PCB007 PRESENTS

November 2014

the pcb magazine

AN I-CONNECT007 PUBLICATION

MEDICAL ELECTRONICS

Noble Metal PCB Manufacturing for Direct Implants

by Daniel Schulze, René Tölke and Ilse Widmann, page 12

NOVEMBER FEATURED CONTENT

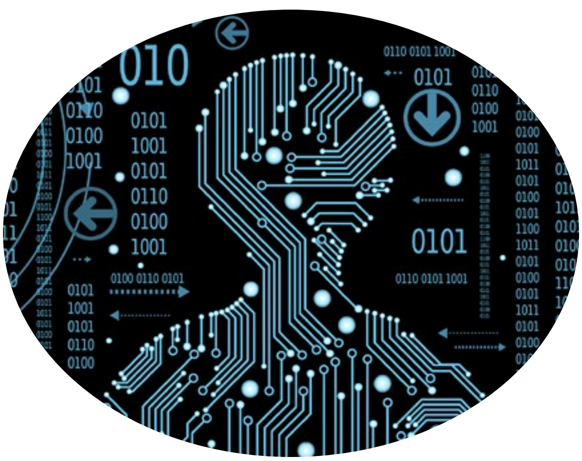
PCB007 PRESENTS

the pcb magazine

AN I-CONNECT007 PUBLICATION

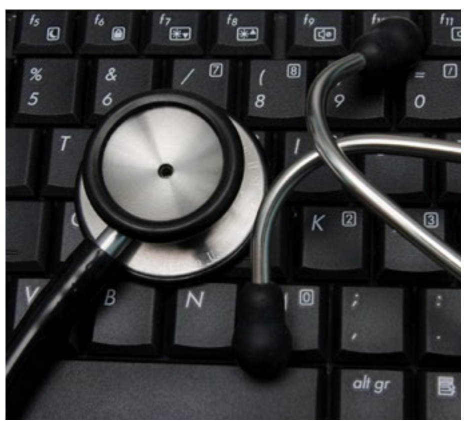
PCB Applications for Medical Electronics

This month, industry experts from Starfish Medical, DYCONEX AG, Micro Systems Technologies, and Nypro are on hand to discuss the latest breakthroughs in medical applications for PCBs, including an in-depth perspective from a medical PCB designer.



FEATURE COLUMN

FEATURE SHORT

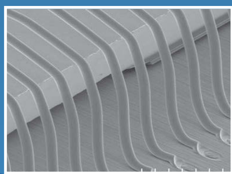


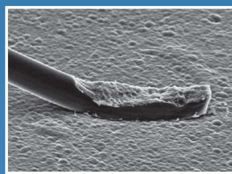
Final Finishing

Universal Finish Solderbond (ENEPIG) – NiPdAu for Soldering and Au-Wire Bond

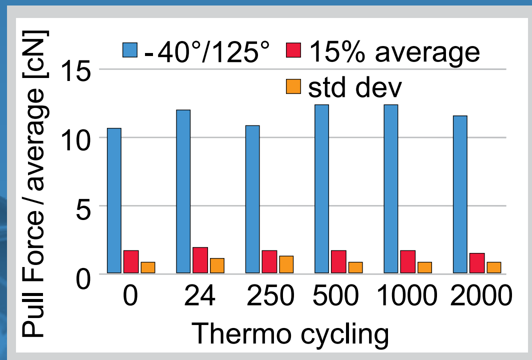
On the same surface

Soldering


Au-Wire


Al-Wire

SolderBond Gold Wire Pull Test after Thermocycling



Pure Pd

Qualified for Aerospace & Satellite Technology

Gold Wire Bond Finish

Process Flow

Cleaner
Pro Select SF

Micro Etch
Micro Etch SF

Dip
Aurodip (Option)

Predip
Pre Dip

Activator
Aurotech Activator 1000

Fine Line Post Dip
Aurotech FL Plus Post Dip (Option)

Electroless Ni
Aurotech CNN mod.

Electroless Pd
PD-Tech

Immersion Au
Aurotech DC 10

Features and Benefits

- Pure Pd-layer without phosphorous
- Softer than phosphorous containing Pd-layers
- Green benefit: EDTA-free process
- Au-wire bondable
- Excellent solder joint reliability
- Good thermo cycling performance

	NiPdAu (ENEPIG)
Deposit Structure	
Electroless Nickel	4.0 µm - 6.0 µm
Electroless Palladium	0.05 µm - 0.3 µm
Immersion Au	0.02 µm - 0.06 µm
Functionality	
Soldering	
Eutectic Sn/Pb	Good
Sn-Ag-Cu Alloy	Excellent
Contact Switching	Yes
Au Wire Bonding	Yes

Atotech Headquarters
P.O. Box 21 07 80 · 10507 Berlin · Germany · www.atotech.com
Tel. +49 30-349 85-0 · Fax +49 30-349 85-777

ATOTECH

PCB007 PRESENTS

NOVEMBER 2014
VOLUME 4
NUMBER 11

THE DEFINITIVE INTERACTIVE MAGAZINE DEDICATED TO THE GLOBAL PCB INDUSTRY

thepcbmagazine.com

the pcb magazine

AN I-CONNECT007 PUBLICATION

MEDICAL ELECTRONICS

IPC Seal of Approval

by Ray Rasmussen
I-CONNECT007

What if we could get the military, aerospace, medical, automotive, and a few other industry groups to agree to accept one site audit and one certification? In other words, one audit, once a year, which covers everything.

Already underway, IPC has ventured into the area of PCB fabricator and EMS provider audits based on IPC-1071 for fabricators and IPC J-STD-001 and IPC-A-610 for PCBA. Called IPC Validation Services, this effort is being driven by IPC's OEM members who want to replace for individual site audits with a single IPC audit as the standard for vendor acceptance.

What Does Compliance Cost?

Randy Cherry is leading this effort at IPC. It's a good first step but it doesn't go far enough IMHO (my kids taught me that). We have an opportunity to remove a huge source of pain and suffering for many companies on both sides of the transaction. These site audits are time-consuming and expensive. After querying, I found one medium-sized EMS company's costs of compliance, including audits, translated to about 0.5% of sales. I'm sure that with some economies of scale, that number would decrease, but it's still a significant number for most companies and another non-value-added expense to doing business. EMS industry wide, that starts to add up. At about $500 billion in sales, the EMS industry's cost of compliance is approximately $1–3 billion annually. If the same number holds for PCB fabs, then we're looking at another $100–300 million. The industry spends a ton of money on this! Get this down to a single audit and the OEM wins, EMS/PCB fabs win, and if IPC does the job, they win as well. Seems like a no-brainer. Here's a list of

likely target certifications which could be addressed with this effort:

- ISO 9001 (Quality)
- ISO 14001 (Environmental)
- ISO/TS 16949 (Automotive)
- ISO 13485 (Medical)
- AS 9100 (Aerospace)
- ANSI/ASQ Q9001
- ANSI/ASQ Q9003
- DSCC (Defense Supply Center Columbus)/ DLA (Defense Logistics Agency)
- NADCAP (list specifications and/or audit type)
- UL
- ITAR (Registered #)
- OHS18001 (Occupational Health & Safety)
- IRIS (International Railway Industry System)
- MIL PRF 55110

I'm sure there are more we could add to this list and some that shouldn't be on it. In any case, you get where I'm going with this.

It Will Take a While

I figure we can make a10-year plan to address the consolidation of all these different efforts. If we start off with a more comprehensive and complete audit with ongoing product and site reviews than what the current agencies do themselves, it might be an easy call for them to sign on. We should be so much better (and cheaper, for them), which will "pressurize" them (I love that word) to join up.

We just need to go slow at first, proving IPC's capabilities to do this properly, one step at a time. The IPC board would have to commit serious resources to this effort and think long-term. There will be a payback for the IPC, but it may take a couple years of investment to ensure this is done properly. A half-assed effort will just add another layer of audits to the industry, which won't do anyone any good. Theoretically and technically, it doesn't seem to be something outside the capability of IPC. They know the industry inside and out. Politically, we may find some roadblocks, as all these certification groups fight to protect their turf. That's why we need to go slow and methodically, based on a long-term plan designed by the stakeholders.

"The IPC board would have to commit serious resources to this effort and think long-term. There will be a payback for the IPC, but it may take a couple years of investment to ensure this is done properly."

Making this Painless

In thinking this through a bit, I believe one of the best ideas is to make this IPC audit and *seal of approval* a no-brainer for companies, sort of like the Malcolm Baldrige National Quality Award. Most companies that go down this path get more out than they put in, meaning this: The improvements they make as a result far outweigh the cost. That's the way this should be. The assessment should provide objective feedback, at some level, to those being audited. A CEO should welcome the audits each year, which should point out problems and possible solutions, provide benchmarking data and a corrective action schedule. It shouldn't be punitive, but rather, constructive. They should know that if they stumble, IPC will work to help them come into compliance. It certainly has to be IPC's goal to get as many members into compliance as possible. And since companies have to be members to be audited and OEMs have to be members to get access to the audit company's information, IPC membership grows as well.

Different Levels

At first, I thought that it might be nice to have two or three different levels: bronze, silver and gold. Then, I reconsidered. What is needed is one comprehensive audit allowing all companies who pass to build any type of product. It should be so good that in order to sell PCBs to any serious OEM, you need IPC's seal of approval. This does a couple things. It ensures that IPC-certified member companies are recognized worldwide as the best of the best. And, it enhances IPC's importance in the eyes of the OEM community, including the military and governments around the world. It sets the standard.

Some Precedence

In 1999, PCB007 was a business built around transaction management for the PCB industry, which I was part of. PCB007 had a similar business model: to audit fabricators worldwide to a standard that would eliminate the need of OEM audits. Both the OEMs and the fabricators loved the idea of single audits to replace the expense of continually auditing or being audited. In fact, many of the very large OEMs we visited hadn't audited their PCB supplier in years. The idea of an annual audit at no cost to them was intriguing as long as we were looking at the right stuff—and we were. At the time, we had Rob Scott, a longtime PCB fabricator, conducting the audits. He knew much more about PCBs than the OEM auditors did. He would see "behind the veil." They liked that.

The transaction side of that business became a casualty of the dotcom bust, but the news site continued on, when my partner and I purchased the assets of the business in 2004.

Conclusion

At this point, I don't see a downside. Sure, there are quite a few potential roadblocks, but IPC has the clout to pull this off. If the IPC board can see the bigger picture and make a long-term investment in this effort, the industry wins. **PCB**

Ray Rasmussen is the publisher and chief editor for I-Connect007 Publications. He has worked in the industry since 1978 and is the former publisher and chief editor of *CircuiTree Magazine.* To read past columns, or to contact Rasmussen, click here.

VIDEO INTERVIEW

Imagineering's Dhanji on Customer-Driven Technology

by Real Time with...SMTAI 2014

Khurrum Dhanji, COO of Imagineering Inc., joins Publisher Ray Rasmussen to discuss the full range of his company's technology offerings—all of which are customer driven.

CLICK TO VIEW

realtimewith.com

Noble Metal PCB Manufacturing for Direct Implants

by Daniel Schulze and René Tölke,
DYCONEX AG,
Ilse Widmann,
MICRO SYSTEMS TECHNOLOGIES (MST)

The market of active implantable devices requires new strategies in designing and manufacturing the relevant components (e.g., the electronic modules). The request for continuously smaller devices to achieve improved patient comfort with even higher functionalities asks as well for further miniaturization on PCBs (Figure 1). Combining existing PCB manufacturing technology with technologies from the thin film industries allows integration of new functionalities while reducing the footprint and the number of components.

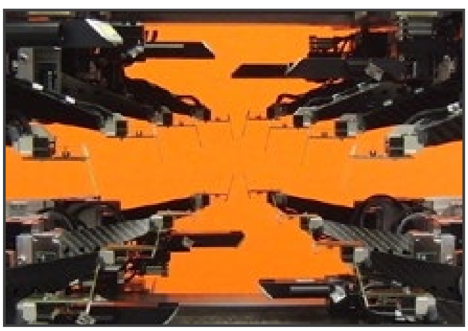
DYCONEX AG has demonstrated the use of PCBs in in-vivo applications without additional sealing or housing by applying noble metals as conductors on biocompatible PCB materials. For short-term implants the use of gold-plated copper traces has proven good results in several clinical studies. For long-term implants the market requests completely copper-free structures (Figure 2).

These copper-free substrates have been manufactured by combining standard PCB processes with thin film deposition methods as used in the MEMS industry. Using thin film technologies, resolutions within the nanometer range can be achieved by combining the know-how between standard PCB and semiconductor industry. As an example a typical high-density

Unrivaled Speed.

A8

Introducing the atg A8-16 with 16 test probes, 8 XGA color cameras, and an unrivaled test speed of up to 275 measurements per second.



Basic specification	16 test probes, 8 XGA color cameras
Test area	610 mm x 620 mm
Smallest test point	25 μm (*with micro needle probes)
Repeatable accuracy	+/- 4 μm
Test voltage	up to 1000 Volts
4-wire Kelvin measurement	0,25 mΩΩ - 1 kΩ (± 0,1 mΩΩ ± 2

A8-16 Video

Get more info

atg

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

LUTHER & MAELZER®

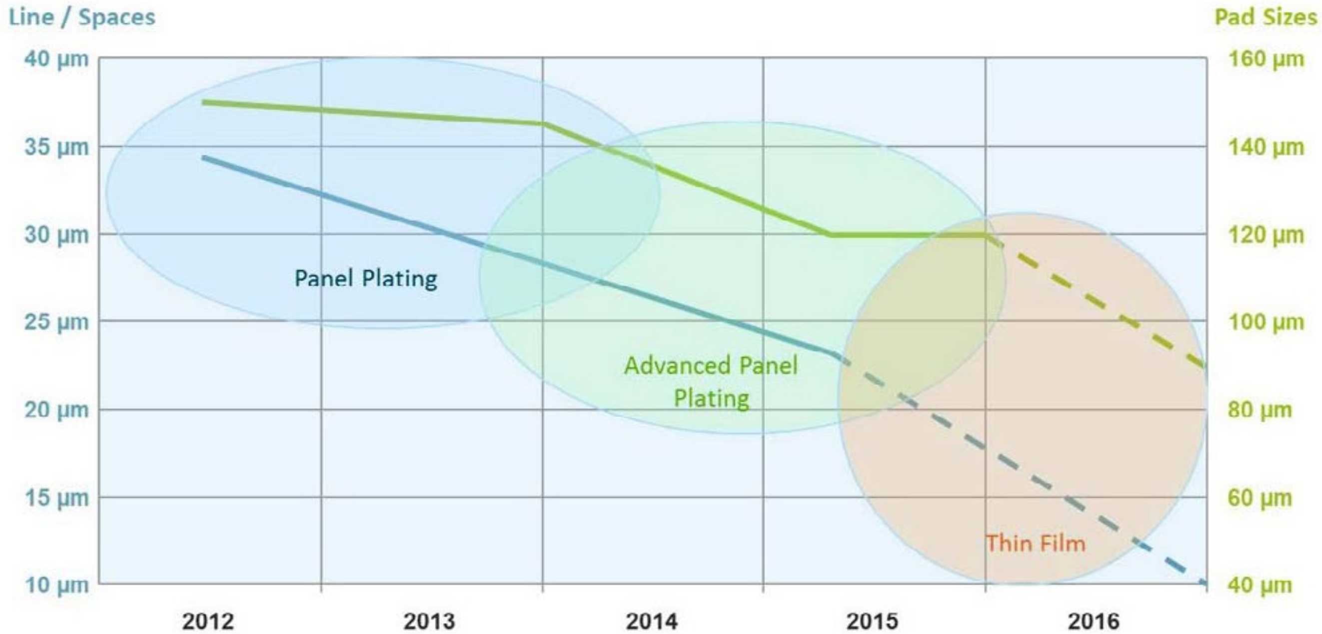


Figure 1: Decrease of the lines, spaces, and pad sizes within the last few years.

