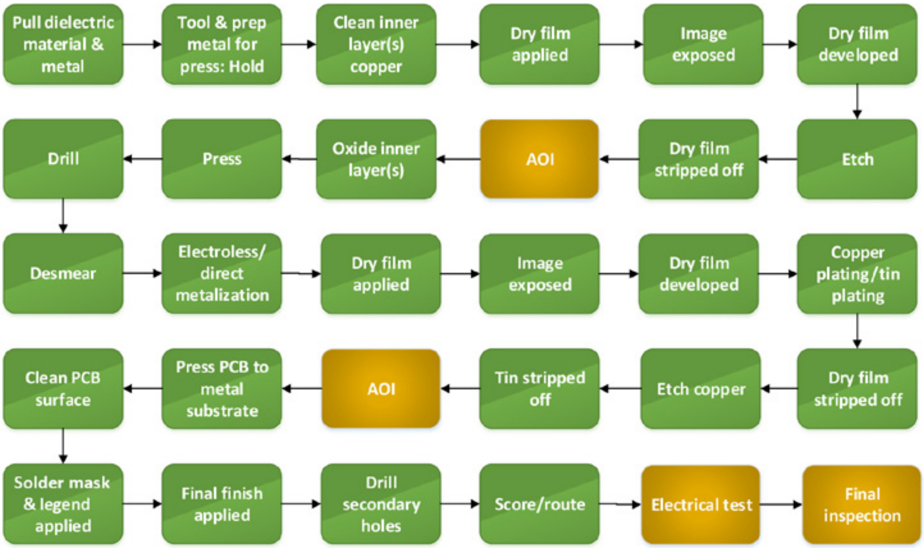
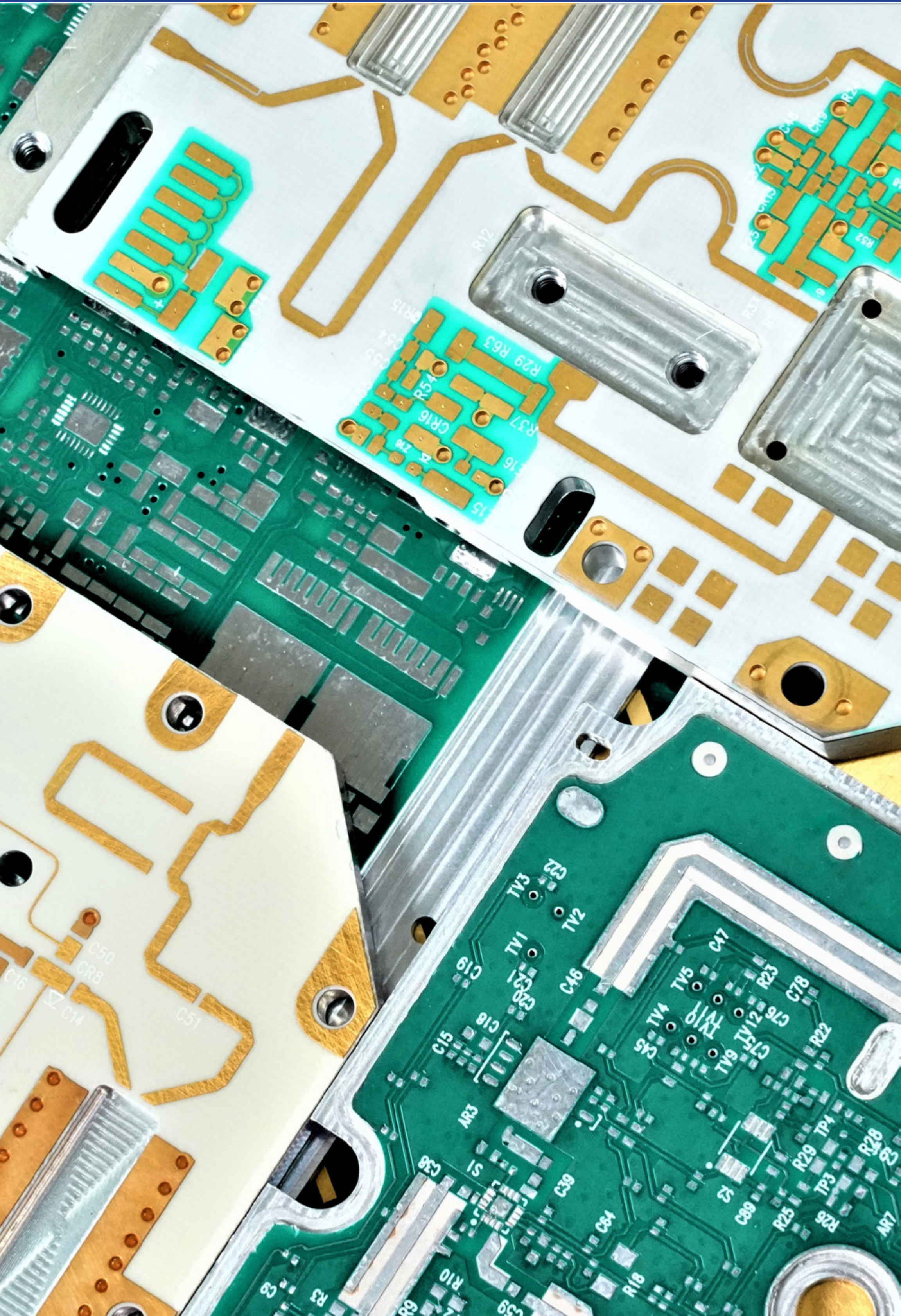


Figure 2-9: Multilayer IMPCB process flowchart.

IMPCB Fabricator Selection Considerations

The supply base for IMPCB suppliers is relatively small, which makes the supplier selection a critical task. Some of the things to consider include the following suggestions:

- The fabricator's experience with manufacturing IMPCB materials
- A fabricator that has manufactured a variety of different types of IMPCBs and materials from a variety of material suppliers
- A fabricator that has a good relationship with the material suppliers in the space
- The fabricator is willing to work in partnership with you keeping an open mind when it comes to your ultimate needs
- If UL is important to your application, ensure that thefabricator has the requisite UL listing(s)
- A fabricator that has the process controls and disciplines in place to consistently meet all your requirements
- Lead-times vary between different fabricators, which is compounded when procuring from a source outside the U.S. If you are interested in multiple regions, choose a fabricator that has a support structure in all regions of the world (supply chain ease)
- Another important factor could be R&D that is being done by the laminate supplier that may support future technology. As an example, there are a couple of laminate suppliers that have developed special laminate materials where the aluminum can be bent/formed without compromising the copper circuitry or the dielectric layer


R12
R29 R63
R54
CR16
R37
C50
CR8
C16
C14
C51
TV3
C22
TV2
TV1
C19
C21
C20
C46
C47
TV5
R23
C78
TV4
C18
C15
C45
TV9
R22
AR3
TP4
R28
R29
TP3
C89
R25
AR7
C38
C39
C64
R10
R18
R3

CHAPTER 3

Metal-Core Boards

Conceptually, a metal-core board is exactly like it sounds—the metal is in the middle of the PCB sandwiched between layers on both sides. Just about any PCBA that will contain active heat-generating components can benefit when designed on a metal-core PCB. On a conventional PCB, the standard FR-4 layers are relatively poor thermal conductors, and heat is normally dissipated from active components using vias and thermal pads, as discussed earlier. A metal core has much greater thermal conductivity, allowing it to easily dissipate heat away from active components. This prevents hot spots that can form in PCBs by dissipating heat evenly across the PCB and increasing performance and lifetime.

One example would be in the LED lighting industry, where LEDs produce a significant amount of energy and heat. A metal-core PCB has two benefits in this application: it provides some natural reflectivity for any light that travels toward the substrate, increasing the device brightness; and the other is extending the life of the product by quickly transferring heat away from the LEDs (Figure 3-1).

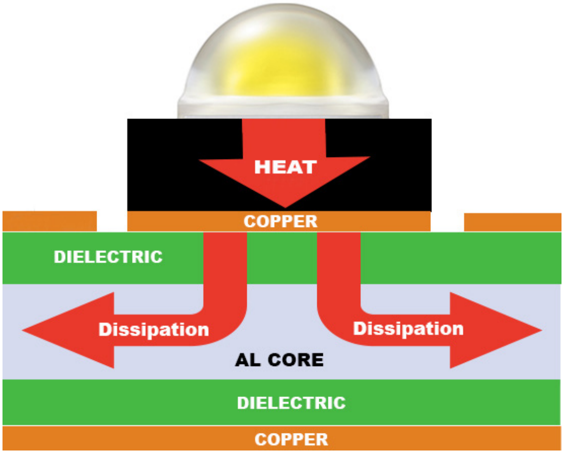


Figure 3-1: LED aluminum core PCB heat dissipation.

Metal-Core Applications

- **Mil-aero**
    - o Extreme temperature and moisture operating environments
    - o Frequent shock and vibration environments
    - o Excessive thermal cycling
- **LED**
    - o Backlight
    - o Street safety
    - o Automotive (conventional and hybrid)
    - o Photovoltaic
- **Power**
    - o High-power output devices
    - o DC-DC converters
    - o High-power converters
- **Consumer**
    - o Flat panels
    - o Motor controls
    - o Printer drivers
    - o Solar energy
    - o Audio components

Metal-core PCBs usually have blind via layers located on both sides of the metal-core substrate. There are also PTHs going through the entire package. From a PCB perspective, it is important to isolate the metal from the through-hole; otherwise, the board would short out completely. To accomplish this, one must start out by drilling the metal core approximately 40–50 mils larger than the PTHs, slots, or cutouts. It then needs to be filled with a non-conductive epoxy filler and then pressed. Figure 3-2 shows a multilayer metal-core board.

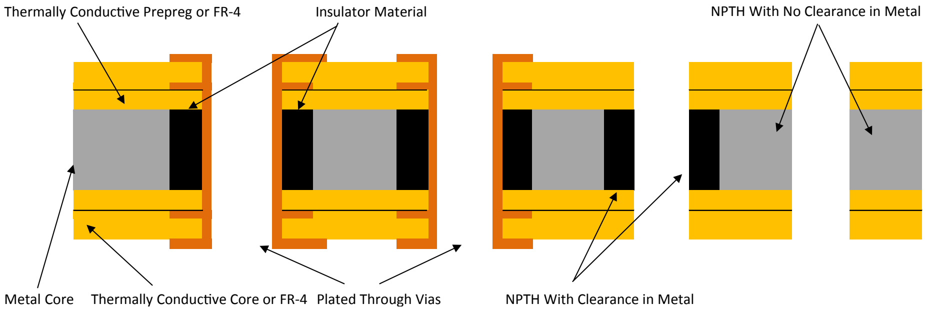


Figure 3-2: Multilayer metal-core board.

After pressing the metal core, it then needs to have the filler compound removed from the surface and prepared for lamination with the inner layer cores. After lamination, the PTHs are drilled and processed through normal manufacturing processes. While thermal management is a factor in these PCBs, one of the other reasons that metal-core boards are used is to help with vibration reduction so that components don't fall off the PCB in high-vibration applications. Figure 3-3 shows a typical metal-core process flow.

Figure 3-3: Metal-core process flowchart.

Some of the design factors that need to be considered include the following.

Core Materials/Prepregs

The core materials and prepregs could be any PCB raw material. We have seen metal-core boards manufactured with polyimide, FR-4, or highly thermally conductive materials, but it really depends on the application and what is desired from an electrical/thermal management perspective.

