PCB007 PRESENTS

12 Flex and Flex-Rigid PCB Technologies and Trends
by Keith Netting

30 Circuit Flexibility (and How to Achieve it)
by Joe Fjelstad

40 National Healthcare Policy Drives Flex Innovation to Meet Demand for Home Medical Electronics
by Dick Pirozzolo

52 Conversations with... Printed Circuits Inc.
by Marcy LaRont

September 2012

the pcb magazine

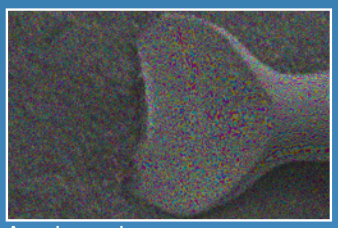
AN I-CONNECT007 PUBLICATION

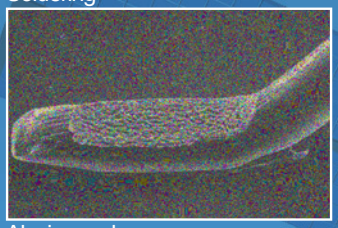
THE RISE OF FLEXiBLE CiRCUiTS

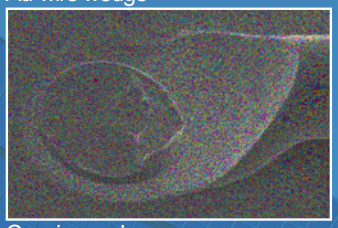
www.atotech.com
ATOTECH
Technology for Tomorrow's Solutions

Final Finishing
Universal Finish SolderBond (ENEPIG and ENEP) for Soldering, Au-Wire and Cu-Wire Bonding

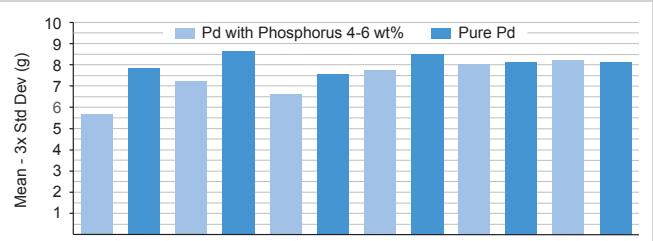
Soldering


Au-wire wedge


Al-wire wedge


Cu-wire wedge

Gold Wire Bond Pull Test

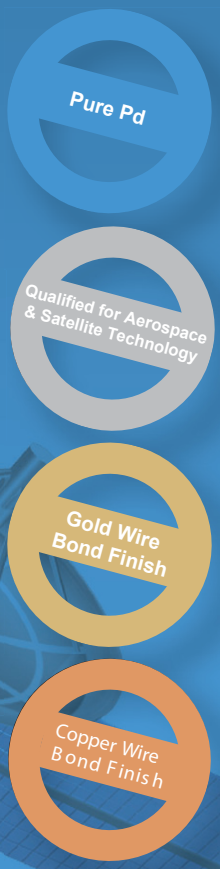


Process	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB
Pd Layer	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd
Nickel µm	6.0	6.0	6.0	6.0	7.0	7.0	7.0	7.0	7.0	7.0	7.0	7.0
Pd µm	0.05	0.05	0.15	0.15	0.25	0.25	0.05	0.05	0.15	0.15	0.25	0.25
Au µm	0.04	0.04	0.04	0.04	0.04	0.04	0.15	0.15	0.15	0.15	0.25	0.15

Universal Finish SolderBond provides also with low Au-thickness excellent results, as pure Pd is 50% softer than Palladium-Phosphor (4-6wt%).

Process Flow

1. Cleaner
   Pro Select UC
2. Micro Etch
   MicroEtch C
3. Dip
   Aurodip (Option)
4. Pre Dip
5. Activator
   Aurotech Activator 1000
6. Fine Line Post Dip
   Aurotech FL Plus Post Dip (Option)
7. Electroless Ni
   Aurotech CNN mod.
8. Electroless Pd
   PD-Tech
9. Immersion Au
   Aurotech SF Plus



	NiPdAu (ENEPIG)	NiPd (ENEP)
Deposit Structure		
Electroless Nickel	4.0 µm - 6.0 µm	4.0 µm - 6.0 µm
Electroless Palladium	0.05 µm - 0.3 µm	0.2 µm - 0.3 µm
Immersion Au	0.02 µm - 0.06 µm	--
Functionality		
Soldering		
Eutectic Sn/Pb	Good	Good
Sn-Ag-Cu Alloy	Excellent	Excellent
Contact Switching	Yes	--
Al Wire Bonding	Yes	--
Au Wire Bonding	Yes	--
Cu Wire Bonding	--	Yes

Features and Bene ts

- Three finishes (ENIG, ENEPIG and ENEP) in one process
- ENEPIG is Au-wire and Al-wire bondable
- ENEP is Cu-wire bondable
- Wider Au-wire bond process window compared to ENEPIG with PdP
- Pure Pd-layer without phosphorus co-deposition
- Pure Pd is 50% softer than PdP
- Superior solder joint reliability
- Excellent reliability regarding thermocycling, hot storage test and humidity
- For wirebond processes of today and tomorrow

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

TOTAL
ATOTECH

the pcb list

The best way to find a pcb fabricator, anywhere.

Features include:

- Free access to the most comprehensive directory of PCB fabricators worldwide
- Intuitive navigation
- In-depth profiles of PCB fabricators
- Multiple search capabilities including board type, material, region and more!
- The only source OEMs and fabricators need to connect!
- Accessible anywhere, anytime!

www.thepcblist.com

PCB007 PRESENTS

the pcb magazine

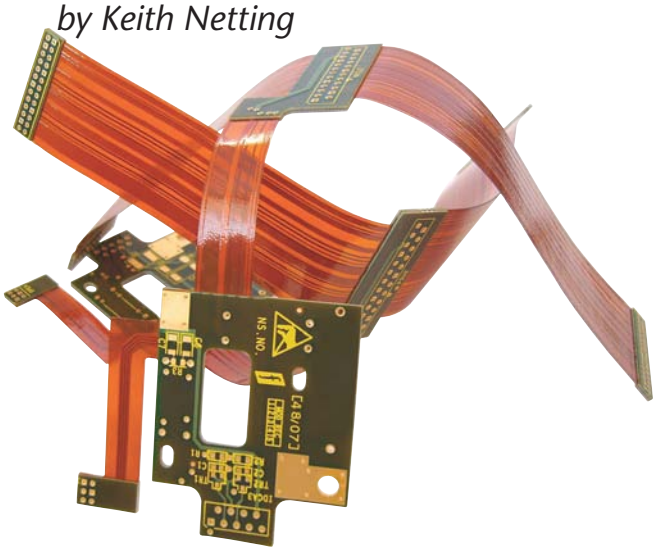
AN I-CONNECT007 PUBLICATION

SEPTEMBER FEATURED CONTENT

FLEX & RIGID-FLEX

They bend! They stretch! They leap tall buildings! Okay, maybe not that last one. Nevertheless, flex and rigid-flex circuits have myriad applications in just about every sector of the electronics industry, where their ability to be formed into complex three-dimensional final geometries, or withstand multiple flexing cycles during their functional life, adds a third dimension to the concept of the interconnecting substrate. This month, experts in the field explore the latest developments in materials and manufacturing processes for flexible circuits, as well as novel design and application opportunities.