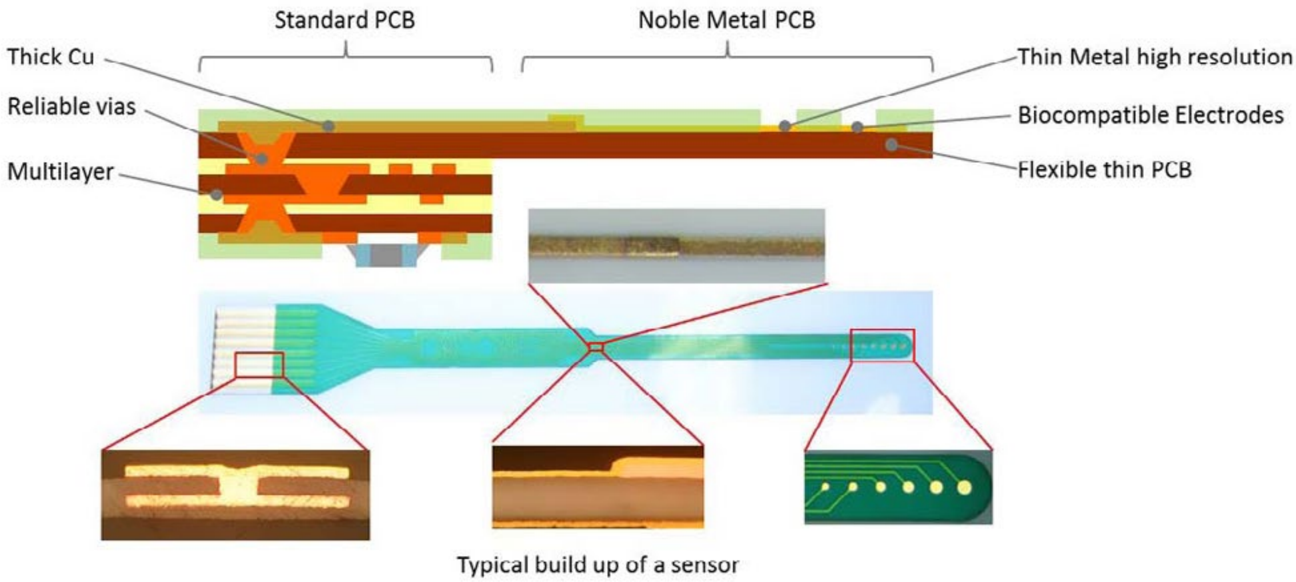
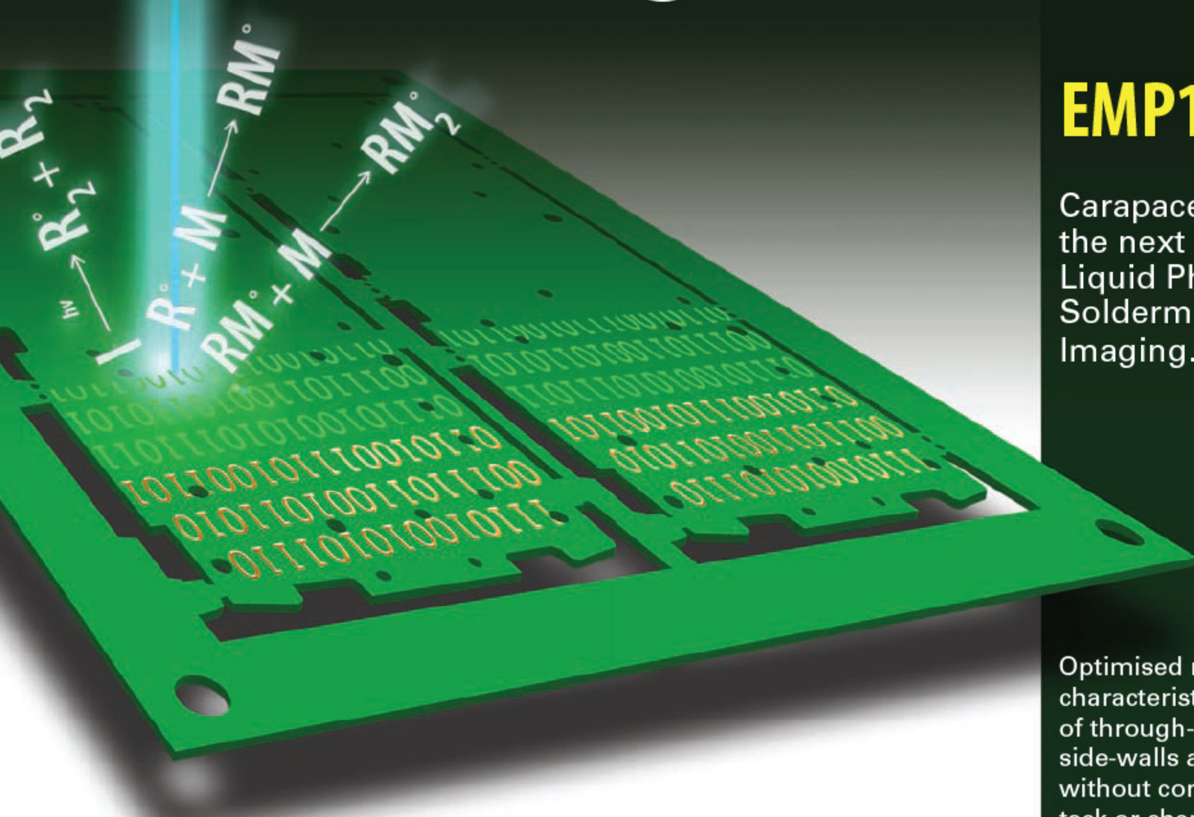


Figure 2: A hybrid PCB combining standard PCB with noble thin film material technology.

PCB has lines and spaces down to 25 µm for signal paths in compare to an ASIC within a 40 nm range. However, the fabrication size in the semiconductor industry is restricted to 12”, while the PCB world manufactures in 12 x 18” large panel sizes. The substrates were structured using standard PCB surface cleaning, activation, photolithography and chemical wet-etching technologies. For the noble metals (Au and Pt), gas phase deposition methods were applied. The material is heated within a high vacuum chamber up to its boiling point. Clusters of atoms evaporate and deposit on the substrate forming a noble metal layer structure. If neces-

ELECTRA 30th
INNOVATION SINCE 1984

Direct Image Solder Mask



EMP110 DI

Carapace® EMP110 DI is the next generation in Liquid Photoimageable Soldermask for Direct Imaging.

Optimised radiation curing characteristics deliver high levels of through-cure and straight side-walls at low energy levels without compromise in surface tack or chemical resistance.

- **Suitable for Laser and LED Direct Image Systems**
- **Versions available for 355nm up to 405nm light sources**
- **Low Exposure Energy (down to 50-80 mJcm^{-2})**
- **Fine Solder-dam Resolution (50 micron, 2mil)**
- **High Chemical Resistance to ENIG & Sn Processes**
- **Screen-print and Spray**
- **Halogen-free, RoHS and fully REACH compliant with no TGIC or other SVHC content.**

www.electrapolymers.com

Figure 3: Assembling of different Pi layer made by adhesives (left); several LCP layer thermally bonded (right).

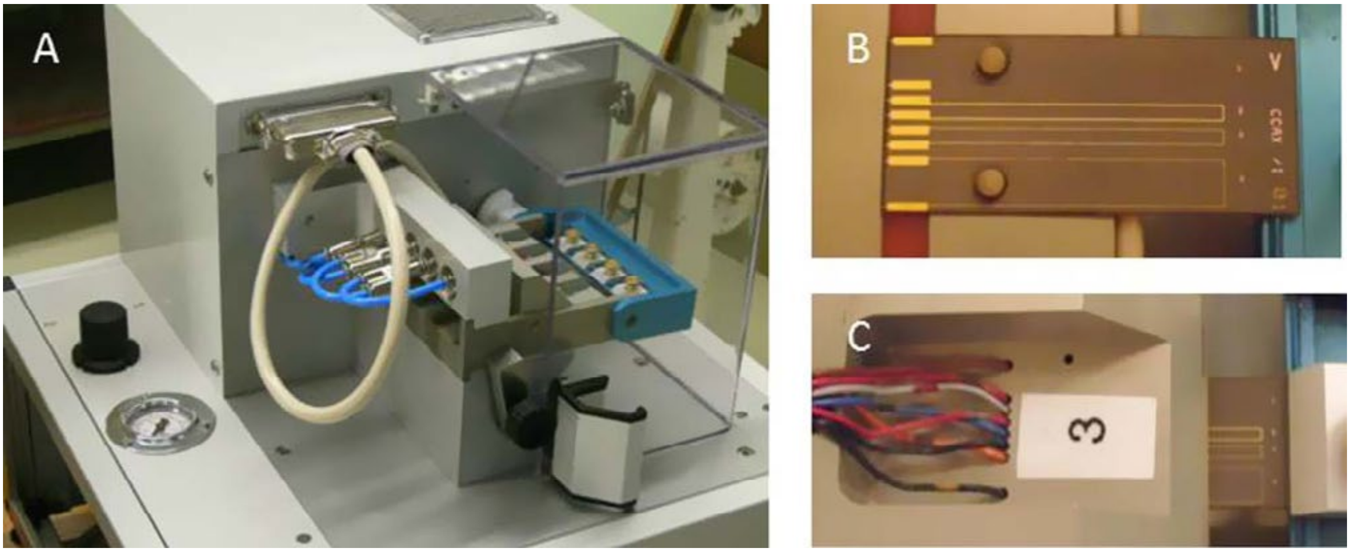


Figure 4: (a) Overview of the bending test machine; (b) test structure; (C) structure under bending including electrical testing.

sary, the noble metal thickness can be increased to several micrometers by electroplating according to the desired properties (ohmic resistance or conductivity). The desired feature sizes and fine lines and spaces (< 25 µm) were achieved by implementing novel technologies like laser direct imaging and drilling with corresponding step and repeat registration as well as new auxiliary materials for thin films.

As a base several bio-inert substrate materials are available. For flexible applications polyimide or LCP (liquid crystal polymer), for rigid applications glass, PEEK or a flexible material with a rigid stiffener can be used. Especially capable are the properties of LCP, a very flexible thermoplastic base material with its biocompatible nature. LCP shows very low water absorption (0.04%) in comparison to standard acrylic adhesives (8%), a temperature stability of up to 190°C (Tg > 280°C, Td > 320°C), excellent high-frequency properties ($\varepsilon R = 2.9$, $\tan \theta = 0.0025$) and low weight (3.2 g/cm^3). For multilayer applications no glue is needed due to its thermoplastic properties (Figure 3).

For applying the pure noble metals on the dielectric base material adhesion is a key feature. To achieve good adhesion adequate surface preparation has to be chosen or an additional adhesion promoter has to be applied. The manufactured PCBs have passed the tape test in accordance with IPC-TM-650 2.4.1. Furthermore the noble metal substrates have undergone a bending test by using a test coupon built up as a resistance loop (Figure 4).

The traces are placed on a 100 µm single sided polyimide. Cracks in the noble metal traces are detected on the first occurrence of resistance changes.

Bend R = 1 mm	**Trace width**			
Au metal thickness: 0.5 µm	20 µm	50 µm	60 µm	100 µm
Cycles [k = 1000]	N/A	2.5k	6k – 12k	> 40k

Table 1: Examples of bending test results.

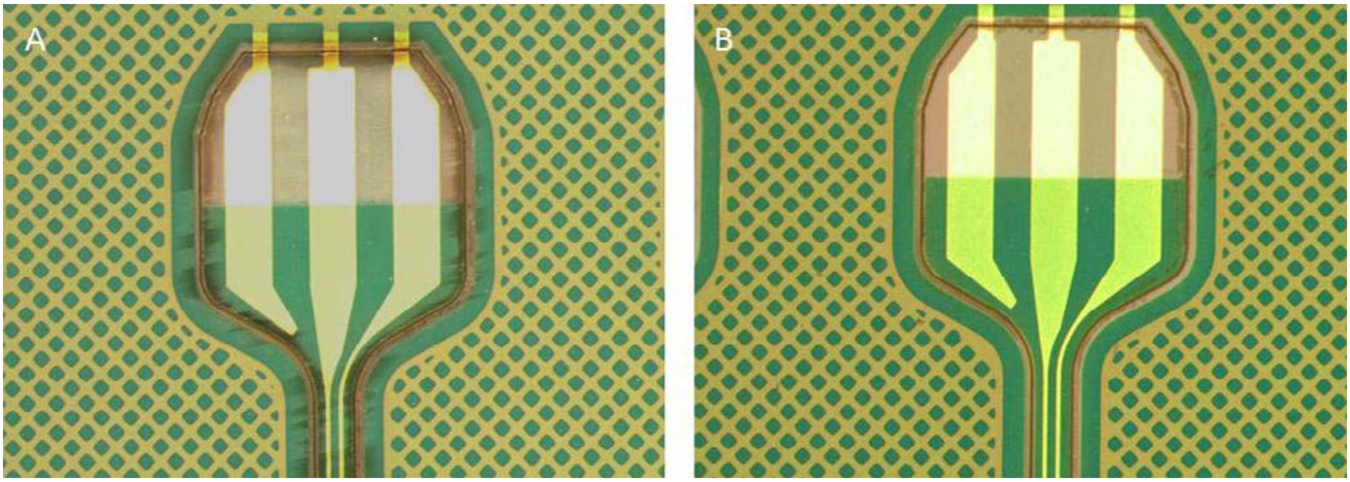


Figure 5: Laser residues on print after final routing (left); print after laser routing including cleaning procedure (right).

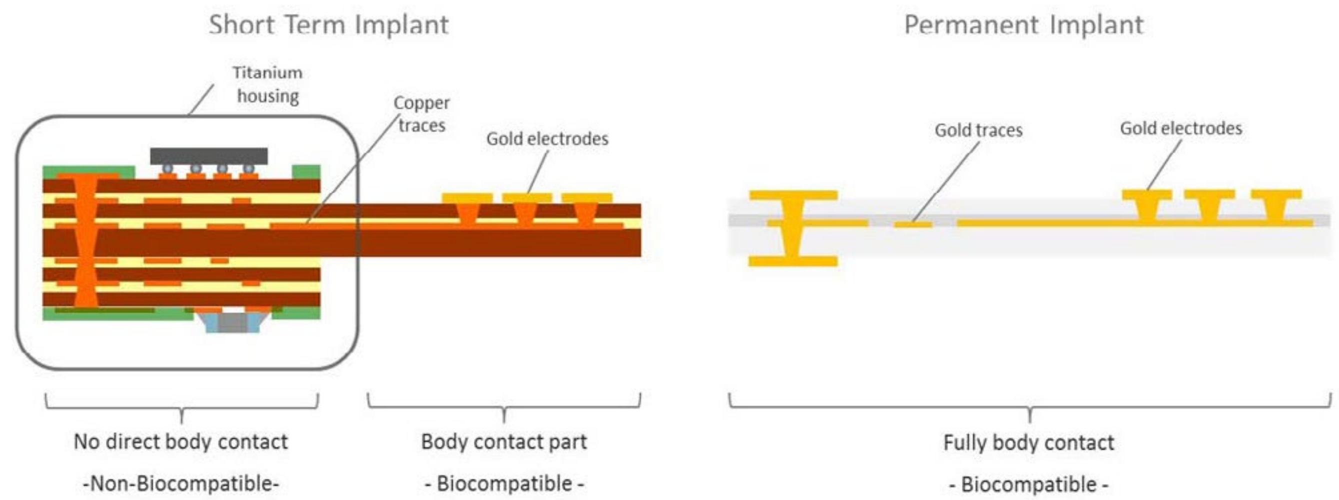


Figure 6: A comparison of a hybrid PCB layout for short term implants and a fully biocompatible layout acting as a permanent implant.

The substrates were bended ± 90° and survived 2,500 cycles at a trace width of 50 µm and more than 40,000 cycles at a trace width of 100 µm (Table 1).

On the final structure a special cleaning procedures to remove laser residues originated by a laser cut for the final outline was used (Figure 5). These residues may cause shorts along the

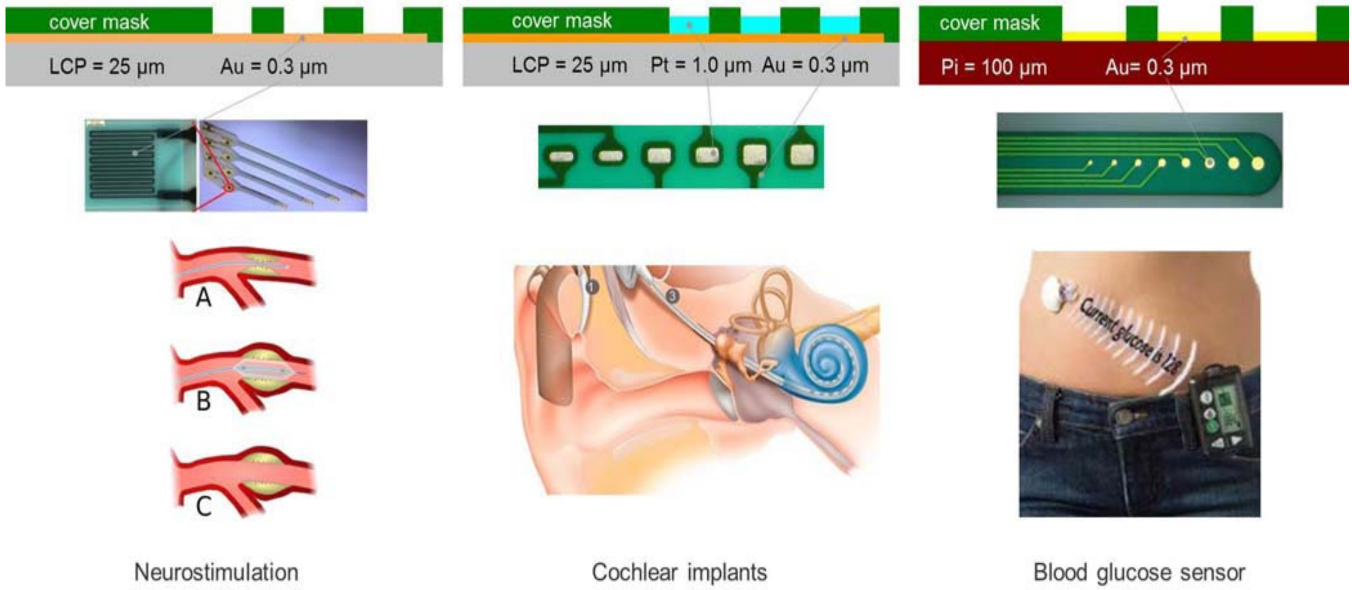


Figure 7: Examples of implantable PCB layouts and their corresponding application.

traces and may contaminate the electrode pads. To guarantee biocompatible PCBs a final cleaning is mandatory. The substrates have been successfully tested according to ISO 10993-5 for in vitro cytoxicity, a standard for evaluating biocompatibility.

The combination of conventional PCB processes and the use of noble metals allows for the manufacturing of directly implantable structures (Figure 6). For short term implants (< 30 days) a hybrid of the already established copper technology for parts without direct body contact and noble metals for body contact parts may be used. For permanent implants (> 30 days) the substrates need to be completely biocompatible by implementing noble metals only. A detailed description of necessary tests for surface devices, external devices or implantable devices can be found in the regulation ISO 10993-1:2009 + Cor 1:2010.

Such devices have been successfully fabricated for short term implants like blood glucose sensors, balloon catheters and diagnostic catheters as well as for permanent implants like cochlear and neurostimulating applications (Figure 7).