Metal Core

Typically, either copper C110 or Aluminum 6061T6 are used with equal frequency, but the cost and weight need to be factored into the decision on which to use.

Drilling the Metal Core

The metal-core boards are drilled oversized with the entire drill pattern associated with the PCB—both the PTHs that go from the top layer to the bottom layer and the non-plated holes going from the top layer to the bottom layer. Occasionally, mounting or grounding holes have no clearance in the metal core.

Insulator/Filler

The insulator/filler material acts to insulate the PTHs from the metal core, so the entire PCB does not short out. The filler is bought in powder form and then is applied to the surface and holes and put in a multilayer lamination press to cure. This is a critical process because it must ensure there are no voids in the filler because once the PTH are drilled and plated, chemistry can leach back to the metal core through a void and cause a short. The core is then planarized (sanded) to remove the excess filler on the surface. The filler material is a ceramic/epoxy combination.

Stackup

It is preferred to be symmetrical in terms of the number of layers on top of the metal core and number of layers below the metal core. Also, copper weight symmetry is preferred between all the layers, like any multilayer PCB, because a lack of symmetry can lead to excessive warpage issues. It is important to note that due to the unique characteristics of metal-core designs, the typical IPC warpage specifications do not apply.

Milling

Most metal-core boards have milling associated with the PCB that results in exposing the metal-core layer on the edges or in cavities (Figure 3-4).

Surface Finish on the Exposed Metal Core

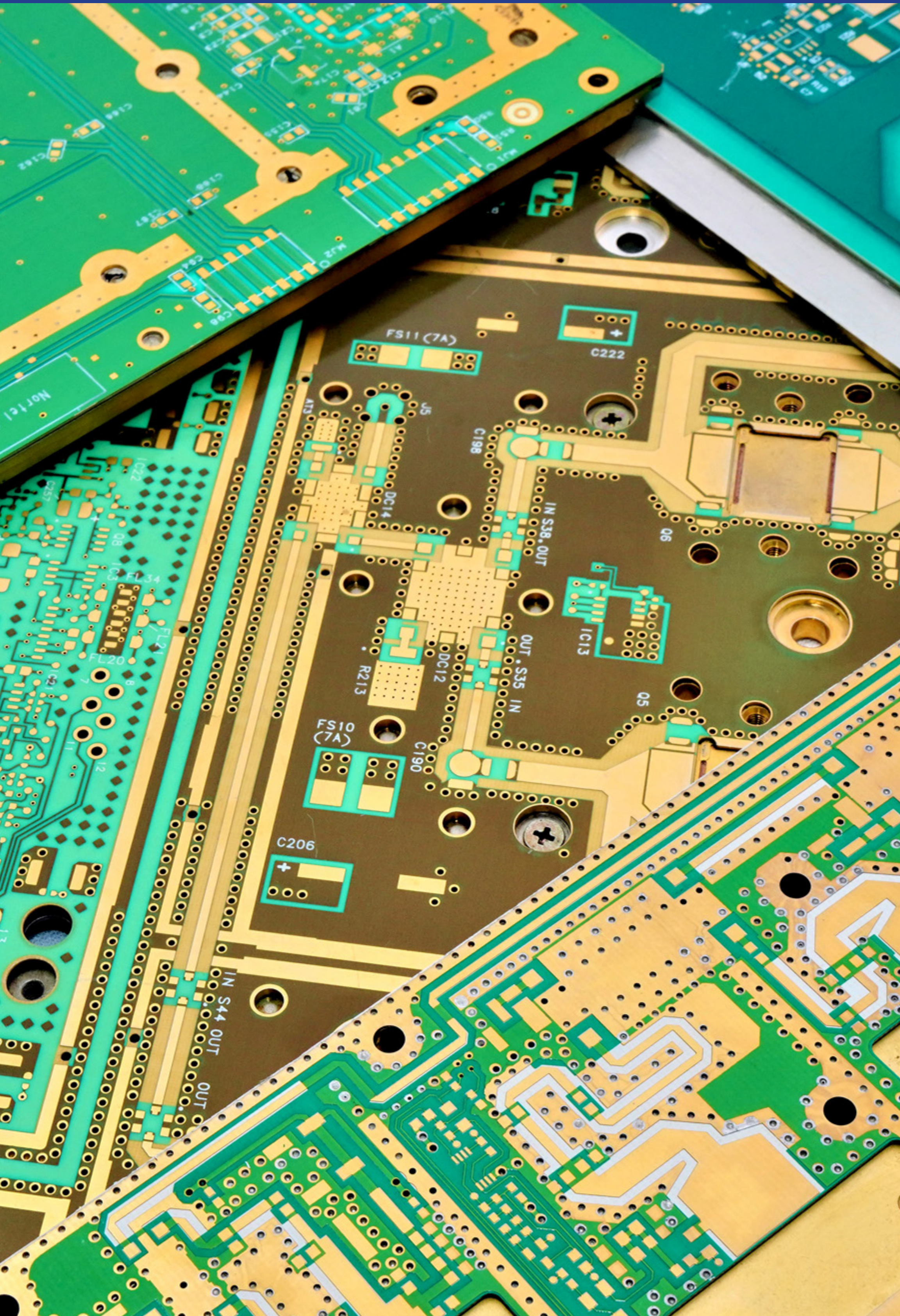
We recommend putting a surface finish on the exposed metal. Typically, for aluminum, we recommend chromate conversion and for copper a minimum of 50 micro-inches of electroplated nickel.

Figure 3-4: Milling on a metal-core PCB.

Benefits of Metal-Core PCBs

Benefits of metal-core PCBs include the following:

- The ability to integrate a dielectric polymer layer with a high thermal conductivity layer for a lower thermal resistance (the higher the conductivity of the material, the faster the heat transfer)
- Much more dimensionally stable than conventional PCB construction
- Reduced weight when using aluminum compared to copper or ceramics
- Absorbed vibrations through the metal cores since metal substrates are robust and durable under adverse operating conditions



CHAPTER 4

RF Thermal Management Fabrication Methods

Earlier chapters primarily focused on the different methods of using metal to enable improvements in thermal management. It is also important to understand that apart from the thermal management function, the metal can also act as a grounding layer and, therefore, a thermal and electrical connection between the circuitry and the metal. At a high level, there are two ways to achieve thermal management of PCBs utilizing metal: the first is pre-bonded, and the second is post-bonded (Figure 4-1).

In a pre-bonded circuit board, the PCB supplier buys the laminate material pre-bonded to the metal. Most of the available high-frequency laminate materials can also be bought in a pre-bonded configuration. The two laminate suppliers that have most of this market are Rogers and Taconic (now AGC Nelco). The PCB manufacturer is then tasked with processing this material and making circuits and machining the metal. In a post-bonded circuit, the PCB supplier manufactures the PCB and the metal separately and then bonds the two together using a variety of methods.

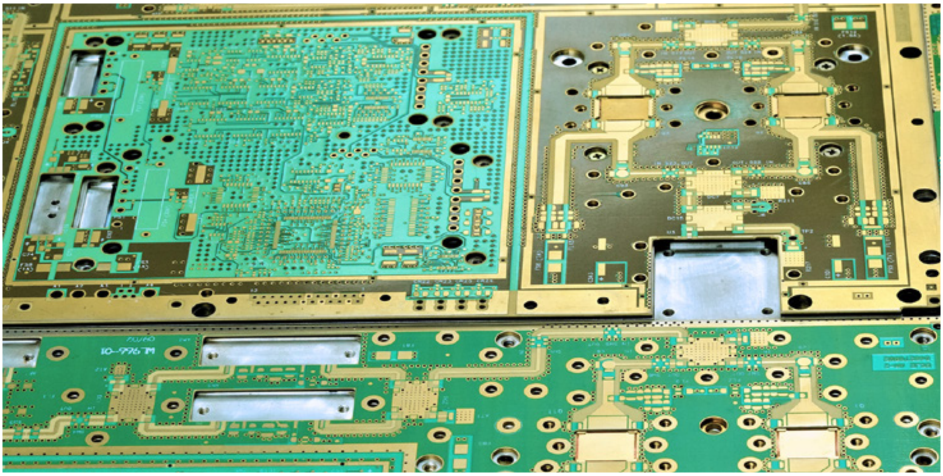


Figure 4-1: RF thermal management PCBs.

There are a number of pros and cons to each of these two methods. Pre-bonded PCBs are typically used in high-reliability, military, aviation, and telecom applications since they offer precise dielectric constant (Dk) control, no risk of delamination, and high reliability. The two disadvantages of this methodology are that it is restricted to a single layer of circuitry and, in general, costs tend to be significantly higher with pre-bonded vs. post-bonded. The reasons that the costs are higher is because the laminate materials are significantly more expensive, processing is more challenging, and any yield issues result in very expensive scrap. There are cases where pre-bonded PCBs can be converted to post-bonded PCBs for cost reduction. Any multilayer applications that require metal for thermal management will need to utilize post-bonding.

Figure 4-2: Material selection decisions.

Pre-Bonded Laminates

There are several design parameters that need to be considered. The first step is to determine the dielectric material, dielectric thickness, and the copper foil weight that is required. The next step is to determine the thickness and type of metal (Figure 4-2). The typical metals used are aluminum 6061-T6 and copper C110, but we have also occasionally seen brass used. Aluminum is usually lighter and cheaper than copper, and each has their own set of properties (Table 4-1). The pre-bonded metal thickness should be at least three times the dielectric thickness to minimize warpage.

In pre-bonded applications, PCB processing is more difficult with aluminum vs. copper. The laminate manufacturer uses lamination temperatures that are hot enough to anneal the aluminum. The 6061-T6 aluminum permanently softens and is more difficult to machine. Once you determine laminate selection, you really should check with the PCB supplier or the laminate manufacturer that your selection is available as you have it configured. Depending on machining requirements, one may have to start with a metal thickness that is higher than the finished thickness.

Property (units)	Aluminum	Copper
Alloy number	6061	110
Tensile strength (kpsi)	20	35
Specific gravity	2.7	8.9
Specific heat (J/kg°K)	960	385
Thermal conductivity (W/mK)	180	390
Thermal expansion (ppm/°K)	24	17

Table 4-1: Relevant properties of aluminum and copper.

Post-Bonding

In a post-bonded application, a double-sided PCB or a multilayer PCB is first manufactured. Typically, the bottom side is mainly a ground layer that may have a few circuits integrated. While the PCB is being manufactured, the metal can be simultaneously machined on a CNC machining center. There is a lot more flexibility in terms of the shape and features in a post-bonded application, as it is processed separately from the PCB. The metal can then be plated.

Once the PCB and metal are completely manufactured, they can be bonded together (i.e., post-bonded). This is typically done in piece form (a single PCB bonded to a single fabricated metal). Interestingly, we have seen several applications where multiple PCBs may be bonded to the same piece of metal. These different PCBs may or may not be built with the same dielectric materials or thickness.

Some of the advantages of post-bonding include:

- Ease of utilizing multilayer structures
- Simplified processing (PCBs manufactured conventionally)
- Metalwork milled and plated conventionally
- Concurrent processing (boards made separately, as well as carriers/pallets/heat sinks manufactured separately)
- Yield (good board joined to a good carrier)
- Cost (reduced through standard/simplified processing)

The PCBs are post-bonded using sweat solder or sheet film adhesive. In both techniques, some sort of a bonding fixture is required to ensure that there is good registration between the PCB and the metal carrier and that the correct amount of pressure is being applied in the bonding process. Table 4-2 shows the requirements for bonding registration holes. Figure 4-3 shows two different examples of how the registration holes can be managed and the impact on overall registration.