12 Flex and Flex-Rigid PCB Technologies and Trends
by Keith Netting



30 Circuit Flexibility (and How to Achieve it)
by Joe Fjelstad

40 National Healthcare Policy Drives Flex Innovation to Meet Demand for Home Medical Electronics
by Dick Pirozzolo

52 Conversations with... Printed Circuits Inc.
by Marcy LaRont

66 Stevenage Circuits Discusses Flex PCB Trends, Issues and Developments
by Richard Ayes

THE Future OF Surface Finish

SPF™ (Semblant Plasma Finish)

- Replaces or Enhances Gold/Silver
- Reduces Process Costs
- Environmentally Responsible
- Mitigate Creep Corrosion
- Pb-Free Solderability
- Flux Containment

Semblant

semblantglobal.com

PCB007 PRESENTS

SEPTEMBER 2012
VOLUME 2
NUMBER 9

THE DEFINITIVE INTERACTIVE MAGAZINE DEDICATED TO THE GLOBAL PCB INDUSTRY

thepcbmagazine.com

the pcb magazine

AN I-CONNECT007 PUBLICATION

FLEX AND RIGID FLEX

CONTENTS

ARTICLES

76 **New Challenges for Higher Aspect Ratio: Filling Through-holes and Blind Microvias with Copper by Reverse Pulse Plating**
by Mike Palazzola, Nina Dambrowsky, Christof Erben, Stephen Kenny, and Bernd Roelfs

90 **A Time-Dependent Analysis of Heat Transfer in a PCB During a Thermal Excursion**
by J. Lee Parker

VIDEO INTERVIEWS

22 **LED Market Drives Demand for New Laminates**

45 **Uyemura Investing and Growing in PCB Industry**

65 **Overcoming Challenges in Laminate Development**

SHORTS

45 **Breakthrough in Ferroelectric Materials**

60 **Low-Cost Micron Copper Ink for Electronic Use**

EXTRAS

98 **Events Calendar**

99 **Advertisers Index & Masthead**

COLUMNS

8 **It's Worse Than I Thought**
by Ray Rasmussen

24 **Critical Placement**
by Barry Olney

34 **Little Things Have the Power to Break Big DAMs**
by Gray McQuarrie

48 **Copper Surface Preparation for Dry Film Lamination, Part E**
by Karl Dietz

62 **Will "Old" PCB Technology Save American Manufacturing?**
by Steve Williams

72 **Interconnect Defect (ICD): Getting to the Root Cause**
by Mike Carano

NEWS HIGHLIGHTS

32 **Market**

46 **Flex007**

60 **PCBDesign007**

70 **Mil/Aero007**

88 **Supplier/New Products**

96 **Top Ten Most-Read News Highlights from PCB007 This Month**

Join the Team!

FujiFilm + Your Company + National Graphic Supply

FUJIFILM

- Superior Dimensional Stability Coefficients
- Exceptional Line Edge Quality
- 4 films to match your imaging needs
- Long Term Commitment to the PCB Industry

- The Best Technical Support
- 9 Warehouses with 14 Technical Specialists
- Programs for:
  - Process control - SPC
  - Silver recovery
  - Image Quality Management

Contact us today to find out how joining the team can support your company's short and long term imaging needs!

National Graphic Supply

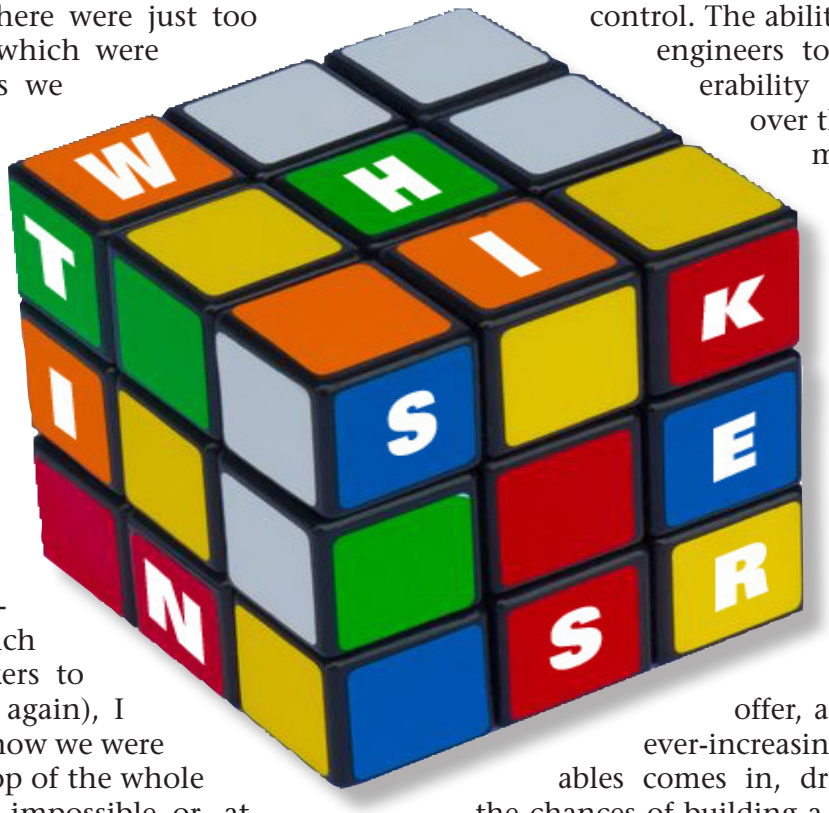
(800) 779-5657 | (909) 989-5200 | www.ngspcb.com

It's Worse Than I Thought

by Ray Rasmussen
I-CONNECT007

SUMMARY: A solution to the problem of tin whiskers, at times, seems a far-off concept, but surely the problems caused by lead-free requirements and SAC solders can be solved, right? A recent phone conversation with an engineer made it clear to Publisher Ray Rasmussen that the situation is quite dire.

A few months ago I wrote a column about the IPC Tin Whiskers Conference I attended in April. The column, "Chasing Our Tails," appeared in the June 2012 issue of *SMT Magazine* and, from my 10,000-foot view, I couldn't see how we were ever going to get this issue under control. There were just too many variables, which were ever-increasing as we learned more about SAC solders and their shortcomings as they relate to high-reliability electronics. Maybe, someday, we'll find a solution to the actual whiskers problem, but as I looked at all the issues surrounding lead-free (which caused tin whiskers to become an issue again), I just couldn't see how we were going to get on top of the whole thing. It seemed impossible or, at least, highly unlikely to me.



As a result of that column, I received a few comments, one of which turned into a phone conversation that left me stunned.

This issue is much worse than I thought.

A recently retired aerospace engineer took the time to walk me through the long history of the problems associated with, and being compounded by, lead-free SAC solders. He and his team had been working on solving these problems well before the lead-free mandate came into effect. What I took away from the conversation was that even with tin/lead solders, the challenges to building these complex and very critical systems were many.

The issues with traditional solders were well known, but a few surprises still lurked. But with lead-free, the challenges were becoming so daunting that things were getting out of control. The ability of these aerospace engineers to guarantee the operability of critical systems over the long term was diminishing. Yes, they are exempt from RoHS, for now, but lead-free regulations, combined with the commercial off-the-shelf (COTS) mandate, have put the high-reliability guys in a box. To build the systems of the future, they need to plug into the latest technology the supply chain has to offer, and that's where this ever-increasing number of variables comes in, dramatically reducing the chances of building a consistently reliable product.