Achieving more complex artwork architectures and connectivity in the Z axes using pure noble metallization is under further investigation, as are ongoing activities focusing on long term reliability, especially with daisy-chained patterned, interconnect stress test designs that accomplish the biocompatible noble metal components. **PCB**

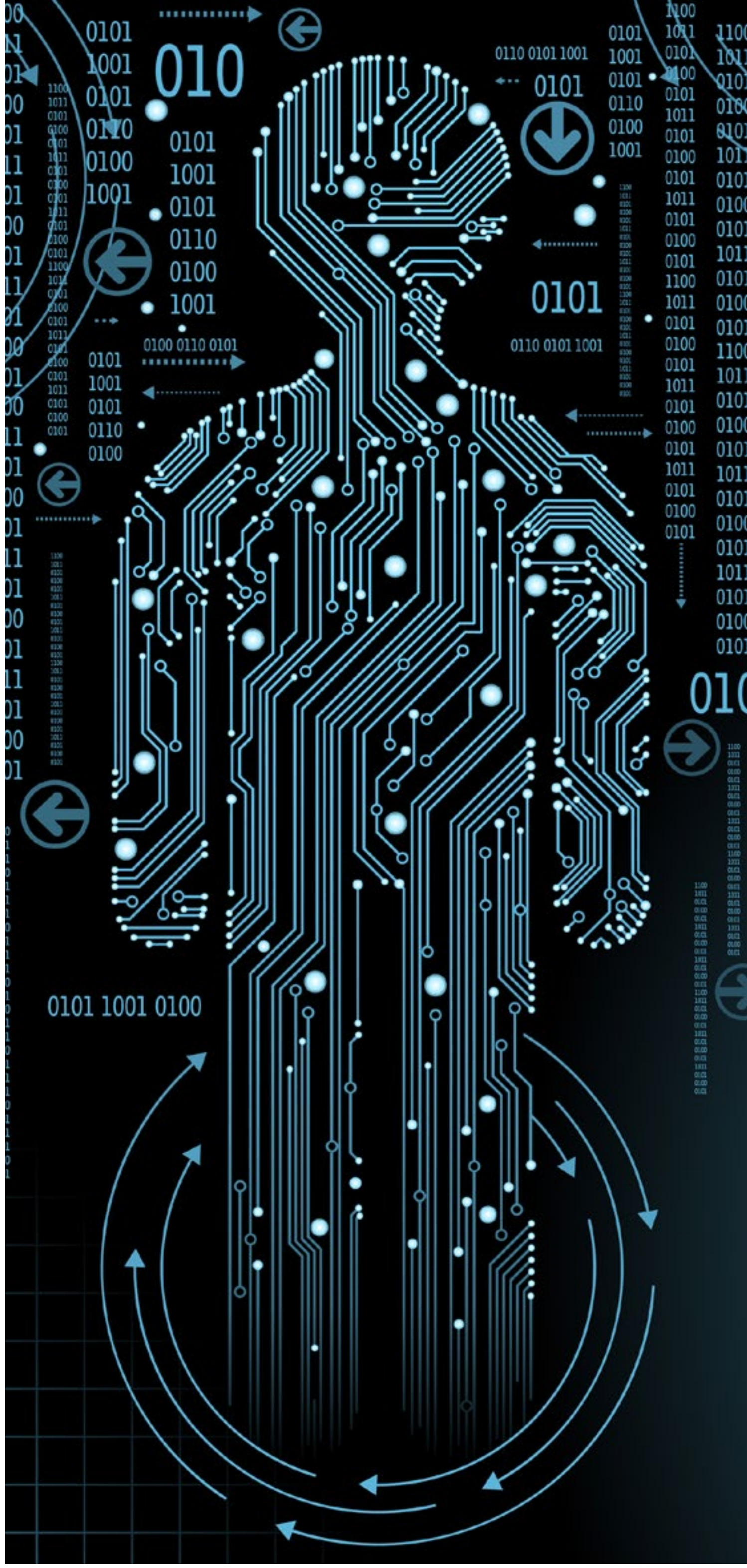

Daniel Schulze is engineering manager at DYCONEX AG.


René Tölke is product engineer at DYCONEX AG.


Ilse Widmann is director of marketing at Micro Systems Technologies.

PCBs for Medical Applications—A Designer's Perspective

by Kenneth MacCallum
STARFISH MEDICAL



Introduction

PCBs are an increasingly important part of modern medical devices—especially electromechanical devices. The opportunities are large, but PCB designers and manufacturers must be aware of and work within the regulations and limitations in the medical device design process. This article will provide an overview of the medical device design process, the role of PCBs in that process and in modern medical devices, and include design considerations and tips for incorporating PCBs into medical devices.

Medical Device Design Process

The medical device development industry is heavily regulated. Not only are the characteristics and performance of the devices regulated, the design and development processes are as well. The focus is on well-defined and understood requirements, quantification of risk of harm to patient and operator, verification that the design meets the specifications and finally validation that the design is effective and safe.

Many medical devices are manufactured in lower volume than consumer electronics and have higher margins; and most have longer design cycles. Notwithstanding these differences, the technologies used in medical devices are often heavily leveraged from consumer electronics. This means the same creep towards higher density circuits and component packaging applies in the medical field. The same techniques of replacing wiring harnesses with PCBs and flex circuits that are found in consumer electronics are found in medical devices, although the drive towards these technologies may be increased reliability and reduced labour cost instead of decreased size and reduced parts cost.

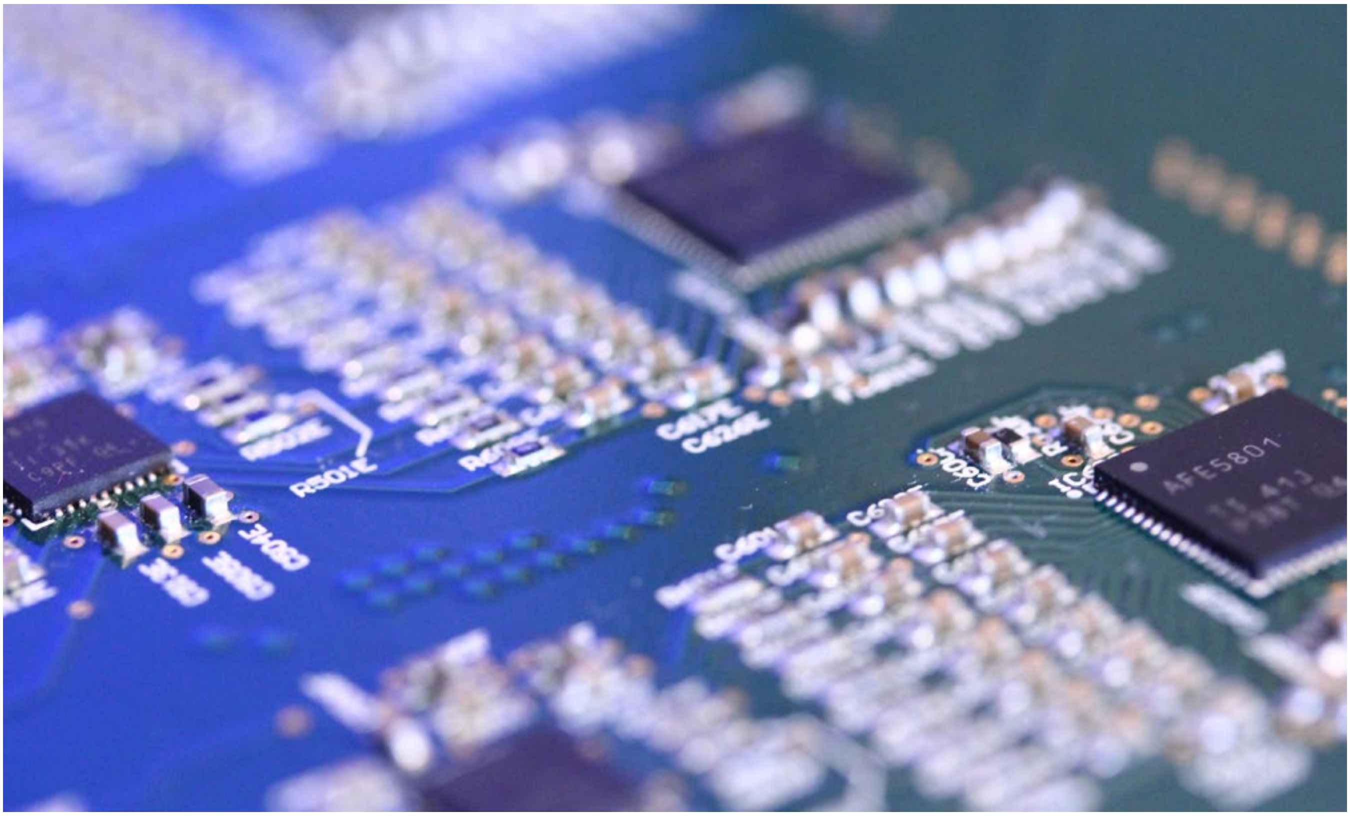


Figure 1: Custom-designed 8-layer PCB for a special purpose, multi-channel ultrasound system PCB.

Figure 2: Light treatment technology non-invasively accelerates early wound healing, implant loading and bone regeneration following oral surgery and dental implants.

Role of PCBs in Medical Devices

Almost every medical device PCB is heavily influenced and constrained by the work of other disciplines. It is rare that a board can be designed without heavy consultation throughout the design cycle with mechanical engineers or industrial designers. There is usually no clear owner to the mechanical constraints of a PCB. A board outline may be proposed, then shoehorned as the housing design gets hashed out. Rearranged as the main, large components, connectors and things get roughly placed; it will be tweaked again by the mechanical engineers or industrial designers. Finally the PCB designer will add their final touches and throw it back. This process is highly iterative and relies more and more on accurate 3D modeling of the board, connectors and wiring.

PCBs are structural elements with all sorts of great mechanical properties: they are fairly rigid, they're pretty strong and their tolerances are often excellent. The collaboration doesn't stop there either. Whoever is doing the firmware, software and logic design will also have some strong opinions about the board. It's best to get everyone together early and often to minimize the chance of later rework or worse.

Figure 3: Accelerated wear tester used to determine the durability or fatigue of prosthetic heart valves and other cardiac devices under pulsatile flow and physiological loading.

Medical devices have some of strictest industry EMC requirements and regulations. Addressing these constraints is primarily achieved at the PCB level. This is yet another factor driving PCB density and manufacturing tolerances.

Design Considerations and Tips

As PCBs in medical devices become more complex and dense, it is more important than ever to ensure that the designs are correct and will function as required. Although a typical medical product will see three or more significant PCB design cycles it is important that the value of each cycle is maximized. The more functionality is designed into a board, the more possibilities for errors and faults there is.

General rules

Electronic design automation (EDA) software now has sophisticated automatic design-rule checking, providing the rules are set up properly. These cannot prevent a bad design from getting into production, but they can prevent very common and frequent mistakes and oversights from getting out.

As PCBs get more complicated, it is helpful to partition designs hierarchically. The top level of a hierarchical design is much like a block diagram of the board. Putting all connectors on the top level schematic sheet if possible, makes that sheet very useful during troubleshooting and design reviews. Child sheets encapsulate lower level functionality and sub-circuits. These can be referenced multiple times if a circuit is repeated, rather than cut and pasted as separate sheets. This allows for easier duplication of layout and is less prone to mistakes creeping in.

Once the schematic design is done, sub-circuits are often routed first, without constraining them to the board outline. This leads to a better overall layout compared to the traditional approach of component placement followed by routing. Once the sub-circuit layouts are routed, these are then pushed and shoved together like puzzle pieces. It is helpful to minimize the number of layers of sub-circuits even

Figure 4: Treatment control device for Age-Related Macular Degeneration (AMD). Custom PCB design enabled device prototyping for evaluation of the technology in clinical settings.

if the board will be manufactured with many. This makes it easier to re-use portions of layouts or adapt to last minute changes in layer counts.

More and more boards have high frequency differential pairs or impedance critical single-ended signals. Use a finite element analysis (FEA) calculator to determine the correct trace width and spacing. FEA gives more flexibility than the analytical solutions although they are often not far off each other. Do not neglect the effects of adjacent ground pours or traces. If you are unable to guarantee the impedance to the closest plane layer then adjacent ground pours are a nice way to control the impedance. Make sure to stitch the pour to the ground plane all along the controlled impedance traces. These calculations can then be compared with the controlled impedance results from the PCB manufacturer.

Unfortunately, the industry standard for PCB manufacturing files is still the old Gerber format. These files never contain quite enough information to be sure the boards will turn out right. They don't contain units, the decimal place can be ambiguous and even the origin is uncertain. It is good practice to disable zero suppression, and use an absolute origin for both Gerbers and drill files. It is also good practice to inspect all files with Gerber viewing software other than the EDA package that generated them. More and more PCB manufacturers now have automated workflows with little human intervention or interaction. Layer misalign-

MAGNA SERIES

THE NEXT GENERATION IN PCB ETCHING

MAGNA FEATURES

- No CF4 for Operation
- Faster Etch Times
- Lower Operating Costs
- Lower Power Requirements
- Incredible Process Uniformity
- PC Control with Touch Screen

PATENT PENDING DESIGN

100% CF_4 FREE

The Magna Series is the world’s first plasma etching system used in the manufacturing production of PCBs that requires no CF4. This new technology from Plasma Etch, Inc. completely eliminates the need for CF4 gas that is presently used by PCB manufacturers using plasma systems for desmear and etch back processing.

PLASMA ETCH
PROGRESS THROUGH INNOVATION

3522 Arrowhead Drive
Carson City, NV 89706

775.883.1336

WWW.PLASMAETCH.COM

ment or scaling issues can sneak right through the manufacturing process and only be found when the boards are done.

Strategies for Planes and Pours

It is a good strategy to flood planes rather than using single net planes. This provides the benefit of allowing the odd trace to route on that layer in a pinch. It also makes it easier to segregate power rails, separate planes and avoid interference with impedance controlled traces. Although it made sense in the days of limited file sizes and taped layouts to route planes as negatives, now this just seems like a source of errors either in review or manufacture.

Due to modern boards having many voltage rails, it is rarely advantageous or even possible to use complete power planes other than for ground. Instead, one or a few layers can be dedicated to power routing using fills and fat traces. As a bonus, signals can be snuck through on these layers as well. When pulling power off onto the component layers, it should first hit one or more bypass capacitors with the other pin drilled to ground, then the destination component pin.

Ensure flooded regions are stitched well to others. This is especially important on a two layer board where planes are impossible and ground is really just a collection of pours and is quite broken up. Pours should be kept away from controlled impedance traces unless they are part of the controlled impedance strategy. If they are, then they should be well bypassed to ground. Note that impedance controlled traces crossing over fill boundaries will experience impedance discontinuities. These boundaries should be stitched either with vias to other layers or with bypass caps across the gap. This stitching should occur as close to the traces as possible.

It is often problematic to break planes into regions like analog signal ground or power ground. Having one good plane that is well decoupled to the chassis can be better than breaking planes up into different power regions. The effects and locations of high frequency switching currents should be carefully considered and controlled. If possible, keep them off the plane. If a plane must be broken into different regions, a line of no-fit caps should be added, just in case.

Consideration should be given to the currents of various sub-circuits that will flow through the plane and their impact on other subcircuits along the way. A slight void or cut in the plane to explicitly corral these currents can be an effective strategy to control them. Sometimes a different arrangement of sub-circuits will give even better results.

"It is often problematic to break planes into regions like analog signal ground or power ground. Having one good plane that is well decoupled to the chassis can be better than breaking planes up into different power regions."

Layout EMC Considerations

Given that medical devices are heavily regulated, at some point electromagnetic compatibility will have to be considered. Add ESD & EMC reduction components like TVS diodes and ferrites, even if the board is a prototype with no plans for EMC testing. This provides a head start to flush out problems early. It also plans ahead for a prototype being used beyond what was initially imagined or intended. Consider designing in the parts without populating them right away. Then they can be easily added later without cutting and gluing.

Consider where noise may come from both on and off of the PCB. Reducing noise at the source is always a better strategy than combatting it only where it is creating trouble. An example of this is the high speed parallel lines to a TFT display. A tidy bank of ferrites right at the driver outputs is often all it needs. This is easiest done at the PCB design stage.

When laying out switch-mode power supplies or filters on PWM circuits, designers should think about high-frequency current

paths—there are usually two of them—and keep the area of these paths minimized. This usually means placing the big components first: switches, inductors, caps and diodes.

Only connect switch-mode supply and PWM filter circuits to the rest of the PCB ground in one place. This includes signal grounds to the control ICs. Use the ground terminal of the main input capacitor. If the connection is done in more than one location, the ground plane and hence other circuits may be polluted with switching currents. Be wary of nearby copper pours accidentally touching the ground net somewhere else.

Simulation

Given the investment in time and effort to design PCBs it pays to simulate in order to learn as much as possible prior to the board order. Usually this does not mean simulating a whole circuit but focuses on key sub-circuits, or even key components of these sub-circuits.

A good candidate for simulation is to validate the choice of switch-mode power supply inductor. If the inductor is poorly chosen the circuit will smoke. Typical equation-based methods for determining the peak current requirements often are optimistic. A quick simulation regularly shows that under certain conditions the current will go just a bit higher. Simulate at worst-case conditions: maximum current, minimum inductance, maximum input voltage, etc. This determines the peak inductor current, which is often much higher than expected.

Simulation is also very useful in developing feedback systems. In medical devices it's not unusual for a control system to have hundreds of Watts at its disposal. To rush to experimentation in such a system is likely to result in damaged parts or even injuries.

"Models rarely take into account actual physical limitations. One can get valid looking simulation results from a situation which would vaporize parts in a microsecond. Similarly, the system modeled might ignore a part's non-ideality or a parasitic signal path that occurs in reality and critically affects performance."