Bonding Medium	Fixture Type	Registration	Preferred Hole Size
Sheet film adhesive	High-temp FR-4, Polyimide, or Metal-AL 6061	Two diagonal holes, or preferably three holes, at the outermost edge of the part.	0.125 or larger
Sweat solder	Metal-AL 6061	Scattered holes throughout the part as equally distributed as possible for even pressure distribution.	2-56 or 4-40 tapped

Table 4-2: Bonding registration requirements.

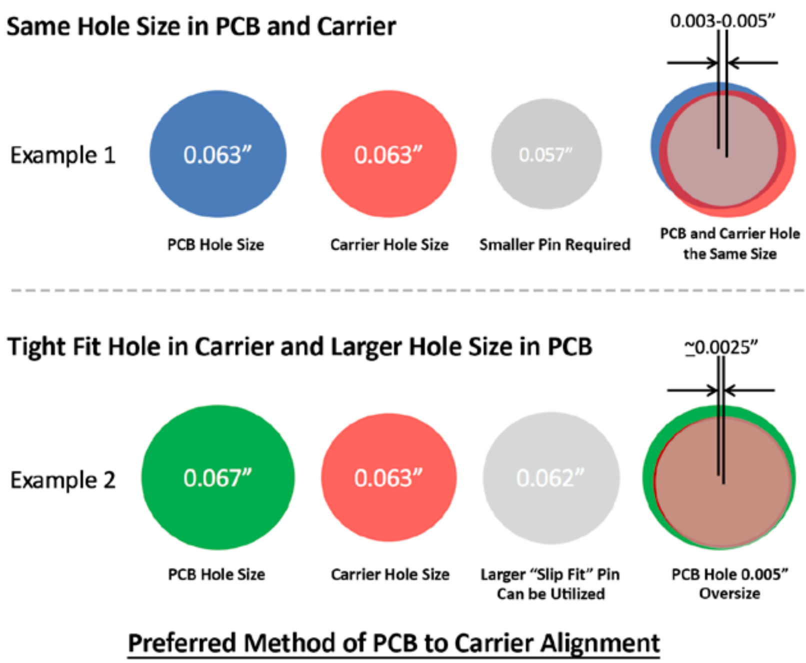


Figure 4-3: Registration example.

Most often, the bottom layer is primarily a ground layer and has very little or no solder mask. Sweat solder and sheet film adhesive are two of the bonding techniques that will be explored in the next section.

Sweat Solder

High-temperature solder is used to bond the PCB to the metal. The solder and temperature used are such that there is no risk of de-bonding in subsequent assembly operations. One of the biggest discussions associated with this technique is void volume. Since the solder paste is a mixture of flux plus solder, the flux creates air gaps as it volatilizes. Typically, we screen the solder paste, but in certain high-volume applications where the void volume control is essential, we have also used solder preforms—which is a more expensive method—to better control the void volume. Figure 4-4 shows how a sweat solder board utilizing solder paste is assembled.

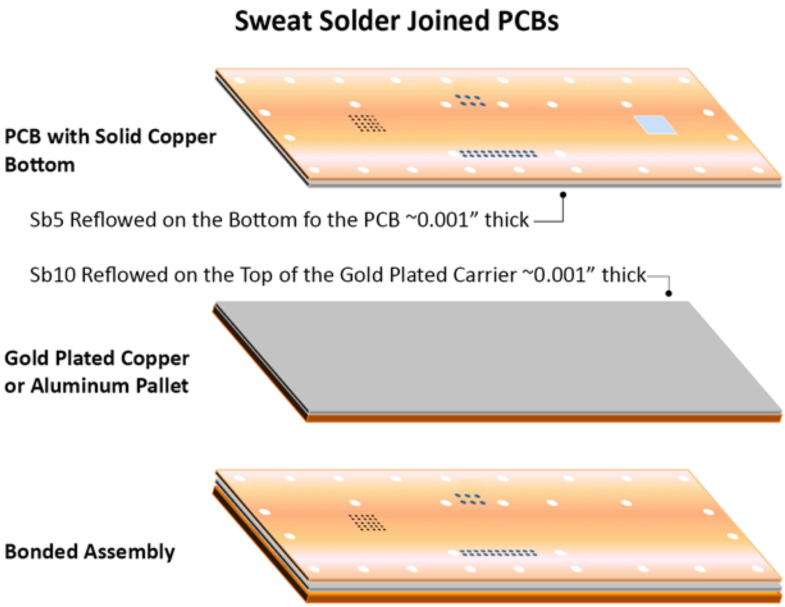


Figure 4-4: Sweat solder construction.

Critical Design Factors

Critical design factors for sweat solder include the following:

- Solder/paste selection: It is important to ensure that the solder utilized does not de-bond in subsequent component assembly operations. Some of the options that we have successfully used on sweat soldering applications include:
  - Eutectic tin/lead
    - Sn63/Pb37
    - Melting point of 183°C (361°F)
  - SAC305 (ROHS-compliant)
    - Sn96.5/Cu3.0/Ag0.5
    - Liquidus temperature of 220°C (428°F)
    - Solidus temperature of 217°C (422°F)
  - Tin antimony (ROHS-compliant)
    - Sn95/Sb5
    - Liquidus temperature of 240°C (464°F)
    - Solidus temperature of 235°C (450°F)
- Sweat solder stencil design: The stencil to screen solder on the PCB or metal must be custom-designed to minimize voids, especially in the critical areas. The PCB fabricator will design this. It is preferred to screen paste on the metal; however, depending on the features of the metal, this may not be practical. In those cases, the solder is screened on the PCB
- Sweat solder fixture design: This is also custom-designed by the PCB fabricator to help minimize solder voids and ensure good registration between the PCB and the metal. It is important to work with a fabricator that has a lot of experience with custom designing both stencils and fixtures
- Metal choice: Typical choices are aluminum, copper, and occasionally brass
- Metal surface finish: If aluminum is the metal that is being used, it needs to be plated at least on the side that is being soldered since it is not possible to solder to bare aluminum
- PCB surface finish: The following surface finishes are preferred—Ag, Au, Sn, and bare copper
- Hot vias on the ground layer: If there are either hot vias or circuit lines on the bottom layer, they should be covered with solder mask. In addition, the metal should either be partially milled out or cut out completely. This helps prevent opportunities for shorts

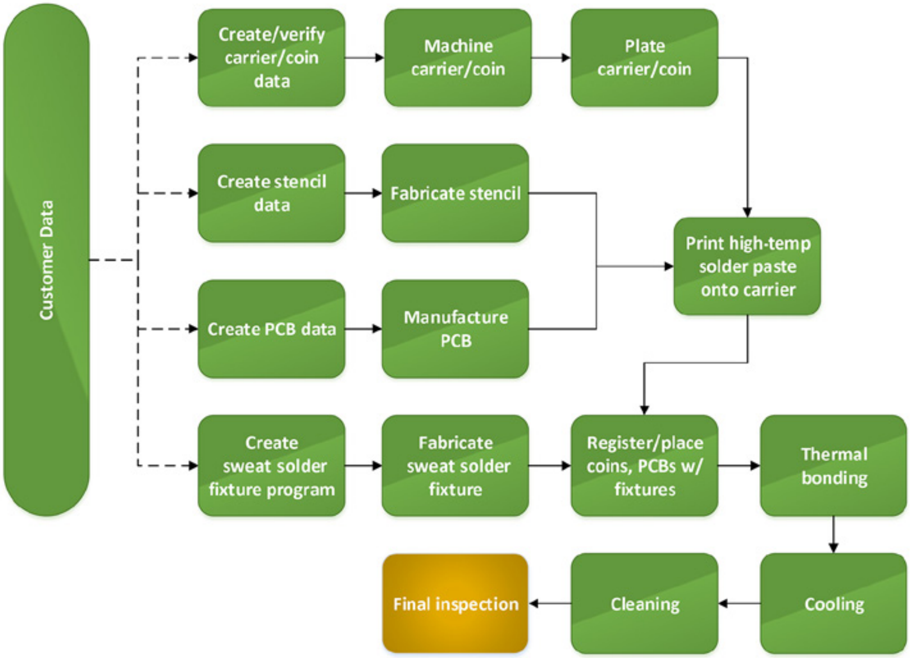


Figure 4-5: Sweat solder process flowchart.

Figure 4-5 illustrates the overall process associated with manufacturing a sweat soldered assembly.

Sheet Film Adhesive

There are two main types of silver-filled conductive films: epoxy and silicone.

1. Silver-filled conductive epoxy films: These are commercially available, and some examples are Henkel CF3350 and Ablefilm 5025E, and Rogers COOLSPAN®. The PCB and metal are bonded using temperature and pressure with a sheet film adhesive.

2. Silver-filled conductive silicone films: These are patented ASC materials and have some differences from the commercially available materials. The PCB and metal are bonded using temperature and pressure with a sheet film adhesive.

Figure 4-6 shows how a sheet film bonded assembly is put together.

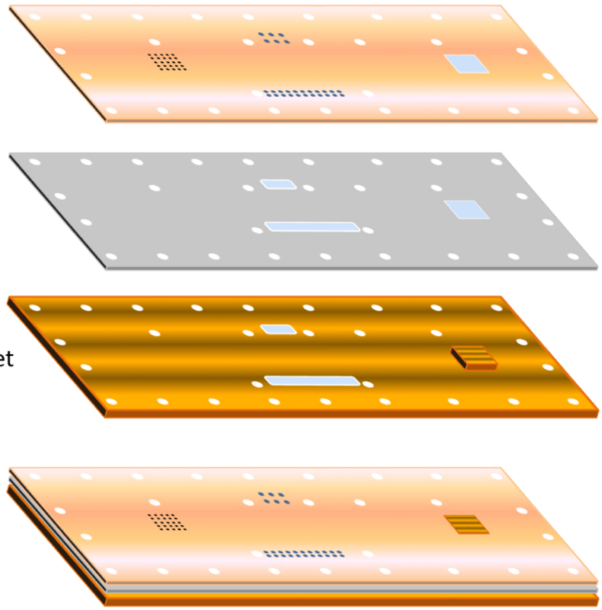


Figure 4-6: Sheet film construction.

Critical Design Factors

Critical design factors for sweat solder include the following:

- **Adhesive selection:** There are several different options that are available, including ASC’s patented material Electrasil-2. Table 4-3 compares Electrasil-2 to commercially available sheet film adhesives from Rogers (COOLSPAN TECA) and Henkel (CF3350 and Ablefilm 5025E) and sweat solder as a reference

Material →	Electrasil-2	COOLSPAN® TECA	E&C CF3350	Ablefilm® 5025E™	Sweat Solder
Thickness	3–6 mils	2–4 mils	2–6 mils	2–4 mils	1–3 mils
Cure Condition	30 min. at 150°C	45 min. at 175°C	30 min. at 150°C	30 min. at 150°C	3–7 min. at 285°C
Thermal Conductivity (W/mK)	25	6	7	6.5	40

Table 4-3: Bonding registration requirements.