Certified Systems

The good news is that aerospace requirements all but ensure that systems won't fail in

What are you losing in lost scrap?

At Stebgo Metals when you send us your manufacturing waste you can trust we'll do everything to get the most precious metal from your scrap to refine and return every ounce back to you with honesty and integrity. Trust the Stebgo advantages for your scrap refining and we'll help you stop throwing money away.

The Stebgo Advantages

- Certified independent scales
- Accurate measurement and reporting
- 99.999% refining of precious metals
- Fully insured
- FREE Stone Removal
- FREE assay on first job
- Advances, E-Payments & Market Lock Options
- Fast Turnaround
  99% of solids jobs settled within 4 days

TRUSTED REFINERS SINCE 1978

Stebgo *Metals*

800-289-0138 / www.stebgo.com

250 Bridgepoint Drive, South St. Paul, MN 55075

the air. We don't have to worry too much about planes dropping out of the sky or military systems failing in the field. Very tough certification processes are required before a new system can come online. That was reassuring to hear. The real trouble will come when the system builders aren't able to meet the requirements of the military aerospace systems of the near future. It's becoming increasingly difficult to get these systems certified. Rather than working with a set of known materials and properties, lead-free variables are causing the industry to reinvestigate just about every piece in the supply chain, which basically means starting over after more than 50 years of known good processes and materials.

The Manhattan Project

It was suggested at a conference in Europe four or five years ago that the high-reliability industry gather and focus on finding solutions to the critical issues by creating a Manhattan-like Project. After some discussions, they decided to move forward. A two-week meeting followed and top engineers from over a dozen major mil/aero OEMs came up with a plan. The only problem was that they needed $125 million to do the work. That's where it sits today: waiting for funding. Just think: this problem would likely be behind us by now if they had acted back then. Too bad.

It's Political

After listening to my new aerospace engineer friend for more than an hour as he peeled back layer after layer of the reliability onion, I offered up that the problem was political in nature, not technical. He agreed. If there were sufficient resources, they—the engineers and scientists—could solve these problems. And, since most materials suppliers cater to the other 90% of the industry with volume production, the high-reliability guys were going to have to solve this on their own. But, with program managers under substantial pressure to keep costs down, they aren't looking to spend too much money investigating new ways to produce their products.

Moving to COTS was supposed to help keep costs down, which would have been the case if RoHS hadn't been legislated. Now things are starting to come unglued. I grabbed the following from a PERM Consortium white paper, which puts the issues in a nutshell. PERM is the aerospace industry's effort to get on top of this.

The reliability of SnPb interconnections is well known and meets the requirements of these more demanding applications. Based on the scientific information available today, there are increased reliability risks in using Pb-free in high-performance electronics. These risks include the spontaneous formation of tin whiskers from Pb-free tin (Sn) based finishes, reduced Pb-free solder joint integrity, reduced reliability by cross-contamination between the different alloys, and the potential component and board damage from the higher Pb-free processing temperatures.

Most of us know about these problems if we're building high-reliability electronics. But even the consumer guys don't want reliability issues. They need their products to be reliable for at least a couple years, so they have an interest in many of the same issues. Of course, people don't die if an iPod stops working (unless it's my daughter), so the consumer industry doesn't face these same pressures. They're more interested in keeping costs down. Most consumer electronics don't have a useful life beyond five years in any case. They aren't going to push their suppliers to ensure reliability 10 to 20 years out. They don't care.

Getting Smaller

The drive to get lighter and smaller yet still be reliable is relentless. This was already putting pressure on the high-reliability industry before RoHS. With the great benefits of *smaller, lighter* and *faster* come problems. We have smaller spaces between leads and pads, which makes it even easier for whiskers to cause shorts and for voids to appear and joints to crack; smaller chip geometries (20nm), I'm told, are leading to long-term reliability issues with the chips themselves, which is something I wasn't aware of. Add to that the problem with tin/lead-based legacy systems and counterfeit components and you can see the complexity of the issues.

The 10% Solution

The high-reliability guys are trapped more and more in the consumer electronics world, which is driving technology and innovation. Unless the high-reliability industry gives up on COTS, they'll be subject more frequently to the shortcomings of these technologies as they try to produce longer-term reliable systems.

If you match up the optimism and future-casting from a Steve DeWaters article on military photonics to this conversation I had with the aerospace engineer working in the trenches, there's a huge gulf. The military has incredible requirements for electronic systems going forward. The systems are very complex, able to crunch tons of data, running complicated battlefield scenarios meshing all types of real-time data into warfighter displays, command and control systems, weapons and much more. And, because of the politics and financial pressures associated, that chasm will get bigger and bigger.

The good news, paradoxically, is that at some point we won't be able to build what's needed. That's when we'll have the attention of top brass at the Pentagon along with the major politicians. Based on that early morning conversation, without a crisis, we aren't going to be able to bring the resources to bear. Maybe that's the way it's ultimately supposed to happen. Only then will we have the complete attention of all the major players and access to all the money needed.

What's Missing?

Courage. There's a lot of bureaucracy in these large defense and aerospace companies. Of course, the Pentagon is one of the inventors of bureaucratic systems that stifle the creative and innovative process. So we're in a tough spot. Nobody wants to rock the boat, especially when it's sinking. It seems like a lot of folks know the boat's taking on water, but are content to just float along, continuing to bail water as needed.

I do hope more folks will step up and talk about this issue. I know it's risky for some, but we have to get on top of it as soon as we can. We literally have a lot riding on this. **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*. Contact Rasmussen here.

Flex and Flex-Rigid PCB Technologies and Trends

by Keith Netting
TEKNOFLEX

SUMMARY: Offering all of the benefit of rigid PCBs and wire harnesses combined, flex-rigid technology provides an improvement in reliability for applications in aerospace, medical, computer, telecom and automotive market sectors.

Flex and flex-rigid multilayer circuits (FRML) allow interconnection to take a 3D format that is not possible with standard, rigid PCB technology. Whilst rigid PCBs can stack many layers, they are not flexible and therefore, cannot be packaged as densely as a flex or flex-rigid multilayer circuit within a fully assembled article. Furthermore, flex and flex-rigid multilayer's can be bent on a repeated basis in dynamic applications, which is currently impossible with the materials utilised for rigid PCBs.

FRMLs offer all of the benefits of rigid PCBs and wire harnesses, combined. Additionally, flex and flex-rigid technology can enable the reduction of the number of interconnections within an assembly. This latter benefit has been shown, over time, to provide an improvement in reliability for a number of applications in the aerospace, medical, computer, telecom and automotive market sectors.

In demanding applications, vibration may affect the integrity of the interconnection between rigid PCBs. For example, when a stack of rigid PCBs are connected using arrays of press-fit pins, the whole assembly can behave as a coupled mass when subjected to vibrational forces. Being metallic, these pins have a higher specific density than the mostly polymeric circuit board materials. As the number of pins in the array increases, so can the mass. Therefore, due to the potential lack of damping within this assembled structure, the press-fit joints or the circuit board itself can become a weak link, resulting in a degradation of the contact integrity over time.