Use Spice, MATLAB, FreeMat or even a spreadsheet to numerically model the behavior of a system, particularly an electronic or electromechanical one. This method allows detailed exploration of system performance over a wide range of conditions prior to building a prototype. It is especially useful to test conditions that occur infrequently or ones you can't easily replicate with the available test equipment.

Note that simulation results are often incorrect. Simulation models are frequently wrong, incomplete or inappropriate for a particular purpose, whether they are made from scratch or they are from an existing library or even a manufacturer's website or datasheet. Models rarely take into account actual physical limitations. One can get valid looking simulation results from a situation which would vaporize parts in a microsecond. Similarly, the system modeled might ignore a part's non-ideality or a parasitic signal path that occurs in reality and critically affects performance.

It is useful to think of simulation like physical bread-boarding. The same thought processes that would be used as physical components are wired up in the lab and the critical thinking that happens right before turning on the bench power supply are helpful during simulation. Special attention should be paid to the operating parameters being simulated and how the parts would behave in those conditions, especially with respect to their absolute maximums. Simulation complements real lab experimentation: model it, make it and repeat as necessary.

Prototyping

Even after simulating a system, it is important that it be physically prototyped. Frequently a key part or characteristic of a system is found to be missing once it is soldered together and

Electronic Chemicals, LLC

Via Dep 4550 Electroless Copper

Electroless Copper for Difficult to Plate Substrates

Ready to Handle... The Most Challenging Surfaces On the Planet.

Advantages:

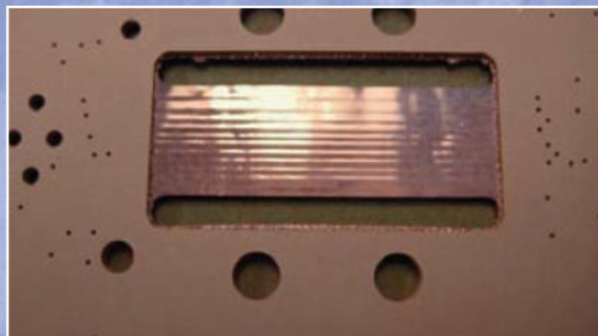
- Blister and Stress Free Copper Deposit
- No Harsh Pre-treatment Chemicals
- Low Temp, No Accelerator

Making Holes CONDUCTIVE

SUITABLE FOR FLEX, RIGID-FLEX, AND RIGID PWBS; COMPATIBLE WITH CONVENTIONAL EQUIPMENT.

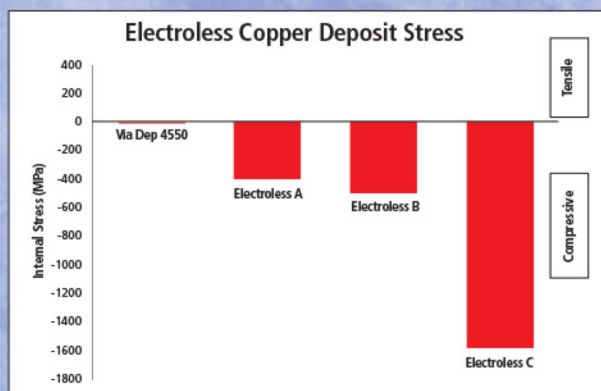
LARGE POLYIMIDE FLEX WINDOW.

DIFFICULT SURFACES CAN BE PROCESSED WITHOUT THE USE OF HARSH CHEMICALS SUCH AS HYDRAZINE.



POLYIMIDE FLEX WINDOW IN RIGID FLEX CIRCUIT BOARD.

VIA DEP 4550 HAS SIGNIFICANTLY LESS INTERNAL STRESS THAN CONVENTIONAL ELECTROLESS COPPER PRODUCTS.



We've Got the Whole Process Covered

OMG Electronic Chemicals LLC

5630 Pioneer Creek Drive
Maple Plain, MN 55359
P: 800-321-9050 • F: 763-479-3344

400 Corporate Court, Suite A
South Plainfield, NJ 07080
P: 800-536-1959 • F: 908-222-3344

scoping begins. In that case, an iterative process of simulation, circuit changes, tests and measurement may be required.

A recent challenge in breadboard testing and early prototyping is the steady industry shift towards higher density packaging and higher pin counts. Many components are nearly impossible to physically prototype with using traditional bread-boarding techniques. How then can sufficient testing and prototyping occur prior to the onset of detailed product design?

Fortunately, a new class of PCB manufacturers has stepped into the market. It is now possible to buy just a handful of high-quality, custom PCBs for less than the cost of the prototyping supplies and time that would have been required for bread-boarding. These manufacturers create opportunities for board layout that were not imagined just a few years ago. It is now possible and economical to make boards for extremely early prototypes. This ability used to be solely available to companies with large budgets or specialized equipment. This change in the PCB manufacturing market will have a significant impact on medical device development.

Conclusion

The medical device industry is a heavily regulated industry with slightly different motivations than consumer electronics. Many of the same design strategies apply and many similar technologies are leveraged. PCB design and use in this industry are growing in sophistication and precision. Similarly, advances in manufacturing capabilities, such as low volume, low cost, quick-turn PCBs are being embraced and leading to noticeable shifts in the design and development process. **PCB**

Kenneth MacCallum is principal engineering physicist at Starfish Medical.

VIDEO INTERVIEW

MacDermid Focuses on OEM Needs

by Real Time with...SMTAI 2014

Publisher Ray Rasmussen and MacDermid's Director of OEM Applications Lenora Toscano discuss MacDermid's focus on understanding the needs, challenges, and roadmaps of OEMs.

realtimewith.com

www.gardien.com
Information
Capabilities
Processes
GO Solutions
AC8 Scanner &
G60 Flying Probe
Fixturing
Testing Services
Electrical Testing
Fixturing
4WK, Hi-Pot, Impedance Testing
CAM and CAD Services
Final Visual Inspection
Automatic Optical Inspection
Automatic Visual Inspection
Equipment Sales and Servicing
Consulting
GO Gardien
Global Quality Assurance Solutions
tailored to your individual needs
GO™
Gardien
Learn more at the International Printed Circuit & Apex South China Fair
at the Shenzhen Convention Center, December 3-5, 2014
Visit us at Booth #1U01

Wearable Electronics: The Shape-Shifting Future of Medical Devices

by Gary Baker
NYPRO

Think about your last doctor's appointment. You probably had your blood pressure taken and blood drawn for routine tests. Your doctor may have listened to your heart with a stethoscope and then referred you to the hospital for an electrocardiogram (EKG). There, a nurse pasted electrodes on your chest so a machine the size of an office printer could record the electrical activity of your heart. All a necessary part of your wellness routine, but you had to take a day off work and endure a little pain in the process.

Now imagine this scenario: You check in at the doctor's office. The receptionist hands you a small, self-adhesive patch that you wear on your skin and it instantly transmits all your vital healthcare data directly to the doctor—before you even get to the exam room. After a brief chat with the doctor, you are sent on your way with a clean bill of health, and this is all accomplished during your lunch break.

This is the future potential of flexible electronics in wearable medical devices: to free both patient and doctor from the bulky and unwieldy technology of the past. When most of us think of electronics on a PCB, we think of a rigid, stiff device. However, advances in electronics, flexible materials and technology are driving development of new wearable electronics that can bend and fold just like paper.

Flexible electronics are lightweight, portable and so thin and supple that they can conform to the human body. They represent a nascent, but fast-growing industry. A report from IDTechEx, Printed, Organic & Flexible Electronics: Forecasts, Players & Opportunities 2013–2023, found that the total market for these technologies will grow almost fivefold in 10 years, from about $16 billion in 2013 to nearly $77 billion in 2023. Likely no industry stands to see more innovation from flexible electronics than the medical device industry.

What if diabetics could wear a contact lens that continuously monitors their glucose levels , forever eliminating finger sticks and bloody test strips? What if, instead of around-the-clock bedside checks that wear on patients and nurses alike, a tissue-thin adhesive patch could report hospital patients' vital signs directly to the nurses' station—or to the patient's electronic medical record? What if a flexible circuit could detect—or even treat—the first signs of recurring malignancy in cancer patients? What if these monitors enable us to gather big data on vitals that through intelligent prognostic analysis could predict the onset of a malady?

"Recent advances in flexible electronics technology has allowed researchers to devise a way to print devices directly onto the skin allowing people to go about normal daily activities for an extended period of time."

Recent advances in flexible electronics technology has allowed researchers to devise a way to print devices directly onto the skin allowing people to go about normal daily activities for an extended period of time. Such systems could be used to track medical conditions and monitor healing near the skin's surface, as in the case of surgical wounds.

Sheila Dharmarajan investigates the outlook for wearable electronics on Bloomberg Television's "Bloomberg West." She reports on a new method to reformat silicon allowing electronics to bend, stretch and conform to the human body.

The possibilities are truly endless: intelligent surgical instruments that give the surgeon real-time feedback to improve the speed and precision of procedures; conformal pacemakers to keep the heart functioning without degrading quality of life; neural implants to control prosthetics; and biosensors to transmit data to remote health care providers, increasing access for rural or movement-restricted patients and reducing or replacing the need for frequent doctor appointments.

In addition to improving health care access, delivery and medical consumers' comfort and convenience, flexible electronics will be cheaper, faster and easier to manufacture than traditional circuitry for medical applications. Wearable electronics relies on the promise of organic materials that function similarly to inorganic materials but can be dissolved in a solution, forming conductive, semi-conductive and dielectric functional inks that allow circuits to be manufactured using printing techniques.

A number of companies have already begun developing flexible medical technologies that will soon be available commercially. As the technology evolves, demand—driven by an aging population, longer life expectancy and the quest for more affordable and less invasive health care—is expected to continue to increase.

What applications do you see for wearable electronics in medical devices? Where do you see the most potential for growth and innovation? Can we do something near term to realize incremental benefits? What stands between these ideas and their realization? **PCB**

Reprinted with permission of Jabil.

Gary Baker is the communications manager for Nypro, a Jabil company.

Research and Markets: *Medical Electronics Market Outlook 2020*

The total value of medical electronics market is expected to reach $56.50 billion by 2020, growing at a CAGR of 5.5% from 2014–2020

In *Research and Markets* new report, Medical Electronics Market Outlook 2020, components (sensor, battery, memory device, display, and microprocessor/MCU) and applications (imaging, medical therapeutics, diagnosis, monitoring, and fitness & wellness) are examined.

The medical electronics market is expected to grow at a rapid pace in most regions of the world due to growing income levels, and the increasing prevalence of chronic and lifestyle disorders such as diabetes, hypertension, and cardiovascular diseases. A growing population coupled with high income levels and awareness about wellness and fitness are some of the factors driving the market growth. Further, initiatives from various governments have led rural areas to adopt advanced medical services, thereby boosting the medical electronics industry.

An increase in lifestyle-related diseases is creating the need to focus on improving healthcare access and providing affordable and preventive healthcare. This report reviews the latest market trends with a perceptive to disclose future growth prospects. An in-depth analysis on a geographic basis provides strategic business intelligence for investments in the medical electronics sector. The study reveals profitable investment strategies for electronics companies, business executives, product marketing managers, new business investors, and many more in preferred locations. In addition to that, the report contains market segmentation by application, component, and geography.

Key Topics Covered:

1. Introduction
2. Research Methodology
3. Executive Summary
4. Premium Insights
5. Market Overview
6. Industry Trends
7. Market, By Components
8. Market, By Application
10. Competitive Landscape
11. Company Profiles
    - Analog Devices, Inc.
    - Cypress Semiconductor Corporation
    - Fairchild Semiconductor International, Inc.
    - Freescale Semiconductor, Inc.
    - GE Healthcare
    - Maxim Integrated
    - On Semiconductor Corporation
    - Philips N.V.
    - Renesas Electronics Corporation
    - Siemens Ag
    - Stmicroelectronics N.V.
    - Tekscan, Inc.
    - Texas Instruments

For more information visit: www.researchandmarkets.com/research/7zc3xh/medical

INTRODUCING COOLSPAN® TECA

thermally & electrically conductive adhesive

Rogers can help by being your reliable conductive adhesive film source

Get the heat out of those high-power PCBs. COOLSPAN® Thermally & Electrically Conductive Adhesive (TECA) Films are ideal for dissipating heat in high-frequency circuits. COOLSPAN adhesives feature outstanding thermal conductivity (6 W/m/K) and reliable thermal stability. Keep things cool, with Rogers and COOLSPAN TECA film.

CONTACT YOUR SUPPORT TEAM TODAY

ROGERS CORPORATION

www.rogerscorp.com

MEET YOUR COOLSPAN® TECA FILM SUPPORT TEAM

Leading the way in...

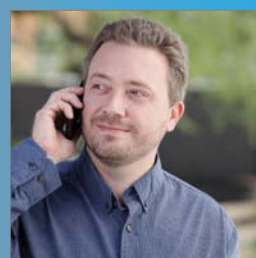
• Support • Service • Knowledge • Reputation

SCAN THE CODE TO GET OUR CONTACT INFO.

Greg Bull
Applications Development Manager
Northern Territory (U.S.) & Canada

Dale Doyle
Applications Development Manager
Western Territory (U.S.)

John Dobrick
Applications Development Manager
Southern Territory (U.S.) & South America



Scott Kennedy
Applications Development Manager
Eastern Territory (U.S.)

John Hendricks
Regional Sales Manager
Europe

Kent Yeung
Regional Sales Director
Asia

If you are unable to scan a VR code please visit our Support Team website at www.rogerscorp.com/coolspan

PCB007 Market News Highlights

Conference Board Consumer Confidence Index Drops

The Conference Board Consumer Confidence Index®, which had increased in August, declined in September. The Index now stands at 86.0 (1985=100), down from 93.4 in August. The Present Situation Index decreased to 89.4 from 93.9, while the Expectations Index dropped to 83.7 from 93.1 in August.

Internet of Things Market to Reach $1423B by 2020

The value of Internet of Things market is expected to reach $1423.09 billion by 2020, at an estimated CAGR of 4.08% from 2014–2020, whereas the Internet of Nano Things (IoNT) Market is forecast to be worth $9.69 billion by 2020.

3D Printing Market Forecast to Reach $8.6B by 2020

North America leads the 3D printing market with approximately 43.9% revenue share in 2013, followed closely by the European region. The dominance of the North American market is attributed to growth in the healthcare, consumer, aerospace, and automobile industries.

Big Data Technology, Services to Reach $41.5B in 2018

International Data Corporation (IDC) forecasts that the Big Data technology and services market will grow at a 26.4% compound annual growth rate (CAGR) to $41.5 billion through 2018, or about six times the growth rate of the overall information technology market.

Medtech Market to See Sales of $514B by 2020

"This is a year of change for the medical device market, particularly within the cardiology and orthopaedic spaces, which have been dominated by megamergers," said Ian Strickland, report author. "If the deal between Covidien and Medtronic goes through, we could see a new market leader. In an industry forecast to be worth more than half a trillion dollars in 2020, that is no insignificant achievement."

Personal & Domestic Robots Sales Continue to Soar

Sales of medical robots decreased slightly by 2% compared to 2012 to almost 1,300 units in 2013, accounting for a share of 6% of the total unit sales of professional service robots. The most important applications are robot assisted surgery and therapy with more than 1,000 units sold in 2013, 2% less than in 2012.

Medical Electronics Market to be Worth $56.5B by 2020

An increase in lifestyle diseases continuously pushes patients for medical monitoring to get their health status to avoid any health related contingencies; medical devices such as cardiac monitors and hemodynamic monitors are continuously gaining market share in the medical monitoring applications.

Wearable Electronics Market at $11.61B by 2020

The global wearable electronics products market revenue is expected to grow roughly at a CAGR of 24.56% and cross $11.61 billion by the end of 2020, with steady sales of wristwear and footwear category, along with the emergence of the small market size for eyewear and bodywear category.

Wearable Device Shipments to Quadruple by 2017

New findings from Juniper Research have revealed that global smart wearable device shipments will more than quadruple by 2017, reaching 116 million units, compared to an estimated 27 million this year. However, this still suggests that less than 5% of smartphones will be used with such wearables by this time.

U.S. Small Businesses Segment Continues Decline

According to D&B's proprietary data, the small businesses segment continues to decline as segments expected to ignite growth, flounder. "While overall business health across the country remains virtually unchanged, small businesses continue to experience inconsistent growth patterns," said Paul Ballew, chief data & analytics officer.

Lead-Free Compatible OSPs: What Does This Really Mean?