- **Bonding fixture design:** This is also custom-designed by the PCB fabricator to help ensure that there is good adhesion between the PCB and metal and good registration between the PCB and the metal. It is important to work with a fabricator that has a lot of experience with custom designing bonding fixtures. Figure 4-7 shows a typical setup of how a PCB is bonded to metal using a sheet film adhesive

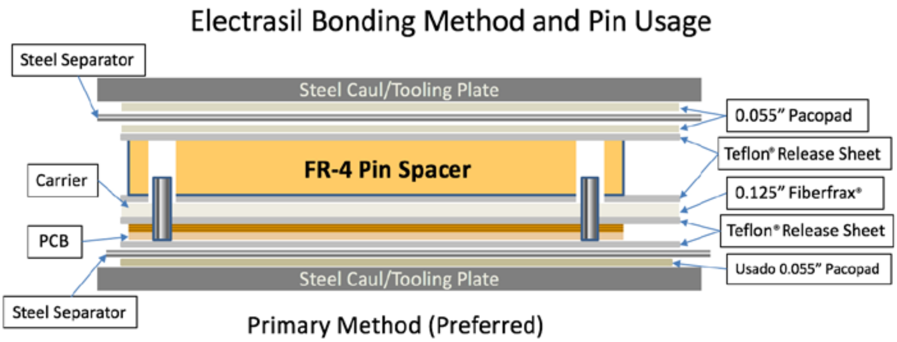


Figure 4-7 : Bonding method and pin usage.

- **Metal selection:** Typical choices are Aluminum 6061-T6, Copper C110, or occasionally brass
- **Metal surface finish:** Table 4-4 illustrates the various surface finish options
- **PCB surface finish:** Gold and silver tend to be preferred surface finishes. HASL or lead-free HASL are not acceptable surface finishes. It is also important to review the datasheets of the sheet film adhesive or discuss with the PCB fabricator any specifics related to surface finish choices
- **Hot vias on ground layers:** If there are either hot vias or circuit lines on the bottom layer, they should be covered with solder mask. In addition, the metal should either be partially milled out or cut out completely

Available Final Finish	
PCB	**Heat Sink**
ENIG	ENIG
Electrolytic Hard Gold	Electrolytic Hard Gold
Electrolytic Tin	Electrolytic Soft Gold
Electrolytic Soft Gold	Electrolytic Silver
Immersion Silver	Electrolytic Nickel
Immersion Tin	Electrolytic Tin
OSP	Alodine/Conversion Coat
Bare Copper	Anodize
HASL	
Lead-free HASL	

Table 4-4: Available surface finish.

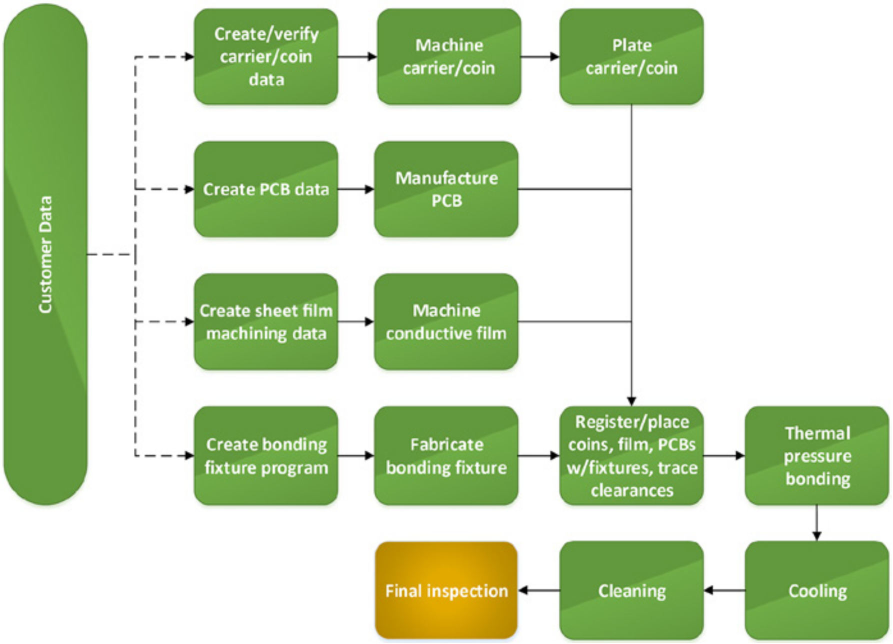


Figure 4-8: Sheet film adhesive process flowchart.

Figure 4-8 illustrates the overall process associated with manufacturing a sweat soldered assembly.

Quality Control of Post-Bonded PCBs

A post-bonded PCB is not a typical PCB, so we wanted to discuss quality control methods that we employ to ensure that the customer gets what they need.

Machining

- 100% mechanical inspection of every feature (utilizing visual and contact inspection techniques)
- Visual inspections derived from CAD data (directly from .iges and .stp files, while inspection from .dxf and .dwg files is possible with interpretation)

Sweat Solder

- X-ray inspection of the solder joint (standard): Each part is inspected for proper reflow and wetting, and void volume is evaluated, not quantified
- Visual inspection (standard): No excess solder
- Peel testing (non-standard): Empirical testing of adhesion on production parts
- Convection reflow (non-standard): Simulation of subsequent convection reflow soldering cycle and validation of product robustness

- Sonoscan (non-standard): Non-destructive test, but we currently outsource. This is a technique that can be used to get an image of the entire part and look at the void volume

Sheet Film Adhesive

- Resistance measurements
- Visual inspection that ensures no excess flow of adhesive material

Embedded Copper Coins

Coin technology is quickly becoming a preferred alternative to internal heat sinks to draw heat directly down and away from the heat-generating device to the backside of the PCB. The phrase "press fit coin" gets used quite often when discussing various embedded coin applications. The fact is that most practical applications where coins are employed use a coin that is bonded into the structure during the multilayer lamination process. In this case, the coin is bonded into place and sealed by the flow of the prepreg resin, which is adjacent to the coin at the time of lamination. The result is a securely mounted but electrically isolated coin. Either ground via structures are added through a flange in the coin, or the cap plating on the top and bottom of the coin provides the grounding connection.

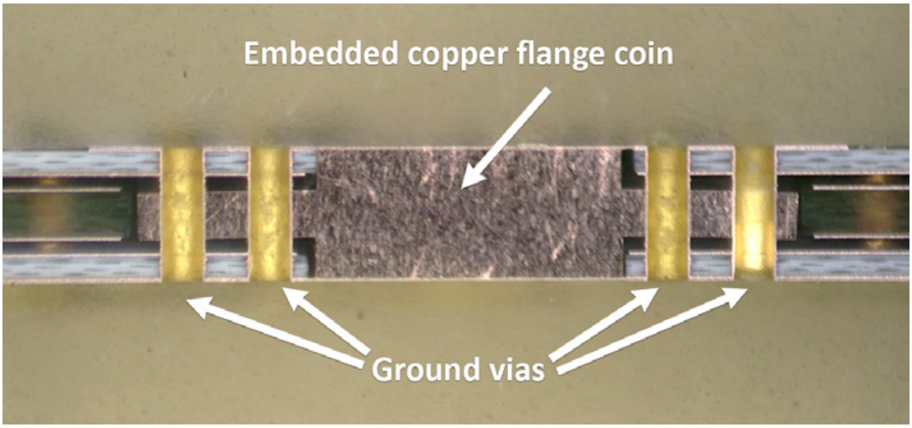


Figure 4-9: Micro-section of a flange coin PCB.

We have utilized a few different kinds of coins: a center-flanged coin, a bottom-inserted coin (or a T-coin), a coin that does not go through all the layers, a U-coin, and a serrated coin. It is important to understand that every embedded coin part number tends to be a unique engineering project for a PCB board fabricator. Figure 4-9 displays a micro-section of an ASC PCB with a flange coin.

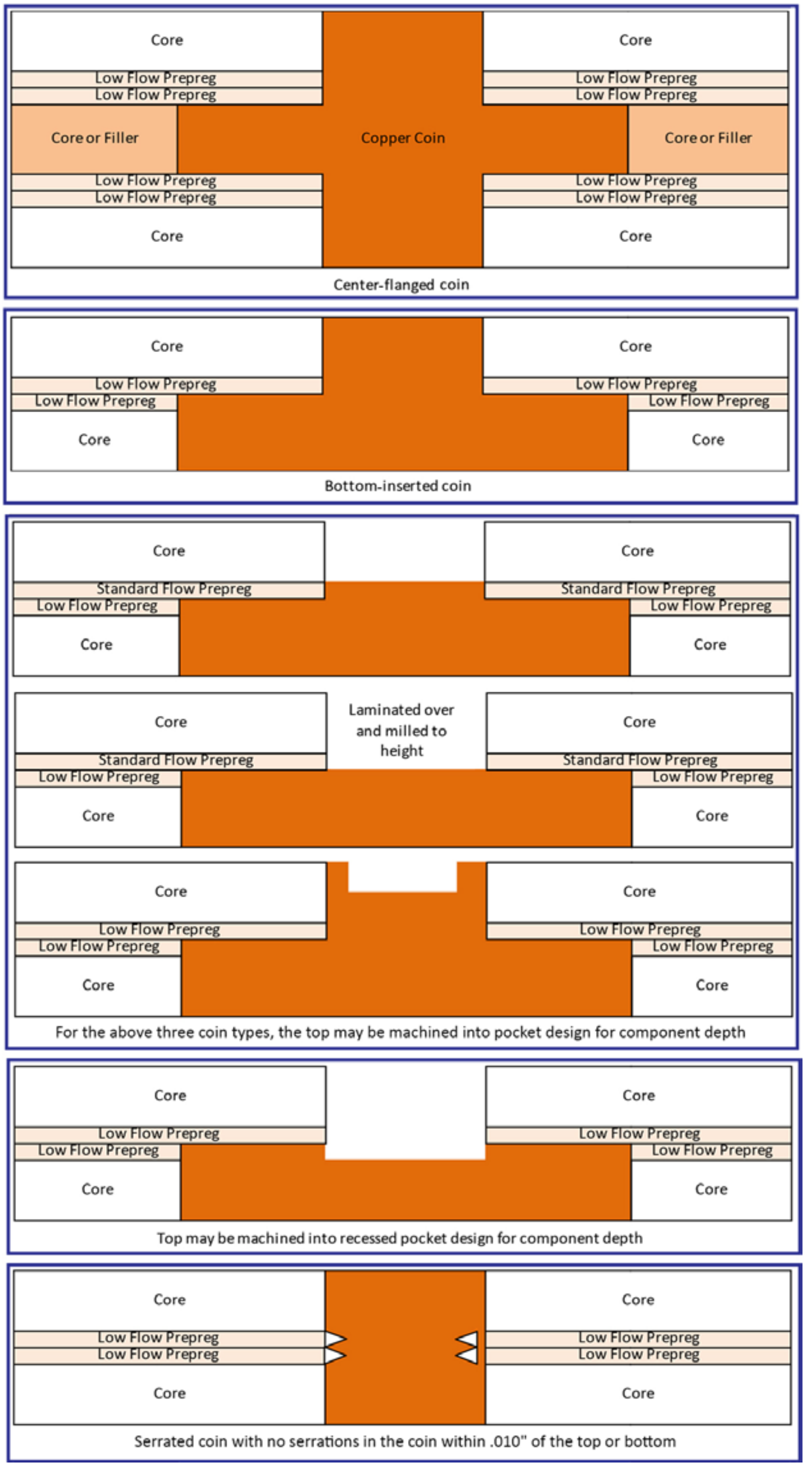
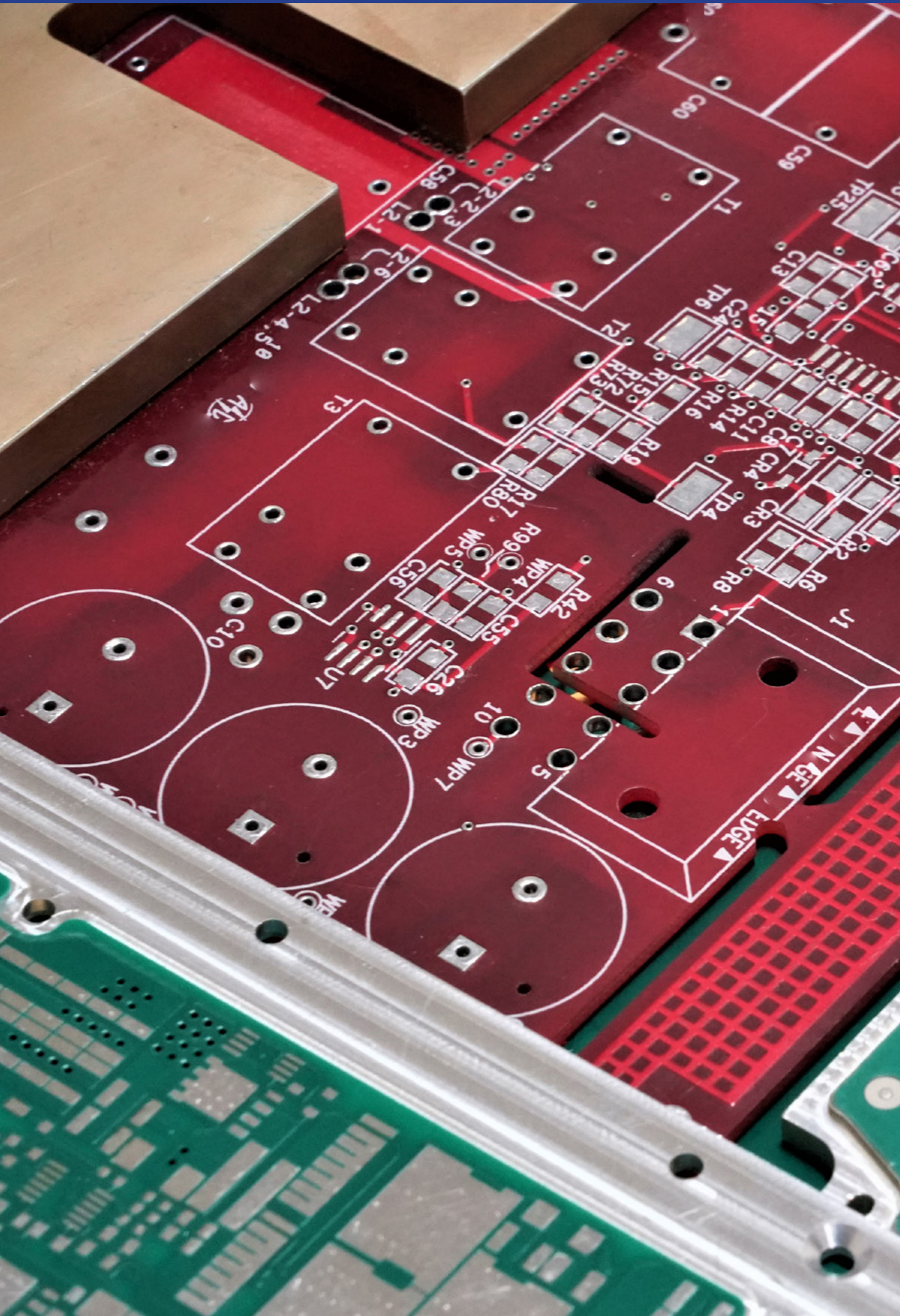


