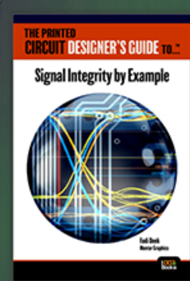
I-Connect007 Presents

December 2017

the pcb magazine

FREE SIGNAL INTEGRITY eBOOK

I-007e Books

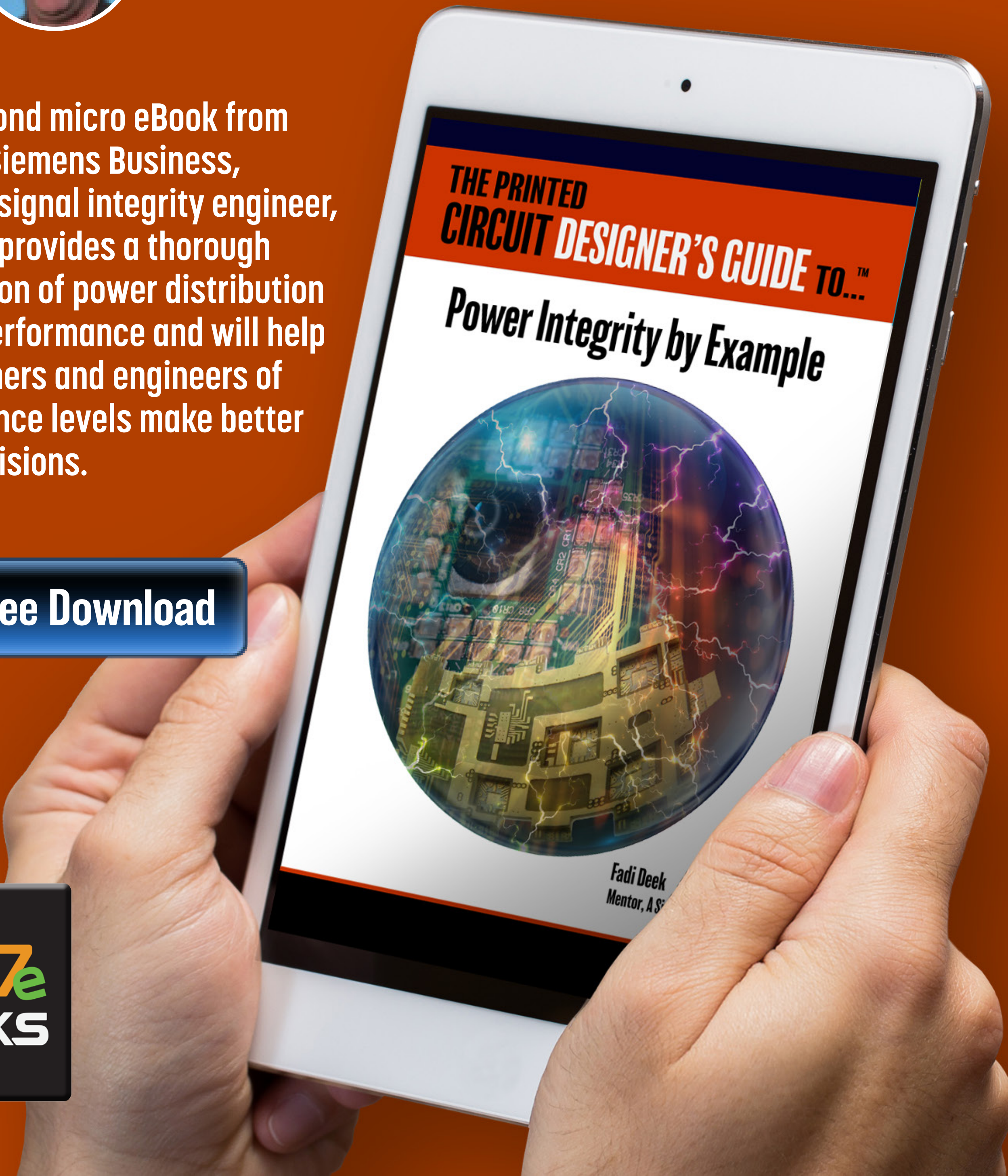


“This micro eBook should be required reading for every PCB designer or design engineer, whether novice or veteran.”

Andy Shaughnessy
Managing Editor, The PCB Design Magazine

In this second micro eBook from Mentor, A Siemens Business, written by signal integrity engineer, Fadi Deek, provides a thorough investigation of power distribution network performance and will help PCB designers and engineers of all experience levels make better design decisions.

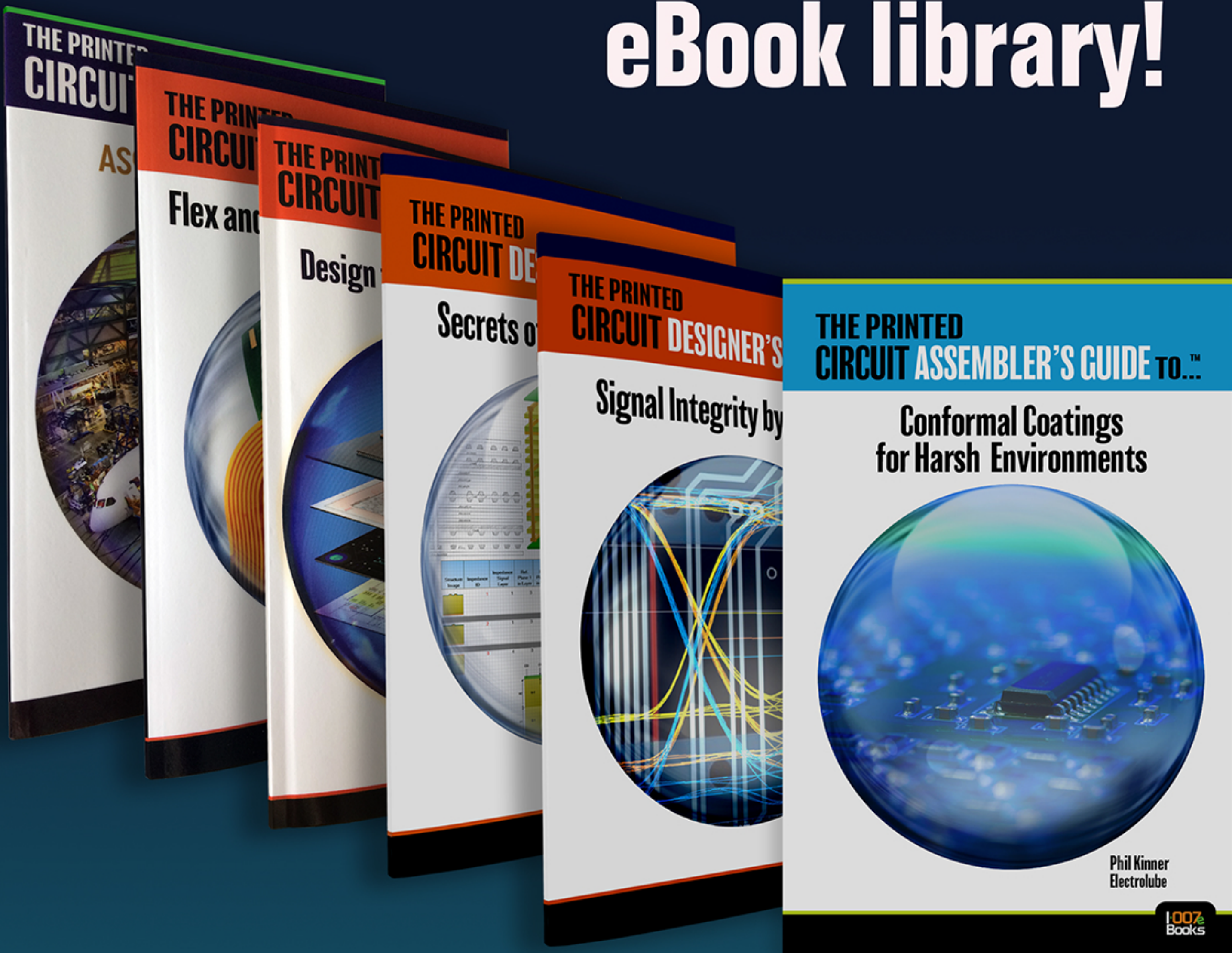
Free Download



I-007e Books

my Premium Content

Download free books from our micro eBook library!



Written by industry experts, these free micro eBooks each contain valuable information on relevant topics in an easy-to-read format.

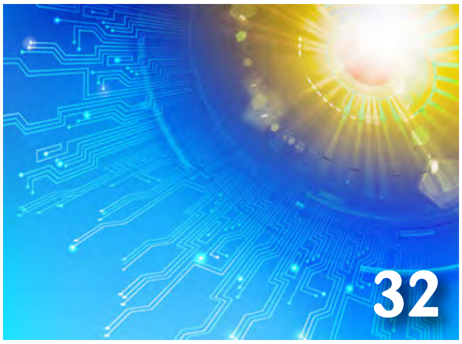
Log in or register to access all of our premium content at: myiconnect007.com

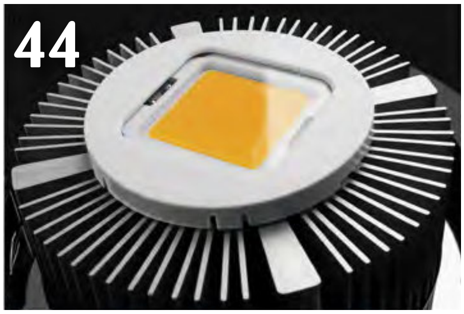
my I-Connect007 GOOD FOR THE INDUSTRY

I-Connect007 Presents

the pcb magazine

December 2017
Featured Content





Thermal Management

Increasing consumer demands for high performance and the need for high reliability in sectors such as automotive, LED lighting, and renewable energy mean that thermal management is now front and center on the priority list for PCB manufacturing. This month, industry experts are in the hot seat to explain how to keep everything cool.

FEATURES:

Tachyon® 100G laminate materials have superior fill and flow compared to competitive products that require very high pressure (500psi) with proven sequential lamination and HDI structure performance.

1000-HOUR PASSED CAF

Tachyon® 100G

Ultra-Low Loss Laminate & Prepreg

Tg: 200°C Td: 380°C

- Ultra-low loss material with Dk: 3.02 and Df: 0.0021
- Stable Dk over temperature range: -55 to125°C and up to 100 GHz
- Scalable solution for next generation designs of backplanes and daughter cards, enabling 10x improvements from 10 Gb/s data rates
- 1000-Hours CAF Tested
- Low moisture absorption
- 0.8 mm pitch capable
- 6x 260°C reflow capable
- 6x 288°C solder float capable
- RoHS Compliant
- Multiple lamination cycles
- HDI technology compatible

Tachyon® 100G

HIGH SPEED DIGITAL

Designed for Very High Speed Digital Applications (>100 Gb/s)

© 2017 ISOLA GROUP. ALL RIGHTS RESERVED.

AEROSPACE & DEFENSE • COMPUTING • STORAGE • NETWORKS • COMMUNICATIONS

isola

FOR MORE INFORMATION, CONTACT YOU YOUR LOCAL SALES REPRESENTATIVE.

Isola Group
3100 West Ray Road
Suite 301
Chandler, AZ 85226
Phone: 480-893-6527
info@isola-group.com

Isola Asia Pacific (Hong Kong) Ltd.
Unit 3512 - 3522, 35/F
No. 1 Hung To Road, Kwun Tong,
Kowloon, Hong Kong
Phone: 852-2418-1318
info.hkg@isola-group.com

Isola GmbH
Isola Strasse 2
D-52348 Düren, Germany
Phone: 49-2421-8080
info-dur@isola-group.com
Website: isola-group.com

I-Connect007 Presents

the pcb magazine

December 2017

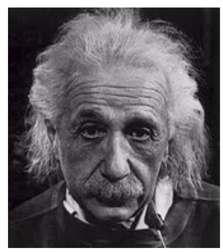
More Content

SHORTS

DEPARTMENTS

SPECIAL SHOW COVERAGE

COLUMNS



HIGHLIGHTS


EXTREME THERMAL CONDUCTIVE
SOLDER MASK
HIGH HEAT
BY TAIYO
PSR-4000 CC01SE
PSR-4000 HS2W / HS3W
SEE YOU AT THE
IPC APEX EXPO 2018
BOOTH #331, SAN DIEGO CONVENTION CENTER
MEETINGS & COURSES:
FEBRUARY 24 - MARCH 1
CONFERENCE & EXHIBITION:
FEBRUARY 27 - MARCH 1
TAIYO AMERICA
Your Ultimate Solution
Phone [775] 885-9959 • www.taiyo-america.com • info@taiyo-america.com
in

Getting the Heat Out!

by Patty Goldman
I-CONNECT007

Increasing consumer demands for high performance and the need for high reliability in sectors such as automotive, LED lighting, and renewable energy mean that thermal management is now front and center on the priority list for PCB manufacturing. This month, industry experts are in the hot seat to explain how to keep everything cool.

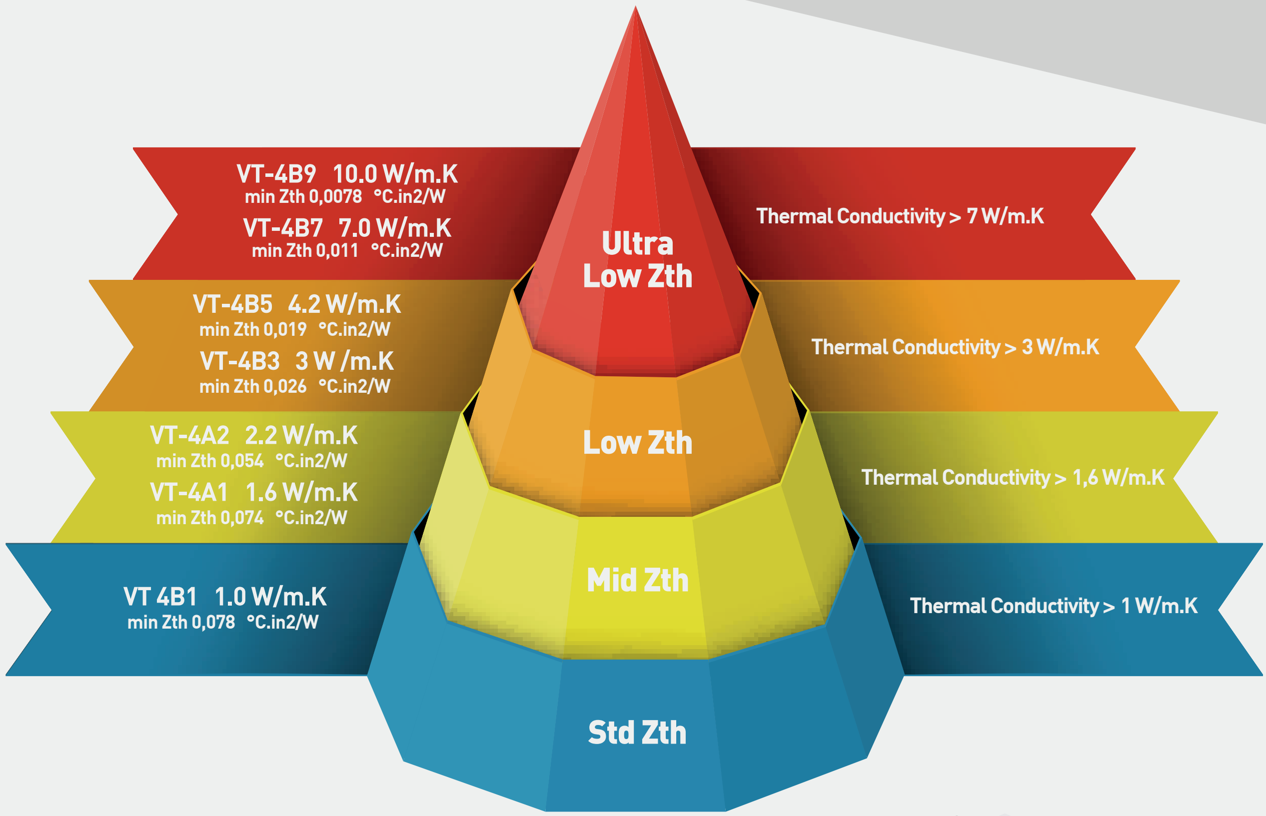
Who would have guessed a few years back that the green movement would so heavily involve the PCB industry—and it's at all not how one would have imagined then. The advent of LEDs in lighting for homes, automobiles, street lamps and much more has created a huge demand for heat-dissipating circuit boards. And the latest high-power chips (power = heat) are requiring innovative ways and materials to keep things cool. Without a doubt the automotive industry is the new big player in power electronics and we are all taking notice.

I am constantly asked or asking the question of articles and columns: Is this design-related or fabricator-related? And truth be told, it's a mixed bag. But I find it increasingly difficult to separate the two as I feel strongly that each needs to know a whole lot more about what the other does. We often hear about lack of communication between designers, fabricators, assemblers and the final OEM customer—and truly this issue always comes up when talking with people in our industry. So if you think "this article isn't for me," think again and give it a read. Whether you benefit from the detail is only part of the takeaway; it's always valuable to understand how your customer/supplier thinks.

We start off with Ventec's Mark Goodwin explaining the surge (pun intended) in power electronics. He presents an excellent, detailed guide to the material choices for thermal management, including new dielectric materials.

Delivering an unprecedented level of thermal performance

ventec
INTERNATIONAL GROUP
騰輝電子



ventec-thermal

Technology leader for high performance thermal management materials

With the market need for ever-increasing thermal performance, reliability and quality, particularly for automotive and other LED lighting, power electronics and DC power conversion applications, the Ventec VT-4B family of thermally conductive IMS (Insulated Metal Substrates) deliver an unprecedented level of thermal performance through their established ceramic-filled halogen-free dielectric technology. Multilayered constructions are made possible through resin-coated foil and resin-coated film options. Contact us to discuss your requirements.

Ventec International Group
21 Water Street, Amesbury
Massachusetts, MA 01913, US
T: 978-521-9700
E: saleseast@ventec-usa.com
@VentecLaminates

www.**venteclaminates**.com

Next, Anaya Vardya, CEO of American Standard Circuits, carefully explains the construction of metal-clad PCBs from single-sided through multilayer. He highlights with options, potential concerns, and material choices along the way.

Our third feature article on this month's topic comes from Jim Barry, PCB Technologies Ltd. He discusses all the different heatsink technologies available touching on thermal vias, coin technology, and copper thickness choices, among others. His first figure succinctly illustrates the options for heat dissipation and is a great "picture is worth 1000 words."

This is followed by a feature by Aismalibar's Gareth Parry. He focuses on LED applications offering practical advice for both design and fabrication.

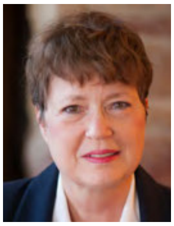
We keep hearing bits and pieces about a looming copper shortage and that is the focus of Elmatica's column authored by CEO Didrik Bech. We are now competing with the lithium battery industry for ED copper, one of the most basic building blocks for PCB laminate material. The other two major components, woven glass fiber and epoxy resin, are also rumored to be in limited quantity. Needless to say, Bech's call to action should be heeded by everyone in the supply chain, like, now.

Even while we wrestle with new technologies, the same old processing problems don't always disappear. RBP Chemical's Mike Carano gives a troubleshooting lesson on mouse bites (yes, you read that right). If you are unfamiliar with this highly descriptive term, talk to the guys in the plating area and QC and they will fill you in.

Next, Tara Dunn, Omni PCB, regales us with a short story to illustrate the many advantages of flexible circuits—including thermal management. While pointing out these advantages she mentions numerous applications that show you why flex is a real growth market.

And rounding out our magazine this month is Steve Williams of The Right Approach Consulting, with a discussion on company culture change as a fundamental part of quality improvement. We all know things don't get easier out there so this is all good info to heed.

So that wraps up another heavy-duty tech issue for you and, hey, it's the end of the year already! How did that happen? I hope you found The PCB Magazine to be a valuable resource for you and your company. We have, of course, been planning issues for next year, starting off with "Choosing the right equipment for your current and future needs" in January. That issue will also include a special IPC APEX EXPO 2018 pre-show section, featuring interviews and other news on the event to be held in San Diego. Looking forward to it and meeting some of you there. In the meantime, go ahead and get subscribed if you haven't been paying attention to me. **PCB**



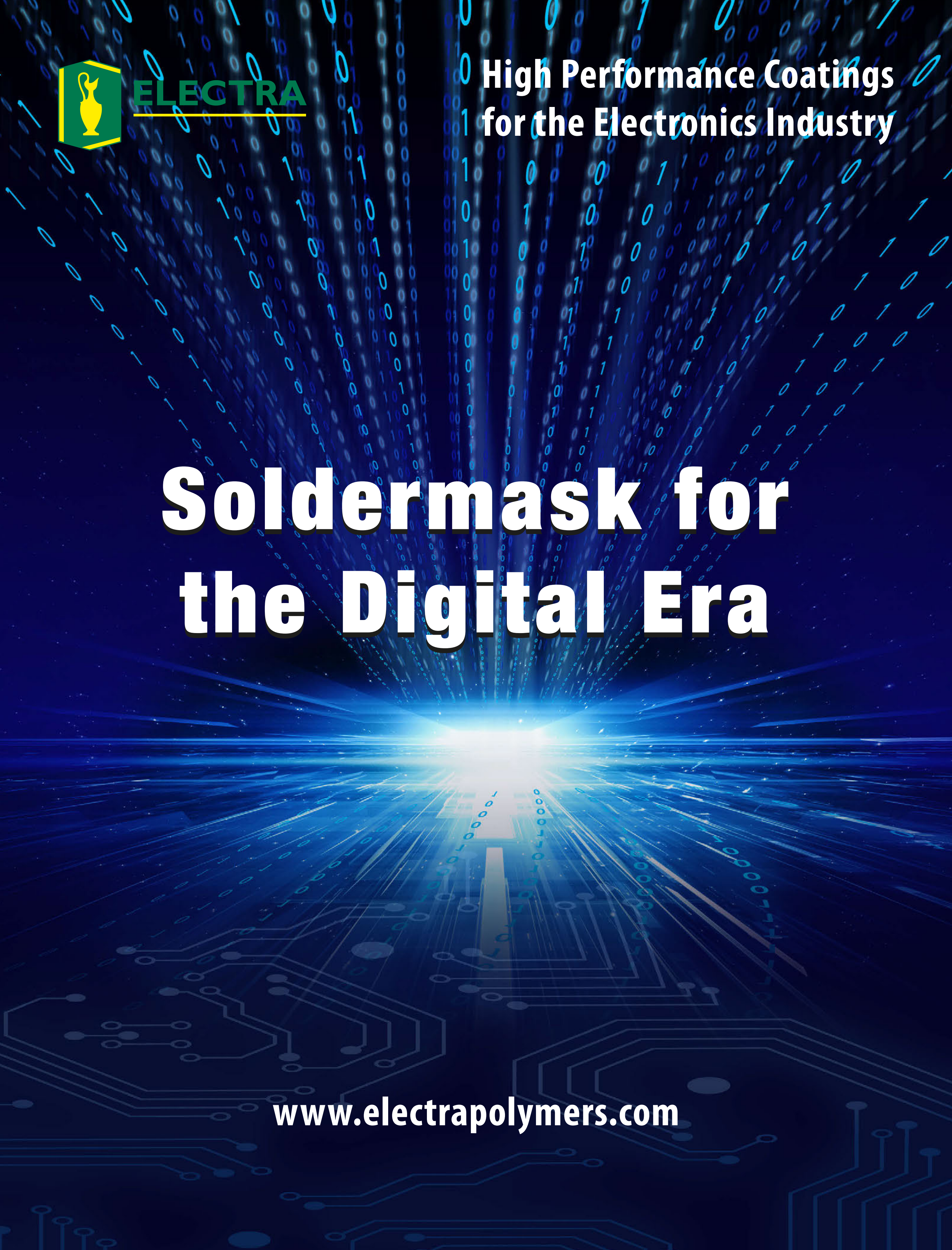
Patricia Goldman is managing editor of *The PCB Magazine.* To contact Goldman, click here.

Deep Learning used to Reconstruct Holograms and More

A form of machine learning called deep learning is one technology behind advances in applications like real-time speech recognition and automated image labeling.

The approach, which uses multi-layered artificial neural networks to automate data analysis, also has shown significant promise for health care. It could be used, for example, to automatically identify abnormalities in patients' X-rays, CT scans and other medical images and data.

In two new papers, UCLA researchers report that they have developed new uses for deep learning: reconstructing a hologram to form a microscopic image of an object and improving optical microscopy.


ELECTRA
High Performance Coatings
for the Electronics Industry
Soldermask for
the Digital Era
www.electrapolymers.com

Understanding Thermal Management and Materials to Boost Power Electronics Reliability

by Mark Goodwin
VENTEC INTERNATIONAL GROUP

Introduction

Understanding how to use advanced thermal materials can help the automotive industry and other sectors, such as LED lighting and renewable energy, deliver greener products that meet consumers' demands for high performance and reliability.

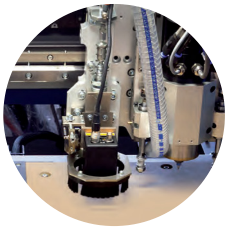
Power Electronics in Reliability-Conscious Industries

Increasing electrification of high-power systems in modern vehicles means additional power electronic subsystems are coming on board to handle functions such as battery management, kinetic-energy recovery, and high-power traction motor control. Peak demands can drive heavy currents through devices and circuits, amounting to hundreds of kilowatts in the highest-power motor drives. As a result, thermal management is vital to keep operating temperatures within acceptable limits.

The lifetime of semiconductor components like power transistors or diodes is known to reduce by about 50% for every 10°C rise in operating temperature. Hence effective thermal management, which helps internally generated heat leave the component efficiently, can extend the lifetime by limiting the rise in die temperature caused by passing current. If the predicted lifetime of the component can be extended beyond the intended lifetime of the vehicle, reliability can be assured.