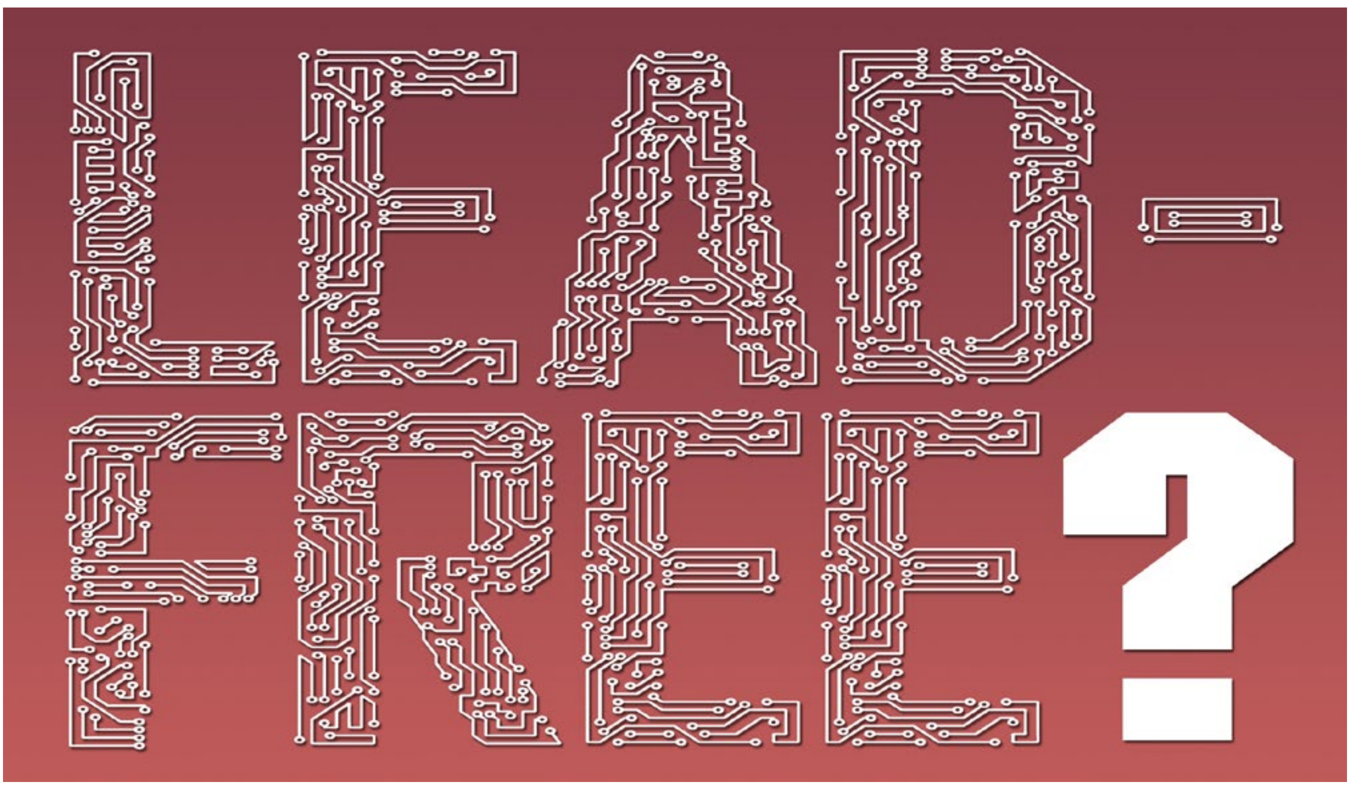
by Michael Carano
OMG ELECTRONIC CHEMICALS

Introduction

The setting: early 2008. Many fabricators and EMS providers are continuing to push programs for RoHs compliance. This, of course, requires electronic assemblies to be, among other things, free of lead-bearing materials. In order to adopt such a program, the surface finish, component finish and the solder used in assembly must be lead-free. While the questions surrounding these issues are many, one in particular appears more frequently: How do we know that a particular finish is truly lead-free soldering compatible? This particular question is being addressed by one of the IPC standards writing committees responsible for developing an industry standard specification of organic solderability preservatives (OSP).

The Nature of the Beast

The IPC committee made up of a cross-section of suppliers, PWB fabricators, OEMs and EMS companies is working to complete the development of IPC-4555 (standard for OSP). This committee is developing a protocol to test OSP coatings for lead-free (high-temperature) compatibility. The protocol requires that copper coupons coated with the OSP would be subjected to three lead-free reflows, then tested for wettability with an appropriate wetting balance instrument. It should be noted that after the coupons are processed through the OSP solution, the coupons shall be rinsed with DI water and dried at 105°C. Following the protocol of the ANSI-JSTD-003B, the samples shall be fluxed using the standard 0.5% activated flux



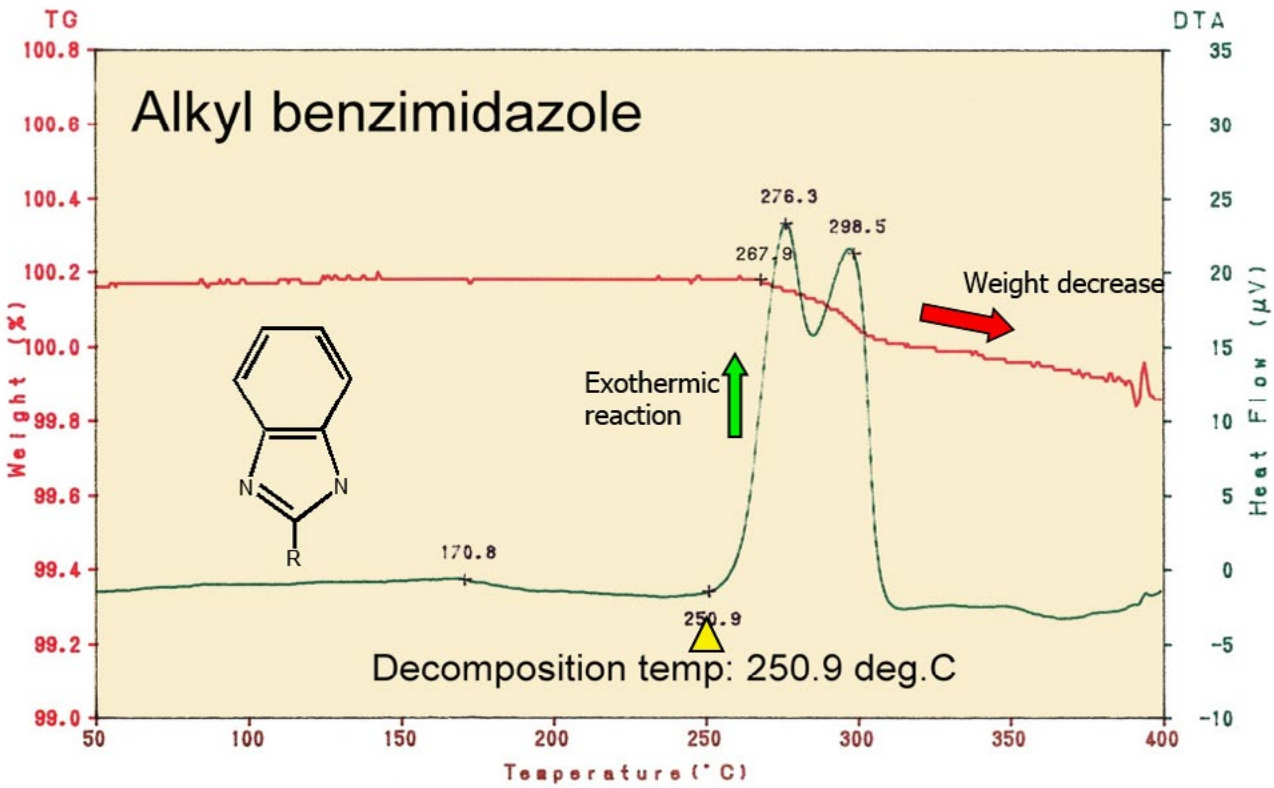


Figure 1: Benzimidazole type OSP.

and immersed in SAC305 at a temperature of 255°C for ten seconds. Further:

1) Ten samples per test shall be run.
2) The mean and standard deviation of the maximum force achieved shall be documented.

Interestingly, the committee has also recommended that a second set of test coupons be processed under the same pretreatment conditions. This time, however, without the drying step. Again, the ANSI-JSTD-003B, wetting balance procedure to measure wettability, will be utilized.

I would also suggest that there is another way with that one can determine whether a particular OSP is lead-free compatible: Does the particular organic film stand up to high temperatures? Further, this relates to the temperature at which the OSP film begins to decompose (Figures 1 and 2). The data shows that depending on the chemical structure of OSP molecule, the resistance to thermal decomposition of the organic film can be different. Obviously, the greater the temperature resistance of the OSP, the better the chance that the film will protect the underlying copper from significant oxidation. Remember, it is all about the oxidation or lack thereof that we are looking for.

Regardless, it is highly recommended that fabricators follow a set procedure to qualify any OSP and its suitability as a lead-free compatible coating.

Before doing that, however, it is critical to realize that the thickness of the OSP film plays a role in its ability to protect the underlying copper surface from oxidation. So, how can one ensure that the film thickness is sufficient to function as a lead-free compatible OSP?

One test that is very simple to implement is the silver nitrate test. The procedure is as follows:

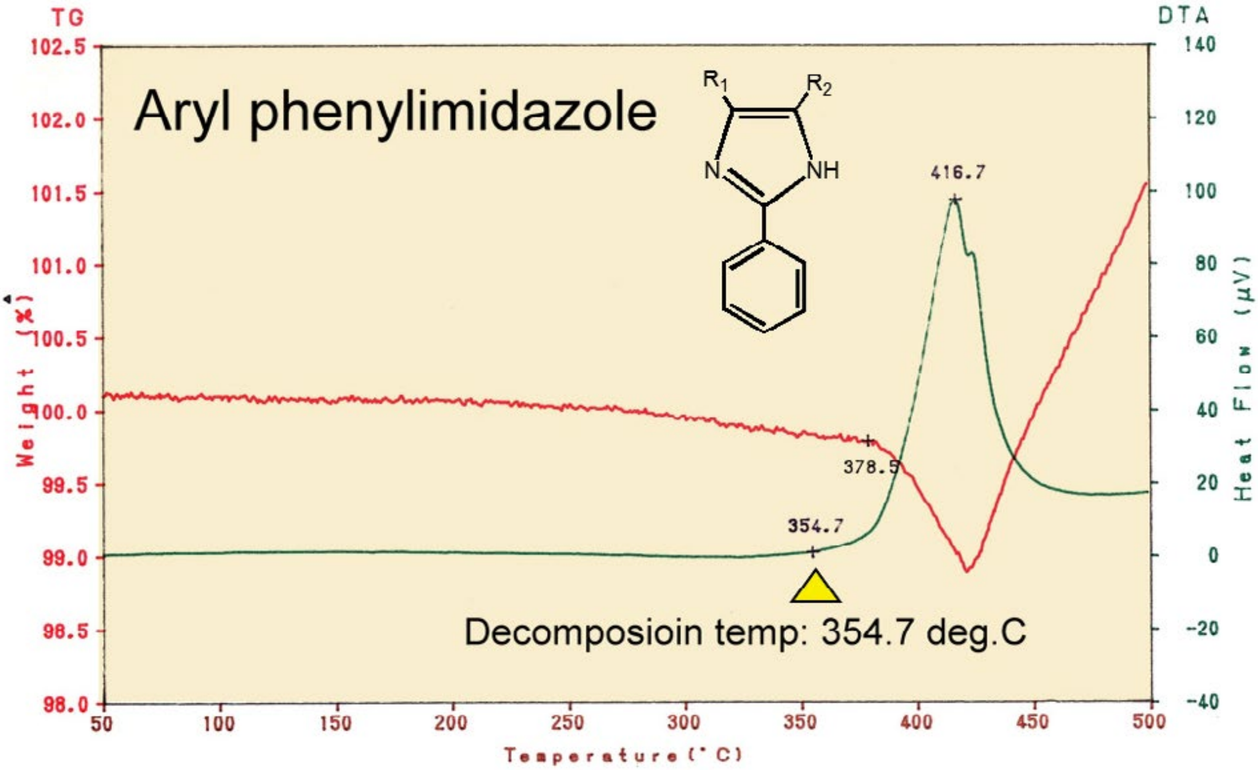


Figure 2: Next-generation OSP molecule—higher temperature resistance than benzyl-imidazole shown in Figure 1.

1. Make up a 0.1 N solution of silver nitrate in water.
2. Select either test coupons or actual PCB treated with OSP.
3. Apply silver nitrate solution with eye dropper (one drop) on area to be tested.
4. Allow 20 seconds to pass and inspect visually.

If there is no discoloration on the copper surface (up to the 20 second interval), the coating is deemed sufficient thickness. I have performed this test many times and consider it reliable check on the process. Of course, as has been presented in past columns by this author, there are other important factors that are part of the OSP reliability equation.

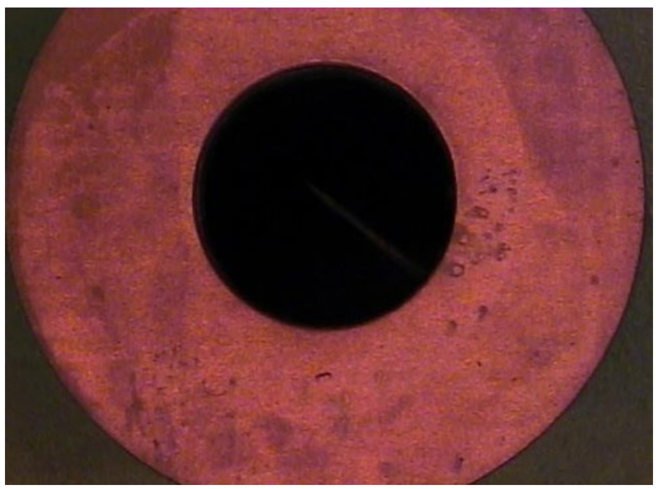


Figure 3: Sporadic discoloration of an OSP treated surface.

The Role of OSP Film Thickness

It is worth mentioning that the author is not advocating increasing the thickness of the OSP film as a means to improve solderability. It is true however that a sufficiently thick and uniform organic film is required at the very least to minimize oxygen penetration to the base metal. The photo in shown in Figure 3 shows spo-

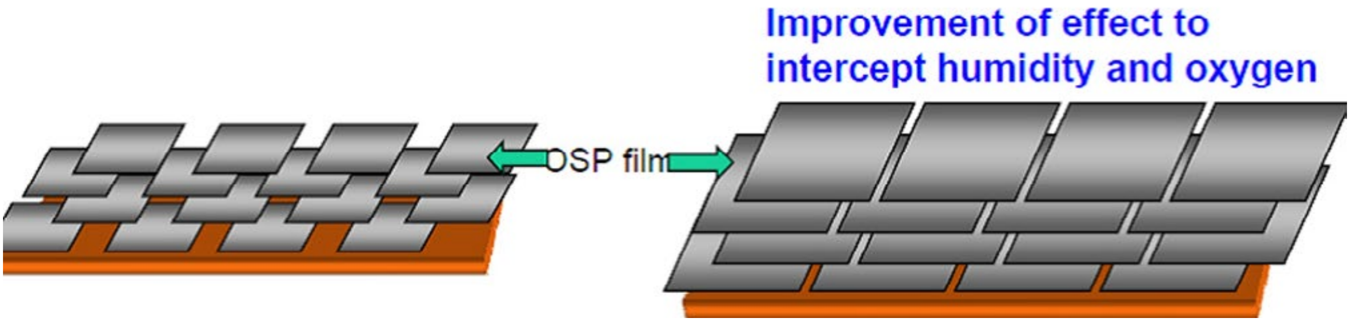


Figure 4: Conventional imidazole (left) and improved imidazole (right).

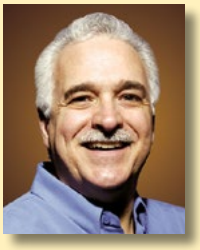
radic discoloration on the surface treated with an OSP process. When the surface was analyzed, the data showed that film thickness was non-uniform and there were areas on the surface where the film thickness was less than 0.20 microns. That, of course, is unacceptable.

The Role of the OSP Film Molecular Structure

The functional groups that are part of the molecular structure of the imidazole molecule play a key role in the organic films ability to minimize oxygen penetration to the base copper. In addition to the amine groups in the azole molecule bonding with the copper of the base metal, next generation OSP technology shows increased cross-linking of other functional groups on the OSP molecule. This allows for a much tighter film that in turn further acts as a barrier against oxygen and humidity (Figure 4).

Summary

While next-generation OSP as a final finish has become the standard for lead-free compatible assembly, one should assume that any new OSP meets the criteria. A number of simple procedures may be followed to qualify any new OSP and ensure it is compatible with these higher assembly temperatures. **PCB**

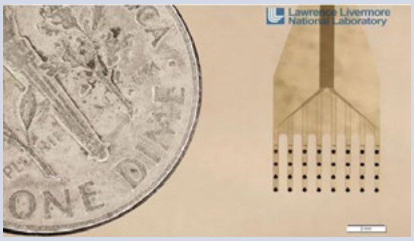


Michael Carano is with OMG Electronic Chemicals, a developer and provider of processes and materials for the electronics industry supply chain. To read past columns, or to contact the author, click here.

Electrode Array System to Monitor Brain Activity

The National Institutes of Health (NIH) awarded Lawrence Livermore National Laboratory a grant to develop an electrode array system that will enable researchers to better understand how the brain works through unprecedented resolution and scale.

Lawrence Livermore is developing a neural measurement and manipulation system—an advanced electronics system to monitor and modulate neurons—that will be packed with more than 1,000 tiny electrodes embedded in different areas of the brain to record and stimulate neural circuitry. The goal is to develop a system that will allow scientists to simultaneously study how thousands of neuronal cells in various brain regions work together during complex tasks such as decision making and learning.



PCB007 Supplier/New Product News Highlights

Technica USA Installs Second Maskless DI Machine at APCT

Frank Medina, president of Technica USA, announced recently that sales and activity for Maskless DI equipment continues to be successful: "We are very pleased with the continued sales activity and interest in the Maskless DI equipment. We have installed four units in the past quarter and have had several successful evaluations during this same period."

Ventec Invests in FOD-avoidance Program

The avoidance of foreign object contamination continues to be a fundamental objective in Ventec Europe's UK prepreg fabrication facility, and the company is proud to announce that its epoxy prepreg cutting and slitting department has been brought to the same standard of clean-room operation as already established for its polyimide prepreg.

Isola Group Withdraws IPO Plan

Isola Group Ltd., a maker of laminate materials, filed with U.S. regulators on Tuesday to withdraw its up to $100 million initial public offering of common shares.

Insulectro Signs Distribution Agreement with DuPont

Insulectro and DuPont Microcircuit Materials (DuPont) recently signed an agreement that allows Insulectro to distribute key DuPont polymer thick film paste products to U.S. customers in the thin film photovoltaic, printed circuit board, smart card, and membrane touch switch markets.