Figure 4-10: Common embedded coin types.

Figure 4-10 shows different types of coins. When designing, it is always preferred that the top and bottom of the coin be in a positive or near-neutral position relative to the top and bottom of the PCB. On the bottom, this ensures that intimate contact can be made with any external heat sinking. Being on the top ensures that the device typically mounted on top of the coin can be adequately solder paste printed and soldered properly to the electrically and thermally conductive structure. When you consider both aspects, it is self-explanatory that a negative condition, one with either the top or bottom of the coin recessed within the circuit board, is less than ideal. Tolerances need to be made to allow for reasonable manufacturing yields. In these cases, a preferred tolerance would be ranging from a slight negative condition of -0.0005” or -0.5 mil to a positive condition approaching +0.002” or +2 mils.

Early engagement and ongoing communication with your fabricator are essential to settle on a specific mass of coin required for the thermal dissipation necessary for device operation within the design temperature requirements. The fabricator will need to adjust the actual coin size slightly to ensure proper fitment and clearance within the pocket in the multilayer. This will also ensure proper prepreg resin fill around the coin as well as ensure that the surface flow of resin is minimized.

Reducing resin flow onto the top surface is important when using thinner copper foils on the top and bottom layers since planarization (sanding or disk grinding) is the most common method for removing excess resin flow. When fine lines and spaces are employed, there is always a tendency to use thinner copper to ease manufacturing. Half-ounce copper will, of course, be more susceptible to damage during planarization than one-ounce copper foil. One-quarter ounce copper foil will be virtually impossible to clean mechanically, so chemical or plasma methods will need to be used—both of which are capable of removing only very slight amounts of resin from the surface.



CHAPTER 5

Mixed Technology

Because of the rapid evolution of electronic products and continual drive for smaller, faster, and cheaper components, printed circuit designs are commensurately increasing in complexity. This is resulting in an ever-increasing amount of new designs that utilize a hybrid of multiple technologies to meet these new design requirements. Technologies that are frequently combined in various permutations include:

- **Material Sets**
    - Rigid FR-4 materials
    - RF/microwave materials
    - Polytetrafluoroethylene (PTFE)
    - Flex/rigid-flex
- **Via Structures**
    - Microvia
    - Buried/blind/stacked/staggered
    - Via-in-pad
    - Insulated/non-insulated
- **Thermal Management**
    - IMPCBs
    - Metal cores
    - Metal-backed
    - Thermal vias

We were recently involved in an R&D project for a customer that evolved into the epitome of mixed technology. The project was later named “The Kitchen Sink Board” by the experts who put their blood, sweat, and certainly tears into developing this product.

Case Study: The Kitchen Sink Board

Table 5-1 shows an explanation of the case study on the kitchen sink board from first addressing the problem to stating the results.

<table>
<tr><th colspan="2">Case Study: The Kitchen Sink Board</th></tr>
<tr><td>Challenge</td><td>To solve a complicated design issue for a customer using an extreme application of mixed technologies</td></tr>
<tr><td>Solution</td><td>Chose a PCB fabricator with world-class engineering support and extensive experience building mixed-technology PCBs</td></tr>
<tr><td>Phase I</td><td>Started with an RF metal-core board
• FR-4 and RF materials
• Top 8 layers FR-4 digital
• Copper metal core
• Bottom 4 layers RF microwave with Rogers material
• Blind/buried vias on both RF and FR-4 portions
• Insulated vias through the copper metal core

Design delivered to the customer</td></tr>
<tr><td>Phase I I</td><td>The customer wanted to change connectors

Converted to a rigid-flex metal-core board
• Top 8 layers FR-4 rigid-flex (layer 6 and 7)
• Copper metal core
• Bottom 4 layers RF microwave with Rogers material
Bonded top and bottom subassemblies to the copper metal core with thermally conductive prepreg

Blind/buried vias on both RF and FR-4/rigid-flex portions (insulated vias through the copper metal core)</td></tr>
<tr><td>Timeline</td><td>• 10 weeks of intense technical discussions with the customer
• 2 weeks of internal engineering and CAM work
• 12 days of internal advanced quality planning (AQP) meetings (no PO coverage during this engineering time)</td></tr>
<tr><td>Results</td><td>After hundreds of engineering hours, the first qualification build had 95% yield</td></tr>
</table>

Figure 5-1: Case study explanation.

Multi-Functional PCB

Layer	Material	Designator	Thickness Inches
	Cu	Plating 2	0.00120
	Cu	Plating 1	0.00070
1	Cu	1/4 oz.	0.00035
PP	370HR (IT-180A, VT-47)	1080	0.00280
PP	370HR (IT-180A, VT-47)	1080	0.00280
2	Cu	1/2 oz.	0.00070
Core	370HR (IT-180A, VT-47)	1080	0.00400
3	Cu	1/2 oz.	0.00070
NF PP	370HR (IT-180A, VT-47)	1080	0.00280
NF PP	370HR (IT-180A, VT-47)	1080	0.00280
4	Cu	1/2 oz.	0.00070
Core	370HR (IT-180A, VT-47)	1080	0.00400
5	Cu	1/2 oz.	0.00070
NF PP	FR406N (VT-47,37N)	1080	0.00280
NF PP	FR406N (VT-47,37N)	1080	0.00280
Coverlay	LF 0110		0.00200
6	Cu	1/2 oz.	0.00070
Core	Flex + Cover Layers	AP8525	0.00600
7	Cu	1/2 oz.	0.00070
Coverlay	LF 0110		0.00200
NF PP	FR406N (VT-47,37N)	1080	0.00280
NF PP	FR406N (VT-47,37N)	1080	0.00280
8	Cu	1/4 oz.	0.00035
	Cu	Plating 1	0.00070
IMS PP	HTC 3.2 (KA04)	3.2 W/m*k	0.00300
IMS PP	HTC 3.2 (KA04)	3.2 W/m*k	0.00300
Metal Core	Cu	17.8 oz.	0.02500
IMS PP	HTC 3.2 (KA04)	3.2 W/m*k	0.00300
IMS PP	HTC 3.2 (KA04)	3.2 W/m*k	0.00300
	Cu	Plating 1	0.00070
9	Cu	1/2 oz.	0.00070
	RO4350F		0.00400
10	Cu	1/2 oz.	0.00070
PP	RO4450F (Itera-MT40)		0.00400
PP	RO4450F (Itera-MT40)		0.00400
11	Cu	1/2 oz.	0.00070
	RO4350F		0.00660
12	Cu	1/2 oz.	0.00070
	Cu	Plating 1	0.00070
	Cu	Plating 2	0.00120
		Total	0.10490