Reliability, of course, is extremely important in automotive markets, and today's car makers have earned their high reputations through painstaking attention to detail through many generations of conventional vehicles. Those reputations are on the line as the industry moves into unfamiliar territory and seeks to master the issues surrounding the reliability of power electronic systems.

MULTIPLE PLATFORMS



Drill/Rout/Optimizer
X-Ray Vision

MORE INFO

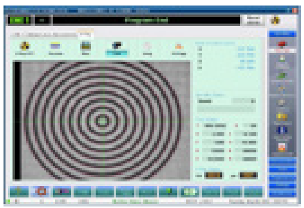
Drill/Rout Multi-Spindle
CCD/RGB Vision

MORE INFO

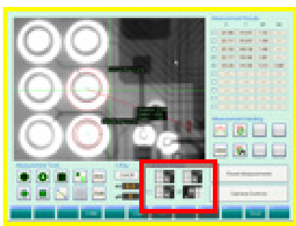
Dual Spindle All in One
Vision System

MORE INFO

SEEING IS BELIEVING



X-Ray - 8X Faster!
with NEW concentric target



See inside your PCB
measure your results!

Drill/Rout/Optimize
Entry level *"Smart Drill"*

MORE INFO

TIME-TESTED FEATURES

Vision Simplifies

Post Etch Slot/Drill

X-Ray Optimization

CCD Camera Optimization
"Smart Drill"

BarCode Job
Data Management

Panel Marking/Serialization

1700 Tool Library per spindle

Control Depth Drilling and
Routing to 15 μm

Lights out Automation

Up to 6 Spindle
Configurations

Automated Panel Prep
(Flash Routing, Edge
Beveling, Corner Champer,
Panel Thickness Measuring)

pluritec

www.pluritec.com | info@pluritec.com

In the wider electronics industry, thermal management has become increasingly important due to rapid growth in markets such as LED lighting and renewable energy. However, the issue remains a secondary concern to many engineers; a recent industry survey[1] showed that thermal management is often considered relatively late in the design cycle, and minimal thermal testing is carried out simply to confirm that the system will not overheat. If a problem is discovered at such a late stage, potentially expensive re-engineering may be required.

In the automotive market, not only is brand image at stake but the extra costs and delays associated with re-engineering to ensure component reliability can be unacceptable. Design for reliability, encompassing proper thermal management for electronic components, is extremely important from the earliest stages of product development.

On the other hand, over-engineered thermal management can be excessively expensive, not to mention heavy and bulky. None of these are acceptable in the automotive business, so engineers need to know how to specify thermal management accurately, using optimal materials and assembly techniques to limit component temperature rise while also meeting tight constraints on size, weight and cost.

Getting to Grips with Thermal Management

Surface-mount power semiconductor packages such as quad flat no-lead (QFN) or land grid array (LGA) are typically designed to maximize extraction of heat from the die to the package underside. Here, a large, exposed, isolated metal heat spreader (Figure 1), or enlarged electrical connections such as metallic drain or source terminations of a power transistor (IGBT or MOSFET), is presented, to be soldered directly to a circuit board or substrate. If space is tight in relation to power demand, such as in a high-power or highly miniaturized traction inverter, devices such as IGBTs or MOSFETs may be sourced as bare dies with metallized terminations and soldered directly on the substrate. Managing the heat from this point is critically dependent on the substrate properties.

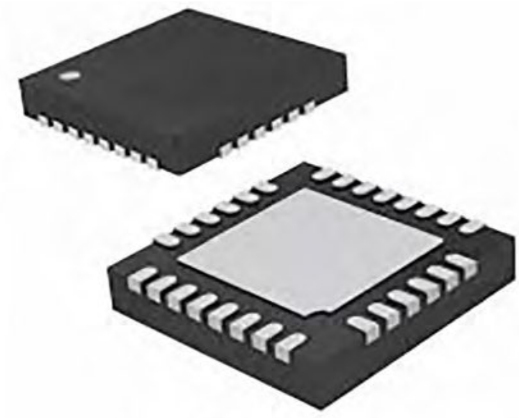


Figure 1: Power packages typically conduct heat to the underside, to be extracted into the circuit substrate.

Designers have a number of techniques at their disposal, utilizing various combinations of metals and engineered thermal materials to achieve the desired thermal performance and mechanical properties such as size, weight and strength, for the right overall cost.

To do this effectively, it is important to understand how to visualize the thermal behavior of the assembly. This can be modelled in a way analogous to an electrical circuit, as a collection of thermal resistances connected in series representing each of the substrate's constituent parts. Figure 2 shows how the stack of materials that thermally couple the transistor die to the substrate are viewed as series-connected thermal resistances.

Note also that components have an associated thermal capacitance, which is defined as the amount of heat energy absorbed or released by unit volume of a material per unit temperature change. This can have an important bearing on the dynamic thermal performance of an assembly. In the context of a heatsink, or heat spreader, it can govern the rate of temperature rise when power components are turned on, the rate of temperature fall when turned off, and the time to reach steady-state temperature in continuous operation.

Referring to Figure 2, the thermal resistance of any of the elements shown is a function of the thickness and surface area of the compo-

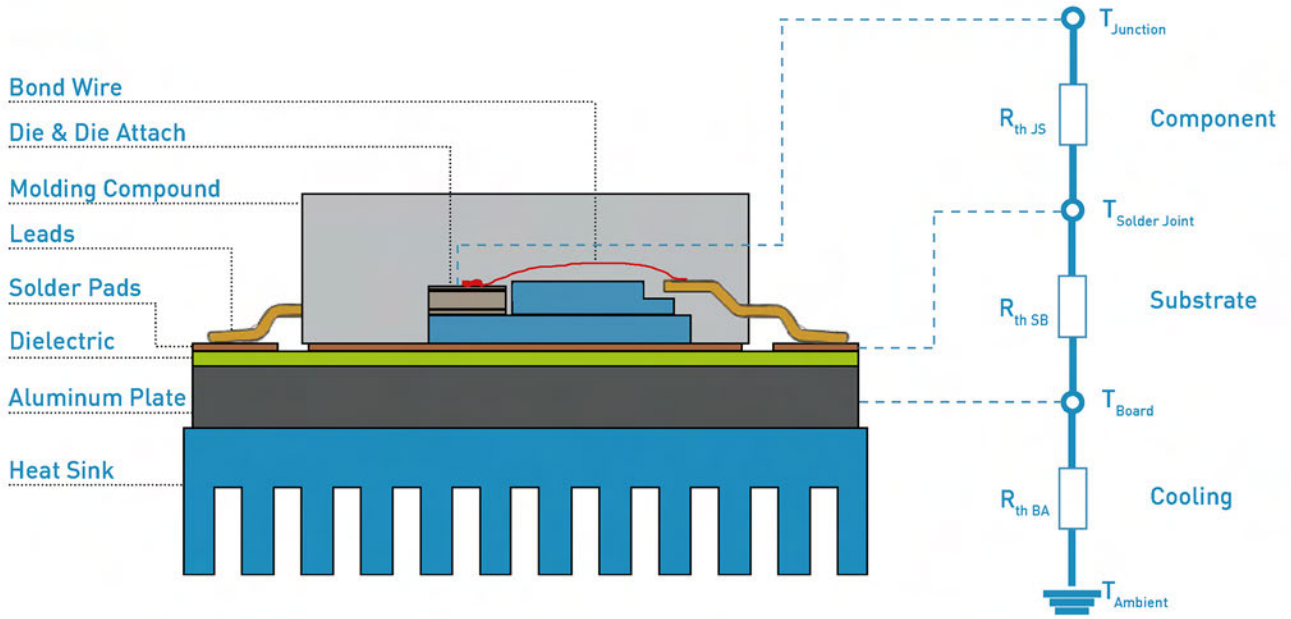


Figure 2: A simple model containing thermal resistances to aid steady-state analysis.

nent, and the thermal conductivity of the material. It can be expressed as:

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

Where: l = thickness, A = surface area, and λ = thermal conductivity of the material.

Clearly, minimizing the thickness and maximizing the surface area can both contribute toward lowering the thermal resistance, in addition to choosing a material with the highest practicable thermal conductivity. Designers can trade off each of these properties against the others, to achieve the desired performance, bearing in mind that material selection can be governed by factors such as cost, electrical properties like isolation, mechanical properties like weight or corrosion resistance, or compliance issues like RoHS or REACH legislation.

It can also be appreciated that parts with the highest thermal resistance have the greatest impact on the overall performance of the stack. Improving these can significantly enhance the effectiveness of the assembly in removing heat from the die.

Enhancing Substrate Performance

The industry-standard FR-4 fiber-glass/resin laminate commonly used as the main substrate of an ordinary printed circuit board has relatively poor thermal conductivity compared to metals or ceramic-filled thermal materials. To enhance the thermal performance of a substrate directly beneath a power component, through-holes, or vias, may be drilled in the FR-4 and metal plated to improve heat transfer into a heatsink or baseplate. Alternatively, the FR-4 may be milled beneath the component, and a metallic "coin" inserted in the recess. This effectively reduces the thermal resistance contributed by the low-conductivity FR-4, by reducing its thickness, and replaces the low-conductivity material with a metal of much higher conductivity.

Although these approaches are effective in reducing the total thermal resistance of the substrate, disadvantages include the fact that the extra processes carried out to create the vias or insert a metal coin are time-consuming and so add to the cost of the assembly.

An alternative is the insulated metal substrate (IMS). These were originally developed in the mid-1960s, and comprise an upper foil layer in which the electrical circuit is formed (usual-

Metal Base Material	Thermal Conductivity W/mK	Thermal Expansion ppm/°K	Density g/cc	Notes
Aluminum 1100	218	23.5	2.7	Pure Al: Good thermal conductivity, worst for CNC machining. Lower cost.
Aluminum 5052	138	25	2.7	Al-Mg-Cr alloy: Best for bending and mechanical forming, punchable. Medium cost. "Most popular choice"
Aluminum 6061	167	25	2.7	Al-Mg-Si-Cu alloy: Best for CNC machining and V-cut scoring. Higher cost.
Copper	386	17	8.9	Pure Cu: Low CTE, high thermal conductivity. High cost.

Table 1: Properties of widely used aluminum alloys and copper, as IMS baseplate material.

ly by etching), a metal baseplate that enhances the thermal performance of the substrate, and a dielectric layer that insulates the circuit foil electrically from the baseplate.

The IMS baseplate is typically either copper or aluminum. Selection can depend on factors such as the required thermal capacity and conductivity, which can affect size, weight, and cost. Copper has higher thermal conductivity and a lower coefficient of thermal expansion (CTE) than widely used aluminum alloys (Table 1), but is also heavier and more expensive. An aluminum baseplate may be deemed more suitable for a cost- or weight-conscious automotive use case, although, if a thin copper baseplate is acceptable then weight and cost can both be lower.

New techniques that are important in the automotive and LED-lighting industries, for purposes such as saving space, complying with existing standard form factors, or saving assembly costs, include forming IMS assemblies by machining the metal substrate to allow bending into various shapes. Figure 3 shows some of the novel forms that have been produced to solve complex design challenges without requiring multiple boards and the associated fixings and connectors. Compatibility with machining pro-

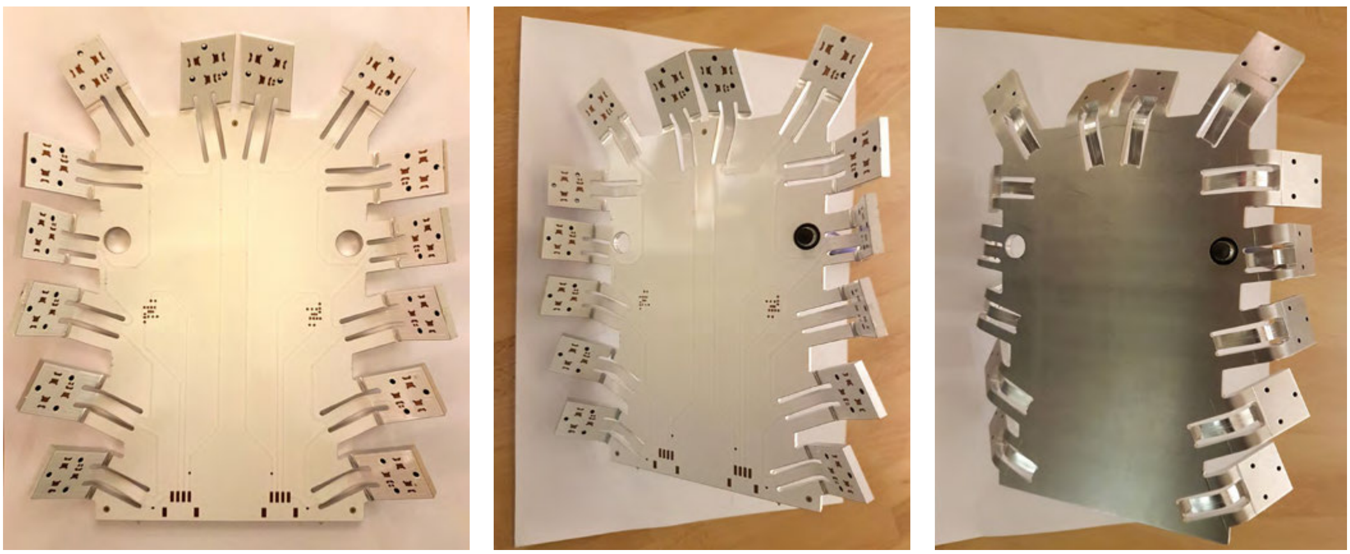
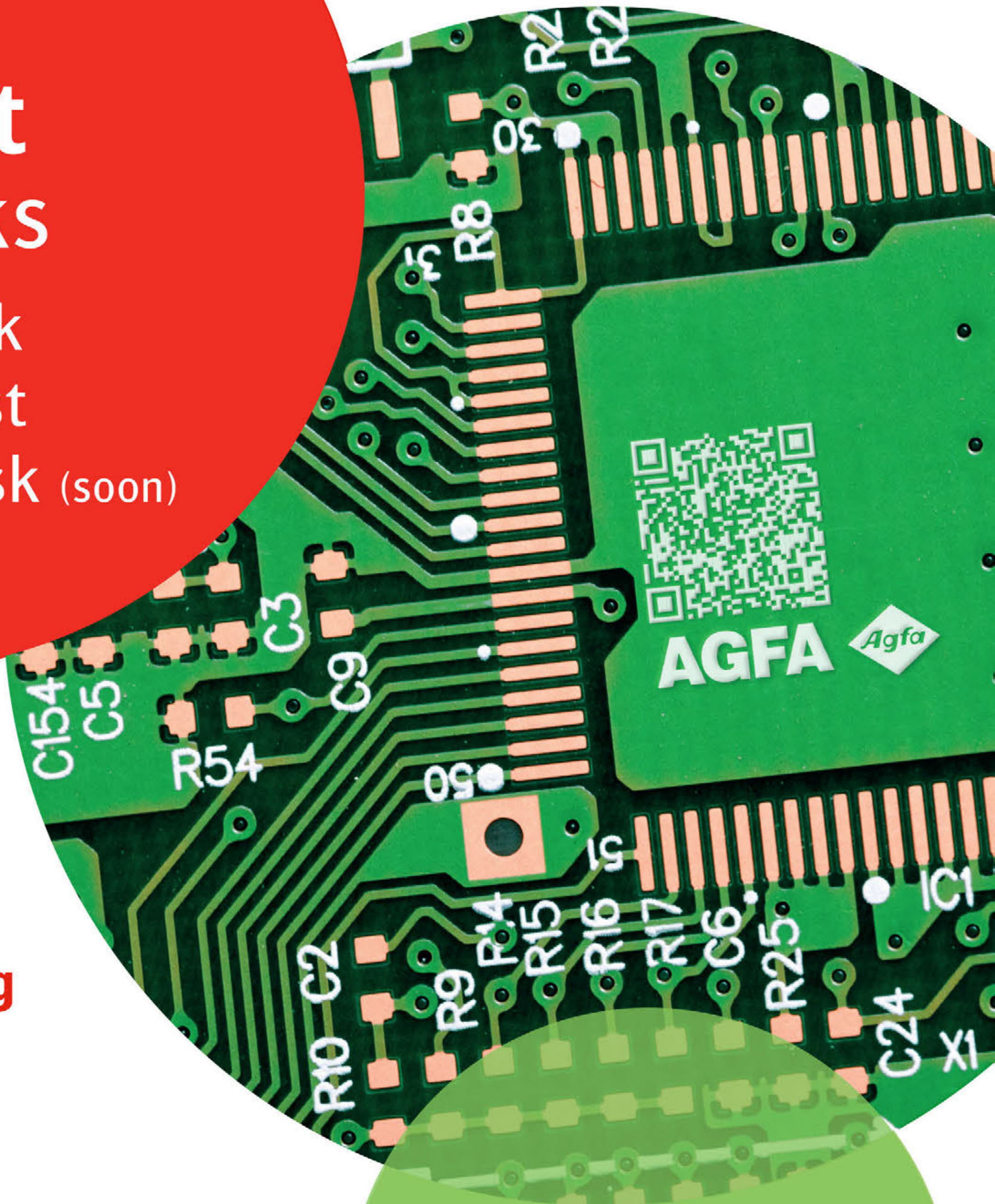


Figure 3: Creative design with IMS allows forming in three dimensions to streamline product assembly while maximizing thermal performance of the circuit board.

DiPaMat
Inkjet Inks

- Legend Ink
- Etch Resist
- Soldermask (soon)



- **Variable data printing**
- **Fast**
- **Ecological**
- **Efficient**

BE FLEXIBLE. GO DIGITAL.

AGFA, your partner for imaging solutions

Traditional Phototooling and UV Inkjet Inks

industrialimaging@agfa.com

AGFA

cesses can be another factor to consider when selecting an IMS baseplate material. As Table 1 shows, 5052 and 6061 aluminum alloys are highly suited to bending or CNC machining.

Advances in Dielectric Materials

The dielectric material may be a woven-glass-reinforced resin composite (prepreg) as in a conventional laminate construction, or a layer of unreinforced resin. Because the nature of the glass reinforcement tends to limit the thermal conductivity, non-reinforced dielectrics have the lowest thermal resistance. However, these demand critical control in manufacture to maintain consistency of dielectric thickness, whereas glass fabric provides a natural mechanical spacer. Extra care is also needed, during manufacture, to ensure uniform distribution of the thermal filler particles and prevent inclusion of foreign matter that may compromise the dielectric strength.

Compared to a conventional FR-4 laminate, thermal resins are loaded with ceramic filler particles to a proportion as high as 70%, which significantly increases thermal conductivity. At the same time, the resin is responsible for the adhesion of the circuit foil to the metal baseplate and must withstand potentially severe thermal cycling. Even when loaded with ceramic particles, the dielectric has lower thermal conductivity than the metal baseplate and circuit foil layer; hence, improvements in the dielectric materials can have a significant impact on IMS thermal performance.

The latest high-thermal-conductivity dielectric materials now provide several options for designers to stack multiple circuit foils, each separated by a thin, thermally conductive dielectric layer, and bonded to a copper or aluminum metal base. This contrasts with historical constraints when designing with IMS designs, which has typically been compatible only with single-layer circuit designs, and hence allows space-saving designs or more complex circuitry within standard size enclosures.

With these new materials, designers of automotive systems can take advantage of other important new options such as hybrid substrates featuring selective localized thermal enhancement. As reliance on autonomous-driving modes increases, moving towards fully self-driving vehicles in the future, we anticipate demand for logic and processing circuitry to be mounted on the same board as high-power converters or inverters in a single consolidated assembly. This could help overcome tight size and cost constraints, and is now achievable using today's highest performing dielectric materials such as Ventec's VT-5A2 as a thin core or as a glass-reinforced prepreg substrate. State-of-the-art manufacturing processes allow this material to encapsulate up to three-ounce copper foils, giving high power-handling capability.

As a guide to the performance that can be expected from today's most advanced thermal dielectrics, thermal conductivity of 2.2W/m.K is eight times greater than ordinary FR-4, and maximum operating resilience to withstand electronic surface-mount assembly processes, including glass-transition and decomposition temperatures (Tg, Td) up to 190°C and 365°C, and extended time to delamination at peak lead-free reflow temperature (T300 > 15 minutes), as measured using standard IPC test methods.

Conclusion

Proper thermal management is critically important in emerging power electronics applications in the automotive, LED-lighting and renewable-energy markets. Effective removal of heat is vital to prevent excessive operating temperatures and so maintain optimum reliability. Considering thermal management early in the development cycle is recommended, to meet cost and time targets as well as reliability. Today's most advanced insulated metal substrates, featuring the latest thermal dielectric materials, give designers more choices in the quest to maximize performance, save space, and simplify design and assembly. **PCB**

References

1. Eurotech: Institute of Circuit Technology Northern Seminar 2016, Harrogate

Mark Goodwin is COO of Ventec.

Excellon

Manufacturers of precision Micro-Machining Solutions for over 50 Years

COBRA II

Hybrid laser system

689

6-station precision drilling and routing system

HS-1L DS

Single-station, dual spindle high-speed system

136L

High accuracy and speed

154L

Large panel applications

HS-2L

2-station, high-speed

129

Single-station precision system

Move into the future with the latest innovations in Fabrication Technology from the Industry Leader!

www.excellon.com

Thermal Management:

A PCB Manufacturer's Perspective on Insulated Metal PCBs

by Anaya Vardya and Dave Lackey
AMERICAN STANDARD CIRCUITS

A designer may choose from among many options to help dissipate the heat generated by various PCB components. This article primarily focuses on options that utilize metal to attach directly to the PCB during the manufacturing process to help with the heat dissipation process. Note that some people refer to these types of PCBs as thermal clad or metal clad PCBs (MCPCBs) while we call them IMPCBs.

When metal is attached to the PCB, the bonding material can either be thermally conductive but electrically isolative (insulated metal PCBs or metal core PCBs) or, in the case of RF/microwave circuits, the bonding material may be both electrically and thermally conductive. RF designers usually have the bonding material thermally and electrically conductive because they are using this not only as a heatsink but also as part of the ground layer. The design considerations are quite different for these different applications.

This article will focus on the IMPCB design considerations and things you should be discussing with your PCB supplier to ensure that you get a good quality PCB. It is not possible to go in great detail so we always recommend collaborating with your PCB supplier about your specific design and how to end up with the most cost-effective solutions.

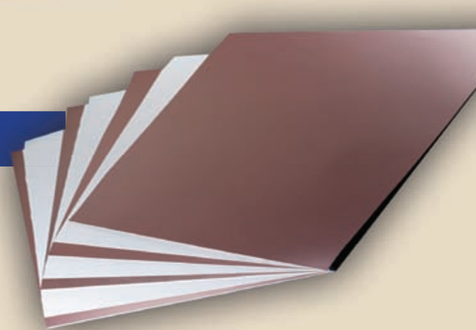
Some of the applications of IMPCBs are:

- **Power Conversion:** An IMPCB offers a variety of thermal performances, is compatible with mechanical fasteners, and is highly reliable
- **LEDs:** Using IMPCBs assures the lowest possible operating temperatures for maximum brightness, color and life
- **Motor Drives:** Dielectric choices for IMPCBs provide the electrical isolation needed to meet operating parameters and safety agency test requirements
- **Solid State Relays:** An IMPCB offers a very thermally efficient and mechanically robust substrate

Don't Let Your Temperatures Rise

Let ROGERS' Thermal Management Solutions *Keep You Cool*

COOLSPAN® TECA Film • 92ML™ StaCool™ Laminates • ML Series™ Laminates & Prepregs



Heat can be damaging, especially when it is not managed. That's why Rogers Corporation invested so much time and energy into creating an array of material-based thermal management solutions to keep heat rise to a minimum in printed circuits. From automotive circuits to LED modules to power supplies, ML Series laminates and prepregs effectively conduct heat away from the source, while COOLSPAN thermally & electrically conductive adhesive (TECA) materials enhance the thermal management of new and existing designs. And for that extra cooling edge, 92ML StaCool laminates feature a thermally conductive metal bottom plate to enhance the heat dissipation.

Product	Thickness	Thermal Conductivity (Z-Axis), W/mK	Thermal Impedance, C-cm2/W	Tg, C	CTE (Z-Axis), ppm/C		Dk, 1MHz	Df, 1MHz	Breakdown Voltage, kVAC	Flammability
					<Tg	>Tg				
92ML	8mils	2.0	0.52	160	22	175	5.2	0.013	>50	HF V-0

Don't let your temperatures rise. Use Rogers' thermal management solutions.

ROGERS CORPORATION
Advanced Connectivity Solutions

www.rogerscorp.com/acs

USA - AZ, tel. +1 480-961-1382
EUROPE - BELGIUM, tel. +32 9 235 3611

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

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 1). Typically, a PCB supplier can buy the copper foil laminated to the base metal from a number of different laminate manufacturers. Some of the key design factors to consider for single-sided IMPCBs are listed here.

Copper thickness

Typical thicknesses range from 1 oz. to 6 oz. The most commonly used ones are 1 and 2 oz. The thicker the copper, the more expensive the PCB.

Thermally conductive prepreg

This is one of the most important elements of the 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 rapid transfer of heat between the two. It ensures that heat generated by the components is dispersed to the base metal (heatsink) 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 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 (watts per meter Kelvin or W/mK) and thermal impedance (Kelvin, meter squared per watt or Km^2/W). The higher the thermal conductivity the better the heat transfer; the lower the thermal impedance the better the heat transfer.

It is important to understand that the better the heat transfer associated with the prepreg, the greater the cost. It is therefore critical not to over-design. To put this in perspective, the thermal conductivity of FR-4 is approximately 0.4 W/mK, whereas the thermally conductive prepregs that are available on the market today range from 1 W/mK to 7W/mK. Apart from thermal conductivity, the thickness of the dielectric can be critical. Typically, the thickness of the dielectric ranges between two and six mils.