ESI Unveils New Laser Micromachining Platform

Electro Scientific Industries, Inc. has announced the μFlex Series multi-axis precision laser micromachining platform, enabling new levels of price-performance in laser applications for consumer electronics, medical devices, automotive components, and many other industries.

Mitsui Kinzoku Expanding Capacity of FaradFlex in Asia

FaradFlex is used as a material for improving power delivery, reducing resonances and noise, major challenges that information and communication equipment must overcome. This enables higher processing speeds and higher capacity, in multilayer PCBs for high-speed transmission routers, servers, and supercomputers.

Enthone Nets ISO/IEC 17025 Accreditation

Enthone Ltd. (Woking, UK) has received accreditation to the ISO/IEC 17025:2005 standard. The United Kingdom Accreditation Service (UKAS) conducted the independent, third party certification and issued TL# 8202.

Rogers Debuts Laminates with Rolled Copper Foils

Rogers Corporation's Advanced Circuit Materials Division, a global leader in microwave PCB materials, recently introduced rolled copper cladding options with its RO3003, RO3035, and RO3203 low dielectric constant laminate materials.

Atotech Launches CupraEtch SR

CupraEtch SR is one of the company's surface pretreatment products, specifically developed to maximize the adhesion of dry and liquid soldermasks and photoresists.

Fuji America, FlexLink Sign Partnership Agreement

Fuji America Corporation and FlexLink Systems, Inc. recently signed a North American agreement to bring board handling equipment and conveyor systems by FlexLink to Fuji's existing product offer, providing turnkey solutions to the PCB industry.

Best Practices 101: Part 4

by Steve Williams
STEVE WILLIAMS CONSULTING LLC

Looking back through the annals of the U.S. PCB industry, when it comes to quality, we have evolved from a reactive, to a proactive mindset. This evolution has led to what is loosely called the zero-defects methodology (ZDM). The old gold standard of three sigma is no longer acceptable and has gone the way of the dinosaur.

Pipe Dream?

Are zero defects an achievable, sustainable goal 100% of the time? Of course not, but with six sigma levels we can come pretty close. Recognizing that we will occasionally fall short of any goal *mandates* that the goal be set at zero defects. The reasons why a zero defect mentality is required can be condensed down to the singular, bottom line principle of reducing costs, and as we all know, it's *always about the dollars*. Costs are always attached to defective product in the form of inspection/test, rework/repair, scrap, and warranty (customer returns). Reducing these costs results in increased customer satisfaction, and quite simply, happy customers mean higher revenue.

It is always an interesting study to compare the advertised capability of a company to its actual capability. The sales force touts world-class quality, which implies that they are operating at a six sigma level. However, an objective on-site assessment of their processes quickly separates the bluster from the facts, typically revealing that most organizations are operating at a true

- 32,000 babies dropped in the delivery room every year
- Unsafe drinking water for 15 minutes each day
- 2 short or long landings at most major airports each day
- 20,000 lost articles of mail per hour
- 5,000 incorrect surgical procedures each week
- 200,000 prescriptions filled incorrectly each year in the US
- 240 defective parts in the average new car
- No electricity for 88 hours every year
- 22,000 checks deducted from the wrong account each hour

Figure 1: Life at 99% Good. (Source: Elusive Lean)

yield somewhere between 93 and 99%—the old three sigma complacency. Statistics are a wonderful tool, but like most things in life, you will only get out of them what you put in. It all boils down to what the organizational objective is; superficial window dressing or honest-to-goodness improvement. Inflating process yields by excluding things like rework, customer waivers, or returns does nothing but mask problems and will not result in true improvement. If window dressing is indeed your goal, then I would suggest tossing this article and immediately picking up a copy of *Extinction for Dummies,* by Peter T. Platypus.

Why 99% is Not Good Enough

I remember the not-too-distant past when a 99% yield rate would earn bragging rights (myself included). Looking at what that really means by today's metrics shows that a 1% scrap rate converts to 10,000 defective parts per million (DPPM). As a customer, imagine a supplier striving to give you only 1% defective parts! World-class six sigma levels allow only 3.4 DPPM. If you are still reading at this point, I would hope that you agree with me that 10,000 DPPM is totally unacceptable and are prepared to do something about it.

(Note: It must be mentioned that the 3.4 DPPM attributed to six sigma levels was developed by Motorola, and based on the assumption that over time, a process is likely to have a shift in the mean of up to +/- 1.5 sigma. This potential shift is factored into the 3.4 DPPM. Statistical purists would argue that a six sigma level is actually .002 DPPM, but since the Motorola interpretation is universally accepted, I use 3.4 DPPM to represent a six sigma level.)

When companies like Motorola and General Electric began communicating six sigma expectations to their suppliers in the early 1980s, what began as a ripple quickly developed into a shockwave throughout the supply chain. To say that this concept was met with some resistance is a monumental understatement. Companies had absolutely no idea how they were going to effect a change of such magnitude that their process defect rate would drop from 10,000 to 3.4 DPPM. Through a slow and painful process, companies began to understand that the way to achieve these quantum paradigm changes was through Lean best practices. The interesting paradox is that none of us would accept 99% in our personal lives, so why do we accept it in our businesses? Figure 1 shows what life would look like if we settled for having things right only 99% of the time in some areas we can all relate to. This kind of changes the perception that 99% is good enough, doesn't it?

Contrast this with a six sigma level in which your local weatherperson's forecast would be correct every single day for 795 years in a row!

What is Six Sigma?

Sigma (σ) is the eighteenth letter in the Greek alphabet, and is defined and used in two different ways:

1) As a mathematical measure of the amount of variation in a process. This is normally referred to as the standard deviation of a process; the lower the standard deviation, the better, and

2) To describe the quantity of defects a process will produce. This is normally referred to as the sigma level of a process and is a measure of process performance; the higher the sigma level, the better. Although statistics are usually associated with six sigma, that is only part of it; six sigma is the problem solving methodology called DMAIC (define, measure, analyze, improve, control). DMAIC is process that uses a collection of tools to identify, analyze, and eliminate sources of variation in a process. Six sigma can be an intimidating concept to grasp, particularly regarding the statistics and math part of the process. The key takeaway is that to achieve a six sigma level, process variation must be cut in half from a three sigma level.

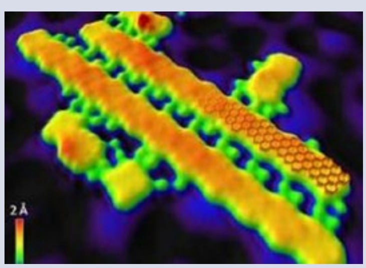
This concept will be explained in greater detail in the next issue, Best Practices 101: Part 5. **PCB**

Steve Williams is the president of Steve Williams Consulting LLC and the former strategic sourcing manager for Plexus Corp. He is the author of the books, Quality 101 Handbook and Survival Is Not Mandatory: 10 Things Every CEO Should Know About Lean. To read past columns, or to contact Williams, click here.

Graphene Nanoribbons to Revolutionize Electronics

Graphene is a two-dimensional material with extraordinary electronic and magnetic properties that can be tailored by cutting large sheets of the material down to ribbons of specific lengths and edge configurations. Scientists have theorized that nanoribbons with zigzag edges are the most magnetic, making them suitable for spintronics applications.

But this "top-down" fabrication approach is not yet practical, because current lithographic techniques for tailoring the ribbons always produce defects.



Now, scientists from UCLA and Tohoku University have discovered a new self-assembly method for producing defect-free graphene nanoribbons with periodic zigzag-edge regions. In this "bottom-up" technique, researchers use a copper substrate's unique properties to change the way the precursor molecules react to one another as they assemble into graphene nanoribbons. This allows the scientists to control the nanoribbons' length, edge configuration and location on the substrate.

This new method is a stepping stone toward the production of self-assembled graphene devices that will vastly improve the performance of data storage circuits, batteries and electronics.

Paul Weiss, a member of UCLA's California NanoSystems Institute, developed the method for producing the nanoribbons with Patrick Han and Taro Hitosugi, professors at the Advanced Institute of Materials Research at Tohoku University in Sendai, Japan. The report appears in the journal ACS Nano.

Mil/Aero007 News Highlights

Ventec Supports ESA Proposal for IPC4101D's Appendix A

The proposed Appendix A, originally drafted in February 2013 and revised in September 2014, aims to define an enhanced quality standard for base materials used in high-reliability PCBs for critical applications, with particular reference to the prevention and detection of foreign material inclusions early in the supply chain.

i3 Nets Contract to Supply Substrates for Military Use

The company has announced that an industry leading aerospace and defense firm has awarded the company an order for the supply of advanced substrates for a military application.

Cutting Dynamics Employs Robot to Transform Business

Cutting Dynamics has joined the roster of advanced manufacturers using collaborative robotics to transform its business, adding a Baxter robot to a critical part of its thermal deburring line in its Avon, Ohio plant. Baxter works alongside the company's team of aerospace manufacturing engineers, automating the finishing process for its line of high-precision parts.

DARPA Unveils Tool to Identify Counterfeit Electronics

"The Advanced Scanning Optical Microscope—one of many IRIS-developed technologies—offers important hardware security and reliability assurance capabilities," said Kerry Bernstein, DARPA program manager. "These tools are optimized to support the mission of ensuring trust in microelectronics in DoD labs such as NSWC Crane."

New JEDEC Subcommittee: Focus on Emerging Tech

JEDEC Solid State Technology Association announced that its JC-13 Government Liaison Committee has formed a subcommittee focused on evaluating new and emerging electronic device technologies for potential future insertion in military, aerospace, and other special use applications.

CEA Lauds FAA's Ruling on Drones

The decision by the FAA is an important milestone as the agency develops rules to allow unmanned aircraft to operate safely in U.S. airspace.

Going Beyond X-rays for Advances in Imaging in the Field

Seeking to expand the nation's capability to detect and identify materials that are not easily visualized by conventional imaging technologies, DARPA released an announcement inviting proposals to develop portable, next-generation imaging tools that combine the complementary benefits of X-ray and neutron radiography.

Electronic Warfare Systems Market: $13.2B by 2024

The global electronic warfare systems market is expected to value US $10.2 billion by the end of 2014, which is estimated to increase to US $13.2 billion by 2024, representing a CAGR of 2.68% during the forecast period.

Military Unmanned Aerial Vehicles Market to Hit $5.26B

New analysis from Frost & Sullivan, "Global Military Unmanned Aerial Vehicles Market Assessment," finds that the market earned revenues of $3.87 billion in 2013 and estimates this to reach $5.26 billion in 2022.

U.S. GDP to Grow 3% by 2016; Defense Spending Rises

UCLA Anderson Forecast's third quarterly report of 2014 indicates that the real Gross Domestic Product for the United States will grow at approximately 3% over the next two years, following a decline of 2.1% in the first quarter of this year and a rebound of 4.2% growth in the second.

Flying Probes Expanded

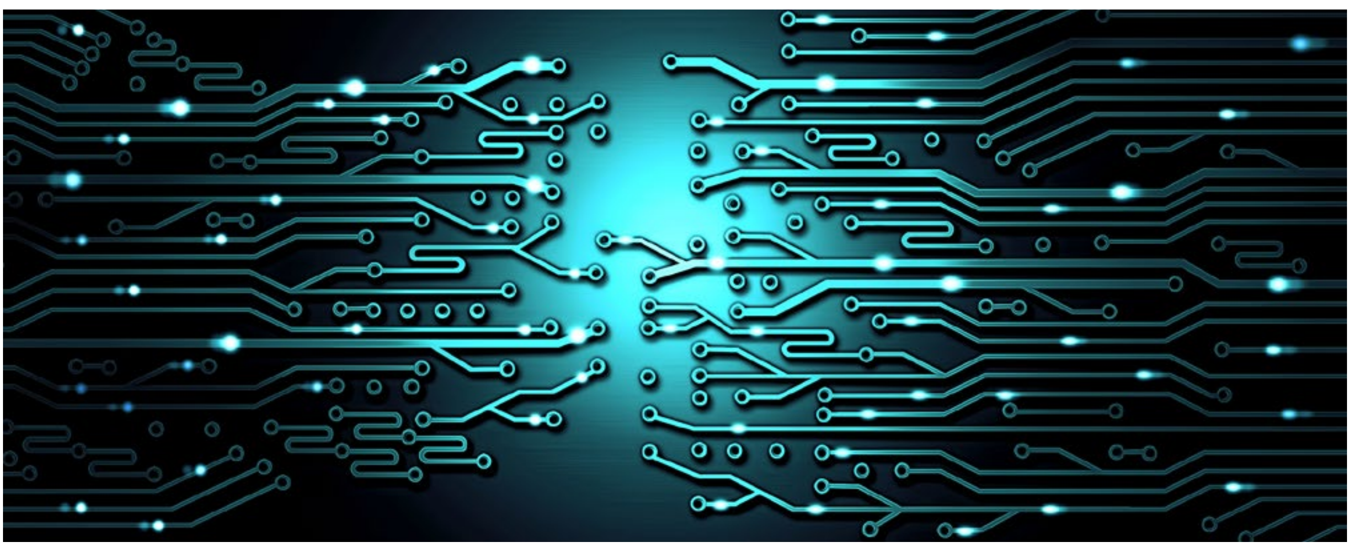
by Todd Kolmodin
GARDIEN SERVICES USA

Flying probes have been around for a long time now, but there are mysteries for some about how these machines really can perform the same as the "grandfathered" bed-of-nails machines. Although the "grid" has been known for years to provide the required electrical test, it has failed somewhat—due to pitch, density and cost—to provide the economical solution for both time and budgets. The flying probe has taken a new stance in the ET world, allowing operators to reduce cost and still provide expedient throughput in today's challenging ET/FA arenas.

This month let us explore the parameters of the flying probe vs. the bed of nails. Using either format we are in the end concerned with the electrical functionality of the individual PCB. In the electrical test theatre we are driven by the customer requirements, fabrication drawings or industry standards. This is a very important point. Fabrication drawings, PO documents or special requirements need to be stipulated in advance so that the PCB manufacturer can ensure that all requirements for the given PCB are met. This alleviates any unnecessary delays in the delivery of the product. There needs to be a full, open communication channel between the OEM and the PCB manufacturer regarding all requirements for the manufacture of the PCB. This is just not limited to the physical manufacturer but the final electrical attributes given copper lengths of any given net. This is crucial when a specific class is called for manufacture and physical attributes of given nets exceed thresholds for continuity for the given class. This leads to delays due to nets on the product outside of specification parameters. I digress, so let us move to the flying probe vs. grid.

Bed of Nails

These workhorses are still widely used today. They provide full simultaneous continuity and isolation (shorts) testing in one single pass. Albeit expensive, they have been the go-to test for a great many years. The negative now is the expense of adding fixture to today's designs. Pitch, density and continuity of design revisions make fixtures a rather large expense in the design cycle of a given PCB. But the bed of nails machines can perform tests extremely quickly. Many OEMs still call out "simultaneous test" in their specifications that require bed of nails test. This term means that the continuity and isolation test is done immediately. With a bed



of nails test, the difference from flying probe is that ALL nets are interrogated against one another regardless. Although extremely fast, it's rather unnecessary.

Flying Probe

Here we enter what could be seen as the new world for many OEMs including military. How is this machine going to give me the same results or test the same as my grid tester? New definitions boggle the mind: direct testing, indirect testing, adjacency and indirect testing with signature analysis! Well, that's why I'm here. Let's walk through this:

Direct Testing

The flying probe is going to perform a full resistance test on every net for continuity at the given threshold. (For reference see IPC-9252A w/Amendment 1.) This is no different than the bed of nails. However, when it comes to isolation testing the flying probe does this a little differently. The isolation test is performed using adjacency, which I will explain shortly.

Indirect Testing (Signature Analysis)

This is how the flying probes can compete against the bed of nails. This is known as capacitance test, discharge test, field test or phase test depending on the type of machine used. Rather than doing a full parametric test on all nodes in the PCB, the flying probe places a reference probe on a plane layer and reads all associated nets to the reference to develop a "master." When the first board is tested the reference is gathered and then a full continuity and adjacency test is performed. If the PCB tested is acceptable the master values are written. The subsequent boards are tested and compared to the master. Deviations from the master will evoke continuity or isolation verification.

The World of Adjacency

This is where the flying probe makes speed gains. Now with my colleagues in the industry this is where we have taken a leap of faith in reference to manufacturing. In the flying probe world we have made some calculations that we do not need to do every "net to net" test for shorts. This has been accepted by the DLA and also the IPC 9252 Task Force. Adjacency testing is defined to testing a single net to all nets adjacent within a given window. The industry standard currently is 1.27 mm or .050". This is where the grid test and flying probe part company. The grid test will test all primary nodes to ALL nodes on the PCB. The flying probe will only check nodes within the adjacency window, aka .050". With that said, we have to drill into the types of adjacency.