Via Structures

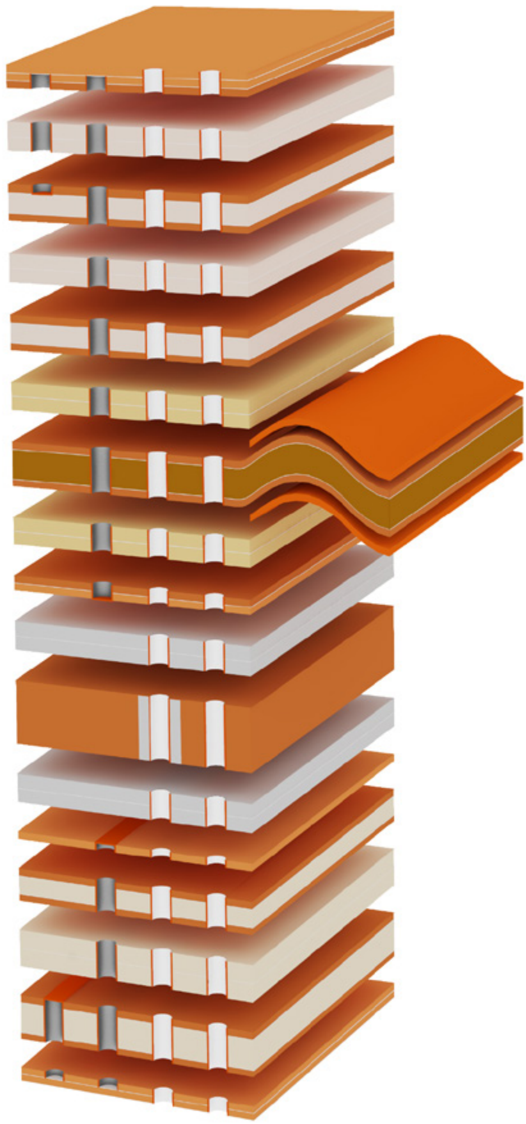
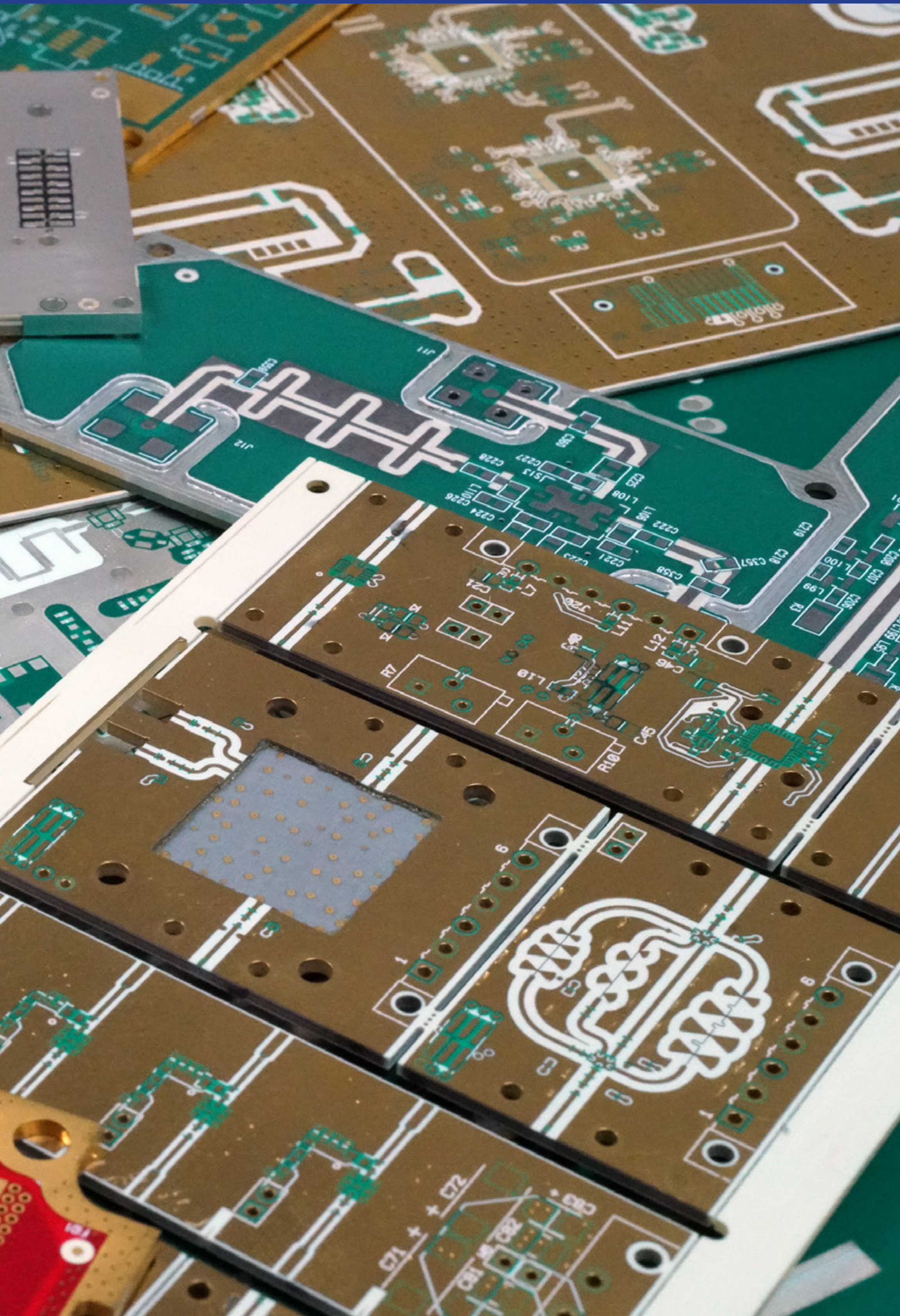


Figure 5-1: Case study of mixed technology dubbed "the kitchen sink" board.


R7
C21
L10
C45
R10
C72
C71
C83

Acknowledgments and Further Reading

Acknowledgments

We would like to acknowledge our customers and designers we have worked with over the years who have challenged us in many ways and have helped us grow in this sector. We would also like to acknowledge our various supplier partners who have helped us to better understand these materials and their applications. We would like to thank John Bushie and David Lackey for input throughout this process and for many of the figures. We would also like to thank Rick Kohn for helping take many of the pictures in this book.

Further Reading

Check out these other books published by American Standard Circuits:

- *The Printed Circuit Designer's Guide to... Fundamentals of RF/Microwave PCBs* by John Bushie and Anaya Vardya
- *The Printed Circuit Designer's Guide to... Flex and Rigid-Flex Fundamentals* by Anaya Vardya and Dave Lackey (visit I-007eBooks.com to download these and other free, educational titles)
- *Fundamentals of Printed Circuit Board Technologies* by Anaya Vardya, Robert Tarzwell, and Dan Beaulieu

Appendix A

Test Methods–Accurately Measuring the Thermal Conductivity of IMPCB or MCPCB Materials

Thermal conductivity (λ) is defined as the ability of a material to transmit heat through itself; for the International System of Units (SI units) it is expressed as W/mK. It can also be expressed as watts per centimeter per degree Kelvin (W/cmK). For imperial units, thermal conductivity is measured in BTU/(h/(ft°F)). Watts is the amount of heat applied to one surface. Per meter or foot is the distance through the material the heat must travel, and per degree Kelvin or °C or °F is one unit of temperature difference.

Measuring the thermal conductance can be done in a few different ways. The results can vary with the method of test parameters, how careful the calibration of the equipment is, and the operator's skill. The most common method for IMS involves placing a disk of the PCB laminate to be measured between two brass disks of known mass. In the disk method, one of the brass disks is heated, and one is cooled. Both heated and cooled sections are encapsulated in insulating foam to keep lab room temperature influence to a minimum. The sample has thermal conducting paste properly applied between the surfaces to minimize micro-voids. Thermal probes inside the two brass disks measure the change in temperature over time. The heavy brass disks take some time to absorb and equalize the heat flowing through the PCB laminate in the Z-axis. Typically, the test runs for half an hour.

The formula shown under "disk-type calculations" is used to calculate the thermal conductance. The disk method is best for thin solid materials, such as printed circuit material. However, the quality of the thermal contact between the brass disk and the laminate will affect the thermal conduction reading but not the thermal radiation movement of heat. For this test to be accurate, the use of high-quality thermal paste gap fillers and or phase change fillers must be done correctly. They should be spread evenly, and pressure should be applied to spread out and minimize the small gaps and air pockets between the laminate and the brass disks before the measurements. Any air pockets will lower the thermal conductance measured.

In the disk test, we measure both conductance (i.e., the movement of heat by excited molecules through the laminate), as well as radiation through the emittance of infrared waves. Most disk measuring units that you can purchase use non-steady-state and measure conductance during the process of one brass plate cooling the other one to equilibrium which results in a quicker test.

The steady-state technique is the measurement when the material that is analyzed is in complete equilibrium (i.e., the end temperatures of the two brass disks are the same). This makes the math analysis easier. The disadvantage, generally, is that it takes longer to reach equilibrium between the two brass disks.

Disk-Type Calculations

Thermal conductivity λ in steady-state is given by the following formula:

$$W/mK = Q * d / (T1 - T2)$$

- Q: The quantity of heat passing through a unit area of the sample in unit time (W/m2)
- d: The distance between two sides of the sample (m)
- T1: The temperature on warmer side of the sample (K)
- T2: The temperature on the colder side of the sample (K)

There are four main types of instruments available to measure thermal conductivity:

1. Insulated or Guarded Hot Plate

Used for thin solid-type materials. It uses two brass disks—one heated, one cooler—with a thin disk of the product to be measured placed between them. The insulation around the two brass disks keeps the heat in and improves accuracy. This is the most used method for IMS material heat conductivity measurements. Low-cost calibrated units that simplify the measuring and calculating of thermal conductance are readily available. This method can be easily purchased as ready to run with very accurate results that can meet ISO22007 standards. Care must be taken as interface gaps in the thermal paste can cause errors.

2. Hot Wire

A heated wire is embedded in the sample to be tested or etched in copper on the surface to provide the heat source. A thermal probe is placed on the other side to measure temperature versus logarithm time as the heat flows through the material. This is designed for low-density foams, fluids, and semisolids. This method is low cost and sometimes used for laminates, but tests have shown it not to be the most accurate. It can be made in the lab with normal PCB materials, but it is not recommended due to inaccuracies.

3. Modified Hot Wire

The hot wire providing the heat is supported on backing, so the wire does not have to penetrate the sample. This design allows for the testing of thin solid samples. The correct density and thermal dissipation of the material under test must be known. This is more accurate than the heated hot wire method; however, it's still not as accurate as the disk or laser flash method.

4. Laser Flash Diffusivity

A laser is used to heat one side of the material with a short pulse of known wattage. An infrared scanner on the back-side measures temperature over time for a specific amount of heat to be passed through the sample. This is not suitable for PCB laminates as it is mostly used for solid metals. Computerized units are sold with software but are relatively expensive.

The various methods differ in the technique, sample mass, type of heat, and testing time. Some are designed for house insulation materials, and some are more suited for measuring thin metals or more solid-type materials.

In addition, IPC is preparing to release a more manual lab method, IPC-TM-650 method 2.4.54, to measure thermal conductivity. It uses a different type of hot wire made by silk-screening resistive carbon paste to create a film heating element added to the fabricated coupon. A T-type thermocouple is embedded within the sample. The test utilizes a constant temperature variable power supply that keeps the material under test at a constant 80°C utilizing an infinite liquid heat sink cooled by a temperature-controlled unit. The amount of power the power supply must input into the surface of the material under test to keep it at a constant 80°C will be measured by the power supply: V x A = watts. Thermal conductance is calculated by the power needed to keep the sample at a constant 80°C by referring to an IPC-supplied chart, which shows the thermal conductance of the sample.

This new IPC test requires five samples be tested, and all must be within specified variances. This new test method is designed to be used only for conformity testing of IMS aluminum and copper metal base laminates of 1.5-mm thickness or greater, including all alloys.