Base metal

Aluminum is the most commonly used base metal. The two most common types are 5052H32 and 6061T6. The former is typically less expensive and a lot more available than 6061T6. The thickness of the aluminum typically ranges between 40 and 120 mils, with 40 mils and 60 mils the most common thicknesses available. There are also cases where copper is used as a base metal. This is a significantly costlier solution. A brief comparison of the various base metals is illustrated in Table 1.

Panel utilization

The IMPCB laminate materials are significantly more expensive than FR-4 materials. It is therefore extremely important to understand how your board/array designs utilize the production panel. You should work with your PCB supplier to help you with this. The most popular size for a working panel on these materials tends to be 18" x 24".

Solder mask

Single-sided IMPCB designs are used for LED lights. Many of these applications require white

Figure 1: Single-sided IMPCB.

Metal Base Material	Thermal Conductivity (W/mK)	Thermal Expansion (ppm / K)	Comments
Aluminum 5052 H32	138	25	Al-Mg-Cr alloy: Best for bending, 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 1: Properties of various base metals.

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

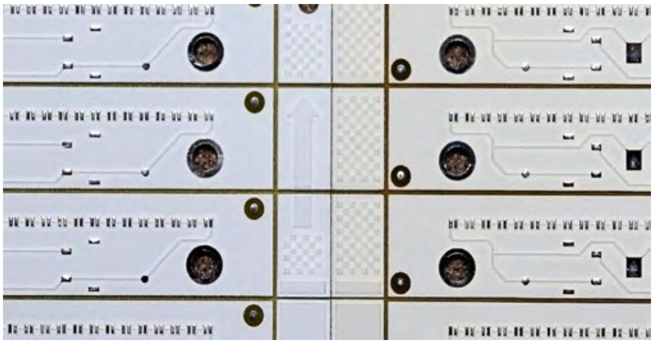


Figure 3: Solder mask "browning" with multiple reflow cycles.

solder mask, which is important to address. 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 a number of different solder masks that are marketed as LED solder masks. 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). Also, the colors look different with one coat versus two coats of solder mask, so this is another decision that will need to be made. In addition, 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. Boards with lead-free HASL tend to become yellower the more heat they are subjected to. It is generally best to permit only one pass through the lead-free HASL process (therefore, no re-work at this station).

Figure 3 illustrates the same solder mask after lead-free HASL versus a board that has been through two assembly reflow cycles. Boards with ENIG after the solder mask process may turn slightly pink with subsequent reflow. This is caused by the formation of a complex between gold residues in the final rinse of the ENIG process with the titanium pigment in the solder mask which can shade the solder mask pink during the high temperature assembly process. It is thus important for the PCB suppli-

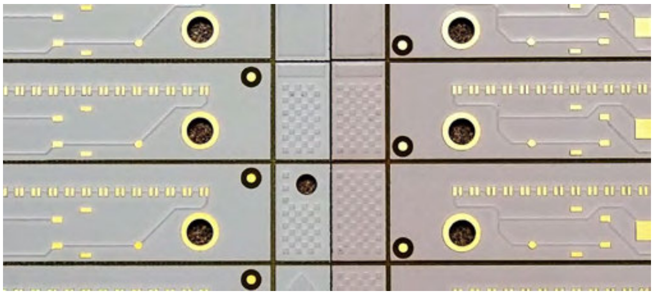


Figure 4: "Pinking" solder mask on ENIG board with multiple reflow cycles.

er to manage the rinses on the ENIG bath very carefully (Figure 4).

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 the material utilization. Make sure that your 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.

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 5). The bonding process is done in the same multi-layer press that is used to manufacture a multi-layer PCB.

Many design factors and considerations that were discussed in the single-sided IMPCB section apply here, plus there are some additional considerations to think about as listed here:

Copper weights on all the layers

The thicker the copper, the more expensive it is. Also remember that the outer two layers will receive additional copper since the vias will need to be plated. Lines and spaces should follow the design guidelines of the PCB shop 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 a PCB supplier or a laminate supplier on constructions that make sense. The prepregs tend to be low flow; it is impor-

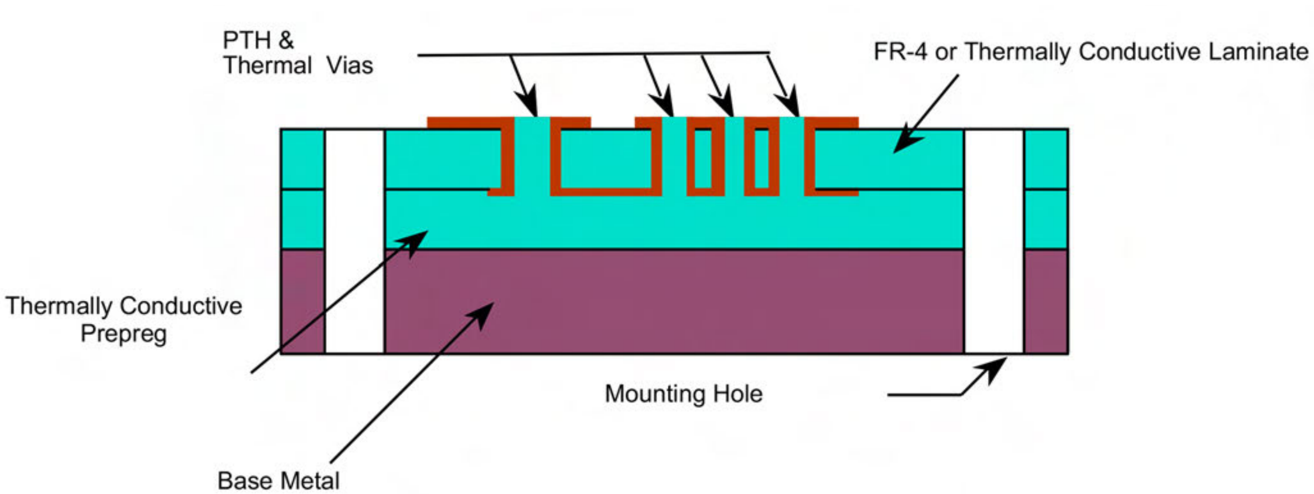


Figure 5: Schematic of a double-sided IMPCB.


Innovation:
it's what leaders do.
For over a decade, the world's top flex PCB processors have relied on ESI's advanced laser processing solutions to drive innovation and stay ahead.
Can you afford to make the wrong decision? Future-proof your investment in flex PCB with ESI's family of laser processing solutions, and maintain your innovation advantage.
Align yourself with the industry leader and let ESI take care of your flex PCB drilling needs.
esi®
Designed for Brilliance.
Engineered for Production.
BE SURE TO VISIT US
INTERNATIONAL PRINTED CIRCUIT & APEX SOUTH CHINA FAIR
HKPCA IPC
2017.12.6 - 8, BOOTH 1Q01 (Hall 1)
ESI's family of FPC laser processing solutions.
5335™ GemStone™
Flex5335™ LodeStone™
5335xi™ RedStone™
For more information, visit us at www.esi.com

tant to work with a PCB supplier that fully understands the lamination press cycles to be used with these materials.

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, a number of different factors need to be accounted for in the process of bonding the PCB to base metal:

- Ensure there is no 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 prepreg through the plated through-holes to the top side; have a method to remove any flow that ended up on the top surface of the PCB
- There are number of mismatched CTEs in this package. It is important to try to balance the copper in the construction as much as possible from a PCB perspective and have a press cycle that helps to minimize warpage

Base metal

Aluminum is the most common metal in use, however there are many applications that will also use copper as the base metal. In general, when aluminum is the preferred metal, the 6061T6 alloy is the best choice for this type of construction.

Metal Core Boards

Conceptually, a metal core board is exactly what it sounds like: the metal is in the middle of the PCB, sandwiched between layers on both sides. Metal core PCBs usually have blind via layers located on both sides of the metal core substrate. There are also plated through-holes (PTH) 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 has to start by drilling the metal core approximately 40–50 mils larger than the plated through-holes, slots or cutouts. These then need to be filled with a non-conductive epoxy filler and then pressed. After pressing the metal core will need to have the filler compound removed from the surface and then prepared for lamination with the innerlayer cores. After lamination, the PTH is drilled and processed in the normal manufacturing process.

While thermal management is a factor in these PCBs, another reason that metal core boards are used is to help with vibration reduction so that components don't fall off the PCBs in high-vibration applications.

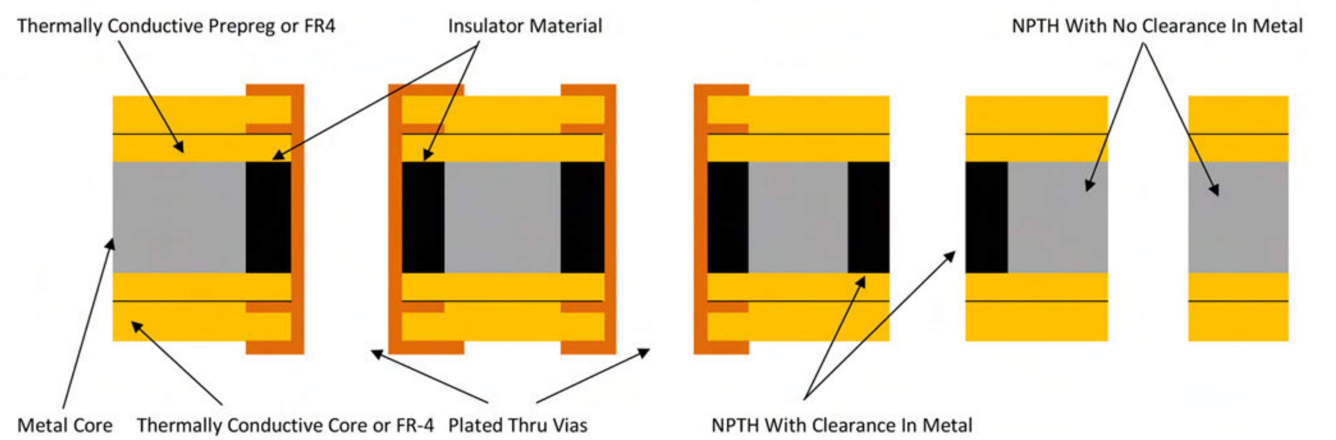


Figure 6: Schematic of a multilayer metal core board.

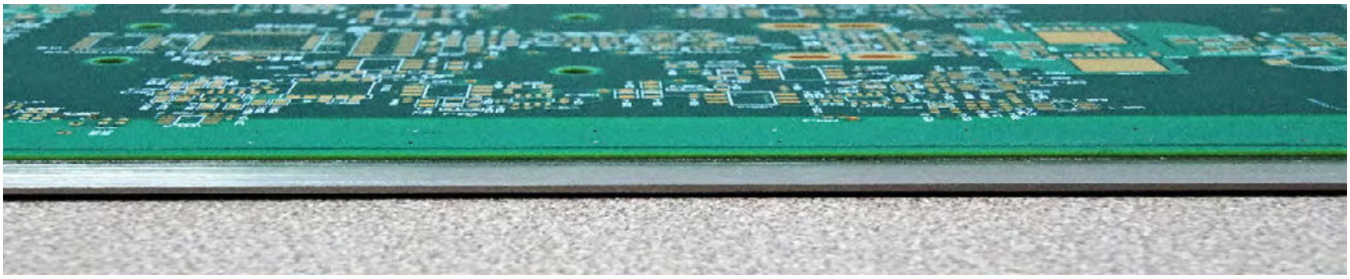


Figure 7: Milling on a metal core PCB.

Design factors to be considered:

Core materials/prepregs

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

Metal core materials

Typically, we see either copper C110 or aluminum 6061T6 used. In the case of metal core boards, we see almost as many designs with copper as we do with aluminum.

Drilling the metal core

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

Insulator/filler

The insulator/filler material acts to insulate the PTH from the metal core so the entire PCB does not short out. The filler is initially in powder form and then applied to the surface and holes and put in a multilayer lamination press. This is a critical process; there can be no voids in the filler or when the PTHs are drilled, as chemistry can leach back to the metal core and cause a short. The core is then sanded to remove the excess filler on the surface. The filler material is a ceramic, epoxy combination.

Stack-up

The stack-up should be symmetrical in terms of 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 just as one would want on any multilayer PCB. Lack of symmetry can lead to excessive warpage issues. In general, the typical IPC warpage specifications do not apply to these types of PCBs.

Milling

Most of the metal core boards have some kind of milling associated with the PCB that results in exposing the metal core layer (Figure 7).

Surface finish on exposed metal core

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

Selecting an IMPCB Laminate Supplier

There are many suppliers, with a majority based in China. In selecting a supplier, several factors should be considered:

- It is important to realize that when one looks at data sheets there are many different ways that laminate suppliers test these materials for thermal conductivity and there are no IPC standards for this. You really need to understand the test methods utilized since all materials advertised as 2W/mK may not result in similar performance

- Some suppliers 100% hipot test their materials, some will test them if you request it at an extra cost, and many don't have the ability to test their material
- Many suppliers will supply single-sided IMPCB materials. If you are thinking of utilizing multilayer constructions, only a handful of suppliers will supply you cores with high thermally conductive prepreg and thermally conductive prepreg that can be used to bond the multilayer PCBs to the metal
- Supply chain ease is important. Lead times vary between the different suppliers. Do you want to source in the U.S. and China or just China? If you are interested in doing both regions, choose a supplier that has a support structure in both regions of the world
- Another important factor could be R&D that is being done by the laminate supplier. As an example, a couple of laminate suppliers have developed special laminate materials where the aluminum can be bent and formed without compromising the copper circuitry or the dielectric layer

Selecting a PCB Supplier

It is important to partner with an appropriate PCB supplier for your IMPCB needs. Some criteria to consider are:

- Level of experience with manufacturing IMPCB materials
- A supplier that is educated and has manufactured a variety of different types of IMPCBs and materials from different suppliers
- A supplier that has a good relationship with the material suppliers in the space
- A supplier that 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 the supplier has the requisite UL paperwork
- Finally, consider a supplier that has the process controls and disciplines in place

Conclusion

We have focused this article around the designs we see most commonly. We have dealt in a number of situations where people's requirements for a variety of reasons deviate dramatically from these norms. Many other options are possible, so the key is to work closely with your PCB fabricator.

As we conclude this discussion of thermal management materials and substrates, it is important to remember that everything in technology is a trade-off. When choosing between the different thermal management substrates it is vitally important to define the end-product in terms of the actual conditions it will experience during its lifetime and deciding which options will be best suited for that product. Finally, selecting the right thermal solution is a like a three-legged stool, with the legs being: the end product, the actual design, and the manufacturing process—none of which can be ignored to create reliable thermally sound PCBs.

Acknowledgements

The authors would like to thank Jim Zeman, director of quality, and John Bushie, director of technology, for their work on the white solder mask evaluation. Our discussions with some of our suppliers—Electra, Taiyo, Aismalibar and Vente—helped to shape our thoughts. Finally, we would like to acknowledge all the IMS customers and designers we have worked with over the years who have challenged us in many ways and helped us grow in this sector. **PCB**

Anaya Vardya is president and CEO of American Standard Circuits.

Dave Lackey is vice president of business development at American Standard Circuits.


MEETINGS & COURSES
FEBRUARY 24 – MARCH 1
CONFERENCE & EXHIBITION
FEBRUARY 27 – MARCH 1
IPCAPEXEXPO.ORG
#IPCAPEXEXPO
APEX EXPO IPC 2018
SUCCEED AT THE VELOCITY OF TECHNOLOGY
THE SHOW FLOOR WILL ACCELERATE YOUR INNOVATION
In 2018, your leading opportunity to stay on top of electronics industry innovation is on the IPC APEX EXPO show floor. Join thousands of your industry peers as you uncover new solutions that will move you in the right direction to capitalize on tomorrow's opportunities!
>> Explore and compare equipment from more than 450 of the industry's top suppliers.
>> Discover new processes to gain greater efficiency.
>> Start your year with new suppliers that can save you both time and money.
VISIT IPCAPEXEXPO.ORG FOR MORE INFORMATION
NEW THIS YEAR!
WIN WITH OUR PASSPORT TO PRIZES GAME
Travel around the show floor collecting stickers from exhibitors, and enter for a chance to win an exciting prize!
SAN DIEGO
CONVENTION CENTER
CALIFORNIA > USA

Supply Lines
Highlights

Final Finishes: Taking Gold Thickness into Account

Martin Bunce was one of several people from MacDermid Enthone who presented at SMTA International this year. He sat down with me for a chat about his paper on final finishes, particularly with regard to controlling gold thickness.

Frontline PCB Solutions' InCAM Software Reaches a Record 1,500 Seats

Frontline PCB Solutions announced at the TPCA show today that its InCAM preproduction CAM solution has achieved a record number of 1,500 seats worldwide.

Orbotech Revolutionizes the AOI Room with 4-in-1 AOI Solution

Orbotech's new Ultra Dimension Series reduces the total cost of ownership of the PCB AOI room by integrating leading pattern inspection, laser via inspection, Remote Multi-Image Verification (RMIV) and 2D metrology into a single AOI solution.

Elmatica Expects Copper Foil Shortage to Last Longer

Printed circuit broker Elmatica encourages open dialogue around the copper situation and its consequences. With several governments desiring a shift towards cleaner new-energy vehicles, and an all-electric automotive industry, the copper shortage will last longer than first anticipated.

Trouble in Your Tank: The Critical Importance of Rinsing, Part 2

In Part 1 of this series on the importance of rinsing, the author presented an overview of the critical aspects of rinsing as it applies to the overall quality of a printed circuit board, with considerable space devoted to water conservation. Thus, we now turn to how one can improve rinsing effectiveness without increasing water consumption and, by default, significant waste treatment costs.

IPC Awards IPC-4101 Qualified Product Listing to Ventec International Group

Ventec International Group is delighted to announce an IPC-4101 Qualified Products Listing (QPL). The IPC's validation services program has qualified VT-90H and VT-901 to specification sheet 41 of IPC-4101E, specification for base materials for rigid and multilayer printed boards.

Protecno Becomes the First French PCB Manufacturer to Invest in Orbotech's Nuvogo Direct Imaging Solution

Orbotech Ltd. today announced that Protecno, a France-based electronics manufacturer and part of Groupe GTID, has purchased the first Orbotech Nuvogo direct imaging (DI) solution to be installed in France.

MacDermid's Metallization Fills Microvias, Plates Through-Holes in One Step

MacDermid Enthone Electronics Solutions, a MacDermid Performance Solutions business, has released the MacuSpec VF-TH 200 process, a high-performance DC electrolytic copper metallization process for simultaneous via filling and through-hole plating.

Video from productronica 2017: Electra Polymers Updates on the Latest Developments in DI Soldermask

From the show floor at productronica 2017, Electra Polymers' Sales Manager Ashley Steers discusses the state of the art in soldermask for direct imaging. Sales and Marketing Director Shaun Tibbals comments on developments in ink-jet soldermask and opportunities in wafer-level packaging.

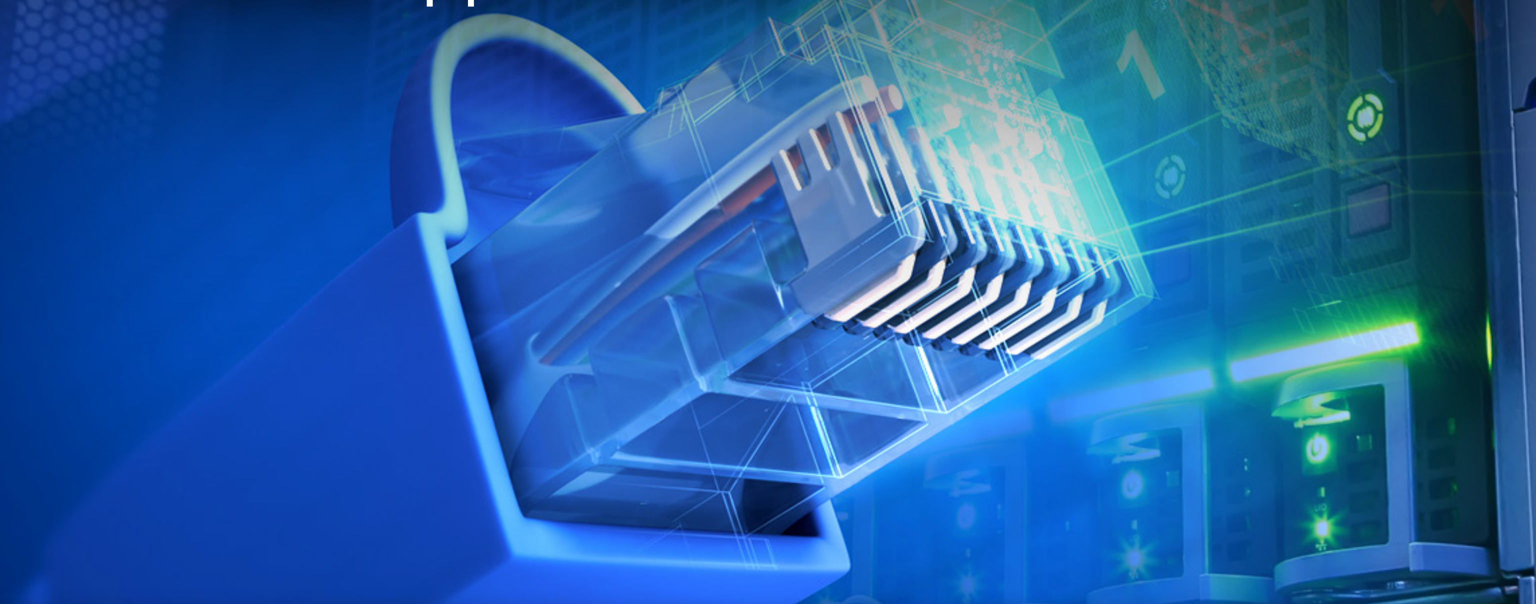
TUC Appoints John Strubbe Vice President of Technology

The position is within the Development and Application Center (DAC) located in Jhubei City, Hsinchu County, Taiwan. Strubbe has been working with TUC since June 2015.

TECHNICA, U.S.A.
Fulfilling Manufacturing Needs Throughout the Electronics Industry

Celebrating 30 YEARS

Halogen-free EM-890/ 890K and Halogenated EM-891/ 891K Ultra High Speed Laminates for 100, 200 and 400Gbps Ethernet Applications



KEY FEATURES

Ideal for multiple lamination cycles (>4x)

Compatible with FR-4 for Economical Hybrid Designs

Ultra Fine BGA pitch (<0.7mm)

Excellent Thermal Properties (10 X LF260 IR Reflow, IST >1K cycles)

Available exclusively through Technica, U.S.A.
1-800-909-8697 • www.technica.com

ELITE MATERIAL CO., LTD

Thermal Management Considerations at the Bare Board Level

by Jim Barry
PCB TECHNOLOGIES LTD (ISRAEL)

Today's PCB/PCBA (CCA) designers, are faced with a much greater task than their predecessors. Our needs today require a great deal of thought that goes well beyond the traditional mindset of layout patterns of circuitry. With continual shrinking of package sizes, as well as the complexity and speed of components becoming ever-more complex, heat dissipation is becoming a major factor. In the past, CAD layout design engineers gave little thought to mitigating thermal events at a component level, which can directly affect not only performance of specific device, but also potentially create a catastrophic failure mechanism down the road. So, the question today that designers face is: How do we remove the heat without adding too much size or weight to the circuit card assembly? Can we find ways to build in some thermal transfer into the bare PCB board whilst balancing size, weight, and performance with cost and capabilities?

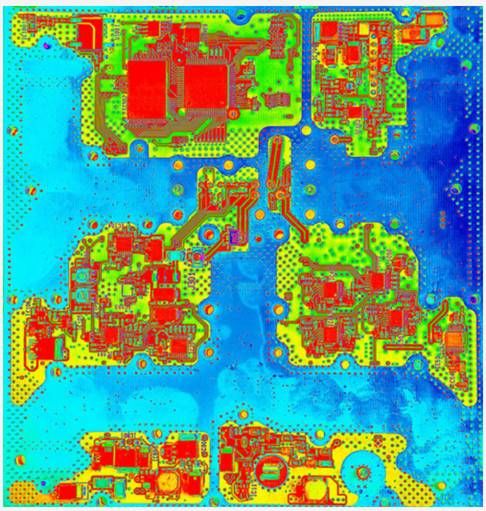
There are many creative ways to remove heat from the PCB, even at the specific component location. Traditionally, heat removal has been done at the component level with finned heat sinks placed over a specific component location. Unfortunately, this creates large, cumbersome designs and/or requires some imaginative design engineering at a box level build. The second most common methodology has been to apply a metal back to the whole PCB which can connect to the chassis and draw away some of the thermal energy. The downside of these two typical techniques is that size and weight of some of these heat removal methods can also pose a challenge, especially when the world of electronic packaging is shrinking, not growing. Smaller packaging and weight are now required for products such as night vision goggles, UAVs and soldier-worn technologies, to name a few.

Technology leader
for high performance thermal management materials

ventec
INTERNATIONAL GROUP
騰輝電子

Ventec VT-5A2 - Designed for the world's most thermally demanding PCB applications

VT-5A2 is designed for the world's most thermally demanding PCB applications such as power converters,hybrid multilayers and high power electronics with heavy copper designs providing enhanced signal stability in harsh environments. With VT-5A2, Ventec has created its highest performance thermally conductive laminate material so far, with a polymer matrix that is fully compatible with other Ventec laminates, epoxy- or polyimide based materials including our IMS tec-thermal® range & tec-speed®. With VT-5A2 total compatibility in the manufacture of hybrid multilayer boards is guaranteed.