Temperature and environmental effects could potentially accelerate these effects. If a flex-rigid multilayer assembly or even flex circuits connecting between rigid PCBs are used to replace this approach, the more mechanically compliant and lightweight materials in the flex hinge will act to dampen any vibrational forces and will reduce the total mass of the circuit assembly into smaller connected masses, thus reducing the stresses the boards are subjected to under the vibrational stress. Furthermore, the modal frequency of the assembly is also disrupted by using this approach, meaning that the assembled unit could be less susceptible to vibration-related failure in a specific

Figure 1: Flex-rigid multilayer for military E-scan radar application with 27 conducting layers.

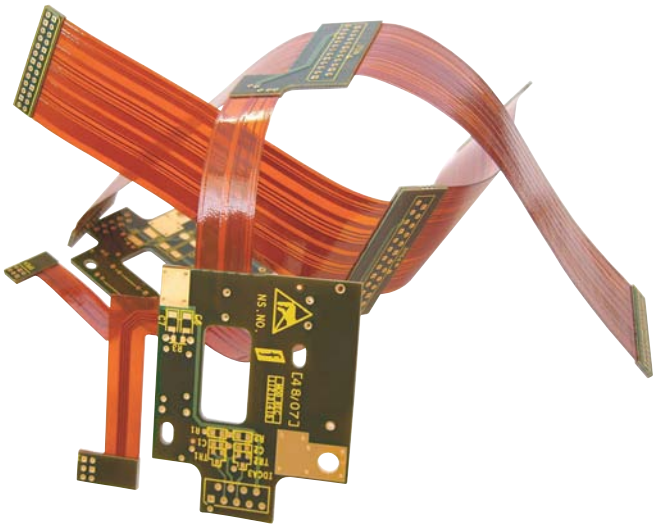


Figure 2: 6-layer flex-rigid multilayer for military sight application.

Circuit Technology	Potential Advantages	Potential Disadvantages
Printed circuit board	Cheap Stable substrate Standard technology Fast leadtime	Only 2D in assembly Only static connections possible Increased interconnections Vibration risks Higher mass
Flex circuit	3D in assembly Static & Dynamic connections possible Potential to reduce interconnections Lower mass (~30% of cables)	Less stable substrate
Flex-rigid multilayer circuit	Lowest number of interconnections Static & Dynamic connections possible Stable substrate Intermediate technology Optimised technology Can combine technologies	Leadtime Cost

Table 1: Relative advantages and disadvantages of various circuit technologies.

frequency range. If the circuit design is unfortunate enough to have a modal frequency in the range of operation of the final unit, then the vibration-induced stresses can become significant and possibly precipitate early failure if not disrupted by the use of flex and flex-rigid technology. The environmentally integrated nature of flex and flex-rigid means that multiple interconnects as used on rigid PCBs can be reduced, thus reducing risks due to environmental degradation such as moisture or thermal oxidation.

The relative advantages and disadvantages of rigid PCB, flex, and FRML technology are summarised in Table 1.

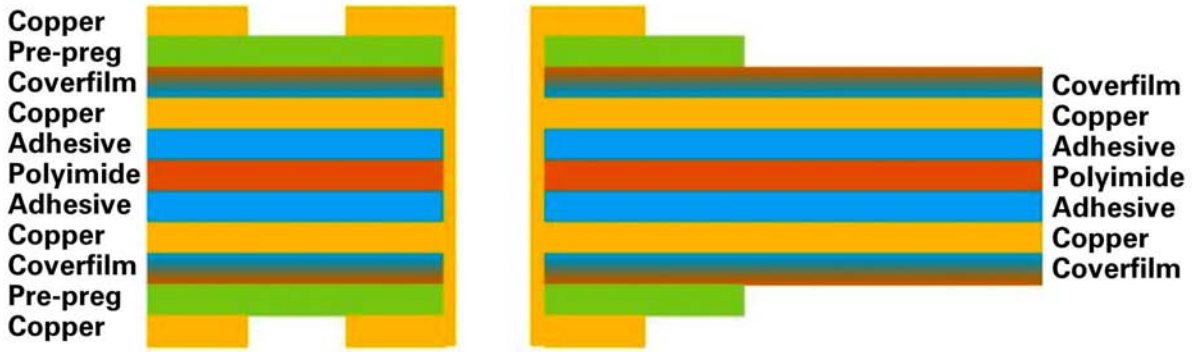
Many electronic applications require both 3D and dynamic capability. Examples include:

- Cameras
- Laptops
- Automotive systems, e.g., wing mirrors and HEV control systems
- Electrical/electronic actuators
- Imaging systems—Industrial, Security and Defence
- Sensors
- Power connections
- High-speed signal connections

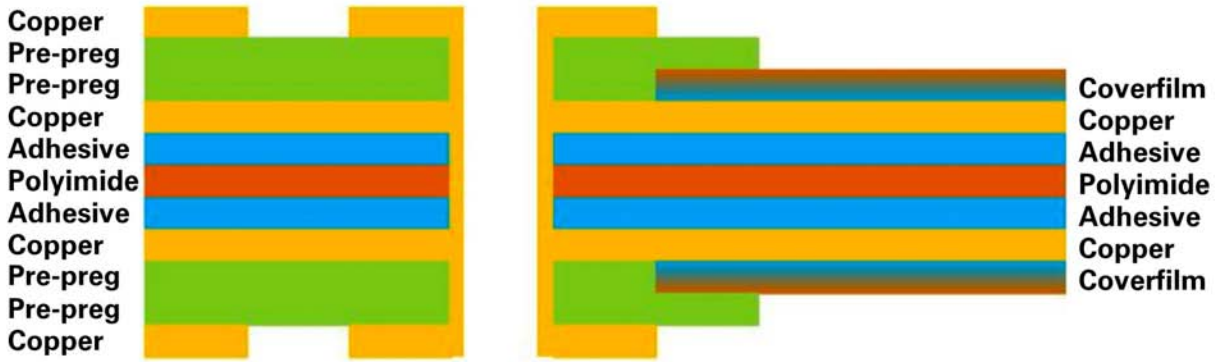
Cables or flat-flex cables are generally heavier, thicker and stiffer than flexible circuits. Such dynamic capability is impossible with a rigid PCB due to the limitations of the materials. Primarily, this is related to the high stiffness of the materials used to manufacture rigid PCBs, which incorporate glass fibres for reinforcement. With flexible circuits and FRMLs, the flexible portions of the circuits are made using unreinforced dielectric films. These films are very flexible and can therefore support the metallic conductors during dynamic bending to avoid the localisation of stresses and premature fatigue failure. Equally, these engineered films are significantly thinner than equivalent fibre-reinforced laminates, which can result in lower bending stresses arising within the circuit/conductor layers, thus further reducing the risk of premature fatigue failure.

A typical epoxy/glass clad laminate is 8-17x stiffer than a clad adhesiveless polyimide flex circuit laminate. The inclusion of glass reinforcement is the main reason why rigid PCBs do not bend and similarly why most semi-flex circuits are potentially less reliable than flexible printed circuits or FRMLs for applications where repeated bending is a requirement.