Capability, Accuracy, and Methodologies of Measurement Systems

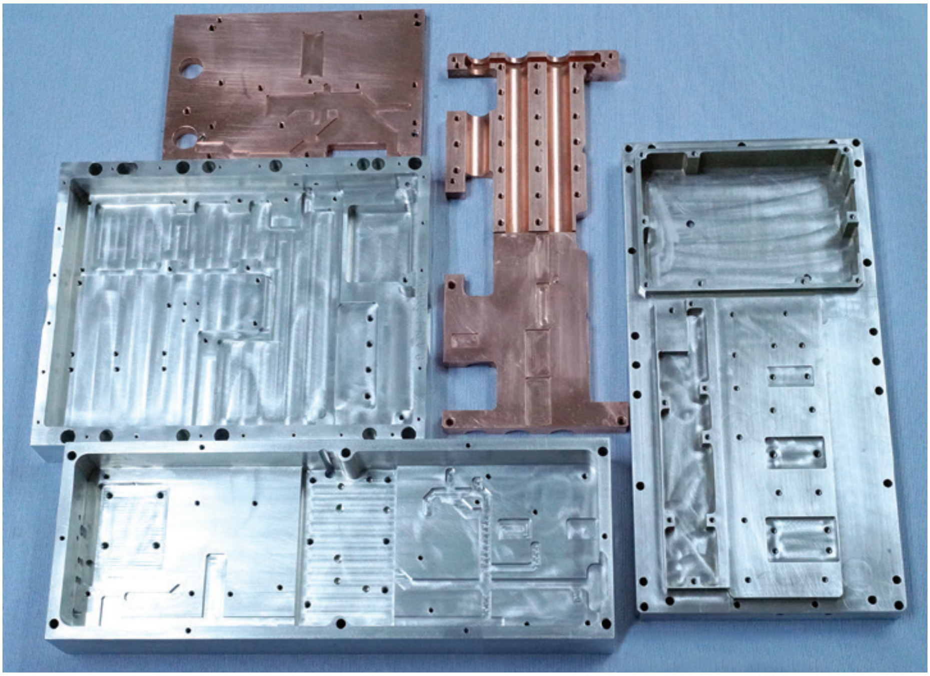
Temperature in the laboratory during the measuring should not vary more than ±2°C, and relative humidity should not exceed 65% or be less than 50%. The sample and the heat sinks should be enclosed in a suitable JEDEC enclosure to limit errors from varying room temperature and air breezes from the air conditioner or heating ducts. The door should remain closed with a sign not to enter. In measuring IMS material for use in LED and heating requirements, the small value of heat conductivity can easily be skewed by inaccurate tests.

Items that can affect the accuracy of the measurement include thermal contact, accuracy and placement of the thermal probes, and moisture content of the laminate (as laminate sits around the lab, it absorbs moisture and increases the measured thermal conductivity). It is suggested to bake the test samples at 225°F or 110°C for an hour before measuring to ensure the sample is low in moisture. High humidity in the lab, breezes or air movement in the lab, and non-constant temperatures in the lab, and the accuracy of your thermal probes and meters may have an impact. ISO equipment testing is required to keep all measuring units in proper calibration.

To ensure your testing is accurate, it is always advisable to have two or three known metal samples, such as thin copper or aluminum of a known thermal conductance and metallurgical number (i.e., 6061 t6 aluminum) to compare your results against. If your standards do not give the same reading as listed in thermal conductance charts, then you need to re-evaluate your test method.

Typical PCB electrodeposited (ED) trace copper is around 388 W/mK but varies slightly with chemical composition. FR-4 laminate varies by resin type and content and glass percentages between 0.25–0.34 W/mK. A typical solid aluminum backing for IMS material is approximately 200 W/mK. An IMS aluminum-based laminate with 1 mil of thermally conductive insulation and 1 oz. copper will be in the 0.4–7 W/mK range.

Comparing thermal conductance lab measurements against PCB layout software, such as Mentor's HyperLynx® Thermal can produce different results than expected. Advanced PCB layout software analyzes board-level thermal conditions on fully routed PCBs by using finite element analyses to simulate thermal conduction, convection, and radiation. The analysts produce temperature profiles, gradients, and excess temperature maps. These maps allow the designer to resolve board and component area overheating early in the design process. Finite element analysis allows engineers to simulate thermal and power integrity analysis, enabling a better understanding of the effect of power distribution network current densities on board temperatures.



Aluminum- and copper-machined heat sinks.

Appendix B: Laminate Selection Guide

Materials	Thermal Conductivity	Thermal Impedance (Dielectric Thickness)					Max Operating Temp	Proof Test (High Pot)	Peel Strength
	°W/m-K	25µ (1.0 mils) °C in^2/W	35µ (1.4 mils) °C in^2/W	50µ (2.0 mils) °C in^2/W	75µ (2.9 mils) °C in^2/W	90µ (3.5 mils) °C in^2/W	°C	V	lb/in
FlexTherm	0.7	0.055	0.078				140	N/A	16.0
VT-4B1	1.0			0.078	0.116		130	2000	11.0
TOTKING T110F Bendable	1.0	0.039		0.078			125	1000	10.5
Cobritherm Alcup G	1.3						130	1000	13.1
Bergquist MP (3)	1.3					0.09	130	N/A	9.0
Cobritherm Alcup G-NT	1.3**						130	1000	10.3
TOTKING T110	1.5			0.052	0.078		125	4500	9.0
VT-4A1	1.6				0.074		90	4500	12.0
Cobritherm Alcup	1.8						150	2000	10.3
TOTKING S111	2.0			0.039	0.0581		115	4500	9.4
TOTKING T111	2.0			0.039	0.0581		125	4500	9.0
TOTKING K12	2.0			0.039	0.0580		130	4500	9.7
Laird Tlam LLD	2.0						150	N/A	6.0
TOTKING M111 7MPA Ultra low Elastic Modulus	2.0			0.039	0.058		130	4500	4.2
VT-4A2	2.2				0.054		90	4500	12.0
VT-4A2H	2.2				0.054		105	4500	12.0
VT-5A2	2.2						150	1000	7.0
Laird Tlam HTD	2.2						150	N/A	6.5
Cobritherm HTC 2.2	2.2					0.063	150	1500/ 3000****	16.0
Bergquist HT	2.2					0.05	140	N/A	6.0
TOTKING S112	3.0			0.026	0.039		115	4500	9.7
TOTKING T112	3.0			0.026	0.039		125	4500	9.5
TOTKING K13	3.0			0.026	0.039		130	4500	9.7
TOTKING T113 high TG180	3.0			0.039	0.058		125	4500	9.0
VT-4B3	3.0			0.027	0.04		130	2000	11.0
Laird Tlam SS 1Ka	3.0						110-130	N/A	5.0
TOTKING F112 800MPA low Elastic Modulus	3.0			0.026	0.039		130	4500	4.2
Bergquist HPL	3.0***		0.02				140	N/A	5.0
Cobritherm HTC 3.2 *****	3.2		0.017	0.024	0.036	0.044	150	1500/ 3000	11.4/ 16.0
VT-4B5L	3.6						130	4000	5.0
TOTKING K14	4.0			0.019	0.029		130	4500	9.7
Cobitherm HTC 4.0 Ultra Thin Tg180°C	4.0		0.014	0.020	0.031		150	750	10.3
VT-4B5	4.2			0.02	0.029		130	2000	7.0
VT-4B5H	4.2			0.019	0.029		150	2000	5.0
VT-4B5SP	4.2		0.015				120	1000	7.0
TOTKING K15	5.0			0.016	0.023		130	4500	9.8
VT-4B7	7.0			0.011	0.017		130	2000	5.0
VT-4B7SP	7.0		0.009				130	1000	5.0
TOTKING K18	8.0			0.01	0.015		130	4500	10.3

Note:
**Cobritherm Alcup G-NT thermal condictivity is 1.3 °W/m-K (ASTM-D 5470 Test Method) and 2.0 °W/m-K (ASA-7540 Test Method)
***Bergquist HPL is not commercially available as a laminate.
****130µ High Pot = 3000V, 90µ High Pot = 1000V
***** Cobritherm HTC 3.2 thickness ≤75µ (2.9 mils) is branded as Cobritherm HTC3.2 Ultra Thin

All Ventec A series materials are glass reinorced and available in copper core and prepres. All B series materials are un supported (no glass) these are not available in prepregs. All TOTKING T/S/K series materials are available in copper/ferrum/stainless steel core.

Chart 1: Thermal impedance 25µ–90µ.

Materials	Thermal Conductivity						Max Operating Temp	Proof Test (High Pot)	Peel Strength
	°W/m-K	100µ (3.9 mils) °C in^2/W	120µ (4.7 mils) °C in^2/W	130µ (5.1 mils) °C in^2/W	150µ (6.0 mils) °C in^2/W	200µ (8.0 mils) °C in^2/W	°C	V	lb/in
FlexTherm	0.7						140	N/A	16.0
VT-4B1	1.0	0.166					130	2000	11.0
TOTKING T110F Bendable	1.0						125	1000	10.5
Cobritherm Alcup G	1.3		0.143				130	1000	13.1
Bergquist MP (3)	1.3				0.14		130	N/A	9.0
Cobritherm Alcup G-NT	1.3**	0.12					130	1000	10.3
TOTKING T110	1.5	0.103	0.124		0.155	0.207	125	4500	9.0
VT-4A1	1.6	0.099	0.123		0.148		90	4500	12.0
Cobritherm Alcup	1.8	0.086	0.103				150	2000	10.3
TOTKING S111	2.0	0.0775	0.0930		0.1163	0.1550	115	4500	9.4
TOTKING T111	2.0	0.0775	0.0930		0.1163	0.1550	125	4500	9.0
TOTKING K12	2.0	0.0775	0.0930		0.1163	0.1550	130	4500	9.7
Laird Tlam LLD	2.0	0.07					150	N/A	6.0
TOTKING M111 7MPA Ultra low Elastic Modulus	2.0	0.078	0.093		0.116	0.155	130	4500	4.2
VT-4A2	2.2	0.072	0.089		0.107		90	4500	12.0
VT-4A2H	2.2	0.072	0.089		0.107		105	4500	12.0
VT-5A2	2.2						150	1000	7.0
Laird Tlam HTD	2.2	0.072			0.107		150	N/A	6.5
Cobritherm HTC 2.2	2.2	0.092		0.092			150	1500/ 3000****	16.0
Bergquist HT	2.2				0.11		140	N/A	6.0
TOTKING S112	3.0	0.052	0.062		0.078	0.103	115	4500	9.7
TOTKING T112	3.0	0.052	0.062		0.078	0.103	125	4500	9.5
TOTKING K13	3.0	0.052	0.062		0.078	0.103	130	4500	9.7
TOTKING T113 high TG180	3.0	0.078	0.093		0.116	0.155	125	4500	9.0
VT-4B3	3.0	0.053	0.067		0.08		130	2000	11.0
Laird Tlam SS 1Ka	3.0	0.05			0.08	0.11	110-130	N/A	5.0
TOTKING F112 800MPA low Elastic Modulus	3.0	0.052	0.062		0.078	0.103	130	4500	4.2
Bergquist HPL	3.0***						140	N/A	5.0
Cobritherm HTC 3.2 *****	3.2			0.063			150	1500/ 3000	11.4/ 16.0
VT-4B5L	3.6	0.043					130	4000	5.0
TOTKING K14	4.0	0.039	0.047		0.058	0.078	130	4500	9.7
Cobitherm HTC 4.0 Ultra Thin Tg180°C	4.0						150	750	10.3
VT-4B5	4.2	0.038			0.058	0.076	130	2000	7.0
VT-4B5H	4.2	0.038			0.057		150	2000	5.0
VT-4B5SP	4.2						120	1000	7.0
TOTKING K15	5.0	0.031	0.037		0.047	0.062	130	4500	9.8
VT-4B7	7.0	0.022					130	2000	5.0
VT-4B7SP	7.0						130	1000	5.0
TOTKING K18	8.0	0.019	0.023		0.029	0.039	130	4500	10.3

Note:
**Cobritherm Alcup G-NT thermal condictivity is 1.3 °W/m-K (ASTM-D 5470 Test Method) and 2.0 °W/m-K (ASA-7540 Test Method)
***Bergquist HPL is not commercially available as a laminate.
****130µ High Pot = 3000V, 90µ High Pot = 1000V
***** Cobritherm HTC 3.2 thickness ≤75µ (2.9 mils) is branded as Cobritherm HTC3.2 Ultra Thin

All Ventec A series materials are glass reinorced and available in copper core and prepres. All B series materials are un supported (no glass) these are not available in prepreg
All TOTKING T/S/K series materials are available in copper/ferrum/stainless steel core.