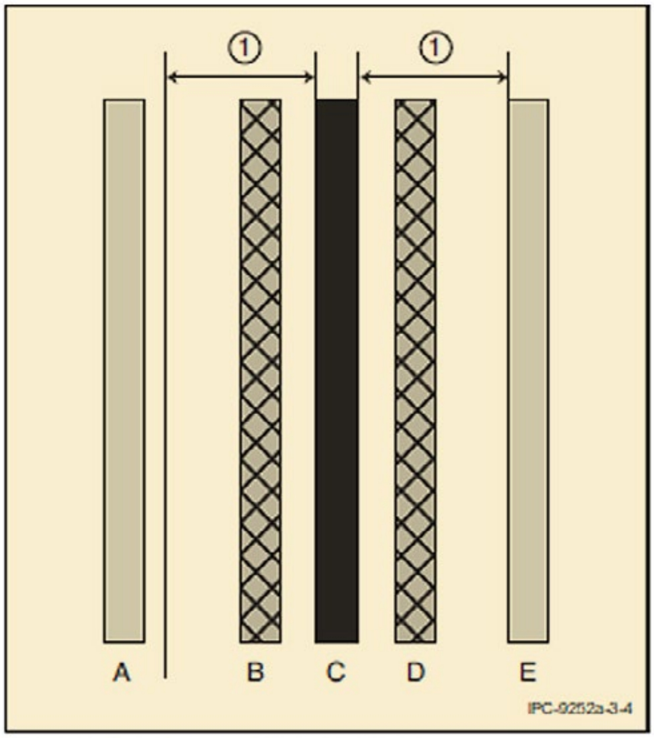


Figure 1: Line of sight adjacency. (Source: IPC)
1. 1.27 mm (0.050 in.).
2. Where line of sight adjacency distance is shown as 1.27 mm (0.050 in.), C is tested to B and D, but not to A or E.
3. A short from C to E should also be reported as a C to D and D to E short.

Line of Sight/Horizontal

When checking for discontinuity (shorts), the system scans the PCB on the horizontal plane. In short, Figure 1 applies. As you can see there is a leap of faith implied as if a catastrophic even were to occur the short from A to E would not be detected.

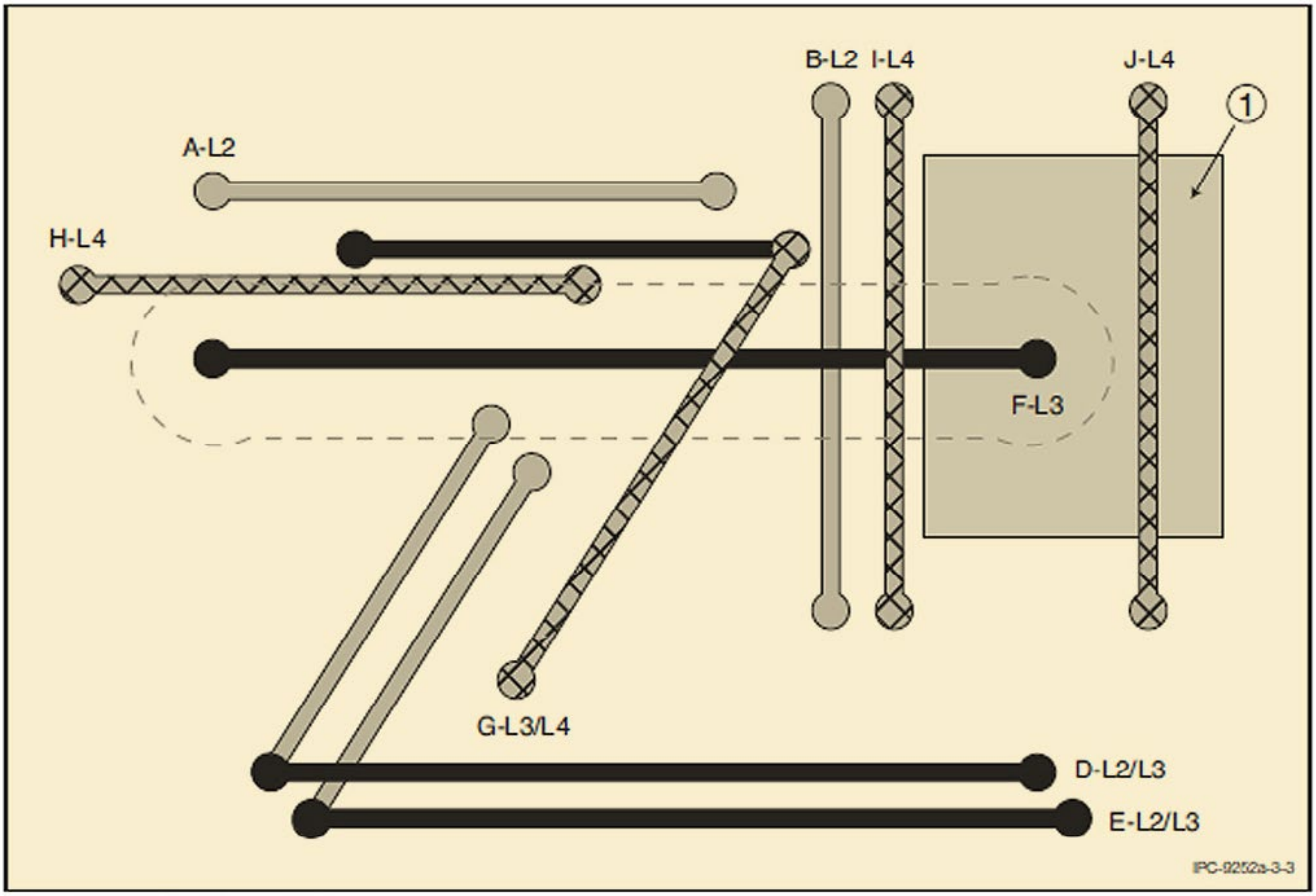


Figure 2: Vertical layer adjacency. (Source: IPC)
1. Grounded plane: C-L2.
2. Dotted area denotes vertical layer adjacency area for net F on layer L3.
3. Nets to be included in the vertical layer adjacency list for net F: B-L2, C-L2, D-L2/L3, G-L3/L4, H-L4, I-L4.

Z-Axis (3-Dimensional Adjacency)

In conjunction with horizontal adjacency, this methodology can be used. This will scan the horizontal (line of sight) with the addition of the Z-axis (3rd axis.) This method requires up-front information of the entire stack-up of the PCB. This is critical as an over measurement in the Z adjacency will have the flying probe perform isolation testing well above and beyond what is required resulting in excessive test time. Figure 2 outlines Z-axis.

From Here

How the OEMs wish their product validated will always be their decision. Currently, the bed of nails is still available but the cost outweighs the benefits of the flying probe. Cost of tooling is greatly reduced, speed of program repair is virtually painless and the quality of the PCB test is strong. **PCB**

References

1. Images copyright by IPC and IPC-9252A. Images appear with permission from IPC.

Todd Kolmodin is the vice president of quality for Gardien Services USA, and an expert in electrical test and reliability issues. To read past columns, or to contact Kolmodin, click here.

Solving a True DAM Problem

by Gray McQuarrie
HEI INC.

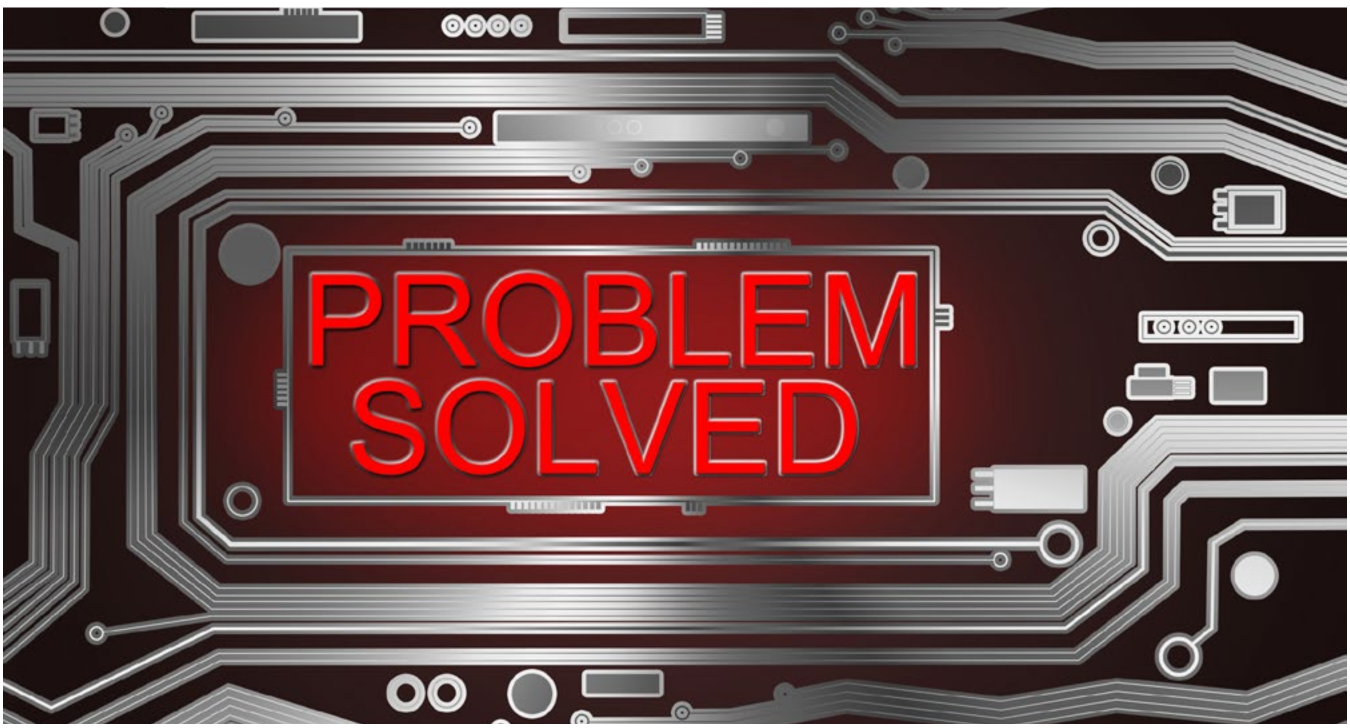
It's one thing to be a consultant where you don't have to really do anything, but quite another to be accountable to solve a truly difficult DAM problem. Anybody who has been on the firing line of a tough problem knows the feeling of vulnerability that results. Will I fail? And if I do fail, will I survive or be crushed? When I feel that way, I reach for the quotation that my cross-country coach posted on my dorm room door at St. Olaf College.

"The man who is actually in the arena, whose face is marred by dust and sweat and blood; who strives valiantly; who errs, who comes short again and again, because there is no effort without error and shortcoming; but who does actually strive to do the deeds; who knows great enthusiasms, the great devotions; who spends himself in a worthy cause; who at the best knows in the end the triumph of high achievement, and who at the worst, if he fails, at least fails while daring greatly, so that his place shall never be with those cold and timid souls who neither know victory nor defeat."

—Theodore Roosevelt

There is something called DMAIC: design, measure, analyze, improve, and control. It doesn't really work on solving truly difficult DAM problems. Now many of you may criticize me on this point, but I offer a counterpoint. There are only a few shops in the United States today that have figured out how to thrive with the onslaught of foreign competition and the resultant relentless price pressure. If DMAIC worked, then I submit most of us would have solved this problem. DMAIC may work for a certain class of problems, but for truly complex problems, where an answer has to be creatively generated out of the thin rarified air of true human ingenuity and innovation, DMAIC isn't enough.

Solving a truly difficult DAM problem requires, believe it or not, understanding humanity, and the quote by Theodore Roosevelt con-



the pcb list

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

EAGLE ELECTRONICS
Schaumburg, Illinois, U.S.A.

Website | Contact Me | RFQ

EAGLE electronics Inc.
MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

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

Eagle Electronics, Inc was founded in 1980. We are a quick turn to medium volume production facility. Layer count up to 32 layers, hybrid materials, sequential lamination for blind/buried vias, and aluminum core. ITAR registered and ISO 9001:2008.

Markets:	Automotive, Communication, Computers, Consumer, Industrial, Medical, Military/Aerospace
Board Types:	Single-sided, Double-sided, Multilayer, Rigid-Flex
Mfg Volumes:	Prototype, Small, Medium
Other Services:	Quick Turn-Around
Specialties:	Blind/buried vias, backplanes, carbon contacts, chip-on-board, controlled depth drilling, Controlled Impedance, Filled/ ugged vias, Heavy copper, Large format PCB, RF, quential lamination, Signal integrity, Sil th le, Thermal clad PCBs
	PC-6012, ISO 9001, ITAR, registere OHS compliant, UL

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

tains exactly the humanity required to solve any complex problem. You could write a book about it, and in fact a book has been written: *Daring Greatly,* by Brené Brown. In the book, Brené states, in essence, it is only when we can allow ourselves to be vulnerable that we discover innovative solutions.

You might ask that, since DMAIC doesn't work, what methodology can we follow to solve a true DAM problem? I present you with three statements, which you can use to develop your own homebrewed problem-solving methodology.

1) People always create problems. In order to solve a problem, you have to address the people part just as much as the technical part.

2) In order to solve a problem, people have to be inspired to solve the problem, knowing in the end that if they fail, they failed while daring greatly.

3) A tough problem cannot be solved with one human mind. It requires the engagement of a mastermind.

Let me explain these three statements in more detail, starting with the people part, and a real-world example taken from the annals of my own recent experience.

The group of engineers that report to me had a problem with a rigid-flex board that had inconsistent plating, and thus scrap due to voids. The reason for the voids was our inabil-

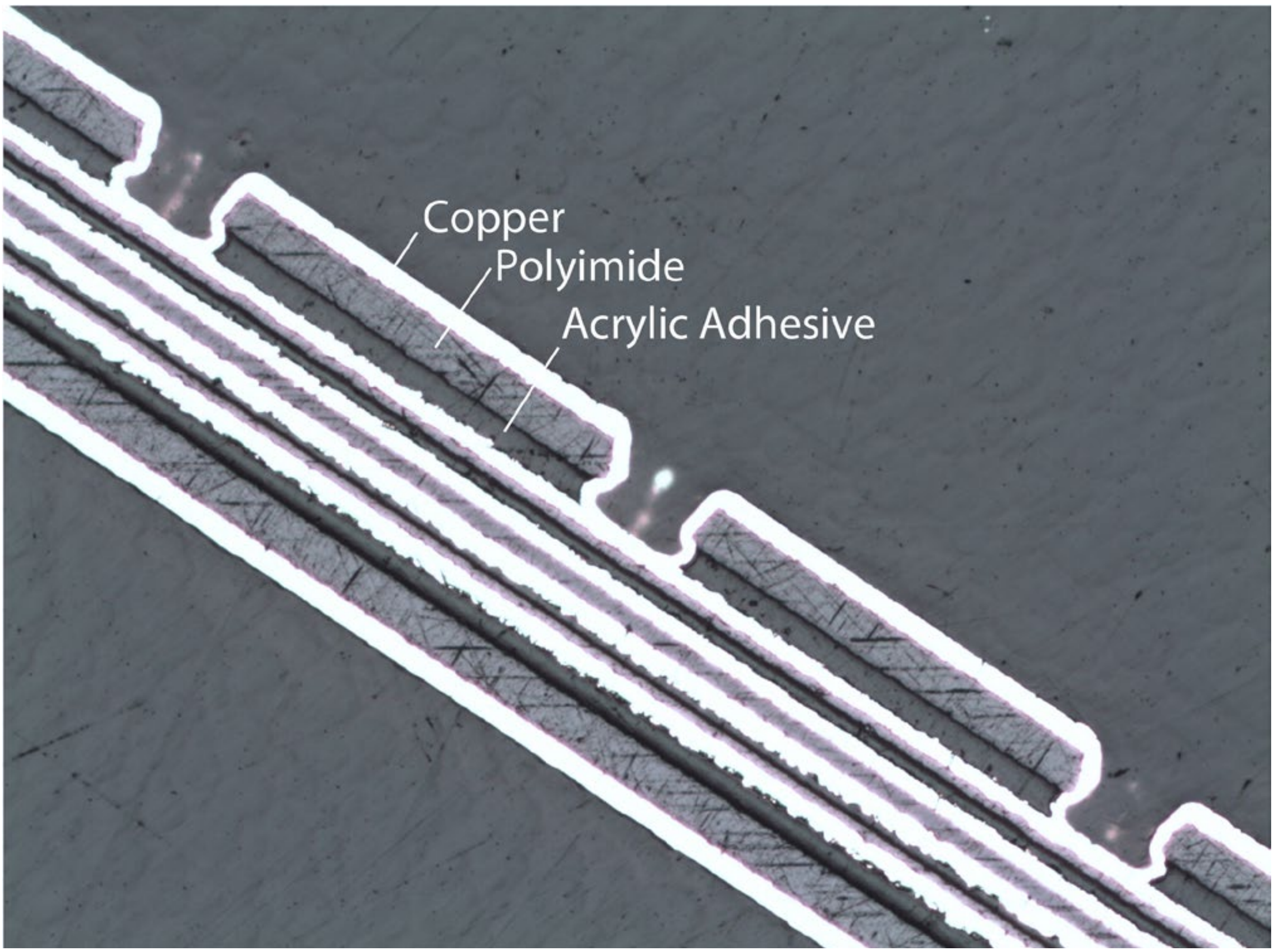


Figure 1: Test panel result showing improved plated vias.

ity to plate consistently on the acrylic adhesive layer. This had been a problem for almost a decade, and it is recognized within the rigid-flex community that this material is really difficult to plate.

Figure 1 and Figure 2 show the outstanding results of test panels that resulted from the discovered solution.

Figure 1 illustrates how the plating is the same on both the polyimide layer and the acrylic adhesive layer. This wasn't so prior to the discovery of the solution. The plating on the acrylic would always be considerably thinner or non existent.

With further improvements on real parts (not shown), the engineering team, behaving as a mastermind, was able to perfect the process such that plating occurred preferentially on the acrylic adhesive.

In order to get the result seen in the two figures, I had to demonstrate how ineffective the team members were as solely individual problem-solvers. For example, I asked them what had been learned from all of the years working on this problem. Eventually, the group admitted that nobody had a clue about the root cause of the problem.