VT-5A2 best-in-class performance:

- thermal conductivity of 2.2 W/m.K (8 times that of FR4)
- in plane thermal conductivity of 3.4 W/m.K
- high Tg of 190°C
- best in class thermal performance (T260 >60 Minutes, T288 >30 Minutes and T300 >15 Minutes)
- M.O.T. up to 150°C
- lead-free assembly compatible
- fulfills RoHS and WEEE requirements
- UL94 V0 compliant
- IMS materials compatibility in hybrid constructions
- choice of thin core and prepreg variants

The VT-5A2 thermally conductive laminate is manufactured by Ventec using strict quality controlled processes that are certified to AS9100 Revision C, ISO/TS16949 and ISO 9001:2015, and, like all Ventec products, is backed by a fully controlled and managed global supply chain, sales- and support-network.

Ventec - Your strategic partner for safety-critical supply chains

Ventec International Group
21 Water Street, Amesbury
Massachusetts, MA 01913, US
T: 978-521-9700
E: saleseast@ventec-usa.com
@VentecLaminates

www.venteclaminates.com

In this article we will take a brief look at each method available (Figure 1).

Typical heat mitigation techniques include:

- Materials
- Copper thickness
- Thermal vias
- External heatsink
- Internal heatsink
- Coin technologies

Materials

One of the simplest and most common ways that is often overlooked as a function of removing heat from components, is to carefully choose the right materials. When designing bare PC boards at the CAD level, we tend to think of material choice as more of a dielectric, impedance or signal speed requirement, neglecting the need for heat movement. Often when a PCB is going to see high temperature in its environment or operation, we tend to think polyimide. Polyimide is a great heat-sustaining material that possesses a higher heat transfer rate than that of FR-4; however, there are others that can do more.

There are many material choices today which can gain one both the signal speed and the thermal conductivity needed, as referenced in Figures 2 and 3. As you can see by these charts, thermal conductivity greatly improves when considering alternative materials. The issue here is to balance all the needs of the design together: electrical, environmental and performance.

Copper Thickness

Copper thickness is another common way of controlling and/or moving temperature into the PCB and away from the component for added performance. It is not unusual to see copper thickness for internal plane layers to be anywhere from two ounces upwards of four ounces or even higher. The problems start to occur when a designer wants to run smaller trace widths on the same plane as the heavy copper weights; if this can be avoided and these smaller traces can be placed on other layers, it is always advised.

One example of a simple copper weight improvement relates to a design of a former customer. The product was a high-speed camera for ultra-slow-motion detection (8-layer rigid-flex design). There were many components generating tremendous heat on both sides of the design. As a fix we simply added two additional

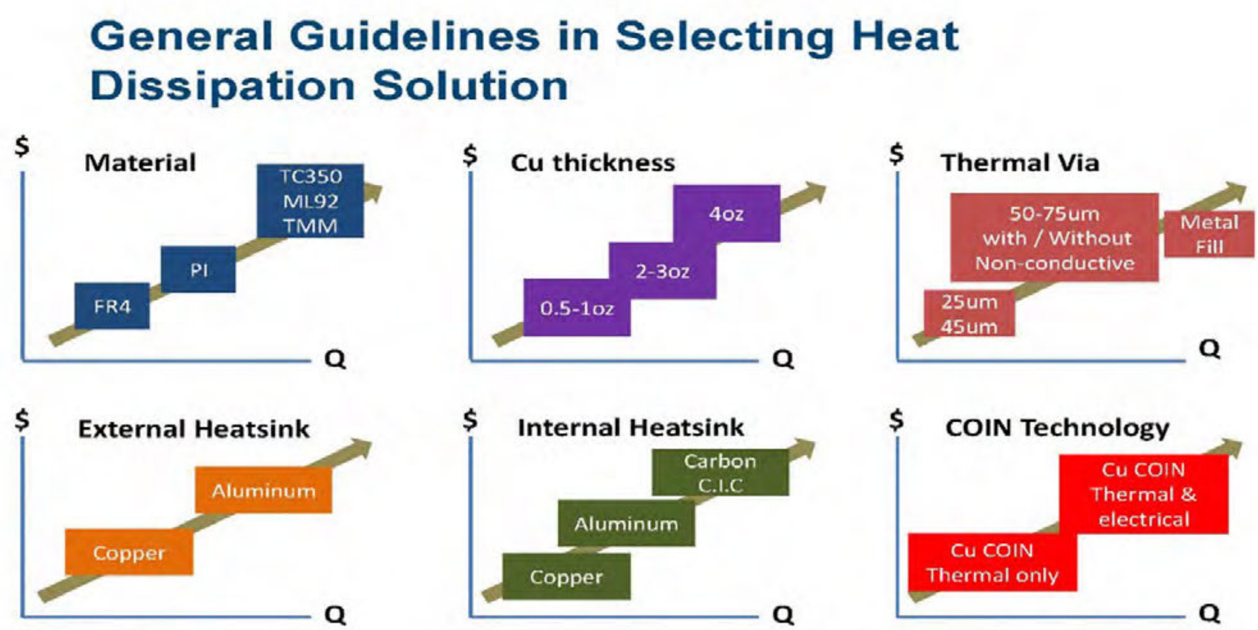


Figure 1: Comparison chart for choosing heat mitigation method.

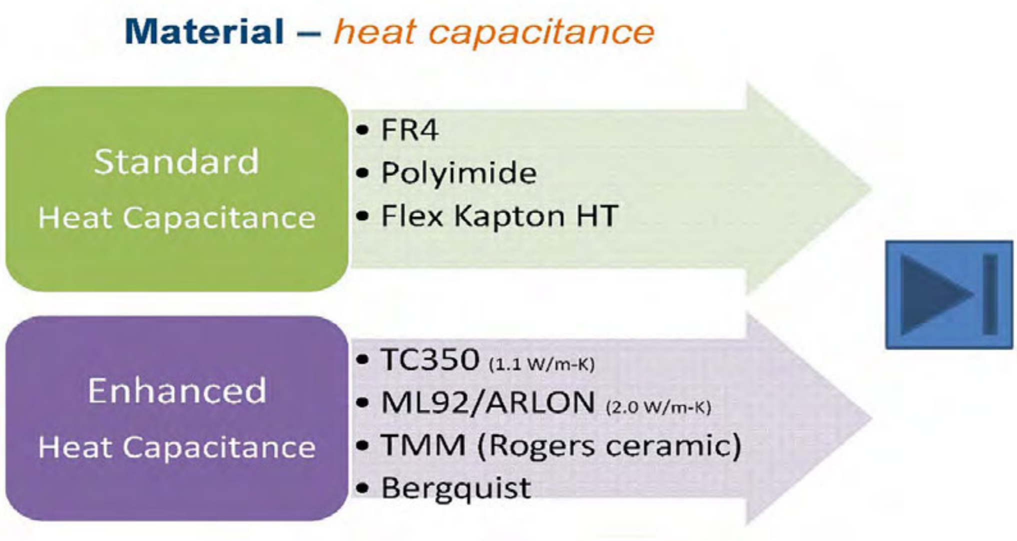


Figure 2: Comparison of materials vs. heat capacitance.

Material – *thermal properties*

Standard | enhanced

	FR4 / PI	TMM / PTFE ceramic	Cu		Multi-crystal Diamond (synthetic)	Single-crystal Diamond (synthetic)
Thermal Conductivity (W/m*K)	0.3-0.4	0.5-0.86	384		900	2200
Specific Thermal Resistivity (m*K/W)	2.5-3.3	1.1-2.0	0.0025		0.001	4.55E-04
Specific Heat Capacity (J/g*K)	0.85 -1.2	0.72-1.0	0.385		0.509	0.509
Density (g/cm^3)	1.3-1.7	2.1-2.8	8.96		3.52	3.52

Figure 3: Details of material comparisons.

2-oz. plane layers under the outer layers, making it a 10-layer design. This simple fix reduced the heat by 10X and gave them a major performance improvement in the optics capabilities.

There are some design considerations. However, when using these thick copper planes, as backfill becomes a possible concern. One must also consider the resin fill of the areas without the copper, as to which materials are used. Too much resin can seriously affect movement in the Z-axis. So once again careful consideration must be given, when using heavy copper inner-layers, to balance the thermal conductivity with fill requirements and trace widths. As an alternative to heavy copper-clad materials one can consider the use of the internal heatsink.

Thermal Vias

Thermal vias can also be a great method to aid in the movement of heat from a specific location(s) or device into the circuit board. The use of thermal vias for heat transfer is a widely used and proven method that does not impact the cost too dramatically. When using the thermal via in conjunction with a heavy copper plane layer or internal heatsink, one can create a thermal Z-Axis highway of sorts. One of the gains of this method is that it can be used in specific problem locations or components versus trying to do it in a mass event. The designer can create a via field under the specified location and simply plate heavier than normal copper, copper fill or plate shut stacked vias to help draw temperature.

Figure 4 shows thermal vias with various plating thickness and/or internal fill, whether conductive or non-conductive. Today, we tend to see about 65% of all designs using stacked or blind via technology. This allows the designer to utilize HDI plated closed stacked vias for density but can also be used to aid in creating a thermal highway directly to the internal heatsink.

External Heatsinks

Probably the most common and widely used method of heat removal is the external heatsink. Simply noted as a metal plate, it is typically made of copper or aluminum placed on the outside of the PCB. These are usually bonded to one side of the PCB using either a b-stage material or some type of thermally conductive bond ply. The heatsink is then either connected to the chassis, box or fanned from some airflow source.

This method is commonly used today as it is one of the simplest, most cost-effective methods. However, you are now drawing the thermal, not from a specific location, but through the entire thickness of the PCB itself and then to the opposing side, dispersing the heat through a mass method. It is not always the most efficient, nor most effective way of accomplishing the thermal dissipation, though it is tried and true.

We have seen this method being used typically in a box level where weight and/or size is not an issue. As stated, the external heatsink can be bonded using a thermally conductive adhesive as well as being somewhat 3-dimen-

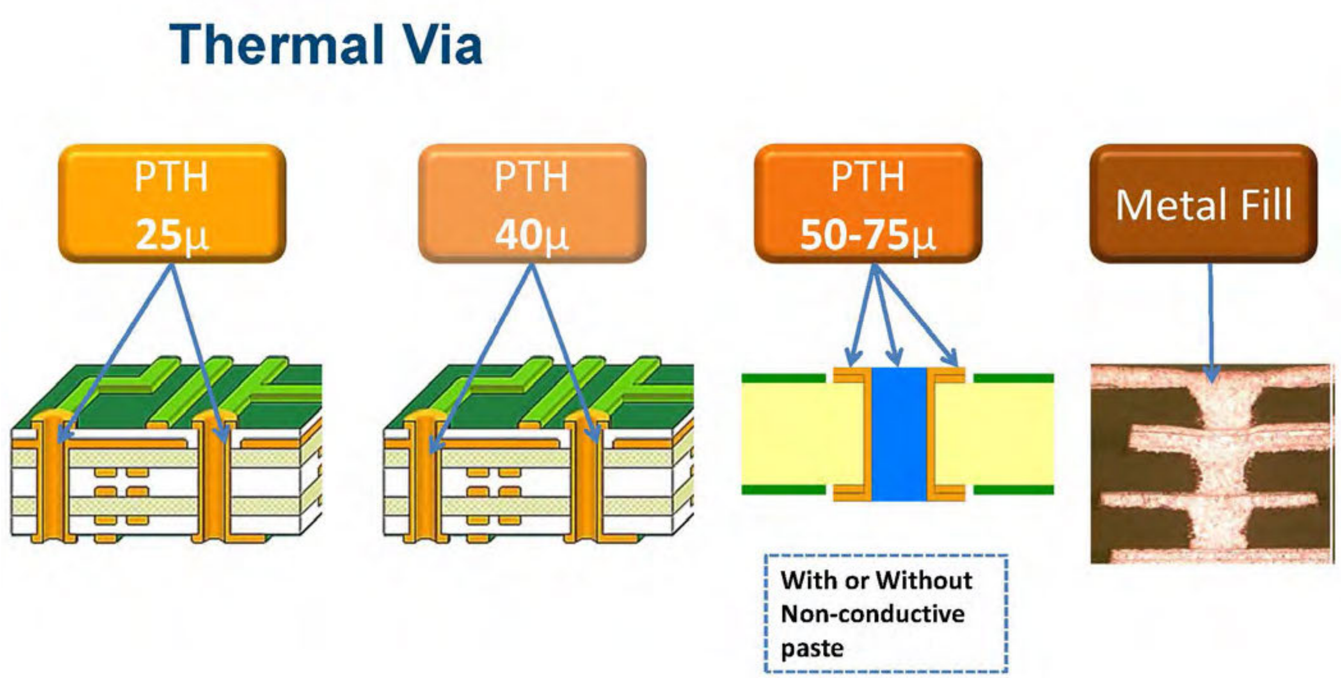


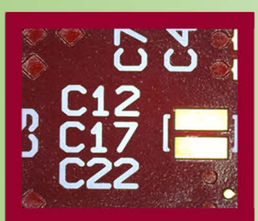
Figure 4: Various via configurations.

MivaTek
Global

MIVA
TECHNOLOGIES

LED Direct Imaging

We give a dam...2.0 mil solder dams



- > 2.0 mil solderdams
- > 4 mil mask defined pads
- > Colors: GRN, WHT, BLK, BLU, RED
- > Excellent camera registration
- > Legend printing using white SM
- > Inner and outerlayer imaging
- > Extensive North American support
- > 20 machines sold in North America

MIVA: LED DIRECT IMAGING

MivaTek Global, LLC formerly *Kuper Technology, LLC*

Now Hiring : *Sales Engineers, Applications Engineers and Technical Support*

MivaTek Global, LLC, PO Box 893, Concordville, PA 19331, Ph: (610) 620-3795

Eastern US: Brendan F. Hogan, M:(610) 620-3795, B.Hogan@MivaTek.Global

Western US: Kurt Weber, M:(714) 767-6367, Kurt@MivaTek.Global

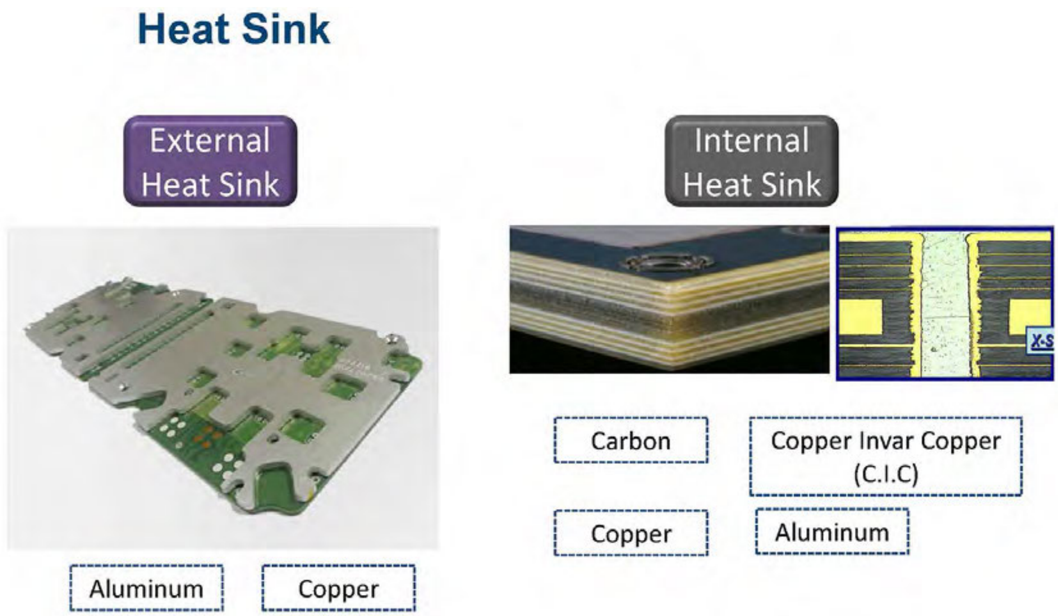


Figure 5: Visual of basic external and internal heatsink methods.

sional, wrapping itself around the finished bare PCB. For some more complex designs we have seen the use of the external heatsink as a center core between two opposing PC boards bonded together like a sandwich.

Internal Heatsinks

Placing the heatsink within the PC board brings the mass of metal closer to the devices, as well as opens both sides for component mounting, but again has its limitations. Yes, it brings the heatsink closer to the device level, but it is still a slow and mass method of trying to dissipate the heat.

Typically, this method has been done through the years by using CIC (copper/Invar/copper), solid copper or aluminum. The issue becomes that now the fabricator must etch and/or machine the inner heatsinks so as to not short out to the metal core. Once the removal of the metal is complete, the heatsink is typically backfilled with a non-conductive resin like that of the buildup material of the PCB.

In recent years there have been many advances in materials such as the use of carbon fiber within the PC board. This too can be used as a thermal highway to draw the heat into the board and out to a source of thermal connection draw such as a chassis. Again, one must consider that having too much resin in a PTH could create its own set of problems. The fabricator must consider using a stable resin system so that Z-axis expansion does not create other problems. Filling and planarization need to be done prior to the lamination to ensure that no air entrapment is evident. This method as with external heatsinks does require some extra processing (Figure 5).

Coin Technology

Coin technology is fast becoming an alternative to the internal heatsink. It is widely used for the component heat management and removal in RF/microwave designs on a regular basis. It is probably the costliest, but it is also the most effective and direct method of pulling heat away from a specified location. Using coin technology allows the designer to place the component directly on top of or within the solid "coin," which allows the thermal draw directly downward and away from the device to the backside, where it can be dissipated (Figure 6). Not all suppliers are willing to offer this technology, but in the RF design world it is used often.

Integrate QA Solution

Because everyone's QA needs are just a little bit different.

Integrate is a fully customized solution, and no two contracts are exactly the same. No matter what size customer you are, we can tailor AOI, testing, final inspection, or a combination of all three services to fit you. And you only pay for the product that actually passes quality assurance, not on panels produced or tested.

Find out if our Integrate solution is right for you!
www.gardien.com/why-integrate

GLOBAL FOOTPRINT
WITH A LOCAL PRESENCE

30+ YEARS
EXPERIENCE

30 YEARS OF PCB
INDUSTRY EXPERIENCE

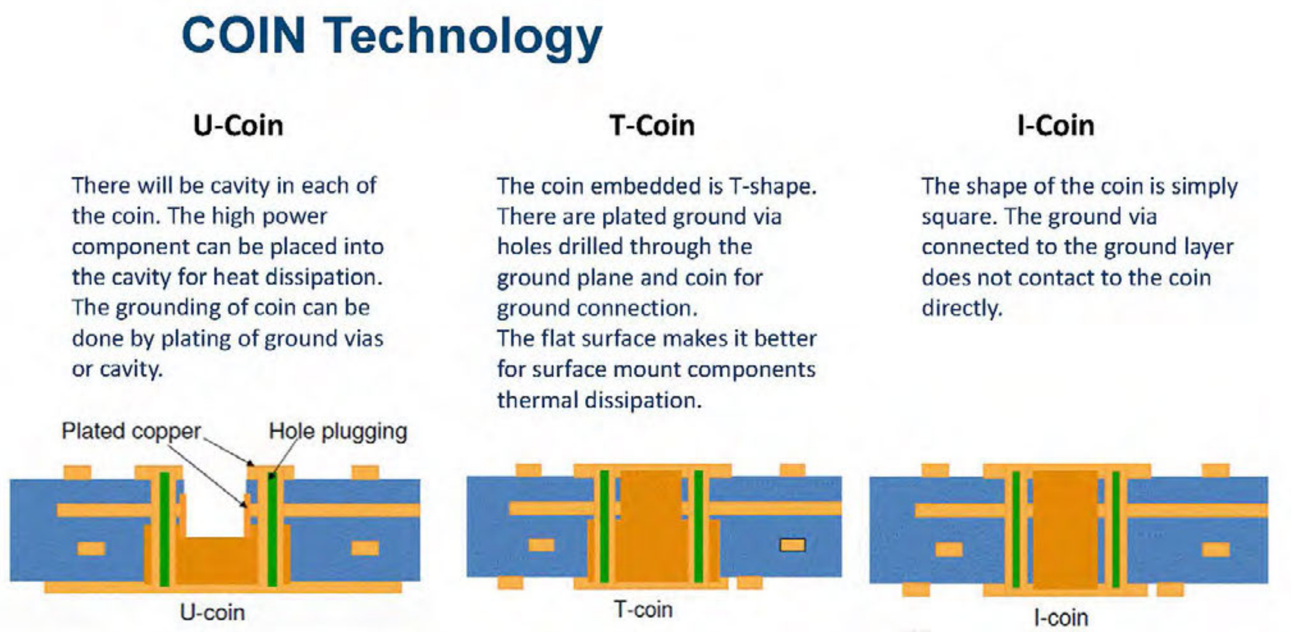


Figure 6: Cross-sectional view of coin technology options.

Conclusion

Thermal cooling at the PCB level is always an option as there are various configurations of finned or bladed heatsinks that set onto a device which then carries the temperature out of a specific location. Open any desktop system and you will still see this used today on the mother board. Again, however, the area needed for the finned heatsink to draw upward and away from the PCB board uses volumes of space (and weight).

In the case of this article we are focusing on what the options are for thermal management considerations at the bare PCB level but there are many options open that can be used to remove heat from a device, a location or from the PCB itself. Designers must consider all the options when choosing one method over the next, or even when combining which methods they are considering.

One important thing to consider when creating a thermally conductive bare board is how this will affect the assembly portion of the build cycle. If a PCB has great thermal absorption, that will affect the soldering process. An example of this might be to have your thermal zone(s) too close to a device that is sensitive to heat which can be damaged during the soldering process due to the potentially increased amount of heat needed to make a solder joint.

There are many options for the PCB designer to choose from. It is not uncommon to see many designs that combine one or more of the heat removal methods discussed above. An example of this would looking at today's RF designs, as they commonly use mixed materials, cavities and coins all within the same structure.

Today the industry is building many types of structures, adding even more elements of enhanced design uniqueness. One of these new technologies, called "air cavity," is something that we see which adds great value for advancing thermal mitigation for the demanding needs of tomorrow's products. Stay tuned as we move forward…

Acknowledgement

The author would like to acknowledge and thank CTO Yaad Eliya and Technical Expert Shlomi Danino, Sr., both at PCB Technologies Ltd, for their valued input to this article. **PCB**

Jim Barry is the solutions architect for PCB Technologies Ltd (Israel) and supports the North American market. He has nearly 40 years of engineering, manufacturing and application experience within the PCB/PCBA industry. To contact Barry, click here.

trac THE RIGHT APPROACH CONSULTING LLC

I guarantee that you will upgrade and simplify your QMS in just 6 months.

In today's increasingly competitive market, Quality Management is more important than ever. I can help you upgrade to ISO 9001: 2015 or AS9100 Rev D in six months - guaranteed!

CLICK HERE TO LEARN MORE ABOUT MY CERTIFICATION GUARANTEE!

TheRightApproachConsulting.com

Electronics Industry News
Market Highlights

Electronic Circuits Reveal When a Plant Begins to Experience Drought Conditions
Forgot to water that plant on your desk again? It may soon be able to send out an SOS. MIT engineers have created sensors that can be printed onto plant leaves and reveal when the plants are experiencing a water shortage.

Berkeley Startup to Train Robots Like Puppets
Robots today must be programmed by writing computer code, but imagine donning a VR headset and virtually guiding a robot through a task, like you would move the arms of a puppet, and then letting the robot take it from there.

Smart Manufacturing Market Size Worth $395.2 Billion by 2025
The growing emphasis on increasing production efficiency and gaining visibility across the entire value chain are the two major factors driving market growth. In addition, the availability of advanced technologies such as 3D printing, Manufacturing Execution Systems (MES), and plant asset management solutions to small and medium enterprises is further accelerating the industry growth.

Top 3 Drivers for IIoT Market in Automotive Industry
Technavio market research analysts forecast the global IIoT market in the automotive industry to grow at a CAGR of more than 33% during the forecast period, according to their latest report.

IPC Pulse Survey: Bullish Outlook for Equipment, PCB Manufacturers
A preliminary survey in IPC's new Pulse of the Electronics Industry global data service reveals a bullish outlook for most segments of the electronics industry, especially for equipment manufacturers and PCB fabricators.

A Robotic Spy Among the Fish
A new miniature robot developed by EPFL researchers can swim with fish, learn how they communicate with each other, and make them change direction or come together. These capabilities have been proven on schools of zebrafish.

Beyond 5G—After the Next Generation
The TERRANOVA team will be working on embedding terahertz wireless solutions into fast fiber optic networks, developing new frequency bands and thereby laying the foundation for a resilient communications infrastructure that is equipped to cope with the demands of the future.

Real-Time Data Drives the Future of Wearables in Healthcare
The proliferation of wearables for health and wellness, and the need for more data about the current and future condition of individuals and patients, are key factors propelling market growth. Future growth opportunities focus on the commercialization and embedding of wearables in skin patches, clothing, and electronic skins.

Telecommunications and Wireless Communication Markets Projected to Grow
According to a research by MarketsandMarkets, the telecom managed services market is expected to grow from $11.90 billion in 2017 to $22.58 billion by 2022, at a CAGR of 13.7%.

New Quantum Materials Offer Novel Route to 3D Electronic Devices
Researchers have shown how the principles of general relativity open the door to novel electronic applications such as a three-dimensional electron lens and electronic invisibility devices.

A New Beginning for PCB Fabrication

The Latest in AUTOMATION

Hakuto 630NP Series

Dry Film Laminator

Lamination Rolls (NO Hot Rolls)

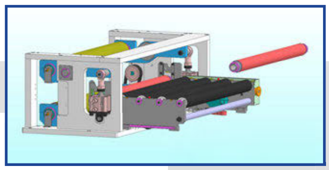
The NP Series Laminators are supplied with 4 standard EPDM (Acid Resistant) rolls which can be "removed and replaced in seconds".