Chart 2: Thermal impedance 100µ–200µ.

Glossary

Additive Process: A process in PCB manufacturing where the circuit pattern is produced by the addition of metal rather than etching metal away.

Air Gap: A routed space between two traces to control creepage.

Aspect Ratio: The ratio of the circuit board thickness to the smallest drilled hole diameter.

Ball Grid Array (BGA): A leadless chip package in which the external terminals form a grid-style array with solder balls that carry the electrical connection to the outside of the package. The PCB design will have round landing pads to which the solder balls are soldered when the PCB is heated in a reflow oven.

Bare Board: An unpopulated PCB with no components assembled on it yet.

Base Laminate: The dielectric material upon which the conductive pattern may be formed. The base material may be rigid or flexible.

Breakdown Voltage: The voltage at which an insulator or dielectric ruptures or at which ionization and conduction take place and creates an arc.

Buried Vias: Vias that start and end in the middle of the board.

Clad or Cladding: A thin layer or sheet of copper foil that is bonded to a composite laminate core to create the base material for printed circuits.

Coefficient of Thermal Expansion (CTE): Thermal fractional change in dimension of a material for a unit change in temperature, expressed as ppm or a percentage.

Computer-Aided Design (CAD): A software program that calculates impedance modeling and provides graphical creation of a PCBs conductor layout and signal routes.

Conductor: A copper area on a PCB surface or internal layer usually composed of lands (to which component leads are connected) and paths (traces).

Controlled Impedance: The matching of substrate material Dk with trace dimensions and locations to create specified electric impedance as required by the designers.

Copper Thickness and Copper Plating: Copper thickness usually specified in terms of the number of oz/sq. ft. 1/2 oz: 17.5 μm or 0.0007"/sq. ft; 1 oz: or 35 μm or 0.0014"/sq. ft. The thickness of copper specified will be the final thickness of base material plus copper plating thickness. Generally, base material comes with 1/4, 1/2, 1, and 2 oz., but finished copper thickness can range from 1/2 to 6 oz.

Core: The copper foil laminated fiberglass panel that PCBs are built upon (also known as substrate panel or interlayer).

Delamination: A separation between any of the layers of a base material or between the laminate and the conductive foil, or both.

Dielectric: An insulating medium that occupies the region between two or more conductors and prevents electrical shorts.

Dielectric Constant (Dk): The ratio of the permittivity of the material to that of a vacuum (referred to as relative permittivity).

Double-Sided Board: A circuit board with conductive copper patterns on both sides with through connected vias.

Embedded: Resistors, capacitors, and small chip die are placed inside the PCB to increase density.

Embedded Copper Coin: An integrated solid copper heatsink embedded in the PCB.

ENIG: Electroless nickel immersion gold final finish.

Etch: The chemical removal of copper to achieve a circuit pattern.

Flex Circuit: A printed circuit made of thin, flexible material.

FR-4: A grade of flame-retardant industrial laminate with a substrate of woven-glass fabric and resin binder of epoxy. It is the most common dielectric material used in the construction of PCBs. Its dielectric constant is 4.4–5.2 at below-microwave frequencies. As frequency climbs over 1 GHz, the dielectric constant of FR-4 gradually drops (Tg 150–175°C).

Glass Transition Temperature (Tg): The temperature at which a polymer changes from a hard and relatively brittle condition to a viscous or rubbery condition. When this transition occurs, many physical properties undergo significant changes. Some of those properties include hardness, brittleness, coefficient of thermal expansion (CTE), and specific heat.

Ground Plane: A copper conductor layer used as a common reference point for circuit returns, shielding, or heat sinking.

High Potential (HiPot) Test: A stress test of the insulation of a device under test (DUT). Such a test applies a voltage to the DUT that is much higher than normal operating voltage; typically, 1000V AC plus twice the normal operating voltage.

Hot Air Solder Leveling (HASL): A method of coating exposed copper with solder by inserting a panel into a bath of molten solder and then passing the panel rapidly past a series of hot air jets to remove excess solder to help detect potential dielectric breakdown failures.

Impedance: A capacitive opposition to the flow of AC electrical current. This term is used to describe how high-frequency circuit boards will react.

Laminate: A product made by bonding together two or more composite layers of material.

Lamination: The process of pressing a laminate in a hot high-pressure hydraulic press.

Land: A portion of a copper conductor usually, but not exclusively, used for the connection and/or attachment of components (also called a pad).

Legend: Silkscreen printed letters or symbols on the PCB, such as part numbers and product, typically in white.

Manufacturability: A term defining the ability of a board design to meet manufacturing requirements.

Mask: A material applied to create selective etching, plating, or the application of solder or solder mask to a PCB.

Micro-Inches: A unit a measurement in millionths of an inch, which is a common unit of measurement in the PCB industry.

Micro-Sectioning: The creation of a specimen for the microscopic examination of the material to be examined, usually by cutting out a cross-section, followed by encapsulation, polishing, ammonia etching, and staining.

Microvia: A via used to make connections between two adjacent layers, typically less than 6 mils in diameter. May be formed by laser ablation, plasma etching, or photo processing.

Mil: One-thousandth of an inch 0.001" (0.0254 mm).

Multilayer Circuit Board: A processed printed circuit configuration consisting of alternate layers of conductive patterns and insulating materials bonded together in more than two layers.

Non-Plated Hole: A hole in the PCB that is drilled after plating, so it is not plated.

Open: An unwanted break in the continuity of an electrical circuit, which prevents current from flowing.

Pad: The portion of the conductive pattern on printed circuits designated for the mounting or attachment of components (also called a land).

Panel: The square or rectangular base material containing one or more circuit patterns that passes successively through the production sequence and from which PCBs are extracted, typically 12" x 18" or 18" x 24".

Passive Component: A device that does not add energy to the signal it passes (e.g., resistors, capacitors, and inductors).

Pitch: The nominal distance between the centers of adjacent features or traces on any layer of a PCB (also known as center-to-center spacing).

Plasma: A highly ionized gas containing an approximately equal number of positive ions and negative electrons. As a whole, it is electrically neutral, though conductive, and affected by magnetic fields. It is used to clean contaminants off a PCB.

Plated Through-Hole (PTH): A hole in a circuit board that has been plated with metal (usually copper) on its sides to provide electrical connections between conductive pattern layers.

Plating: Chemical or electromechanical deposition of metal on a pattern.

Prepreg: Sheet material consisting of the base material impregnated with a synthetic resin, such as epoxy or polyimide, partially cured to the B-stage, which is an intermediate stage (short for pre-impregnated).

Quality Control (QC): A precise system of measurements to ensure the PCB meets the desired specifications (also called quality assurance or QA).

Quick Turn: The ability to produce a small lot of a product in a relatively short time (i.e., fabricating a PCB in 24 hours from receipt of the design data).

Registration: The amount of conformity of the true position of a pattern with its intended position to that of any other point.

Resist: Coating material used to mask or to protect selected areas of a pattern from the action of an etchant, solder, or plating.

Resistivity: The ability of a material to resist the passage of electrical current through it.

RF: Radio frequency.

Rigid-Flex Circuit: A PCB construction combining flexible circuits and rigid multilayers to provide a direct connection or to make a three-dimensional form that may include components.

RoHS: Stands for Restriction of Hazardous Substances, which is part of the European Union Directive 2002/95/EC1. This directive on the restriction on the use of certain hazardous substances in electrical and electronic equipment bans or severely curtails the use of lead, chromium, mercury, polybrominated biphenyls, cadmium, and polybrominated diphenyl ethers in all products from automobiles to consumer electronics.

Scoring: A machine in which grooves are cut on opposite sides of a panel to a depth that permits individual boards to be separated from the panel after the component assembly.

Single-Sided Board: Circuit board with copper conductors on only one side and no PTHs.

Solder Leveling: The process of dipping PCBs into hot solder and leveling with hot air.

Solder Mask: An ink or dry film coating applied to a circuit board to prevent solder from flowing onto any areas where it is not desired or from bridging across closely spaced conductors.

Surface-Mount Technology (SMT): Defines the entire body of the process and components that create PCB assembly with leadless components.

Td: Temperature of decomposition, where the circuit loses 5% of its volume due to outgassing.

Tg: Glass transition temperature (°C), or the point at which the material starts to become soft and plastic-like. It's also the point where the Z-axis starts to expand non-linearly.

Thermal Conductivity: A material property (W/mK).

Through-Hole: A plated hole on a circuit board used for component pins leads. The holes are plated, creating a circuit between multilayers.

Trace: A common term for the copper conductors.

V-Scoring: A board profiling process that involves cutting straight tapered lines from both sides of the board. Suitable for medium- to large-volume production with panels requiring only straight-line cuts. With this process, minimum space is needed between units, and the panel can be assembled as a larger board.

Via: A PTH that is used as an inner layer connection but doesn't have a component lead in it.

Void: The absence of substances in a localized area (e.g., air bubbles).

Warping: Generally refers to finished board warp and twist. All boards may have a certain degree of warp as a result of manufacturing. Customers will specify the warping tolerance.

Wicking: The migration of conductive copper chemicals into the glass fibers of the insulating material around a drilled hole.

About American Standard Circuits

There isn't anything we won't do for our customers.

Our goal is to provide our customers with the best quality and value for all their PCB requirements. First and foremost, American Standard Circuits wrote this book for our customers. Our intention in providing this guidebook to our customers is to make sure that they know everything possible to design the very best and most efficient and economical thermal management PCB solutions possible.

Our ongoing commitment to leading-edge, high-level interconnect technology, cost-effective manufacturing, and unparalleled customer service has put us at the forefront of advanced technology circuit board fabrication.

We manufacture quality rigid, metal-backed, flex and rigid-flex printed circuit boards on numerous substrates for a variety of applications, including:

- Military/Aerospace
- Industrial
- Commercial
- Medical
- Telecommunications
- Consumer Electronics
- RF/Microwave
- Transportation
- Advanced Technology

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

Our ongoing R&D work in both thermal management and embedded passive components allows us to keep up with ever-increasing complexity, various clock speeds, power consumption, and heat output computing devices. Our specialties include hybrid circuits (made from multiple materials), exotic materials, flex / rigid-flex circuits, blind and buried vias, controlled impedance, and thermal management solutions (for which ASC holds several patents).

Quality

ASC has state-of-the-art process control systems, and our certifications include AS9100 Rev. D, ISO 9001:2015, IATF 16949: 2016, MIL-PRF-50884 and MIL-PRF-31032. We have rigorous SPC controls, automated optical inspection (AOI), metallographic cross-section, and electrical test facilities. These provide the quality assurance needed to meet the high standards of our customers.

Quick Turns

We have developed the ability to provide quick turns for all the technologies that we manufacture. Our world-class front-end engineering (CAM) systems and processes, coupled with our rapid-response manufacturing processes, enable us to provide quick-turn options for our customers.

You can get more information about us from our website asc-i.com, or stop by our facility in West Chicago, Illinois, for a detailed tour.