Engineers are probably the most difficult people to manage, because too many of them will not ask for help, which is often related to their high level intelligence combined with their competitiveness. If they don't know the answer,

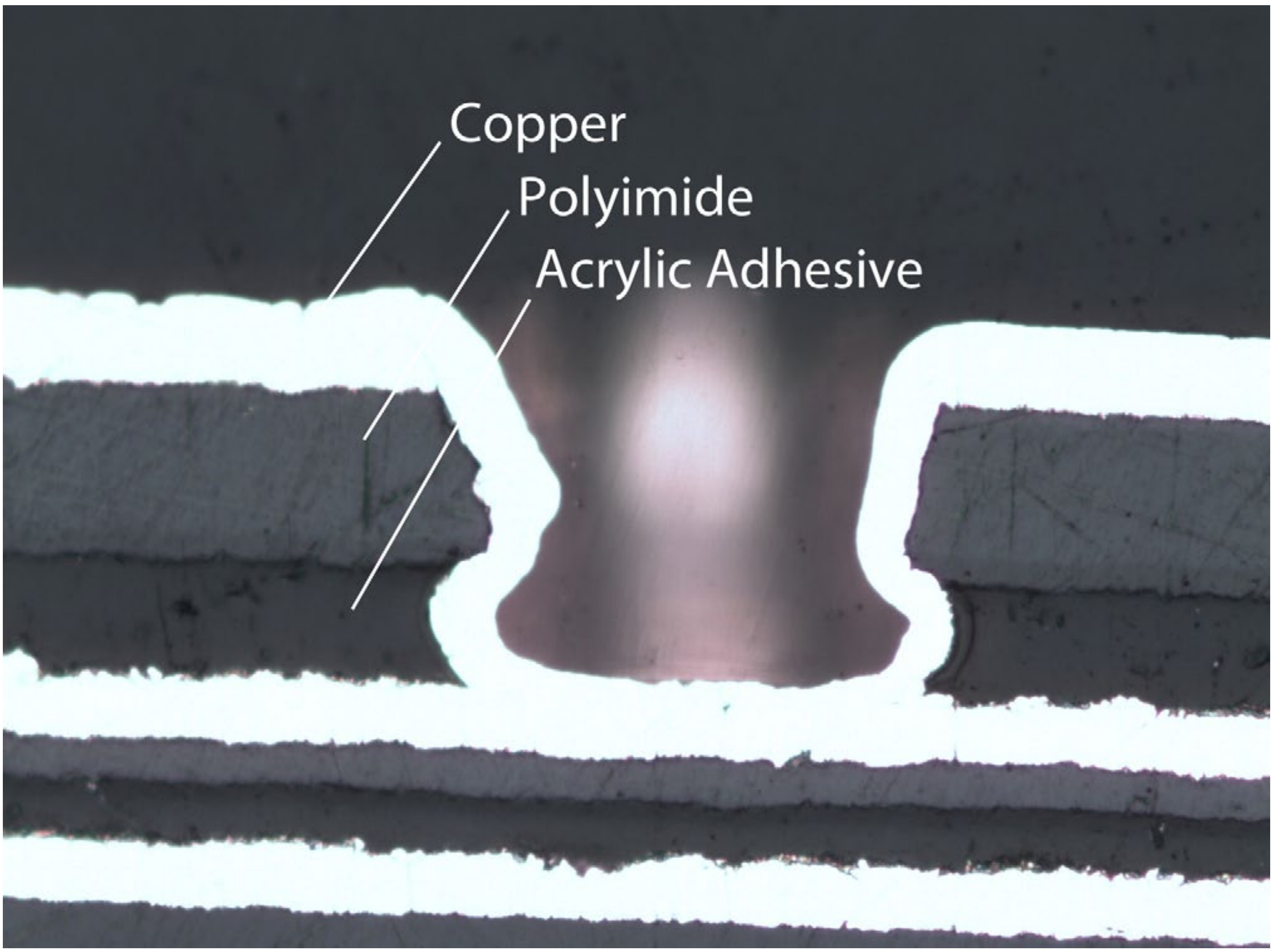


Figure 2: Close-up of a via.

they will often demonstrate their knowledge and intelligence by describing all sorts of facts and figures that have nothing to do with the problem, though it might sound like it. The intent is to show how much smarter they are than you. Then with this proof, QED, no solution exists. My first order of business was to ban this type of discussion.

Let's move on to the vulnerability part. If people don't trust each other, if they feel unsafe because something that they do or say could be used against them, then there will be no hope, because there will be nobody willing to stick out their necks and make an effort that is full of errors and shortcomings. Finding a solution to a complex problem entails going down a rough road of dead ends, false starts, mistakes, and blunders. But with each error-prone experience comes new learning and eventually the discoveries that will crush the problem. When people are unwilling to be vulnerable, because it is unsafe to do so, you will not have a team of problem-solvers. Instead you will have a collection of tortoises hidden away in their shells unable to do anything but collect their paychecks.

> ***"Finding a solution to a complex problem entails going down a rough road of dead ends, false starts, mistakes, and blunders. But with each error-prone experience comes new learning and eventually the discoveries that will crush the problem."***

Once the vulnerability issue was solved (and this took a great deal of work with my engineering team, which I will save for a future article), I had to inspire the group; we chose to call ourselves the cavemen. We physically knocked down walls in the plant, removed all of the cubes, put in tables, and made one large wall a giant whiteboard. This new cavern was a place where we could all meet for our problem-solving adventures as a band of cavemen brothers. And we absolutely saw what we did as an adventure. To get this thing rolling, I explained to each of them that they were about to embark upon a quest, not unlike Joseph Campell's "Hero's Journey." They would be very different people after they had slain their problem. In this adventure, they would experience individual growth. They would emerge smarter and be more confident.

At first the engineers were dubious, but with the natural interaction that occurred in our cave, and after finding that each failure led to a new understanding and perspective, they started to see for themselves that there was a solution to the DAM problem that they previously thought was completely and utterly unsolvable. With the first success came a contagious optimism, a belief that anything was possible by working and thinking together.

Let's look at the mastermind. Clearly, it is my belief that tough problems require asking for help, because one person can't do it. You might say that Albert Einstein is an example of someone doing it all by himself, but this is not true. Einstein asked for the help of Marcel Grossmann, a highly skilled mathematician, and they collaborated in the development of the general theory of relativity.

In order to have a mastermind with my engineers, I had to explain what the mastermind was and how to do it. I am not going to share all of the steps here, but I will mention a few things. First, a mastermind is a meeting of the minds. Second, why-based questions are banned not only in the mastermind, but in any discourse. Third, all of the facts concerning the problem must be presented accurately and be verifiable.

A while ago I wrote an article concerning my thoughts on why-based questions and how they do not get at the facts, but instead put people on the defensive. Suffice it to say that why-based questions are personal. On the other hand, what-based questions focus on the gathering of facts. If you don't believe me, just ask an operator, "Why did you do that?" Observe what happens and note how much information you received. Ask the same operator, "What did you do?" Again observe what happens and note how much information you received. I can't

Why choose Fein-Line?

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

After more than 44 years in the industry, just about everyone knows the name Dan (Baer) Feinberg.

There are few in our industry that have his experience and expertise in just about every aspect of the business. Dan knows what works, and perhaps more importantly, what doesn't.

When additional expertise is needed, or when senior management just needs an independent opinion, companies large and small alike get the guidance of Fein-Line Associates, especially in the areas of New Product Market Entry and M&A Due Diligence.

In our business, there's a lot on the line, not just sometimes, but every time. The line between success and the alternative can be razor thin. That's why winning companies choose Fein-Line time after time.

Dan (Baer) Feinberg
Fein-Line Associates

Sales Training General Marketing Assistance · New Product Market Entry
Merger and Acquisition Due Diligence · Printed Circuit Technology assessment
Market Research (It may work but will it sell) · Expert Witness assignments
Technology Transfer

Fein-Line Associates, Inc.
P.O. Box 73355, San Clemente, CA 92673-0112
(949) 498-7866
baer@feinline.com
www.feinline.com

stress it enough how important it is to stop with the whys, stop with knee jerk solutions, and spend time collecting all of the facts. If the facts going into the mastermind are junk, the solution will be junk too.

Another rule of the mastermind is to include supplier and vendor experts as full participating members of the mastermind. Too many engineers don't listen to their suppliers. This is an extremely arrogant thing to do and it often makes for stupid decisions that compound the problem. The wider the net you make the mastermind, the greater the intelligence of the mastermind. In other words, including operators, supervisors, managers, and other people who may know little about what we do is a very wise thing to do.

Finally, the last rule I want to mention is to ban any discussion about the past, other than documented facts. We have all been in meetings where this was heard: We tried this before and it didn't work. Our memories are extremely terrible. Watch this YouTube video where psychologist Elizabeth Loftus talks about false memories. One of the more provocative things she says is this: "Our memories are constructive. They are reconstructive. Memory works more like a Wikipedia page. You can go in there and change it. But so can other people."

For the astute reader, you may have noticed that I am now in the line of fire. I am no longer looking at things from the outside, but like you I am on the inside trying to make a difference in our industry. After all, the solution to all of our DAM problems is inside of our four walls. **PCB**

Gray McQuarrie is chief technology officer for HEI Inc. To contact McQuarrie, or to read past columns, click here.

Bioinspired Coating for Medical Devices Repels Blood, Bacteria

From joint replacements to cardiac implants and dialysis machines, medical devices enhance or save lives on a daily basis. However, any device implanted in the body or in contact with flowing blood faces two critical challenges that can threaten the life of the patient the device is meant to help: blood clotting and bacterial infection.

A team of Harvard scientists and engineers may have a solution. They developed a new surface coating for medical devices using materials already approved by the FDA. The coating repelled blood from more than 20 medically relevant substrates the team tested—made of plastic, glass, and metal—and also suppressed biofilm formation in a study reported in Nature Biotechnology.

The team implanted medical-grade tubing and catheters coated with the material in large blood vessels in pigs, and it prevented blood from clotting for at least eight hours without the use of blood thinners such as heparin, which is notorious for causing potentially lethal side-effects like excessive bleeding, yet often a necessary evil in medical treatments where clotting is a risk.

"Devising a way to prevent blood clotting without using anticoagulants is one of the holy grails in medicine," said senior author Donald Ingber, the Judah Folkman Professor of Vascular Biology at Harvard Medical School and Boston Children's Hospital, Professor of Bioengineering at Harvard School of Engineering and Applied Sciences (SEAS), and Founding Director of Harvard's Wyss Institute for Biologically Inspired Engineering.

Ucamco
Former Barco ETS

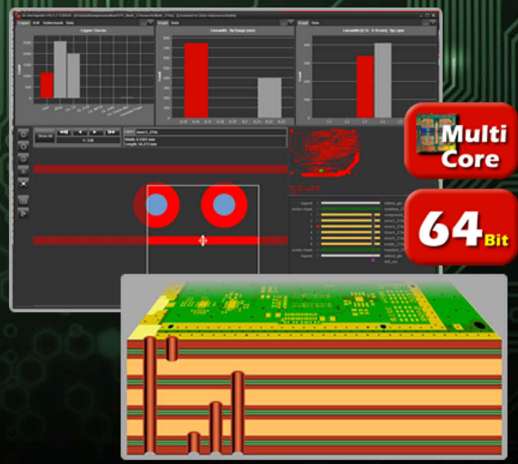
SOFTWARE | PHOTOPLOTTERS | DIRECT IMAGERS | NEWS | SERVICES | COMPANY | DOWNLOADS | CONTACT

SOFTWARE

Integr8tor

- Ucamco's 8th Generation Sales & Engineering Tool
- Automates data entry and design analysis in a pre-CAM environment
- Field proven with more than 250 installations in daily production
- Seamless integration into any environment

READ MORE



SOFTWARE

UcamX

- New ground-breaking CAM Software for Rigid, Flex and HDI PCB's
- Wait times on complex algorithms slashed by up to 80% (8-core)
- State-of-the-art intuitive user interface
- Top-notch performance by using parallel processing on today's multicore processors

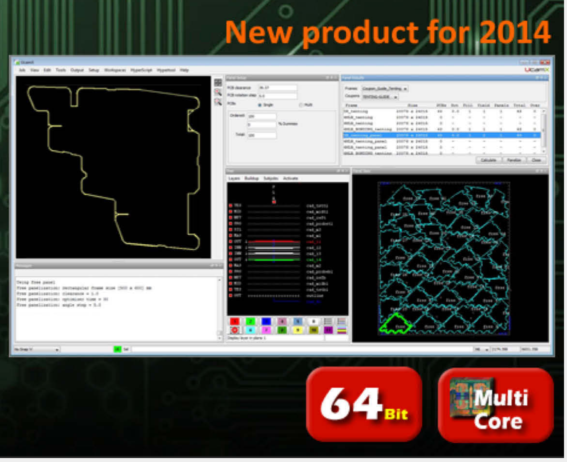
READ MORE

SOFTWARE

UcamX Paneliz8tor

- Create Panels with optimized nesting of irregularly shaped Flex and Rigid PCBs
- Save time and reduce your material cost
- Browse and select your optimum yield Panel
- Update Panels quickly en easily with last-minute layout changes

READ MORE



For more information please contact info@ucamco.com
Call (32) 9-216-9900 or check out our new website at www.ucamco.com

TOP TEN

PCB007 News Highlights This Month

1 PCB Industry Slowdown Continues in August

"August business results for the North American PCB industry continued slightly below last year's levels," said Sharon Starr, IPC's director of market research, "but the PCB book-to-bill ratio continued hovering near parity and actually strengthened a bit in August. Flat growth is still expected for the remainder of 2014."

2 Colonial Assumes Vermont Circuits' Customer Base

Mark Osborn, president and founder of Colonial Circuits, announced recently that his company has acquired Vermont Circuits' customer base and will be offering a smooth transition for those former customers.

3 Wurth Elektronik's Flex PCBs Achieve UL Mark

After a time-intensive and close coordination with the UL engineers, an enormous amount of test patterns and high costs, Würth Elektronik was able to record a big achievement: Four flex-rigid PCBs received the UL Listing.

4 Viasystems Challenges EPA's CDR Cycle

Congressman Bill Johnson (R-OH) sent a letter to U.S. Environmental Protection Agency (EPA) Administrator Gina McCarthy requesting an analysis of reporting data pertaining to by-products sent for recycling collected during the 2012 Chemical Data Reporting (CDR) reporting cycle.

5 IPC: PCB Market Down 2.2%; Production Value at $59.4B

The world market for PCBs declined an estimated 2.2% in real terms in 2013, although real growth in North America was positive at 0.8%, according to IPC's "World PCB Production Report for the Year 2013."

6 Q3 PCB/EMS Market Snapshot

I-Connect007's latest market survey—a one-minute survey designed to provide a snapshot of the current state of the PCB and EMS industries—offers a good sampling to give us a snapshot of where we've been and where we're headed.

7 FTG's Circuits Segment Sales Up $1.9M in Q3

"FTG's momentum has continued in Q3 2014 with strong results across the corporation, particularly at our Circuits business and the two new Aerospace facilities in Tianjin and Chatsworth where we continued to see progress on qualification activities, strong orders, and increased shipments," stated Brad Bourne, president and CEO.

8 IPC Report Shows Moderate Growth Through 2017

Among PCB product types, HDI/microvia boards experienced the highest growth rate in 2013. Data on vertical markets for PCBs show that communications and military/aerospace are the two largest PCB markets in North America, together accounting for approximately 57% of the North American PCB market in 2013.

9 Germany Records Highest PCB Sales Since 2011

The automotive electronics sector drove Germany's PCB market in July, helping the industry record its highest sales since 2011, according to ZVEI.

10 Aspocomp Consolidates Operation: May Close Teuva Plant

As part of Aspocomp's strategic transformation, the company has issued a notice on statutory labor co-determination concerning all its personnel at the Teuva plant regarding the consolidation of production in Finland and the possible closing of the Teuva plant.

EVENTS

For the IPC Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For the iNEMI Calendar of Events, click here.

For the complete PCB007 Calendar of Events, click here.

LaserJob Technology Forum
November 6, 2014
Furstenfeld, Germany

LA/Orange County Expo & Tech Forum
November 6, 2014
Long Beach, California, USA

36th International Electronics Manufacturing Technology Conference
November 11–13, 2014
Johor, Malaysia

International Wafer-Level Packaging Conference
November 11–13, 2014
San Jose, California, USA

TSensors Summit 2014—San Diego
November 12–13, 2014
La Jolla, California, USA

Wearable Sensors and Electronics 2014
November 12–13, 2014
Santa Clara, California, USA

ELECTRONICA 2014
November 11–14, 2014
Messe Munchen, Germany

High-Reliability Cleaning and Conformal Coating Conference
November 18–20, 2014
Schaumburg, Illinois, USA

Graphene LIVE! 2014
November 19–20, 2014
Santa Clara, California, USA

International Printed Circuit & APEX South China Fair
December 3–5, 2014
Shenzhen, China

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

PUBLISHER: **RAY RASMUSSEN**
(916) 337-4402; ray@iconnect007.com

SALES: **ANGELA ALEXANDER**
(408) 915-2198; angela@iconnect007.com

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

EDITORIAL:
GROUP EDITORIAL DIRECTOR: **RAY RASMUSSEN**
(916) 337-4402; ray@iconnect007.com

MANAGING EDITOR: **LISA LUCKE**
(209) 304-4011; lisa@iconnect007.com

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

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **MIKE RADOGNA**
mike@iconnect007.com
MAGAZINE LAYOUT: **RON MEOGROSSI**
AD DESIGN: **MIKE RADOGNA, SHELLY STEIN**
INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**
COVER: **SHELLY STEIN**

PCB007 Presents

The PCB Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138

November 2014, Volume 4, Number 11 • The PCB Magazine© is published monthly, by BR Publishing, Inc.

MASTHEAD

ADVERTISER INDEX

Coming Soon to *The PCB Magazine:*

December:
Industry Outlook: 2015
In our end-of-the-year issue, experts from around the globe will weigh in on what to expect in 2015. Don't miss it!