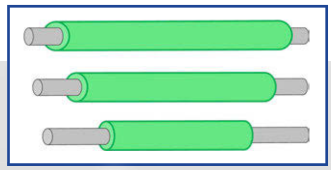
Engineers can now use many combinations of rubber styles for various applications: thickness, durometer and rubber length.

EVEN Pressure and EVEN Heat

(Essential for the transfer of sub 5 mil features)

The new design eliminates traditional roll bending ensuring even heat and pressure across the entire surface of the panel.





EC24 Panel Cleaner

New Design for the PCB Industry

- Electrostatic Touchless Cleaning
- No Tacky roll consumables
- 4 Electrostatic Neutralizer Bars
- 8 High Pressure Air Knife Jets
- Panel Thickness (0.001" to 0.500")
- Panel Widths: 24", 30", custom
- Dust Particle collection

RL-160

Kuttler Automation

New Robotic Technology

Example - **160 Series**

- **RL-160** Loader
- **RU-160** Unloader
- **RLU-160** Loader & Unloader
- **RLE-160** Loader with Exchange
- **RUE-160** Unloader with Exchange
- **RLEP-160** Loader with Exchange & Interleave

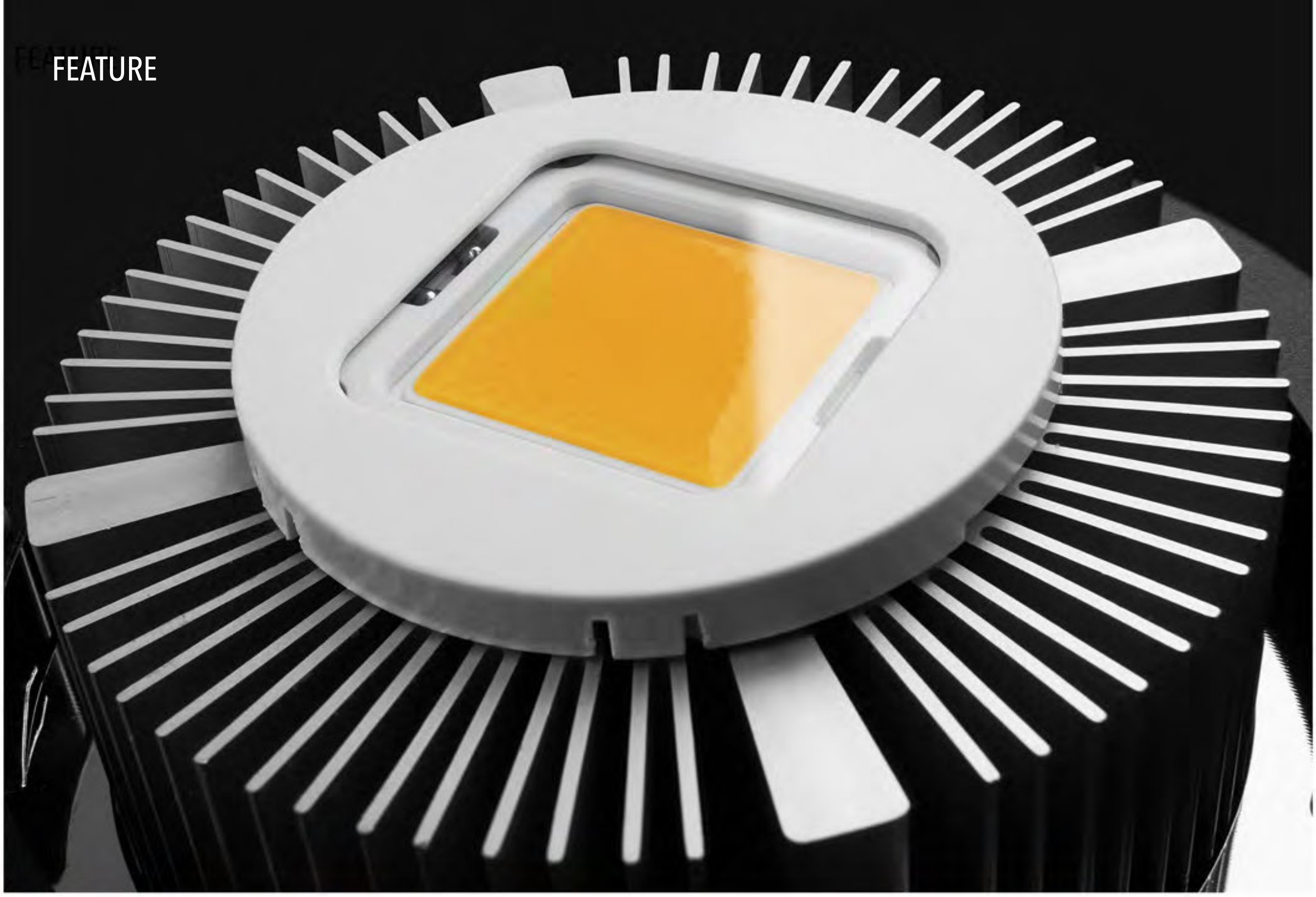
Standard & Custom Options

- Loading
- Unloading
- DI Place/Remove
- Buffering
- Flip, Roate, Divert
- AOI
- Myler Peeling

MATRIX
CIRCUIT BOARD MATERIALS

1-800-668-5447 • www.matrixusa.us

TORONTO • SANTA ANA • SANTA CLARA • CHICAGO



Evolution of Thermal Management in PCBs

by Gareth Parry, P.Eng
AISMALIBAR

Thermally conductive dielectrics used in the PCB substrate have been steadily growing in demand and application, with a surge following the development of LED lighting.

Many thermally conductive dielectrics are similar to the standard FR-4 dielectrics used in the PCB industry: epoxy (or polyimide) reinforced with woven glass with the addition of a conductive filler. The type, size, geometry, and concentration of the filler dictate the thermal/mechanical properties of the dielectric. Typical conductivity values are 1.0–3.2 W/mK. In comparison, standard FR-4 has a thermal conductivity of 0.3W/mK, and copper is 398 W/mK.

Why the need for a thermally conductive dielectric?

The simple answer is thermal management of the system. Components mounted to the PCB or IMS need power to operate and the bi-product is thermal energy. Current running through a copper conductor will generate heat due the resistance of the copper. The greater the resistance, the more thermal energy that must be dissipated. As the temperature of the conductor increases, so does the resistance. In an uncontrolled design, one can experience a runaway condition where the increase in temperature drives an increase in resistance which drives an increase in temperature and so on, eventually destroying the circuit. In most situations this is not the case, but for high-power applications, and high-speed signals, thermal management is a real concern.

It is true that LEDs generally do not produce much heat, and are cool to the touch. Then why the exponential growth in demand for thermally conductive dielectrics? To put it in perspective, the energy consumed by a 100-watt incandescent bulb produces around 12% heat, 83% IR and only 5% visible light. The typical low watt LED might produce15% visible light and 85% heat[1]. But with high power LEDs (>1 watt), it is essential to remove the heat through efficient thermal management. Without good heat sinking, the internal temperature of the LED rises, and this will cause the LED charac-

A BETTER **IDEA**

Valor® NPI
DFM JUST GOT
EASIER!

Valor NPI software is industry's only automated PCB DFM solution, seamlessly connecting PCB design, fabrication, assembly and test processes.

With the latest version of Valor NPI, setup time for DFM rules can be reduced from weeks to days, and DFM results viewing time can be reduced by up to 50 percent.

LEARN MORE AT **HTTP://WWW.MENTOR.COM/NPI**

Mentor®
A Siemens Business

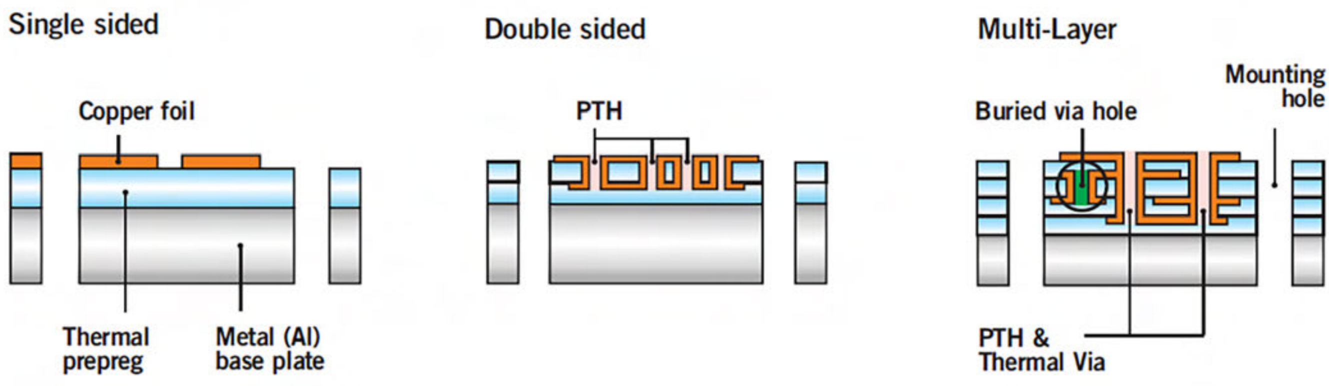


Figure 1: Standard IMS builds.

teristics to change. As the junction temperature of an LED increases, the lumen (brightness) output decreases. The output wavelength also shifts with a change in junction temperature. The optimum way to address this heat passively is by using materials with a high thermal conductivity to move heat away from the junction as quickly as possible. The dielectric should have high thermal conductivity both laterally (x and y directions) as well as vertically through the base of the device.

The circuitry routing for most LEDs is very simple, requiring only one layer of traces. A single-sided IMS would incorporate a thin thermally conductive dielectric sandwiched between a copper foil and aluminum heatsink. The dielectric helps to conduct the thermal energy away from the copper circuit to the aluminum heatsink where the energy is dissipated through convection to the air, or by conduction via a mechanical mounting system. More complex designs will have multiple layers of copper circuitry mounted to a heatsink. Components can be mounted on both sides by routing the aluminum prior to lamination. Very rarely will the aluminum heatsink be used as an electrical ground for the circuit. If electrical contact to the aluminum is a requirement for the design, it is a complex process to manufacture a plated via with connection to the aluminum. The aluminum interacts negatively to the electroless copper bath, and although feasible, few PCB shops offer the process. The easier approach is to use a copper heatsink versus aluminum.

Copper is an exceptional thermal conductor (~400W/mK), but cost and weight considerations drive aluminum to be the preferred heatsink for most applications.

As discussed earlier, aluminum reacts negatively to electroless copper. This is also true with some surface finishes, for example, electroless nickel/immersion gold (ENIG). There is an

Metal	Thermal Cond W/mºK	CTE ppm/ºC	Density gr/cc	Cost USD/lb
Copper	400	17	8.9	2.87**
Aluminum	170	25	2.7	0.87**

*** c/o InfoMine One week price 07/17*

Table 1: Aluminum and copper properties comparison.

UCE WET PROCESS LINE

Etchers

Scrubbers

Developers

PTH

Oxide Replacement

THIN CORE HANDLING • PC CONTROL • RELIABLE • LOW COST

CLICK TO VIEW OUR UCE WET PROCESS LINE

The leading European PCB Equipment and Parts Supplier

www.vikingtest.com

sales@vikingtest.com

Viking

$$\text{Thermal impedance (ºC/W)} = \text{Aluminium} + \text{Copper} + \text{Dielectric layer}$$

$$\underset{\text{(total IMS)}}{\text{Thermal impedance (°C/W)}} = \frac{\text{Thickness Aluminium (m)}}{\text{Al conductivity (W/mK) x sample area (m}^2\text{)}} + \frac{\text{Thickness Copper(m)}}{\text{Cu conductivity (W/mK) x sample area (m}^2\text{)}} + \frac{\text{Thickness Dielectric layer (m)}}{\text{conductivity of Dielectric layer (W/mK) x sample area (m}^2\text{)}}$$

Equation 1.

adverse reaction if the immersion gold bath is exposed to the aluminum. The work-around is to mask the exposed aluminum with a photo polymer and/or tape. Most fabricators do not want to take the risk of a scratch or pinhole in the protective material, and will avoid offering ENIG or ENEPIG as a viable surface finish for IMS product. Lead-free hot air leveling (HAL) is the most common finish, but the designer needs to verify their finish of choice with the fab house to be safe.

The thermal conductivity of most laminates is reported as the thermal conductivity of the standalone dielectric. Some specification sheets report the thermal conductivity of the laminate (copper + dielectric + aluminum). This can be misleading. As an overall performance measure of the system, the designer may want to focus on the thermal impedance vs thermal conductivity. The thermal impedance (or thermal resistance) is the inverse of the thermal conductivity times the material thickness. Thus, the thinner the dielectric, the lower the thermal impedance. For the total system, or at a particular joint, the thermal impedance can be calculated (see equation 1).

One can see that the benefit from going from a 5-mil dielectric with 3.2 W/mK thermal conductivity, to a 1.5-mil dielectric with a thermal conductivity of 2 W/mK will have a 2x improvement in thermal impedance of the dielectric. The thinner the dielectric, the lower the thermal impedance. Dielectric withstanding voltage becomes a greater consideration as designs specify thinner dielectrics. To ensure quality and reliability, it is recommended that IMS laminate be 100% high pot tested by the laminate manufacturer prior to shipping. This will ensure there are no pinholes or foreign material in the thin laminate that could cause a high pot failure.

Traditionally, thermally conductive dielectrics have been used in PCB designs demanding high current loads. Motor controls, power supplies, AC/DC converters and solid-state relays were the major industry applications. As discussed, the LED industry has been the driving factor for the growth experienced over the last 10 years.

Recently, we have seen a growing interest for thermally conductive dielectric solutions in full build multilayer PCBs. The thermal energy output of components can be more effectively managed within the PCB versus use of expensive fans, heat pipes and liquid immersion cooling. In partnership with Advanced Micro Devices Graphic Card Division, Lazer-Tech PCB and Cartel Electronics, 6-layer and 12-layer, HDI, impedance-controlled PCBs were built using 2.2 W/mK and 3.2 W/mK thermally conductive dielectric systems throughout the stack. A 16% reduction in the thermal resistance was realized with the PCBs made with the thermally conductive dielectric as compared to the PCBs made with FR-4. This improvement in thermal impedance allows tremendous opportunities in cost savings long term.

Much more work is needed to fully appreciate the benefits of these materials in full build complex PCBs. As technologies continue to evolve the need for thermal management solutions in PCB design are only going to increase. **PCB**

Reference

1. Fact or Fiction: LEDs don't produce heat, Tim Whitaker, LEDs Magazine.

Gareth Parry is president of Draig Technologies Inc, technical consulting services for the electronics, PCB and design industries.

The Road to Outsourcing CAM Engineering is Paved with VALUE!

	In-House	With Entelechy
Availability	48 weeks	52 weeks, 24/7
Overhead Cost	>30%	0%
Scalability	Limited	Unlimited
Redundancy	Variable	100% Bulletproof
Operational Efficiency	Status Quo	Continual Improvement
Total Cost	Fixed	Significantly lower

"In our company, the CAM department was the bottleneck. As a result of our working with Entelechy, we can now accept orders that we had to refuse in the past."

-Thomas Hofmann, CEO/Owner

HOFMANN LEITERPLATTEN

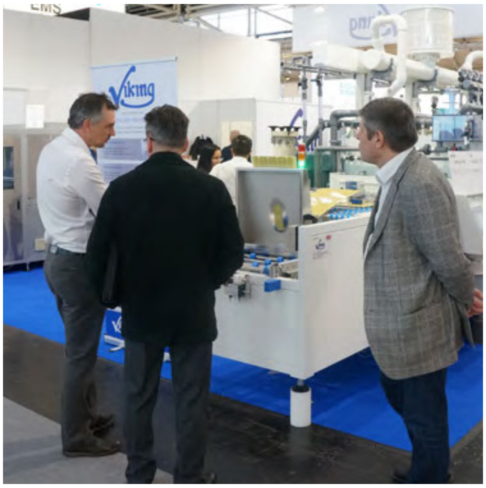
Click here to check out our risk-free Quick Start program»

We deliver the highest quality PCB CAM and Mechanical CAD engineering services to customers around the world.

ENTELECHY GLOBAL

www.entelechyglobal.com

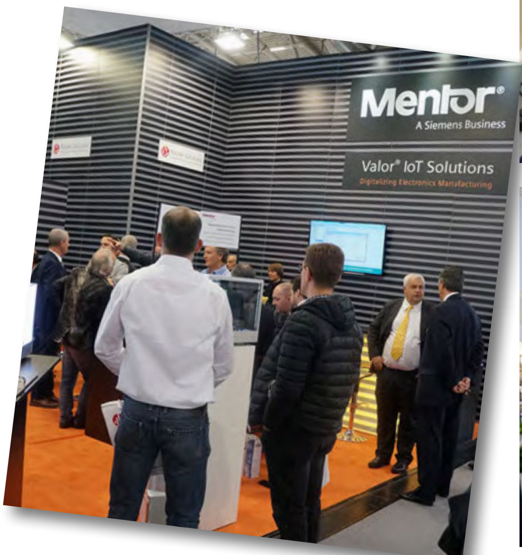
productronica 2017

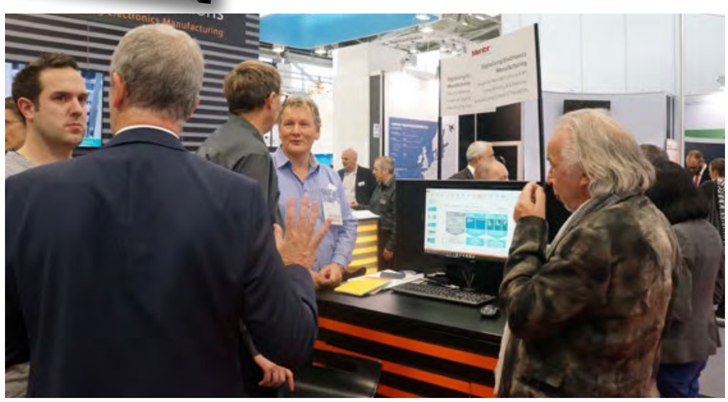


This was my first visit to productronica and it was all they said it would be. Hall after hall were filled with technology, though only one was focused on PCB manufacturing. All of the familiar equipment and chemistry manufacturers were there, along with a number of ones new to me.

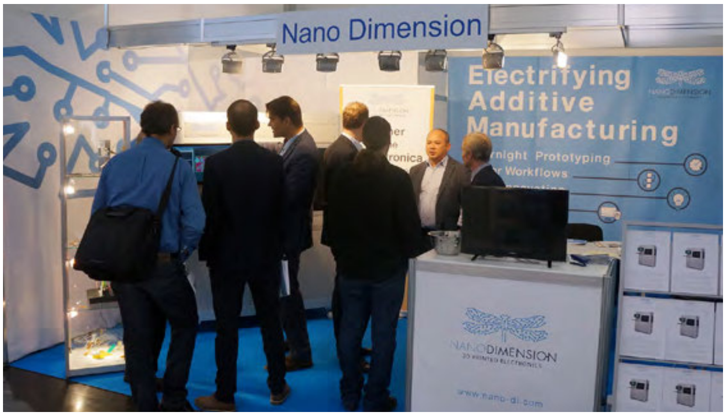
Hopefully these photos help to capture the flavor—literally, as food was the big giveaway at most booths—no pens, no gadgets, just wonderful food and pretzels! Click here to see our entire photo gallery.

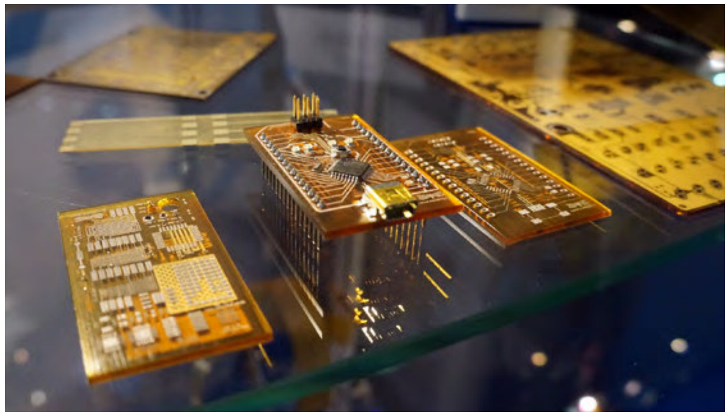
—***Patty Goldman***

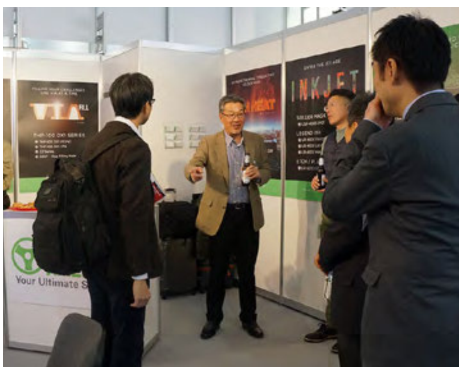


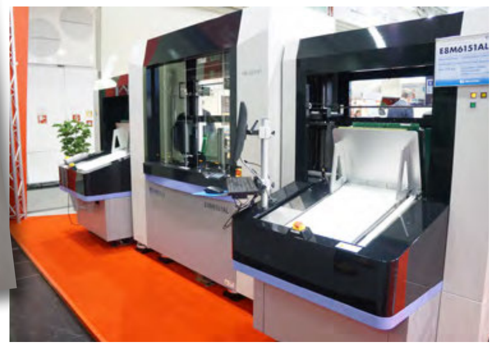


productronica 2017

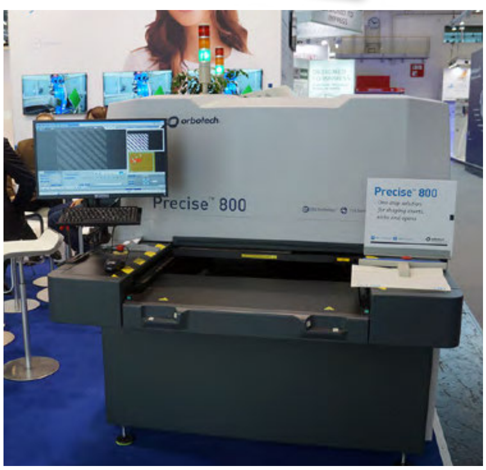


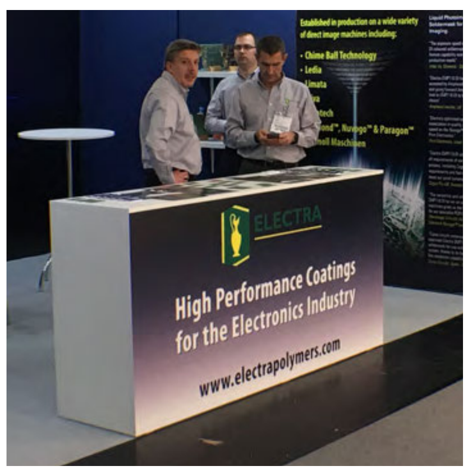




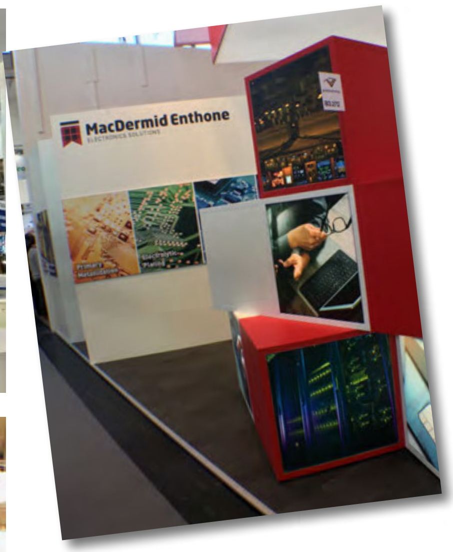


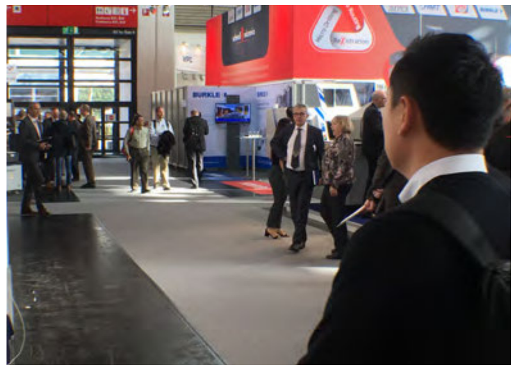
productronica 2017

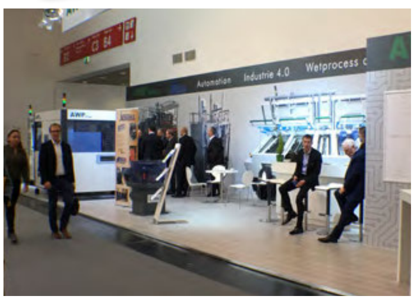


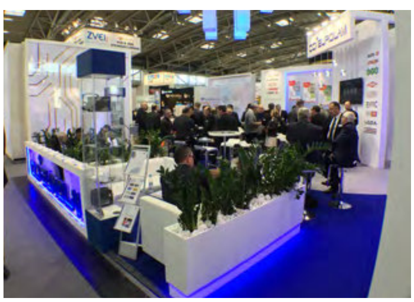


productronica 2017









The Copper Foil Shortage: Why, How and What to Expect

by Didrik Bech
ELMATICA

Earlier this year we wrote in our blog about the elephant in the room. You know, the one everybody knows is there, but nobody talks about—the copper foil shortage. Nine months later, the copper shortage is still present but now perhaps the elephant is less visible. We have encouraged open dialogue and information-sharing in order to minimize the effect of unwanted situations.

The present global lead time development for components, with lead time increase for some components from 26 to 52 weeks' lead, has resulted in a state of temporary stabilization of copper-clad demand. However, once the component market stabilizes and/or in conjunction with the increased lithium battery production, one can expect a constrained supply situation.