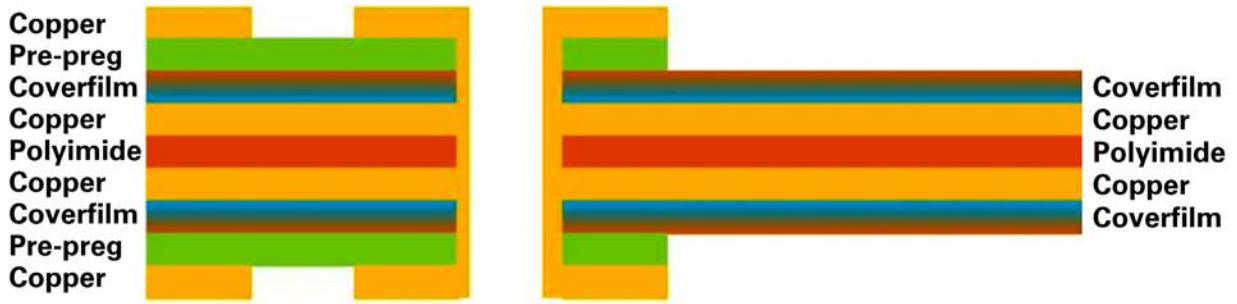
Typical constructions for FRML circuits are shown below.



Type 1A—Flex-rigid with flex adhesive in rigid and flex zones.



Type 1B—Flex-rigid with adhesive-based base laminate and patched coverfilms.



Type 2—Flex-rigid with adhesivesless-based laminate and non-patched coverfilms in all areas.

Type 3—Flex-rigid with adhesiveless-based base laminate and patched coverfilms.

MATRIX
CIRCUIT BOARD MATERIALS

Quality and Service

Panasonic Multilayer and Flex Laminates ..... Panasonic PCB Materials Partner in North America MATRIX USA

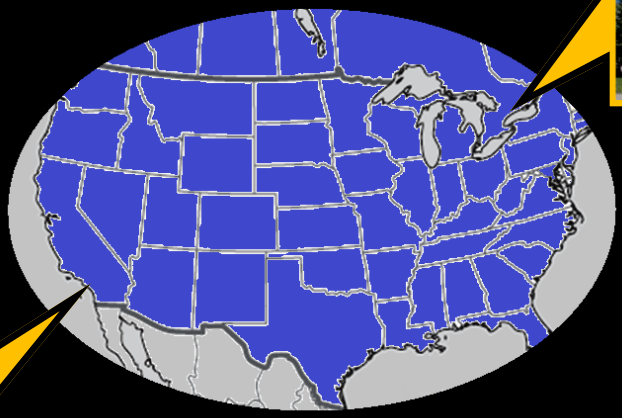
..... Jinzhou Drills and Routers

PerfectPoint Drill Sharpening ..... PERFECT POINT DRILL SHARPENING CENTER

..... Kingboard Rigid Laminate

Copper and Aluminium Foils .....

..... Hakuto Film Laminators



Also in Canada:

DuPont .....

..... MacDermid

ACS Chemicals .....

..... Pacothane

CA Picard .....

..... Argus International

Micronutrients .....

USA: 301 West Dyer Road, Suite "E", Santa Ana, CA, 92707 Tel: 714-825-0404 Fax: 714-825-0406

Canada: 1124Midway Blvd., Mississauga, Ontario, L5T 2C1 Tel: 800-668-5407 Fax: 905-670-8420

www.matrixelectronics.com

Surface-bonded FRML—two-layer version shown.

REGALFLEX 5—Flex-rigid with no flex laminates in rigid zone.

Table 2 compares the relative benefits of each of these FRML types.

Many manufacturers promote their favoured constructions for FRML circuits according to their processing capability. However, each should be considered on its own merits for the particular application. For example, low-cost FRMLs with constructions in the range of three to eight layers can be made using adhesive or adhesiveless-based flex laminates for the flexible layers. Similarly, flex coverfilm layers can be complete through the FRML structure,

Flex-rigid Construction	Relative Benefits of Each FRML Construction Type				
	Cost	Reliability	Space	Weight	Ease of Component Assembly
Type 1A	✓✓✓✓✓	✓✓✓✓	✓✓✓	✓✓✓	✓✓✓
Type 1B	✓✓✓✓	✓✓✓✓	✓✓✓	✓✓✓✓	✓✓✓✓
Type 2	✓✓✓✓	✓✓✓✓	✓✓✓	✓✓✓✓	✓✓✓✓
Type 3	✓✓✓✓	✓✓✓✓✓	✓✓✓	✓✓✓✓	✓✓✓✓
Surface bonded FRML	✓✓✓✓✓	✓✓✓✓	✓✓✓	✓✓✓✓	✓✓✓
REGALFLEX 5	✓✓✓	✓✓✓✓✓	✓✓✓✓	✓✓✓✓	✓✓✓✓✓

Table 2: Relative benefits of each FRML construction type (✓✓✓✓✓ = Best).

i.e., including the rigid zone, or they can be patched to only have these materials present in the flexible zone. Ultimately, the patched coverfilm approach is potentially more resilient to barrel cracking failure modes that can arise due to the greater Z-axis thermal expansion of the flex/coverfilm adhesive compared to the other materials in the circuit. However, in terms of manufacturing cost, the patched approach is potentially more expensive. For many applications that allow the choice of suitable material combinations, a cost-effective and reliable structure suitable for lead-free and mixed technology (through-hole and surface mount) assembly can be utilised.

For very high reliability, the approach has been to use more exotic laminate materials such as polyimide/glass. However, the REGAL-FLEX technology approach can yield a highly reliable construction, as used in aircraft engine control systems using some of the newer lead-free laminates and adhesiveless or high-speed flex materials. The reduced material set in this type of construction could also yield lower outgassing from the finished assembly during use.

> ***It is interesting that most of the flex-rigid circuit patents granted in the last five years are from Japan or China.***

Future Technology and Trends

Flex circuit technology is being driven constantly by the demands of device miniaturization—or apparently so as we are advised by the CAD software developers. Whether it is for surface mounted components or embedded components within the circuit, this causes component pad footprints to be ever denser; consequently, this drives reduction in track, gap, via dimensions, complexity and cost.

It should be noted that this is not a requirement in all areas of circuits and in many cases designing to the minimums can increase risks of manufacturing and reliability issues, especially as volumes and/or circuit sizes increase. The materials employed to manufacture flexible and flex-rigid circuits are inherently thin and flexible. Therefore, their dimensional stability is worse than that of most rigid printed circuit materials and consequently, due to the more complex circuit patterns/layouts, these material-related movements can be localised and unpredictable in advance of manufacture. As a result, they can affect yield as designs get more complex. This may not be apparent for prototype batches, but can arise more seriously in series production.

Similarly, the risk of damage to the generally softer flex laminate materials can lead to reduced yield, especially as dimensions are reduced. Such effects can also occur within the circuit during drilling or laser via processing. Hence, there is the need to reiterate this point: Do not design to minimums unless essential. If it is needed, keep these complex areas as localised as possible and work closely with your flexible circuit supplier. The use of microvias and supporting track and gap features is well established for rigid PCB production. However, flex and FRML technology has to cope with a different and more diverse material set, plus many more flexible substrates as mentioned earlier. Consideration of these factors with your supplier will ensure that yield and cost are optimised when using flexible circuit substrates and technologies compared to the relatively more rigid (and reinforced) PCB circuit technologies.