With several governments now desiring a shift towards cleaner new-energy vehicles, and an all-electric automotive industry, the copper shortage will last longer than first anticipated. One thing is certain: Now is the time to strategize, inform and plan. Copper output has not increased and the demand for copper in lithium batteries is not decreasing, so the copper foil shortage will persist. There are no signs in the near future that it will change, especially now with the recent governmental statements from France, England, Scotland, Norway and China, who have all announced a possible future ban towards fossil-fueled vehicles.

China has set a goal that a minimum one-fifth of all new cars in 2025 shall be electric; that's four million cars per year in China alone and as of November 2016, a total of one million pure electric cars had been produced globally. The electric car is also unique in relation to copper not only because of batteries and copper demand but the car is practically a computer on wheels rather than a car with a computer, and the PCB demand is consequently significantly higher.

A new set of rules for PCB production

Less process steps. CAM to etch in minutes instead of hours. Maximum yield. Minimal environmental impact.These are just a few of the PCB production rules that Lunaris has redefined.

We've put together a complete set of all the new rules made possible by Lunaris ...get yours today!

Download all the new rules:
The Official Lunaris Rulebook

CLICK HERE TO DOWNLOAD

mutracx
THE BETTER WAY

www.mutracx.com

China already has a quite aggressive zero-emission mandate, resulting in automakers significantly increasing their investments in electric vehicle production in that country. As the biggest car market in the world, it's not hard to understand that a massive increase in production of electric cars for China will affect the copper shortage, and the amount of copper foil reserved to produce printed circuits. It is estimated that there will be a 10 million ton copper foil deficit by 2028; that is around 40% of the copper demand in 2017.

> "Adding further tension to the copper shortage is China's plan to ban waste imports, which is a significant source of recycled copper."

Adding further tension to the copper shortage is China's plan to ban waste imports, which is a significant source of recycled copper. Other actors will continue this work; however, it will only create a temporary decrease in supply. These aspects and other elements such as mines located in unstable countries, few new copper mines and a number of closed mines will affect the supply chain of copper and in turn the copper-clad foil situation for the PCB industry.

No Time to Lose

In May, Mark Goodwin, Ventec COO for the USA and Europe, said, "Our market assessment is that the copper foil shortage has resulted in approximately 2.8M sheets per month CCL global material shortage. This is approximately 2x the total rigid demand of USA and Europe combined. We expect the situation to last at least until the middle/end of 2018. Now is the time to work closely with your material supplier to secure your supply, and to pass on the inevitable increases in cost in your finished board prices."

Now is the time to plan. The automotive industry will not shift overnight; the production line takes time to adjust. But the PCB industry needs to start communicating about this challenge and open dialogue is critical, since we all are facing the same issues.

An additional concern is the continued shortage of glass yarn which also reduces the output of glass cloth used in laminates and prepreg. One of the world's largest CCL manufacturers has recently issued a notice to its customers, informing them of a reduced output until the end of 2017 due to insufficient glass cloth.

Feeling Stable

The current apparent stability of the CCL supply is most likely due to a decrease in orders placed and/or postponement of delivery of printed circuits in recent months. This is due to a lack of components and hence, a short-term decrease for PCB. This stable situation is not due to an increased supply of CCL. The question is, what will happen over the next months with Chinese New Year coming, copper scarce, and if the supply of components stabilizes?

If everyone starts placing orders for printed circuits at the same time, then a temporary shortage of CCL can result. We have for a long time advised our customers to prepare an accurate forecast, place orders in advance, and plan for longer lead-times. Another strategy is to approve more than one type of laminate for a product and consequently increase the flexibility of supply.

The whole supply chain needs to work together now. This is not a challenge that will suddenly disappear. It has built up over the last years, and I am certain that it will stay for quite some time. **PCB**

References

1. Global and China Copper Clad Laminate (CCL) Industry Report, 2016-2020, ResearchandMarkets.

Didrik Bech is CEO of Elmatica.


Get the spark you need
to tune up your sales team.
When an engine doesn't fire on all cylinders, performance is limited. The same goes for your sales team. We provide the spark to get your sales to the next level ...and beyond.
We've helped dozens of companies big and small improve their bottom line.
db
MANAGEMENT
GROUP
Learn how we can help YOU
207-649-0879
danbbeaulie@aol.com

MilAero007 Highlights

[All About Flex: More on UAVs and Flexible Circuits]

The use of drones or unmanned aircraft vehicles (UAVs) is growing at a nearly exponential rate. This includes drones used by the government, private companies and hobbyists. If you have $100, you can find a long list of drones for purchase.

Nine Dot Connects: Good Design Instruction is a True Value-Add

Nine Dot Connects has certainly blazed an interesting trail. The company started out as an Altium reseller, but in less than a decade, Nine Dot Connects has also become a design service bureau and a provider of PCB design instruction, training, and consulting services.

Coast to Coast Names Ed Porter President and CEO

Coast to Coast Circuits has named Ed Porter president and CEO. Porter has served in a variety of managerial and executive leadership roles over the last 20 years within leading organizations in the printed circuit board industry, including TTM Technologies and Sanmina SCI.

NPL and SMART Group to Hold Design, Process & Reliability Seminar

The National Physical Laboratory and SMART Group will hold a seminar November 9, 2017, that will showcase the latest research and results from NPL projects looking at solder joint and contamination failure, coating thickness measurement, solder joint reliability, and high temperature reliability for alternative solders and substrates materials.

American Standard Circuits Appoints Tony Monaco to Sales Team

American Standard Circuits has recently added industry veteran Tony Monaco to the company's sales team.

Leveraging Industry Shows to Create a Presence

At the recent SMTA International show in Rosemont, Illinois, I had an opportunity to meet with Mark Osborn, president and owner of Colonial Circuits, based in Fredericksburg, Virginia. Colonial Circuits is a supplier of PCBs, mainly for the defense industry.

Catching up with...Brigitflex

Today, Brigitflex is building unique custom-made boards for companies all over the world, from large defense and aerospace companies to small incubator companies inventing new products. They have become well-known as the shop to go to when nobody else can solve your problems.

'Lighten Up' - Deep Space Communications via Faraway Photons

A spacecraft destined to explore a unique asteroid will also test new communication hardware that uses lasers instead of radio waves.

New RoboBee Flies, Dives, Swims and Explodes out the of Water

New floating devices allow this multipurpose air-water microrobot to stabilize on the water's surface before an internal combustion system ignites to propel it back into the air. This latest-generation RoboBee, which is 1,000 times lighter than any previous aerial-to-aquatic robot, could be used for numerous applications.

Marine Communication Systems Market Witnessing Surge in Advanced Technology

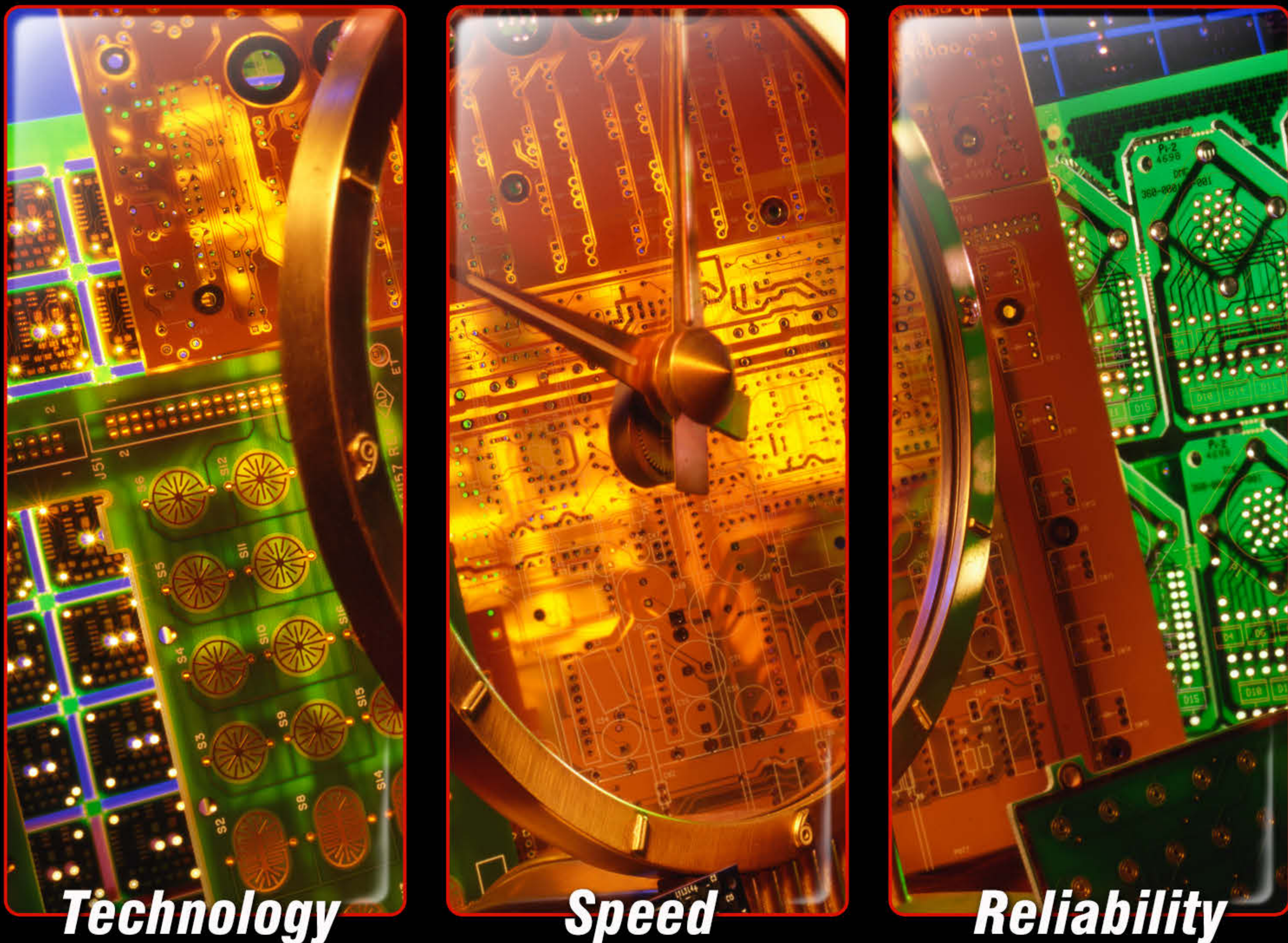
Technavio market research analysts forecast the global marine communication systems market to grow at a CAGR of more than 8% during the forecast period, according to their latest report.

prototron.com

We're Hiring!

We Take the Time to do it Right

because you don't have the time to do it over.



Prototron Circuits

High quality quick turn manufacturing, just another reason we are

America's Board Shop

Prototron is Hiring! We are currently seeking candidates to fill positions for both a midwest sales professional and an experienced PCB process engineer. To apply, contact us at 425-823-7000 or email Russ Adams at russa@prototron.com for sales or Kirk Willams at kirkw@prototron.com for engineering.

Case Study: Solving Plating Pits and Mouse Bite Issues, Part 1

by Michael Carano
RBP CHEMICAL TECHNOLOGY

Introduction

It happens when you least expect it. You are processing a highly critical job for a key customer. Everything is looking good so far. You have your best team on this job. And then, after all the plating is complete and expensive boards are being inspected, the QC team calls your attention to pitting and what looks like someone had taken a bite out of the copper traces and pads. This is, in a nutshell, a significant process indicator at the very least and perhaps a non-conforming defect (depending on whether trace widths and annular rings are reduced below requirements). So, let's look at an actual situation that occurred and how the problem was resolved. In Figure 1, an example of the defect is shown.

Background

The PCB fabricator had been using the same photoresist for at least two years and had not had any reported issues with pitting or mouse bites. The company employed a standard pattern-plating process. After electroless copper was applied, the boards are laminated with photoresist, exposed and developed. From here, the boards were processed through the pattern plating process which included an acidic soak cleaner, followed by micro-etch, followed by acid copper and tin plating. The remaining processes included resist strip, etch and tin strip. It was during the inspection that pitting and mouse bites were noted. So, with the group now brainstorming as to possible causes, several possibilities as to the origin of the pits and mouse bites were discussed. These are:

1. Review of lamination procedures
- Vacuum draw-down
- Lamination pressure and temperatures
- Unintended exposure

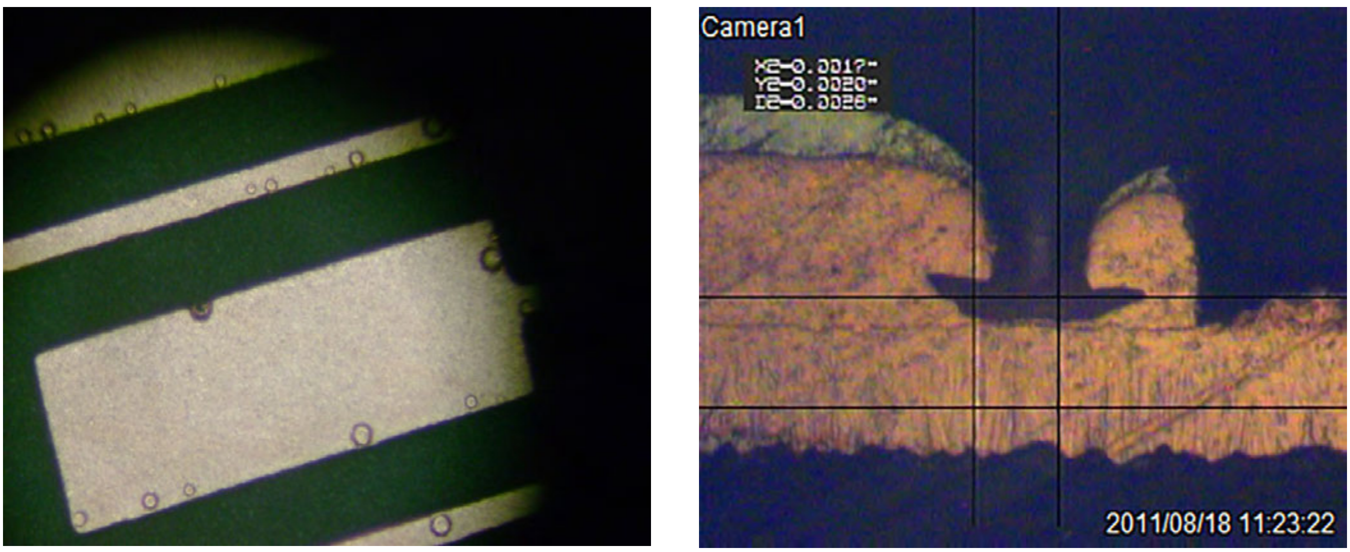


Figure 1: Pitting and mouse bites on copper circuit pattern. Example of two defects shown: Pitting on left photo around area of resist sidewall and on the right example of a pit that goes down to the base laminate.

Panasonic

DesignCon 2018
WHERE THE CHIP MEETS THE BOARD

EXPO: JAN 31 - FEB 1, 2018
SANTA CLARA CONVENTION CENTER

Visit our booth # 654

MEGTRON7

Laminate **R-5785(N)**
Prepreg **R-5680(N)**

H-VLP2 Cu and Low Dk Glass-Cloth Specifications

Features

- H-VLP2 copper foil and Low Dk glass-cloth allows for 30% loss improvement than MEGTRON 6
- Glass transition temperature (Tg): 200°C (DSC)
- Reduce process cost (Improve drill processability and ENIG (Electro-Nickel Immersion Gold))

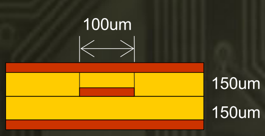
	Dk	Df
R-5785(N) + H-VLP2 Cu	3.4	0.002 @ 12GHz
R-5775	3.6	0.004 @ 12GHz

Applications

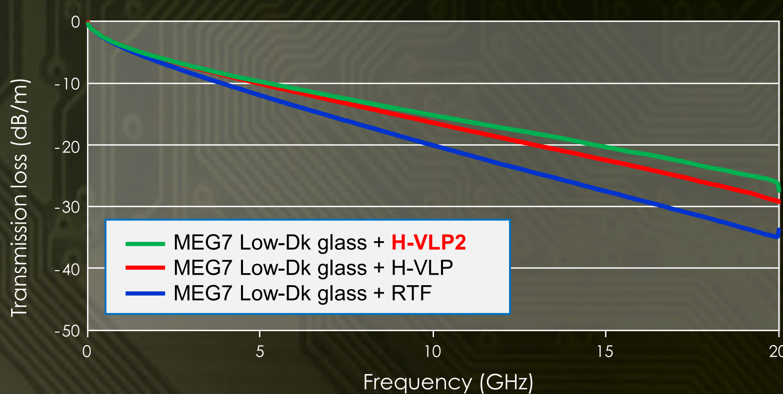
- High-end servers, High-end routers, Supercomputers, and other ICT infrastructure equipment, Antenna (Base station, Automotive millimeter-wave radar), etc.

Trasmission Loss

- Evaluation Sample



Line Length	1000 mm
Impedance	50 Ω
Copper Thickness	18 μm
Inner Cu Treatment	No-surface Treatment
Core Type	#1078 (RC67%) x 2ply
Prepreg Type	#1078 x 2ply



Contact us

More information

DISTRIBUTOR IN NORTH AMERICA
MATRIX USA INC.
TORONTO • SANTA ANA • SANTA CLARA • CHICAGO • MINNESOTA
Visit our website at www.matrixusa.us

Partnering to go beyond.
Electronic Materials
Panasonic Corporation

2. Effectiveness of developing step
- Developer pH
- Developer loading
- Breakpoint in developer

3. Rinse water quality after developing
- Hardness of water
- Rinse water temperature

4. The electroplating process (pattern plating, cleaning)
- Effectiveness of acid cleaner
- Acid copper plating solution (organic contamination, air bubbles)
- Solution filtration

Unintended exposure is often overlooked as a potential cause of pitting and mouse bites. During the exposure process, there can easily be some amount of UV light that may find its way to areas that are not intended to be exposed. Unintended exposure will lead to partial polymerization of the resist. Consequently, this leads to incomplete development of the resist in the areas that require complete development. Resist residues left behind will increase the risk of multiple plating defects such as pits, mouse bites, domed circuit traces, ragged edge plating, and can even result in the plated copper lifting from the base.

The factors contributing to this variable are degree of collimation and declination of the UV radiation, the distance between the phototool and the resist, which in turn depends on the vacuum drawdown, the coversheet thickness, the presence or absence of a protective coversheet over the phototool, the degree of the UV scatter that the beam experiences while traveling through the media, and how much, and at what angle, UV radiation is reflected by the substrate under the resist. More on this later.

The Troubleshooting Approach

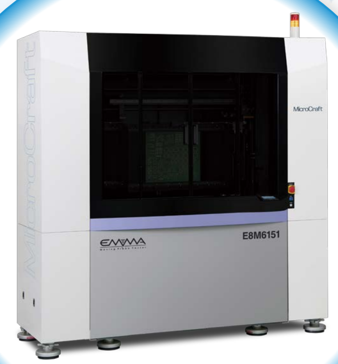
The team felt fairly certain the root cause of these defects rested in one or more of the process areas listed above. So a divide and conquer approach was taken in order to speed up the process of identifying and correcting the root cause. When using this approach, list all process steps in sequence. Examine product halfway through process for defects or its possible cause (is the problem there yet?). Keep on dividing until the exact process step causing the problem is located. Also, make use of other information sources, such as suppliers and available literature. This may also include cross-check-

Variable	80% Range/Spec. 10% Range/Spec.	Process Effect	Test Method (Frequency) and Notes
Radiation intensity; Lamp power (kW)	**80%:** 1–8 **10%:** 5–10	•Exposure time •Productivity •Fine-line quality yield?	Radiometer: excessive exposure intensity will lead to partial polymerization in the unintended areas
Exposure intensity (at the surface of the resist), (mW/cm^2)	**80%:** 5–10 **10%:** 10–20	•Exposure Time •Productivity •Fine Line Quality Yield?	Radiometer
Exposure energy at resist surface, (mJ/cm^2)	**80%:** Range: 30–100 Spec.: Depending on resist type/ thickness	•Line-width uniformity •True feature size reproduction •Resolution capability	Radiometer (Integrator)
Spectral output characteristic (radiation intensity vs. wavelength); peak wavelength	Range: 340–440 nm Peak: 365 nm Output in 400–440 nm region should be low	•Effective radiation intensity ⇒productivity ⇒line width reproduction	
Phototool alignment accuracy	Repeatability: +/-2 µm top to bottom (with glass to glass to fixture) Phototool <-> reference target (with CCD camera): +/-20 µm repeatability	Registration (top/bottom; hole/pad)	If tool is misaligned, it is very possible that areas designed to be left unexposed are receiving some UV energy.
Collimation (half angle)	**80%:** 5° **10%:** 1.5°	•Resolution •Off contact performance	
Declination (half angle)	**80%:** 5° **10%:** 1.0°	Resist sidewall shape	Visual—microsections
Cooling (heat rise in °C at the panel surface/phototool during exposure)	**80%:** 2°C **10%:** ²2°C	•Image dimension •Dimension reproducibility	
Cooling mode (closed loop; other)		Clean room environment maintenance	
Vacuum draw down time (adjustable) in seconds	**80%:** 10–45 sec. **10%:** 10–45 sec.	Phototool/board contact (air escape)	

Table 1: Process variables for dry film lamination and exposure. (Source: author's personal experience and IPC-5001 document.)

NEED FASTER THROUGHPUT?

MicroCraft's Latest Solutions for Quick Turn Shops



E8M6151

3x faster than previous models

CPT6151

Both solder resist and legend inks are printed with 1 machine.

SC6151

10x faster final bake

www.usamicrocraft.com

ing other processes and possibly even having panels processed outside the facility having an issue.

For the first process check, the lamination and exposure parameters were scrutinized. Table 1 lists the variables along with the outcome of the process checks.

With assistance from the photoresist supplier, the lamination and exposure parameters were reviewed. Only minor process issues required any attention. As an example, there was concern as to the exposure intensity. That is,

(Exposure Energy) x (Time of Exposure) = Exposure Intensity

It was determined that, while overall exposure of the resist looked effective, there was an issue of perhaps too high an exposure intensity, which may allow for partial polymerization of the resist that is not intended to be exposed. As an adjustment was made to correct the increase in intensity, the team then processed panels through the developing process. Interestingly, the defects (mouse bites, pitting) remained. The team then decided to concentrate on the developing portion of the imaging process and the pattern plating operation. This will be presented in a future column as Part 2.

Summary

It was important during this exercise to gather the team together and brainstorm over the multitude of possible causes for the pitting and mouse bite problem described herein. For Part 1, the possible process areas that could contribute to the defect were identified and a systematic approach to finding the root cause was begun. **PCB**

Michael Carano is VP of technology and business development for RBP Chemical Technology. To reach Carano, or read past columns, click here.

NASA to Test Advanced Device for Returning Small Spacecraft to Earth

NASA launched the Technology Educational Satellite, or TechEdSat-6, to the International Space Station on Orbital ATK's Cygnus spacecraft from NASA's Wallops Flight Facility in Virginia on November 12. This bread loaf-sized satellite is part of a continuing series to demonstrate the "Exo-Brake" parachute device, advanced communications and wireless sensor networks.

TechEdSat-6 was released into low-Earth orbit from the NanoRacks platform on November 20, to begin wireless sensor experiments that will be the first self-powered tests, expanding the capabilities of sensor networks for future ascent or re-entry systems. This is the fourth TechEdSat satellite carrying an updated version of the Exo-Brake that will demonstrate guided controlled re-entry of small spacecraft to safely return science experiments from space.

"The Exo-Brake's shape can be changed to vary the drag on the satellite. With the help of high-fidelity simulations, we will demonstrate a low-cost, propellant-less method of returning small payloads quickly, and to fairly precise locations, for retrieval," said Michelle Munk, NASA's System Capability Lead for Entry, Descent and Landing.

The Exo-Brake is funded by the Entry Systems Modeling project within the Space Technology Mission Directorate's Game Changing Development program.

UPCOMING Programs

December 6–8, 2017

HKPCA International Printed Circuit and Exhibition & APEX South China Fair

Conference

Shenzhen, China

December 13, 2017

Wisdom Wednesday

Strategies for Reducing Product Warranty and Liability Risk in Manufacturing Services Agreements

IPC Members Only

Webinar

February 24–March 1, 2018

IPC APEX EXPO 2018

Conference & Exhibition

San Diego, CA, USA

March 1–2, 2018

The Pb-Free Electronics Risk Management (PERM) Council Meeting No. 35

held in conjunction with IPCAPEXEXPO

IPC Members Only

Meeting

San Diego, CA, USA

IPC TECHNICAL WEBINAR

December 12, 2017 | Cleaning Before Conformal Coating

Do you conformally coat your electronic assemblies? If so, cleaning prior to conformal coating is a must to ensure the long-term reliability of your products. This webinar provides an overview of potential failure mechanisms resulting from coating over residues. Topics to be discussed are:

- Cleanliness requirements for electronic assemblies before coating
- Overview of standards
- Cleanliness monitoring
- Methods for reliability testing and quality assurance

Join us for this educational opportunity on **12 December at 14:30pm CET**.

Cost:

IPC Members: €55

Non Members: €69

www.ipc.org/cleaning-coating-webinar

For more information, **visit www.IPC.org/events**

The Power of Flex

by Tara Dunn
OMNI PCB

Fall in the Midwest is one of my favorite seasons. The leaves are changing color, the air is crisp, but not yet cold, and those unexpected days when the weather turns warm in late September and early October bring a smile and cheerfulness. Somehow it is like cheating the winter season just a little. Fall in the Midwest is also a busy time for farming. Fields are thick, dense and ready to harvest.

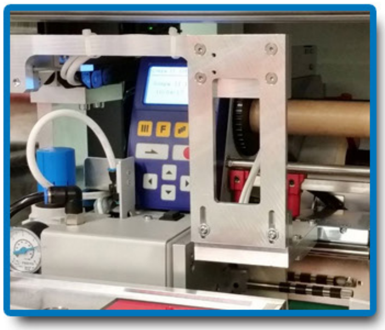
I have a story to share. Picture a beautiful fall day filled with sunshine and warm winds. Just after dinner a family is out in their yard. The father and older son are finishing chores before dark and a little boy is playing in the yard with the family puppy. In a split second that puppy ran into one of those thick, dense cornfields and this four-year-old boy followed, chasing after his favorite pet. It was one of those split-second moments that every parent fears. If you have ever been in a cornfield in the Midwest, you know—your visibility is limited to maybe a row or two in front of you and maybe a row or two to the sides.

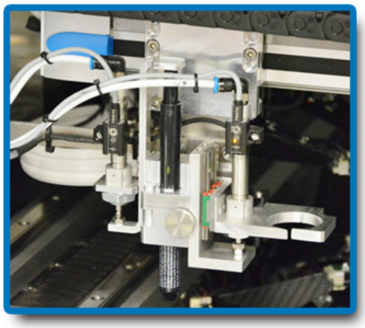
It quickly became apparent that this family was going to need help and law enforcement was called. Within an hour there were 160 trained volunteers from the surrounding communities and a command center was set up. Ten years ago, this would have been what you would typically think of as a manhunt with chains of people walking through the fields. The scene looked significantly different on this fall day. The command center utilized drones enabled with infrared and heat-sensing technology, a helicopter with similar technology to cover a larger area, the GPS from the tractor that planted that field was able to accurately display where each and every stalk of corn was and all 160 of those searchers were able to communicate in real time.

So, why do I tell you this story in a flexible circuit column? I tell this story because each of those items just listed contains a flexible circuit. Our industry accomplishes some pretty amazing things. We regularly hear that flex and rigid-flex are a significantly growing portion of the

Unrivaled Test Speed with Full Automation

Introducing the newly designed atg A8a with 8 test probes and a new high speed “lights out” automation for unrivaled throughput.





Highlights:

- Small footprint (6 square meters)
- Dual shuttle pick & place automation
- High accuracy combined with high test speed
- Pen or label marking option

Watch video

Get more info

atg®

LUTHER & MAELZER®

an Xcerra company

atg Luther & Maelzer GmbH
Zum Schlag 3 • 97877 Wertheim • Germany
Phone +49-9342-291-0 • Fax +49-9342-395 10
sales@atg-lm.com • www.atg-lm.com

world-wide PCB market but, speaking only for myself, I don't always take time to really think about the end applications that flex enables. Let's look at a few of the known benefits of flexible circuits and what type of products we may interact with that have been, or are being, developed to take advantage of this.

Advantages of Flexible Circuits

Solve Product Packaging Problems

Flex allows for a 3-axis connection. It is able to be bent and folded around corners eliminating the need for discrete pieces. It is easy to think of products that take advantage of the space-saving benefits of flexible circuits: portable medical devices such as insulin pumps or heart rate monitors, hearing aids, smartphones

"As consumers, we are requiring our electronics to be smaller, lighter and at the same time have increased functionality. Flexible materials allow designers to meet those demands."

and tablets, and cameras. As consumers, we are requiring our electronics to be smaller, lighter and at the same time have increased functionality. Flexible materials allow designers to meet those demands.

Reduced Assembly Cost

Flex eliminates hand-wiring and provides additional cost savings when purchasing costs for multiple wiring and component pieces are factored in. Home monitoring bracelets and wearable electronics are good examples. The product needs to be lightweight and durable; wired and flexible circuits are both options. A simple flex circuit eliminates time for assembly, purchasing costs and inspection costs by solving the problem with just one unit.

Reduces Weight and Volume

This is a big one. Bulky wire harnesses and solder connections can be replaced with thin, lightweight rigid-flex. It is not uncommon to see studies showing that this savings in weight and space can be near 60%. Aerospace is a perfect example of an industry that benefits from reduction in weight and volume. With aircraft, rockets, missiles, etc., weight is an expense. Any opportunity to reduce weight and space translates to a product that is less expensive to operate. The fun little TV screens that are being built into aircraft to keep us entertained, lighting systems in the airplane, engine controls, braking systems—all are products that have taken advantage of flexible materials.

Dynamic Flexing

This is easy. Anything with a hinge! The one I use every day is my laptop. Let's not forget printers, disk drives, cameras, and robotic arms.

Thermal Management

Flexible dielectrics offer a greater surface-to-volume ratio than round wire and this extra surface facilitates getting the heat away from the circuit. Rigid PCB dielectrics often act as a thermal insulator inhibiting the flow of heat. One area of significant growth in flexible circuit designs is the LED lighting market. Automotive and aircraft applications, especially with the combined benefit of lighter weight and improved thermal management, are increasing the usage of flex. Examples include headlamps, interior lighting, and interior electronics, just to name a few. One of my favorite applications is LED lights in a pair of high-top tennis shoes. This application is not just your typical shoe that lights up when you walk; this high-top was designed with an artistic LED lit pattern throughout the shoe. It might not be the most high-tech application, but it is eye-catching and fun.

Improved Aesthetics and Bio-Compatibility

Appearance can impact decisions when the end user is exposed to functional elements of the product. For example, a simple hand-held medical device being used in a doctor's office

Ucamco
Former Barco ETS

SOFTWARE | PHOTOPLOTTERS | DIRECT IMAGERS | NEWS | SERVICES | COMPANY | DOWNLOADS | CONTACT

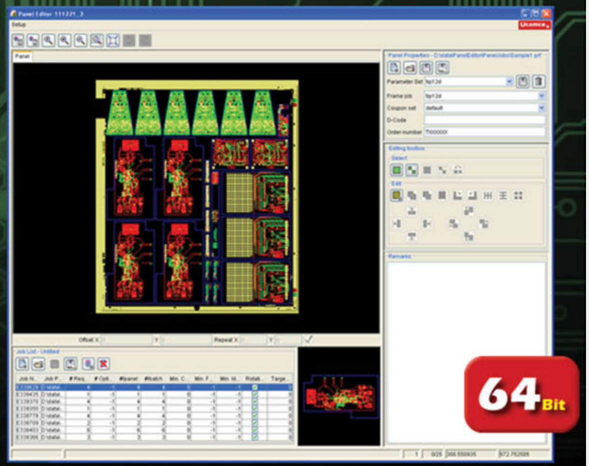
Software

Multi Job Panelizer

High-yield multi-job panel generation made easy ...

- Combines different PCBs into a production panel
- Optimizes material utilization
- Reduces material costs
- Interactive or 24/7 automated operation
- Hotlinkable to ERP or Planning systems

Read more



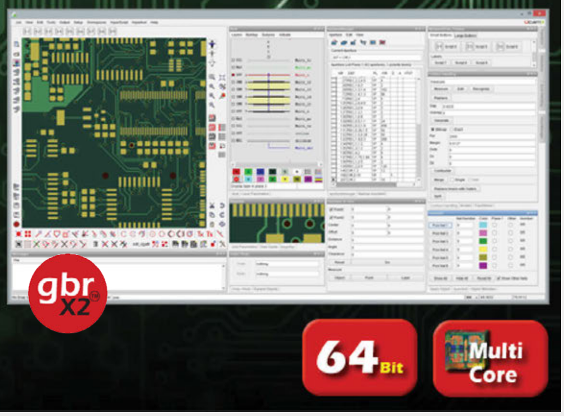
Software

UcamX

New ground-breaking CAM Software

- For Rigid, Flex and HDI PCBs
- New Graphical User Interface with Workspaces
- Optimal performance using Parallel Processing
- Multi Job Processing via load balanced Sessions
- Support for Gerber version 2 (X2) format

Read more



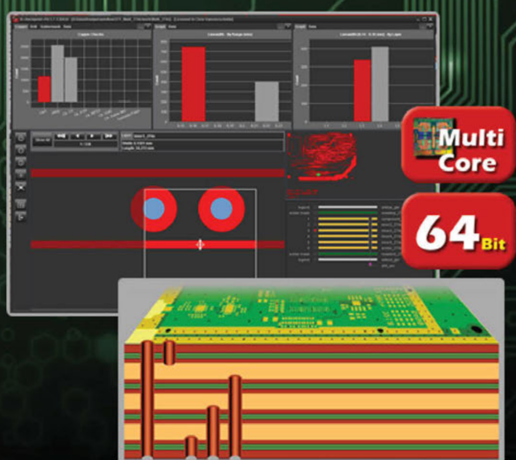
Software

Integr8tor

Automated data entry and design analysis

- Ucamco's 9th Generation Sales & Engineering Tool
- Field proven with more than 250 installations in daily production
- Seamless integration into any environment

Read more



For more information please contact info@ucamco.us,
call (415) 508-5826 or check out our website at www.ucamco.com

had a wire that was visible to the patient. Although the medical device was working perfectly, patients' perception of and confidence in the procedure was not high. This was traced back to patients not being comfortable with the perception of the wire. That simple wire was replaced with a very simple flexible circuit, so simple, there were only two traces. But, by making this simple change, the patients' perception and confidence in the medical device skyrocketed.

Polyimide is also bio-compatible. Most often, the polymide material is fully encapsulated before being inserted into the body. New developments are exciting. Polyimide laminate with gold, rather than copper traces are fully biocompatible and being tested as sensors to be implanted into the human body. This development is also aided by additive technology that allows trace size in the 5- to 10-micron range, significantly shrinking the package size as well. There are exciting things on the horizon.

Intrinsically More Reliable and Reduced Opportunity for Operator Error

Flexible circuits can significantly simplify the system design by reducing the number and levels of interconnection required. Because the design is controlled by the artwork, the opportunity for human error is eliminated. Aerospace is great example. Spacecraft are subjected to many kinds of dynamic forces, especially during take-off. In traditional PCBs these vibrations contribute to failure. Rigid-flex circuits are made to twist and flex and are a benefit in these harsh environments. Solder joints, crimps, etc., are also at risk for failure in these conditions. Flexible circuits can remove this concern by eliminating connections.

Yes, our industry has developed so many interesting, life-enhancing and life-saving products and for that we should all be proud to be a part of the growth in this market. To finish the story I started earlier, this little boy emerged from the field, a little tired, very muddy and mostly angry that he still had not found his puppy. Guess what. The person stationed at the edge of the field that spotted him was able to notify his parents and the command center immediately with his cellphone, which, you guessed it, also contains a flexible circuit. **PCB**

Tara Dunn is the president of Omni PCB, a manufacturer's rep firm specializing in the printed circuit board industry. To read past columns or to contact Dunn, click here.

Old Phones Get New Life in High-Powered Computer Servers

While most consumers don't pay much attention to the fate of dead smartphones, Princeton University researchers are envisioning a way to breathe new life into them. Instead of tossing old phones in a junk drawer or burying them in a landfill, the researchers want to turn them into high-powered computer servers.

In a recent paper, graduate student Mohammad Shahrad and David Wentzlaff, an assistant professor of electrical engineering, demonstrated that it is possible to build servers out of ranks of old smartphones. Servers are specialized computers that provide data, storage or computing power to other computers. The research showed that not only are smartphone servers feasible, they are often cheaper to build and operate than conventional high-end servers. In part, this is a result of supply costs.

"You can get decommissioned smartphones at low cost because used phones are inexpensive and no one else wants them," Wentzlaff said.

The researchers said that reusing smartphones not only makes economic sense, it is a good environmental policy. Electronic waste, of which smartphones play a substantial part, is a growing concern worldwide.

Why Choose Fein-Line?

Because there is a Fine Line between winning and the alternative.

Fein-Line Associates is a consulting group serving the global interconnect and EMS industries, as well as those needing contact with/information regarding the manufacture and assembly of Printed Circuit Boards. The principal of Fein-Line Associates, Dan (Baer) Feinberg, formally president of Morton Electronic Materials (Dynachem) is a 50+ year veteran of the printed circuit and electronic materials industries. Dan is a member of the IPC Hall of Fame; has authored over 150 columns, articles, interviews, and features that have appeared in a variety of magazines; and has spoken at numerous industry events. He covers major events, trade shows, and technology introductions and trends.

Mr. Feinberg and his associates specialize in:

- management consulting
- technology transfer
- new product market entry
- merger and acquisition due diligence
- market information and market research
- expert witness assistance and seminars regarding all aspects of printed circuits
- electronic assembly manufacturing and marketing

Dan (Baer) Feinberg

Fein-Line Associates, Inc.
P.O. Box 73355
San Clemente, CA 92673

Telephone: (949) 498-7866
Email: baer@feinline.com

Fein-Line Associates, Inc.

www.feinline.com

Culture Shift is Key to Quality Improvement

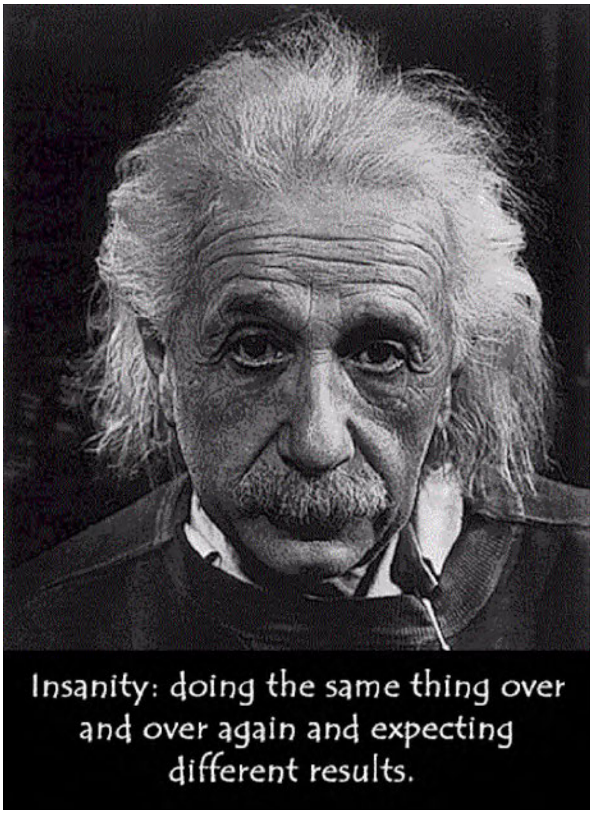
by Steve Williams
THE RIGHT APPROACH CONSULTING LLC

Introduction

Any major initiative, whether implementing ISO, lean manufacturing or introducing a new product, requires culture change. How this change is managed will be the difference between success or failure of the project. This column will offer some fundamental elements that will help navigate your next major implementation by shifting the culture.

Sage Advice

I was enjoying a fine cigar and cognac with an old friend a number of years ago when the discussion turned to the challenges of initiating a major change like implementing ISO. I asked my friend, whose opinion I greatly respect, "How do I begin to steer a company in a new direction that is so transformational it is sure to meet a heavy dose of resistance, skepticism and attitude?" My friend said, "You have to change the culture; and that is no small task." He went on to say, "Steve, have you heard the definition of insanity? Insanity is doing the same thing over and over again and expecting different results." As he has a talent for, my good friend had distilled a very complex issue down to a single, critical point: culture shift. As the evening wound down and we began to go our separate ways, I said, "Good talk Al, by the way, how's your theory of relativity thingy going?"



Insanity defined by Albert Einstein.

Culture Shock

Why is organizational change needed? Because in America, people tend to accept that a certain amount of error is normal. You expect the plane to be late. You expect some mail to get lost. You expect consumer electronics to break down right after the warranty runs out. This translates to our manufacturing operations as well; we expect a certain amount of our process output to be defective, and we plan for it. This mindset extends further into our quality systems, where many times the focus is on detection and not prevention. Quantum operational improvement requires a cultural shift that not just expects, but demands, exceptional quality to be the norm, with perfection as the goal. As the great Vince Lombardi once said, "Perfection is not attainable, but if we chase perfection we can catch excellence."

Fundamentally, most quality systems all want the same thing: a documented process that focuses on continuous improvement. All systems require strong senior leadership commitment to be successful be-

Productive Fabrication Equipment

MDI-DT

WorkCell²

Hybrid CombiDrill UV/CO_2

schmoll optronics

BÜRKLE
PROCESS TECHNOLOGIES

schmoll optronics

MDI Micromirror-Digital-Imaging

- *Cost-efficient & high-quality DDI*
- *High-power UV-LEDs & DMD-technology*
- *Operator-friendly concept w/compact footprint*
- *Modular machine concept & field upgradable*
- *Flexible number of heads and head distances for shortest exposure times*
- *Multiple wavelength concept*

Vacuum Lamination System

- *Even pressure across the platen surface*
- *State-of-the-art heating system*
- *All-steel construction*
- *Modular machine concept & field upgradable*
- *User-friendly visualization system*
- *Affordable initial investment*

Laser Drilling/Cutting System

- *Microvia drilling 50-300µm diameter*
- *Space saving compact design*
- *Best power/size ratio in market*
- *Pulse Stabilizer*
- *European technology center and USA support center*

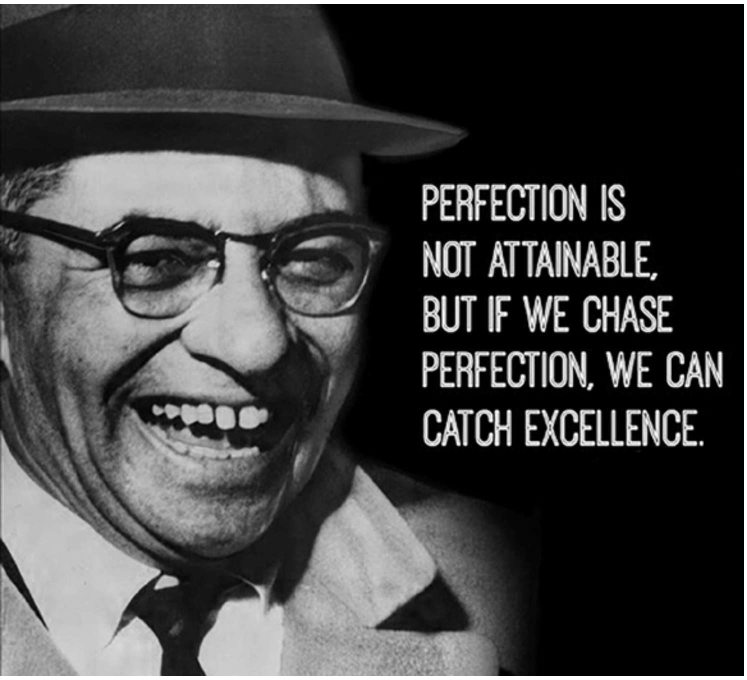
Bürkle North America supplies a full complement of productive equipment for fabricating printed wiring boards. From imaging through metrology inspection, Bürkle North America's equipment offering sets the standard in the industry for Imaging, Registration, Lamination, Mechanical or Laser Drill/Routing and Feature Metrology Inspection.

BÜRKLE
NORTH AMERICA, INC.
11105 Knott Ave., Cypress, CA 90630

Contact Dave Howard
for more details.

BURKLE NORTH AMERICA
25 year
ANNIVERSARY

www.burkleamerica.com **Phone: 714-379-5090**



Vince Lombardi's famous quote.

cause, contrary to popular belief (and with apologies to Philip Crosby), quality is not free! Quality as a way of life may require a fundamental cultural shift in some organizations, and extreme commitment from the person that signs the checks is absolutely the requisite first step down this transformational path.

Current Culture

Changing an organization's culture is one of the toughest tasks that can be undertaken. In most cases, the current culture has formed over many years of interaction between employees at all levels within the organization. Organizational cultures form for a reason, and changing the accepted norm can sometimes feel like swimming upstream. In small, privately owned businesses, the current culture often matches the style and values of the company founder. Organizational culture grows over time and frequently mirrors the prevailing management style. Since managers tend to hire people "just like me," the established culture is carried on through new employees for generations. People are creatures of habit, and basic human nature dictates that they will typically be uncomfortable with change. A key to overcome this is to acknowledge this human nature predisposition and actively manage the change process through the tools and techniques discussed in this column.

Sense of Urgency

For real change to happen, leadership must demonstrate how important the effort is; in other words, they must walk the walk. Dr. Joseph M. Juran recognized this truth over 40 years ago when he said, "Every successful quality revolution has included the participation of upper management. We know of no exceptions." This means a sense of urgency around the need for the culture change must be created that goes far beyond lip service by establishing an open and honest dialog about the importance of the change and how it will benefit all by improving organizational performance. Urgency is contagious; getting people to start talking about the change being proposed will spread like wildfire and feed on itself.

Unfreeze–Change–Refreeze

There are many models for change management, but I prefer the three-stage model developed by physicist and social scientist Kurt Lewin back in the 1940s known as Unfreeze–Change–

Unfreeze–change–refreeze.

Refreeze. This model explains organizational change by using the analogy of changing the shape of a block of ice (Figure 3). Visualizing change as a process with distinct stages will allow organizational leaders to prepare for what is coming and develop a plan to manage the transition. One of the major reasons change fails is that organizations rush into change blindly and without a plan to effectively manage change.

Stage 1: Unfreeze

The critical task during the first stage is to establish an environment that challenges the current norms and set the stage for change to occur. The goal is to move people from the current frozen state to a change-ready or unfrozen state. The unfreeze stage is important to beginning to break down barriers to change and move away from the "we have always done it this way" mentality.

Stage 2: Change

During this transitional stage, people are aware that the old ways are being challenged, but at this point there may be no clear understanding of the new ways which will replace them. Care must be taken to keep fear of the unknown from paralyzing the effort, and strong leadership during this phase will be the difference between success and failure. The goal of this stage is to get, and keep, people in the unfrozen state while establishing new ways and norms. Convincing people that it is okay to break away from traditional norms and adopt new ways of doing things is crucial before moving into the refreeze stage.

Stage 3: Refreeze

This goal of this stage is all about reconnecting people to their new comfort zones and establishing this state as the safe, familiar environment. Refreezing seeks to reinforce the new behaviors and enable permanent change through rewards, celebration of each success, and continuous communication.

Communicate, Communicate, Communicate

When it comes to changing the culture of a company, there is no such thing as over-communication. Involving the entire workforce from day one will be key to successfully change the current culture. Secrecy is the enemy of success; this process calls for full disclosure, employee participation and individual empowerment.

Effective communication will allow the creation of a type of controlled mayhem; in other words, an environment which can build a strong motivation to seek out a new equilibrium. Without this motivation, it will be difficult to get the employee buy-in that will be required to facilitate any meaningful and permanent change. This journey will be an emotional one that will challenge the status quo, oftentimes evoking strong reactions and resistance (remember the human nature thing?). As overused as the word paradigm is, I don't have a better word for describing "the way things are done," and this will be a paradigm shift that can only be overcome with effective, thorough and constant communication.

Change is Constant

The Greek philosopher Heraclitus had it right 2,500 years ago: The only thing constant

"Everything changes and nothing stands still."

— Heraclitus of Ephesus
535 BC – 475 BC

Image source: Wikimedia Commons

with change is change itself. As I mentioned earlier, people are creatures of habit and will be uncomfortable with moving out of their comfort zone. That being said, know that change is difficult and some people will not adapt; this is normal. As with any major cultural change (ownership, management, quality system), the only thing that is certain is that not everyone will "drink the Kool-Aid." And while this can be a trying experience, stay the course. Shifting organizational culture is difficult enough with a willing and able workforce; to be successful it will absolutely have to be an "all in" proposition. **PCB**

Steve Williams is the president of The Right Approach Consulting LLC. To read past columns, or to contact Williams, click here.

Smartphones: A Significant Challenge for Thermal Management Companies

How can smartphones deal with growing heat management challenges? What are the current solutions? Who are involved in this industry? Yole and System Plus Consulting analysts offer you today a snapshot of this promising industry.

"The importance of thermal management in smartphones is due to the growing number of smartphone functionalities and raised customer requirements for processing speed, leading to increased heat dissipation," explains Dr Milan Rosina, senior analyst for Energy Conversion and Emerging Materials at Yole. And he adds: "Additional components needed to ensure new smartphone functions desired by customers, including wireless charging, high-resolution cameras, 3D gaming, security, authentication, and high-speed streaming, also result in denser component integration, making thermal management even more difficult."

Actually, smartphones contain several components that generate heat, and components whose performance and lifetime is negatively impacted by heat. Excessive heating of some components, such as lithium ion batteries, must be carefully handled for safety reasons. The processor is the most important and hottest component in a smartphone.

Amongst the other heat-generating components in a smartphone, Yole's analysts mention image sensors, light sources and batteries. Suitable thermal management solutions are now sought to avoid hot spots in smartphone and keep the component temperature at acceptable levels. The enclosure temperature, or skin temperature, must be also kept relatively low to avoid users feeling uncomfortable when using the smartphone.

According to Yole's analysts, there are different approaches for thermal management, based both on hardware and software solutions. Software thermal management (STM) has several advantages. It enables additional design flexibility and an optimal reaction to a given thermal event and can be improved by a software update in existing products. Contrary to hardware solutions, such as heat pipes, STM does not take additional space in a smartphone.

PCB BUYERS - compare nearly 1900 manufacturers now at *The* PCB List.

Quick Search

Advanced Search

thepcblist

The best way to find a PCB fabricator, *anywhere*.

www.thepcblist.com

Recent Highlights from PCB007

1 New I-Connect007 Team Members Tour American Standard Circuits

Recently, Anaya Vardya, CEO of American Standard Circuits, invited two of I-Connect007's newest team members, IT coordinator Jonathan Zinski and Editor Kiersten Rohde, to tour his facility in West Chicago, Illinois. Happy Holden, resident PCB expert, also joined the newbies on their field trip to ASC. In the following articles, Jonathan and Kiersten describe their experience touring ASC.