This trend of greater packaging density is likely to continue, although its progress will most likely be stepwise as new device sizes emerge every year, which subsequently drives circuit manufacturing equipment and process technology. Japan and China appear to drive a significant amount of device development and manufacture in the world. It is interesting that most of the flex-rigid circuit patents granted in the last five years are from Japan or China. Maybe this is a reflection of the reduction of wafer fabrication in the rest of the world (ROW) and it could explain why the number of ROW patents for circuit technology appear to have dropped.

Direct component assembly onto and through flex and FRML is possible today and

enables many advantages to be realised within the electronic system. Manual, through-hole, surface mount, flip-chip and direct chip attach technologies can all be employed. For multilayer flex and FRML, the interaction between the materials, the design and the assembly processing should be considered to avoid potential issues with differential expansion effects, laminate flexibility, material movement and moisture. The National Physical Laboratory (Teddington, UK) has published some very good work on the topic of moisture retention within rigid and FRML circuits during assembly. They highlight the need to consider such design features as large ground or power planes within the circuit and the material build when defining drying bakes before component assembly.

In the LED market, many designs utilise rigid PCB technology with metal backing. Flex has a role to play here, potentially. The inherent thin nature and thermal stability of some flex laminates make them well suited to flexible LED applications.

In general, the processes for manufacturing rigid PCBs, flex and flex-rigid multilayers have not changed that rapidly over time. In recent years, the processing technology has evolved to enable the creation of smaller vias between layers, utilising laser drilling and laser direct imaging techniques.

The other processes of etching, plating, bonding and profiling have changed little during the last five years, by comparison. The use of additive circuit technology for flex is happening, but how this will integrate with multilayer processes, assembly and reliability requirements remains to be seen. This is primarily because the success of any additive processing relies upon the ability of the conductor layer to adhere sufficiently onto the dielectric substrate. Over recent years, processing technology using inkjet printing techniques have enabled catalytic layers to be added at high resolution to the polymer film substrate. These can then be plated to create signal tracks. The lower adhesion of tracks created in this manner has not changed significantly and this is one potential limiting factor for the further evolution of additive circuit processing into more complex, more stressful, and higher-reliability multilayer constructions. Current activity in additive circuit processing is around its use in circuits, which form a part of accurate linear or rotational position control systems. This in itself is an interesting technology area which is expanding rapidly with the use of electric/electronic actuation in flight systems, for example.

Flexible printed circuits offer the prime benefit of being thin and light. In

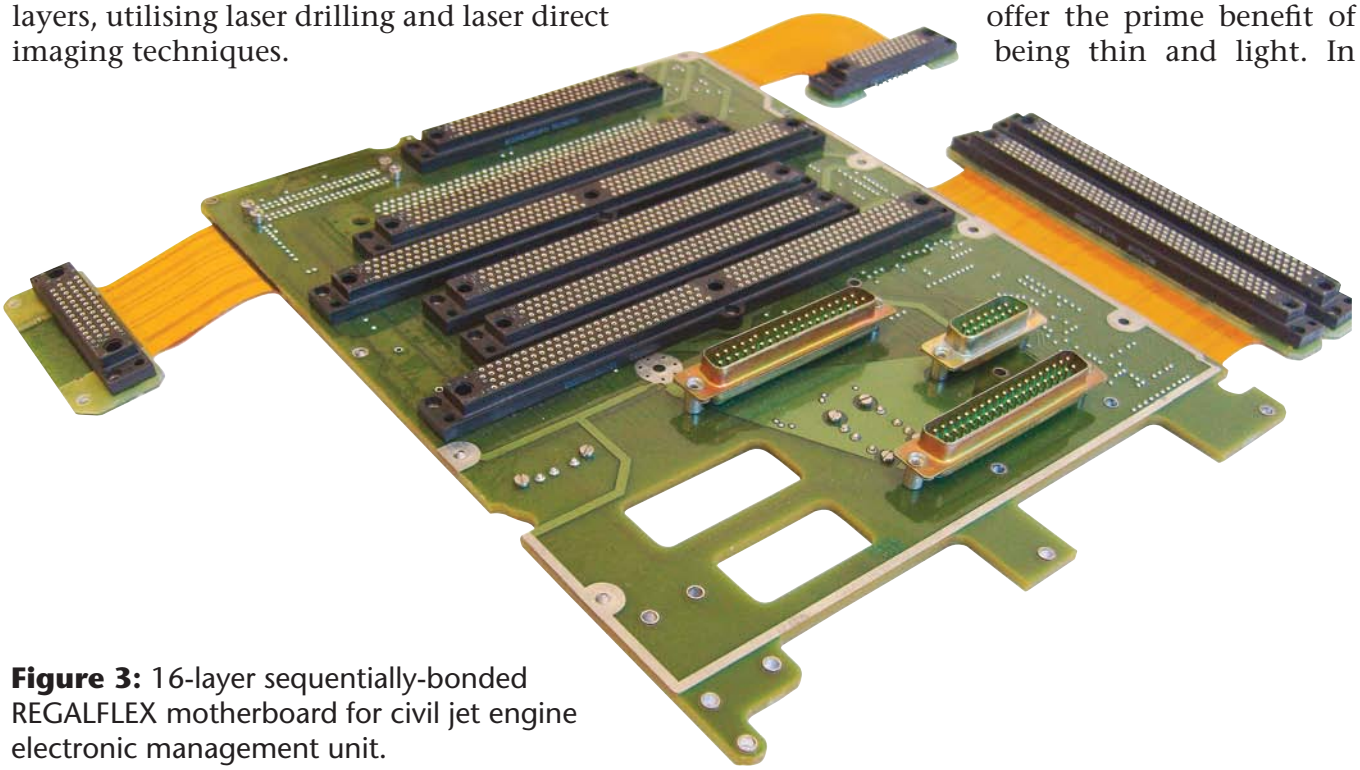


Figure 3: 16-layer sequentially-bonded REGALFLEX motherboard for civil jet engine electronic management unit.

Mentor Graphics®

REALIZE LOWER COSTS THROUGH PCB DESIGN

Despite increased complexity and shortened timelines,
86% of best-in-class companies are meeting their product cost targets
while realizing an 11% decrease in product costs.*

What are they doing differently from your organization?

*Boucher, Michelle. 2010. "Why Printed Circuit Board Design Matters to the Executive." Aberdeen Group

Manage an increase in PCB design complexity,
While reducing development

LET US SHOW YOU HOW

DESIGN THROUGH MANUFACTURING

COMPLEXITY MANAGEMENT

INTELLECTUAL PROPERTY MANAGEMENT

Follow us

reality this means that for equivalent current-carrying capacity, a flexible circuit, when compared to a cable harness, will be approximately 70% lighter than the cable. This has huge potential considering most vehicles (motor cars, trains, ships or aircraft) are powered by some means for propulsion and that the propulsion systems range and efficiency is directly related to the weight of the vehicle. Most vehicles contain extensive cabling and therefore, there is great future potential to replace at least some cable harnesses with flexible circuit technology.

A variation on this theme is that for vehicle engines to get more efficient, greater electronic sensing and control is required. This potentially raises the operating temperature requirement of such circuits from typically 100 to 125°C to >200°C and maybe even higher for aircraft engines. This brings some unique challenges for the polymeric materials typically employed for the manufacture of rigid, flex and flex-rigid multilayer circuits.

Embedded components within rigid PCBs are becoming a useful technology today. However, with pure flex products the bending stresses involved can cause issues mechanically and electrically with these types of components. Therefore, currently we see the next stage is the evolution of such technology into FRML circuits within the rigid areas primarily. Some passive components, e.g., resistors or capacitors, can be embedded into flexible circuit regions; however, care must be taken in their design and the tolerances which are applied.

> ***"Materials will play a significant role in the development of flex and FRML technology in the future."***

Flex and FRMLs are used in chip scale packaging and stacked die packages for module production. This is primarily because of their ability to enable packaging in very tight spaces.

Materials will play a significant role in the development of flex and FRML technology in the future. This is because the requirements of these circuit technologies are expanding, as indicated above. With the replacement of cabling with flexible circuit harnesses comes a need for faster signal transfer and higher temperature operation. DuPont de Nemours has recently introduced a new range of high-speed flex laminates under the Pyralux TK range. These materials combine the excellent properties of polyimide films with the excellent dielectric constant and loss tangent properties of fluoropolymer films into one composite laminate clad with copper for the conductor layer.

Other materials are emerging in copper and adhesive clad laminate form for high-temperature applications such as PolyEtherEthylKetone (PEEK).

Both of these material types will open up new opportunities for flex and FRML applications that require their special properties. However, experience initially with each will be limited, whilst design and processing guidelines are developed by suppliers, processors, and users. Neither material is as yet defined as applicable for dynamic applications, although this may be a possibility moving forward.

Most electronics today involve the interconnection and distribution of signal and/or power around an assembly to fulfill the functions of the unit. For power distribution, the copper layers being used today are thicker than any that have ever been used before. In some cases these can be combined in multilayer constructions with signal layers, provided suitable screening is undertaken. However, there may be areas of electrical, mechanical, thermal and component assembly incompatibility that could arise in such a structure. This is where segmenting the technology, e.g., instead of one FRML using signal and power layers with components assembled and for example some static and some dynamic flex limbs, it may be more suitable to design a combined approach with either rigid PCBs and flex or FRML and flex or rigid PCB/FRML/flex. Where major issues arise between signal and power lines, a growing development is the use of optical connections within rigid, flex or FRML circuits. Today, only a few companies have the ability to undertake such technology and it is often limited to rigid PCBs or FRMLs.

Mentor Graphics

INNOVATE TO COMPETE

Market demand for feature-rich products only continues to grow,
increasing complexity and the need
to fit more functionality into smaller spaces.
Does your organization have what it takes to maintain pace?

Create product differentiation
that directly results in increased market share

LET US SHOW YOU HOW

VIRTUAL PROTOTYPING

COMPLEXITY MANAGEMENT

DESIGN THROUGH MANUFACTURING

Follow us

The spectrum of design, materials and manufacturing processing requirements for flex and flex-rigid multilayer circuit production is significantly wider than for rigid printed circuit boards.

In this paper we have tried to highlight some of these differences and how they can be considered, together with the potential future material and technology trends. It is clear that users need to consider these differences when planning their projects and purchasing flex and flex-rigid multilayer circuits as they are not designed, manufactured and assembled using precisely the same processes as for rigid PCBs.

Flex and flex-rigid multilayer circuits have a great future and we predict that they will potentially develop in many of the following areas:

- Thinner circuit constructions
- Higher-speed signal processing
- High-temperature applications
- To enable more compact and lighter box designs with greater functionality
- Larger and more dynamic circuits
- Embedded devices
- Electro-optical circuits
- Printed electronics
- Further use of hybrid flex and flex-rigid technologies

The world of electronics is ever evolving, and precisely how the flex and flex-rigid technologies will develop remains to be seen. **PCB**

Keith Netting is the technical director at Teknoflex and has 25 years experience in flexible circuit design, manufacture and assembly. Netting is a specialist in materials and processes across a range of markets, including packaging, circuit manufacture, assembly, metallic materials, polymeric materials, adhesives, composites and quality/reliability testing. Contact the author at knetting@teknoflex.com.

VIDEO INTERVIEW

LED Market Drives Demand for New Laminates

by Real Time with...IPC APEX EXPO 2012

Ventec's Bruce Hurley discusses how the LED market has driven the demand for materials that transfer heat efficiently. Standards need to be further developed in this area. Apples-to-apples product comparisons are a challenge right now and long-term reliability studies need to continue.

CLICK TO VIEW

realtimewith.com

Mentor Graphics

GET YOUR PRODUCTS TO MARKET FASTER

When it comes to looking for a competitive advantage,
your PCB design strategy can ultimately prove to be your secret weapon
as you move into an uncertain future.

Are you providing your design teams
with access to the right technology to make that happen?

Speed time-to-market today
with the PCB technology of tomorrow

LET US SHOW YOU HOW

VIRTUAL PROTOTYPING

DESIGN FLOW CONCURRENCY

DESIGN THROUGH MANUFACTURING

Follow us

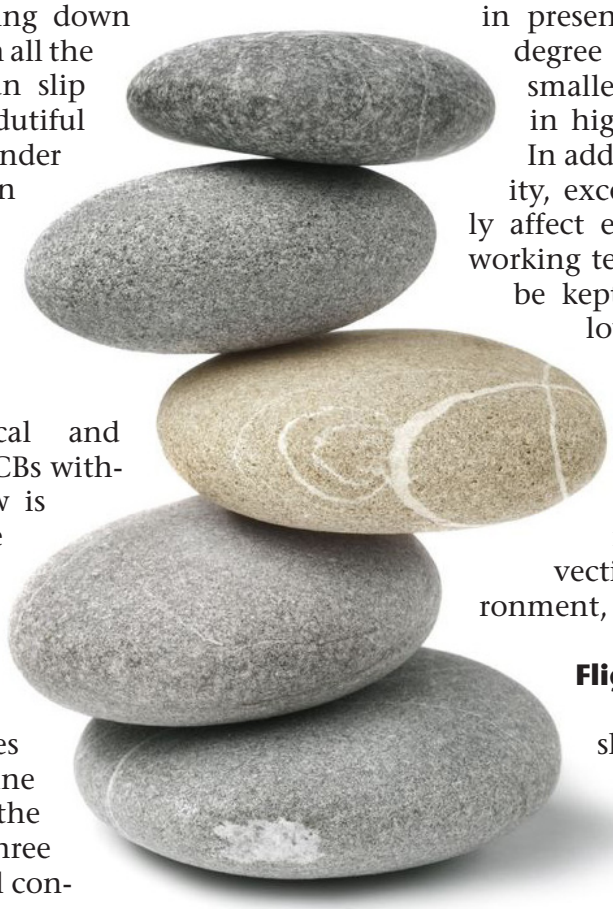
Critical Placement

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD

SUMMARY: Controlling the placement of devices limits maximum trace length, reduces flight time delay and skew, and assists in compliance to timing specifications.

Today's high-speed digital products use high frequencies and fast rise times that demand careful attention to PCB layout to ensure that the system works not only in the prototype phase, but also in mass production, and under all possible operating conditions.

An insidious problem with high-speed boards, in fact, is that their failure mode may simply be flakiness or intermittent behavior across multiple manufacturing runs. In these cases, the proper layout of the PCB can mean the difference between a reliable product and a board that may perform intermittently. A single PCB design error can bring down part or all of a board. Given all the possible mistakes that can slip through even the most dutiful design process, it is a wonder that we ever get the design to work properly. Maybe some of the advice that follows will help ensure you get it right first time.