2 It's Only Common Sense: A Frustrated PCB Customer Vents

I recently received an e-mail from a good friend at a top mil/aero company. Here's just one snippet: "Why do I have to call and ask, 'Where is my stuff?' If it were just one board shop, I would say good riddance and move on. But it is almost universal among our supplier base that late deliveries only get discovered when we ask…"

3 Rex Rozario's Next Big Thing, Part 1

Barry Matties joins Rex Rozario at Lympstone Manor, one of Rex's investments outside of the electronics industry. The property, once an old country house in the Exeter countryside, has been transformed into a splendid hotel and restaurant.

4 RTW SMTAI: NCAB's Perspectives on HDI and Miniaturization

John Piccirilli of NCAB Group discusses the trends in HDI and miniaturization he's seeing around the world. He also talks about other interesting technologies such as metal-backed PCBs and stretchable materials.

5 All Flex Hires Experienced CFO and Sales Manager

All Flex has hired two key individuals to bolster the management and expansion of its high-performing business areas.

6 Walt Custer's Annual Update from productronica 2017

Industry consultant Walt Custer of Custer Consulting sat down with me at productronica 2017 to once again inform our readers with his yearly update and perspective on what's happening in the industry. And guess what? It's good news!

7 The Right Approach: Steve's Particular Set of Skills (to become a World-Class Quality Manager)

Being a quality professional today is nothing like it was 20 or 25 years ago; on a personal level, I can attest to this fact. It is no longer adequate to appoint a quality manager simply based on a person's command of acceptance criteria and industry specifications; in the 21st century, a truly hybrid executive is needed.

8 Planning a PCB: Signal Integrity and Controlled Impedance Considerations

Knowledge and experience are the two key elements when planning a PCB. Today's PCB designers must have far more knowledge and understanding of the PCB production process than in the past. This is especially important when they plan and how they plan the stackup, via span, routing and power distribution.

9 Punching Out! PCB/PCBA M&A Top 10 FAQs

We frequently talk with owners about the possible sale of their businesses. Here are the top 10 questions asked by PCB/PCBA shop owners about the process.

10 Ladle on Manufacturing: Fabricating for Signal Integrity

Signal integrity! In a world which is increasingly high-speed and digital, the chemical-dependent and mainly analogue-controlled world of PCB manufacturing is not always a comfortable partner.

For the latest PCB news and information, visit: PCB007.com

Pssst! Are You Looking for Someone?

Place your notice in our Help Wanted section.

For just $500, your 200 word, full-column—or, for $250, your 100 word, half-column—ad will appear in the Help Wanted section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs and suppliers.

Potential candidates can click on your ad and submit a résumé directly to the email address you've provided. If you wish to continue beyond the first month, the price is the same per month. No contract required. We even include your logo in the ad, which is great branding!

To get your ad into the next issue, contact:
Barb Hockaday at barb@iconnect007.com or +1.916.608.0660 (-7 GMT)

I-Connect007
GOOD FOR THE INDUSTRY

SMT007 pcb007 pcb design007

SALESPERSON WANTED • MAINLAND SOUTH CHINA

Work where you live!

The I-Connect007 China team is seeking an experienced salesperson to generate and manage a revenue stream for our Chinese publications.

Key Responsibilities include:

- Sell advertising contracts for monthly magazine
- Develop and cultivate new business
- Keep timely and accurate records
- Generate and follow up on all leads
- Manage contract renewals
- Account management: work with local and international team to provide customer support
- Phone and email communications with prospects
- Occasional travel

Qualifications

Successful candidates should possess a university degree or equivalent, experience with managing and cultivating leads, projecting, tracking and reporting revenue. We are looking for positive, high-energy candidates who work well in a self-managed, team-based, virtual environment.

Compensation

This is a base salary-plus-commission position. Compensation commensurate with experience.

Requirements

- Must be located in China Mainland, South China area preferred
- Good command of Chinese language, proficient with English speaking and writing
- Able to follow established systems and learn quickly
- Able to maintain professional external and internal relationships reflecting the company's core values
- 2-5 years' sales experience
- Experience with Microsoft Office products
- Must be highly motivated and target-driven with a proven track record for meeting quotas
- Good prioritizing, time management and organizational skills
- Create and deliver proposals tailored to each prospect's needs
- Experience in the electronics industry desirable

QUALIFIED CANDIDATES: CLICK HERE TO APPLY

I-Connect007
GOOD FOR THE INDUSTRY

I-Connect007 pcb007中国 线上杂志 CHINA MAGAZINE

I-Connect007 pcb007中国 每周通讯 WEEKLY NEWSLETTER

WWW.ICONNECT007CHINA.COM

SAKI
The Future in Focus

Position:
Field Application Engineer

Saki America Inc., headquartered in Fremont, CA, a leader in automated inspection equipment, seeks two full-time Field Application Engineers (FAE), one in the Fremont headquarters and the other for the Eastern and Southern United States.

The FAE will support the VP of Sales and Service for North America in equipment installation, training, maintenance, and other services at field locations. The FAE will provide technical/customer support and maintain positive relationships with existing and future customers.

Strong analytic abilities and problem-solving skills are a must in order to understand customer applications and troubleshoot issues. The FAE will perform demos and presentations for customers and agents as well as assisting in trade show activities. Candidate must have a minimum of a two-year technical degree, experience in AOI, SPI, and X-ray inspection, and strong verbal and written communication skills. The position requires the ability to travel about three weeks per month. Must be a US citizen and be able to lift up to 40 lbs.

apply now

eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor at EPTAC

Job Summary:
We are growing! EPTAC, a leading provider in the electronics training industry is looking for some great people to join our team. If you love teaching people, choosing the classes and times you want to work, and basically being your own boss, this may be the career for you. We are looking for instructors that have a passion for working with people to develop their skills and knowledge. If you have a background in electronics manufacturing and an enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Opportunities available across U.S. and Canada, especially in our growing markets of California, Chicago, Minnesota and New England. Some travel involved. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

Qualifications and Skills:
- A love of teaching and an enthusiasm to help others learn new concepts and skills
- Background in electronics manufacturing
- Previous soldering and/or electronics/cable assembly experience
- Previous IPC Certification a plus, but will certify the right candidate

Benefits:
- Ability to operate from home: no required in-office schedule
- Flexible schedule: control your own time, work as often as you like
- IRA retirement matching contributions after one year of service
- Training and certifications provided and maintained by EPTAC

apply now

ACE
ACCURATE
CIRCUIT
ENGINEERING

PCB Assembly Supervisor—full time
Accurate Circuit Engineering—Santa Ana, CA

Position Summary: Responsible for all assembly processes to ensure continued growth as directed by management.

Essential Job Functions:
- Create, implement, and supervise in-house manufacturing facility
- Recruit, hire, train, and supervise assembly floor personnel
- Extensive hands on experience with all aspects of PCB assembly
- Understanding of IPC-A-610 standards
- Research and acquire additional assembly resources
- Gather data on product shortages, lead times, price changes, etc.
- Coordinate the assembly activities with sales to ensure 100% on-time delivery
- Create, implement, and supervise daily quality processes to ensure 100% accuracy
- Document, monitor and review progress of the business unit
- Respond to internal and external customers in a timely manner
- Coordinate walk-through, site audits, etc.

Qualifications:
- Minimum 3 years as operations supervisor of electronics assembly house
- 5+ years' experience in the electronics industry
- Previous experience as a quality or operations supervisor preferred
- Ability to solve practical problems using pre-established guidelines
- Strong facility in Microsoft Office applications
- Excellent verbal and written communication skills
- Ability to work with people of diverse backgrounds
- Highly organized/excellent time management skills
- Ability to perform at the highest level in a fast-paced environment
- Valid California driver's license.

apply now

ACE
ACCURATE
CIRCUIT
ENGINEERING

PCB Process Planner

Accurate Circuit Engineering (ACE) is an ISO 9001:2000 certified manufacturer of high-quality PCB prototypes and low-volume production for companies who demand the highest quality in the shortest time possible. ACE is seeking a skilled individual to join our team as a PCB process planner.

Responsibilities will include:
- Planning job travelers based on job release, customer purchasing order, drawings and data files and file upon completion
- Contacting customer for any discrepancies found in data during planning and CAM stage
- Consulting with director of engineering regarding technical difficulties raised by particular jobs
- Informing production manager of special material requirements and quick-turn scheduling
- Generating job material requirement slip and verify with shear clerk materials availability
- Maintaining and updating customer revisions of specifications, drawings, etc.
- Acting as point of contact for customer technical inquiries

Candidate should have knowledge of PCB specifications and fabrication techniques. They should also possess good communication and interpersonal skills for interfacing with customers. Math and technical skills are a must as well as the ability to use office equipment including computers, printers, scanners, etc.

This position requires 3 years of experience in PCB planning and a high school level or higher education.

apply now

CHEMCUT

BOUNDLESS INNOVATION | UNBEATABLE PRECISION

Chemical Process Engineer

Chemcut, a leading manufacturer of wet-processing equipment for the manufacture of printed circuit boards for more than 60 years, is seeking a Chemical Process Engineer. This position is located at Chemcut's main facility in State College, Pennsylvania. Applicants should have an associate degree or trade school degree, or 4 years equivalent in chemical process engineering.

Job Responsibilities Include:
- Developing new industrial processes
- Providing process criteria for both new equipment and modifying existing equipment
- Testing new processes and equipment
- Collecting data required to make improvements and modifications
- Assisting in investigating and troubleshooting customer process problems
- Ensuring that equipment works to its specification and to appropriate capacities
- Assessing safety and environmental issues
- Coordinating with installation/project engineers
- Ensuring safe working conditions and compliance with health and safety legislation

Key Skills:
- Aptitude for, and interest in chemistry, IT and numeracy
- Analytical thinking
- Commercial awareness
- Ability to perform under pressure
- Communication and teamwork
- Problem-solving

Experience with circuit board processes is a plus.

Contact Arlene at 814-272-2800 or by clicking below.

apply now

CHEMCUT

BOUNDLESS INNOVATION | UNBEATABLE PRECISION

Field Service Technician

Chemcut, a leading manufacturer of wet-processing equipment for the manufacture of printed circuit boards for more than 60 years, is seeking a high-quality field service technician. This position will require extensive travel, including overseas.

Job responsibilities include:
- Installing and testing Chemcut equipment at the customer's location
- Training customers for proper operation and maintenance
- Providing technical support for problems by diagnosing and repairing mechanical and electrical malfunctions
- Filling out and submitting service call paperwork completely, accurately and in a timely fashion
- Preparing quotes to modify, rebuild, and/or repair Chemcut equipment

Requirements:
- Associates degree or trade school degree, or four years equivalent HVAC/industrial equipment technical experience
- Strong mechanical aptitude and electrical knowledge, along with the ability to troubleshoot PLC control
- Experience with single and three-phase power, low-voltage control circuits and knowledge of AC and DC drives are desirable extra skills

To apply for this position, please apply to Mike Burke, or call 814-272-2800.

apply now

orbotech

Electronics Expert Engineer

Orbotech is looking for an Electronics Expert Engineer to handle various hardware activities, including communication, data path processing, device interfaces and motion, as well as system supporting functions in a multi-disciplinary environment.

What Will Your Job Look Like?

- Providing cutting edge hardware solutions for challenging product line needs
- Developing board design and Logic in VHDL
- Defining and managing interfaces (software, algorithm, mechanics and electricity)
- Successfully integrating hardware with other product disciplines
- Supporting the product needs during and following release

What Do You Need to Succeed?

- BSc in electronics engineering
- At least 5 years of R&D experience in complex board design, mainly FPGA (communication interfaces, DDR controller, algorithm implementation)
- Experience in an Altera/Xilinx development environment
- Experience in ECAD design tools (DxDsigner, ModelSim) is an advantage
- Knowledge in laser interfaces, RF and analog is an advantage

Who We Are

Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

orbotech

Electronics Team Leader

Orbotech is seeking an Electronics Team Leader to join our electronics team, which develops multi-disciplinary systems, including vision/laser, image processing, and control and automation missions.

What Will Your Job Look Like?

- Lead a team of electronics engineers in a multi-disciplinary environment
- Lead electronic activities from requirement phase to development, integration and transfer, to production
- Be the focal point for other disciplines and projects managers
- Maintain and improve existing electronics platforms

What Do You Need to Succeed?

- BSc/MSc in electronic engineering/ computer science from a well-recognized university
- 5+ years' experience in digital board design, high-speed links, computing embedded systems, and HW/SW integration
- 2-3 years' experience in leading a team of engineers
- Solid skills in complex FPGA design with multi-modules
- Solid skills in high-speed board design, DDR3/4, PCIE, USB, IO, and optic links
- Ability to design and execute end-to-end solutions

Who We Are

Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting-edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

INDIUM

Technical Content Specialist

Indium Corporation is seeking a technical content specialist to guide the development of data-rich, high-level content for the company's semiconductor and advanced assembly materials (SAAM) sales and technical literature. The technical content specialist will work with multiple departments to ensure that all externally-facing technical and sales collateral and internal training materials are consistent in format and of superior quality.

The technical content specialist will:

- Assist in the development of key content and ensure consistency of message and format across platforms
- Develop a technically-detailed understanding of Indium Corporation materials and offerings to the SAAM industry
- Curate a library of technical conference papers and associated materials, including content related to Indium Corporation materials and their performance
- Assist in the development of, and ensure consistency for SAAM promotional materials, such as product datasheets (PDS), images, brochures, whitepapers and presentations (technical and sales)
- Attend at least one technical conference and its paper session per year

Requirements:

- Technical undergraduate degree (BS in Chemistry/Physics/Metallurgy/Materials Science or Engineering discipline)
- 5 years of work experience in semiconductor assembly or advanced electronics assembly
- Excellent written and spoken English language skills; fluency in Chinese desirable
- Proven ability to work independently with verbal or written instructions

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

Ventec Seeking U.S. Product Manager for tec-speed

Want to work for a globally successful and growing company and help drive that success? As a U.S.-based member of the product and sales team, your focus will be on Ventec's signal integrity materials, tec-speed, one of the most comprehensive range of products in high-speed/low-loss PCB material technology for high reliability and high-speed computing and storage applications. Combining your strong technical PCB manufacturing and design knowledge with commercial acumen, you will offer North American customers (OEMs, buyers, designers, reliability engineers and the people that liaise directly with the PCB manufacturers) advice and solutions for optimum performance, quality and cost.

Skills and abilities required:

- Technical background in PCB manufacturing/design
- Solid understanding of signal integrity solutions
- Direct sales knowledge and skills
- Excellent oral and written communication skills in English
- Experience in making compelling presentations to small and large audiences
- Proven relationship building skills with partners and virtual teams

This is a fantastic opportunity to become part of a leading brand and team, with excellent benefits.

Please forward your resume to
jpattie@ventec-usa.com and mention
"U.S. Sales Manager—tec-speed"
in the subject line.

www.venteclaminates.com

apply now

BLACKFOX
Premier Training & Certification

IPC Master Instructor

This position is responsible for IPC and skill-based instruction and certification at the training center as well as training events as assigned by company's sales/operations VP. This position may be part-time, full-time, and/or an independent contractor, depending upon the demand and the individual's situation. Must have the ability to work with little or no supervision and make appropriate and professional decisions. Candidate must have the ability to collaborate with the client managers to continually enhance the training program. Position is responsible for validating the program value and its overall success. Candidate will be trained/certified and recognized by IPC as a Master Instructor. Position requires the input and management of the training records. Will require some travel to client's facilities and other training centers.

For more information, click below.

apply now

Prototron Circuits

Experienced PCB Sales Professional

With more than 30 years of experience, Prototron Circuits is an industry leader in the fabrication of high-technology, quick-turn printed circuits boards. Prototron of Redmond, Washington, and Tucson, Arizona are looking for an experienced sales professional to handle their upper Midwest Region. This is a direct position replacing the current salesperson who is retiring after spending ten years with the company establishing this territory.

The right person will be responsible for all sales efforts in this territory including prospecting, lead generation, acquiring new customers, retention, and growth of current customers.

This is an excellent opportunity for the right candidate. Very competitive compensation and benefits package available.

For more information, please contact Russ Adams at 425-823-7000, or email your resume.

apply now

Process Engineer (Redmond, Washington)

With more than 30 years of experience, Prototron Circuits is an industry leader in the fabrication of high-technology, quick-turn printed circuits boards. We are looking for an experienced PCB process engineer to join the team in our Redmond, Washington facility. Our current customer base is made up of forward-thinking companies that are making products that will change the world, and we need the right person to help us make a difference and bring these products to life. If you are passionate about technology and the future and believe you have the skills to fulfill this position, please contact Kirk Williams at 425-823-7000 or email your resume.

apply now

orbotech

FPGA Design Expert

Orbotech is seeking a FPGA Design Expert to join our electronics team, which develops multi-disciplinary systems including vision/laser, image processing and electro-optics.

What Will Your Job Look Like?
- Lead image acquisition and processing activities in the team
- Engage in all aspects of FPGA design activity: requirement phase, coding, synthesizing, verification support and LAB bring up
- Participate in system definitions for current and next generation products
- Collaborate with other teams: SW, algorithm and QA

What Do You Need to Succeed?
- BSc/MSc in Electrical Engineering from a well-recognized university
- Extensive knowledge of VHDL
- 5+ years of FPGA development experience (requirement, architecture, RTL coding, simulation, synthesis, timing analysis, P&R, board level integration and verification)
- Experience in designing and implementing low-latency, high-throughput FPGA designs utilizing PCIe Gen2/3, Gigabit Ethernet, SERDES, DDR3/4
- Experience in complex FPGA such as Altera Stratix-II and Arria 5&10 devices
- Authoring documentation experience such as FPGA specifications and FPGA verification plans

Who We Are
Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting-edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

Altium

Application Engineer

The application engineer is the first contact for our customers who have technical questions or issues with our product. We value our customers and wish to provide them with highest quality of technical support.

Key Responsibilities:
- Support customer base through a variety of mediums
- Log, troubleshoot, and provide overall escalation management and technical solutions
- Create various types of topic based content, such as online help, online user guides, video tutorials, knowledge base articles, quick start guides and more
- Distill complex technical information into actionable knowledge that users can understand and apply
- Continually develop and maintain product knowledge

Requirements:
- Understanding of EDA electronic design software, schematic capture and PCB layout software
- Bachelor's degree in electronics engineering or equivalent experience
- Sales engineering and/or support engineering experience
- Circuit simulation and/or signal integrity experience
- Understanding of ECAD/ MCAD market segments
- Understanding of micro controllers, SoC architecture and embedded systems market
- Database experience preferred (i.e., MySQL, PostgreSQL, Microsoft Access, SQL, Server, FileMaker, Oracle, Sybase, dBASE, Clipper, FoxPro) etc.
- Experience with PLM/PDM/MRP/ERP software (Program Lifecycle Management) preferred
- Salesforce experience a plus

Salary based upon experience. Comprehensive benefits package and 401k plan. Openings in USA, UK, and Germany.

For more information, contact Altium.

apply now

MacDermid
PERFORMANCE SOLUTIONS

Do you have what it takes?

MacDermid Performance Solutions, a Platform Specialty Products Company, and daughter companies manufacture a broad range of specialty chemicals and materials which are used in multi-step technological processes that enhance the products people use every day. Our innovative materials and processes are creating more opportunities and efficiencies for companies across key industries – including electronics, graphic arts, metal & plastic plating, and offshore oil production. Driving sustainable success for companies around the world, and at every step of the supply chain, takes talent. Strategic thinking. Collaboration. Execution.

The people of MacDermid Performance Solutions stand united by a guiding principle: If it doesn't add value, don't do it. This belief inspires a unique culture where each team member has opportunities to imagine, create, hone and optimize. Do you have what it takes? Join our growing team of over 4,000 professionals across more than 50 countries with openings in research, finance, customer service, production and more.

MacDermid Performance Solutions and its affiliates are Equal Opportunity/ Affirmative Action Employers.

apply now

NCAB GROUP
Integrated PCB Production

Outside Sales/ Key Account Managers

NCAB Group USA is adding to our existing outside sales team in various U.S. locations:

- Ontario, California
- Itasca, Illinois
- Vancouver, Washington

This is a sales position that requires the ability to convert those cold calls into high-value customer meetings. What we are looking for:

- A "hunter" mentality
- The ability to create solid customer relationships
- A desire to excel and not settle for mediocrity
- 5+ years of experience in the PCB or semiconductor industry
- An excellent ability to present a product and do the "deep dive" during customer visits by asking open ended questions and identifying customer pain points
- The energy to move from prospecting to cold calls to getting the win
- Knowledge of "SPIN" selling
- A college degree
- Willingness to travel, domestically and globally
- U.S. citizens with a valid U.S. passport

Interested? Send your resume.

apply now

Visit us at www.NCABGroup.com

ARLON

Arlon EMD, located in Rancho Cucamonga, California is currently interviewing candidates for **manufacturing** and **management positions.** All interested candidates should contact Arlon's HR department at 909-987-9533 or fax resumes to 866-812-5847.

Arlon is a major manufacturer of specialty high performance laminate and prepreg materials for use in a wide variety of PCB (printed circuit board) applications. Arlon specializes in thermoset resin technology including polyimide, high Tg multifunctional epoxy, and low loss thermoset laminate and prepreg systems. These resin systems are available on a variety of substrates, including woven glass and non-woven aramid. Typical applications for these materials include advanced commercial and military electronics such as avionics, semiconductor testing, heat sink bonding, high density interconnect (HDI) and microvia PCBs (i.e., in mobile communication products).

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2008 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customer's requirements.

more details

YOUR JOB AD HERE

For information, please contact:
BARB HOCKADAY
barb@iconnect007.com
+1 916.365.1727 (PACFIC)

I-Connect007
GOOD FOR THE INDUSTRY

thepcblist

THE **best way to find a PCB fabricator, anywhere.**

PRINTED CIRCUITS LLC.

Minneapolis, Minnesota, U.S.A.

Website | Contact Me | RFQ

Printed Circuits

THE FUTURE OF RIGID FLEX CIRCUITRY

Overview | Contact | Specs | About | Videos | Photos | Brochures | News

Printed Circuits LLC. specializes in manufacturing high reliability rigid-flex and multilayer flex boards for medical, military and other critical electronics packaging applications.

We have the expertise to bring these components together, and to help companies bring their advanced designs to market. Both our customer base and the markets that we serve have expanded. Rigid flex is becoming a popular solution for difficult packaging solutions, due to size, weight and reliability constraints.

Markets: Communication, Computers, Industrial, Medical, Military/Aerospace

Board Types: Flex, Rigid-Flex

Mfg Volumes: Prototype, Small, Medium, La e

Specialties: Blind/buried vias, Controlled I da Embedded Components, HDI, Se lamination

O 9001, ITAR registered, MIL-PRF- L-P-50884, ROHS compliant, UL, Oth 6013 Class 3

Featured Showcase

Click here to see a demo

Why YOU should Showcase:

- Capabilities listing for advanced search functionality
- Specialties and certifications listing
- Ability to upload brochures and videos
- Quick and easy "Contact" and "RFQ" buttons
- News, web and contact links

Click to see a partial list of registered OEMs!

www.thepcblist.com

Events

IPC Calendar of Events, click here

SMTA Calendar of Events, click here

iNEMI Calendar of Events, click here

PCB007 Calendar of Events, click here

HKPCA/IPC International Printed Circuit & South China Fair
December 6–8, 2017
Shenzhen, China

47th NEPCON JAPAN
January 17–19, 2018
Tokyo Big Sight, Japan

DesignCon 2018
January 30–February 1, 2018
Santa Clara, California, USA

EIPC 2018 Winter Conference
February 1–2, 2018
Lyon, France

MD&M West
February 6–8, 2018
Anaheim, California, USA

2018FLEX
February 12–15, 2018
Monterery, California, USA

IPC APEX EXPO 2018 Conference and Exhibition
February 27–March 1, 2018
San Diego, California, USA

China International PCB and Assembly Show (CPCA)
March 20–22, 2018
Shanghai, China

KPCA Show 2018
April 24–26, 2018
Kintex, South Korea

Thailand PCB Expo 2018
May 10–12, 2018
Bangkok, Thailand

Medical Electronics Symposium 2018
May 16–18, 2018
Dallas, Texas, USA

2018 EIPC's 50 Years Anniversary Conference
May 31–June 1, 2018
Bonn, Germany

JPCA show 2018
June 6–8, 2018
Tokyo, Japan

PUBLISHER: **BARRY MATTIES**
barry@iconnect007.com

SALES: **ANGELA ALEXANDER**
(408) 489-8389; angela@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@iconnect007.com

EDITORIAL:
MANAGING EDITOR: **PATRICIA GOLDMAN**
(724) 299-8633; patty@iconnect007.com

TECHNICAL EDITOR: **PETE STARKEY**
+44 (0) 1455 293333; pete@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **HAPPY HOLDEN**
(616) 741-9213; happy@iconnect007.com

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **SHELLY STEIN**

MAGAZINE LAYOUT: **RON MEOGROSSI**

AD DESIGN: **MIKE RADOGNA, SHELLY STEIN, TOBEY MARSICOVETERE**

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **BRYSON MATTIES**

I-Connect007 Presents

the pcb magazine

The PCB Magazine® is published by BR Publishing, Inc., PO Box 1908, Rohnert Park, CA 94927

©2017 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

December 2017, Volume 7, Number 12 • The PCB Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to *The PCB Magazine:*

JANUARY 2018
Equipment:
How to choose for current and future needs.
BONUS: IPC APEX EXPO 2018 Pre-Show Section

FEBRUARY 2018
Knowing your Customer:
External and Internal!

MARCH 2018
New Technologies:
What's around the corner?

I-Connect007

GOOD FOR THE INDUSTRY

myiconnect007.com

mediakit.iconnect007.com

EDITORIAL CONTACT

Patty Goldman
patty@iconnect007.com
+1 724.299.8633 GMT-4

SALES CONTACT

Barb Hockaday
barb@iconnect007.com
+1 916 365-1727 GMT-7

www.iconnect007.com