Mechanical Constraints

Interactive mechanical and electronics co-design of PCBs within a true concurrent flow is becoming common these days, as inevitably every board has to fit into a box or enclosure of some sort.

As the first step, the mechanical designer uses the MCAD system to define the shape of the PCB in the mechanical product in "three space," setting mechanical constraints and placing components such as connectors that have 3D positioning requirements. After completion of the mechanical product design, the mechanical designer extracts the PCB shape and height restrictions from the design, and exports them to the PCB database. Once the PCB designer has completed component placement, the layout can then be transferred back to the MCAD system for a final 3D clearance check.

Electrical isolation and electrostatic requirements may also have to be adhered to. These may include cut-outs, keep-outs, vertical clearances, mounting holes and thermal heatsinks that need to be defined.

Thermal Analysis

Power dissipation is an important issue in present-day PCB design. To the degree that smaller dies enable smaller packages, this can result in higher concentration of heat. In addition to the issue of reliability, excess heat will also negatively affect electrical performance. The working temperature of an IC should be kept below the maximum allowable, worst-case limit. In general, the temperatures of junction and ambient are 125°C and 55°C, respectively. Consideration should be given to thermal conduction and convection in the operating environment, as well as radiation.

Flight Time and Skew

Flight time delay and skew are key pillars in high-speed PCB design. One of the driving factors for flight time and skew performance

...More than Immersion Tin

...More than Immersion Silver

Uyemura ENEPIG

Provides a Final Finish that is as close to Indestructible as a Final Finish can be!

Uyemura's Electroless Nickel prevents Cu diffusion, strengthens thru-holes and has the **greatest line definition for tight lines and spaces of any EN.**

Plus: Exclusive Uyemura NPR nickel eliminates bridging and operates at 176°F for best possible soldermask compatibility.

Uyemura Electroless Palladium allows finish to be solderable and wire bondable, without damaging the nickel.

Uyemura's Exclusive Immersion Gold provides a low-porosity deposit – and further boosts wire bondability.

UYEMURA USA

Tech Center:
Southington, CT • ph: (860) 793-4011
(800) 243-3564 • fax: (860) 793-4020

Corporate Headquarters:
Ontario, CA • ph: (909) 466-5635
(800) 969-4842 • fax: (909) 466-5177

www.uyemura.com

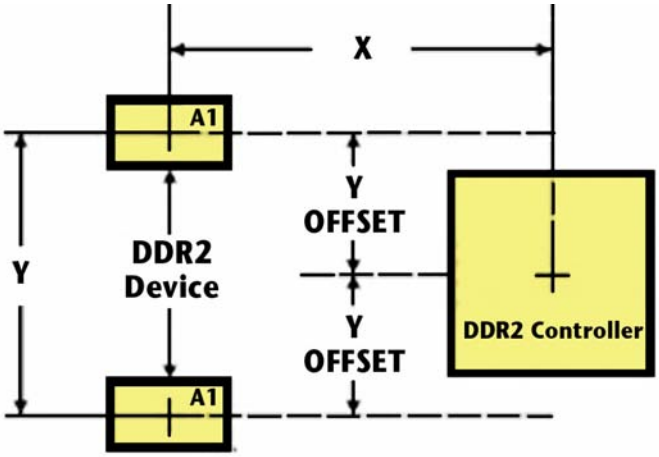


Figure 1: DDR2 processor and memory placement requirements.

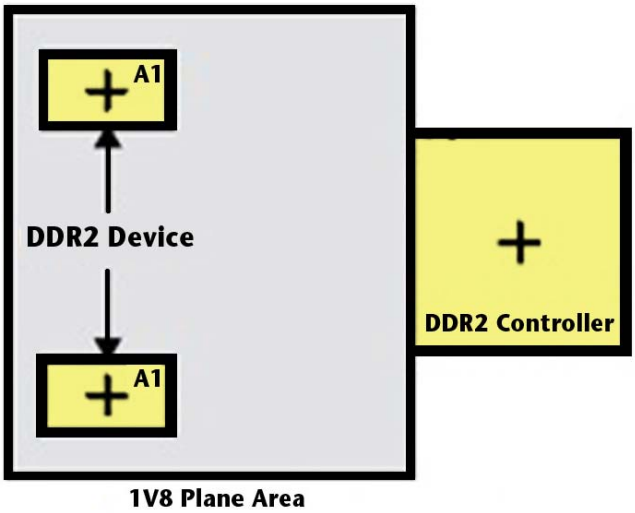


Figure 2: 1.8V plane and keep-out area.

is the placement of components. Maximum placement refers to the placement in which the distances between the devices are the maximum distance permitted. Controlling the maximum placement of devices, combined with the assumption that good general design practices are adhered to, limits maximum trace delay to roughly the longest Manhattan distance of the signals contained in a specific clock domain. Why the longest Manhattan distance?

This is due to skew matching requirements:

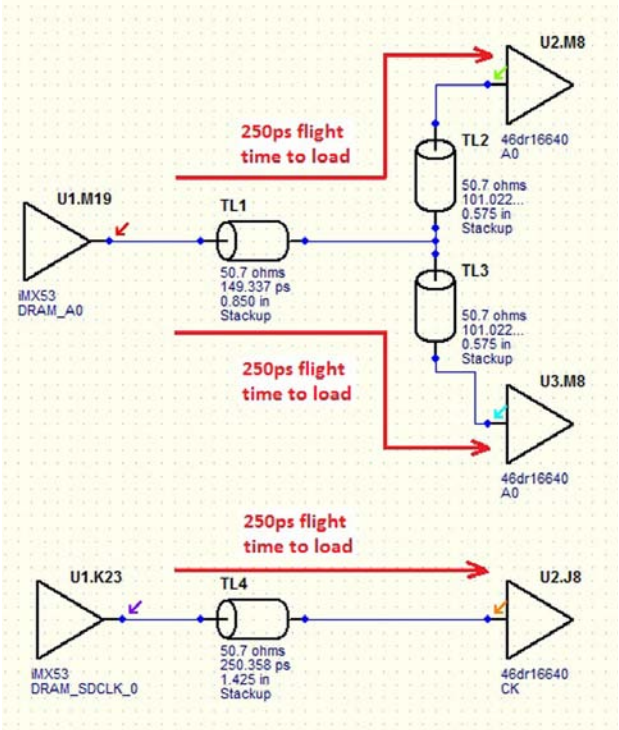


Figure 3: Flight time from processor to loads.

All of the shorter nets in a clock domain must be lengthened to skew match to the longest run length. Therefore, flight time and skew—for an entire clock domain—are governed by the maximum placement, along with the routing rules that constrain the matching of the trace lengths.

In the classic high-speed design flow, timing specifications simulation results are compared to determine placement and routing constraints. Given a length constraint, a designer can control signal integrity by controlling the PCB trace topology of the various parts of an interface. Included in this topology are any terminations.

Figure 1 shows the required placement of the processor and DDR2 memory chips. The purpose of the placement guide is to limit the maximum trace lengths and to allow for routing and via space—which can be a challenge. (Note that this placement does not restrict whether these devices are placed on the top or bottom of the board.)

The region of the PCB used for the DDR2 circuitry must be isolated from other signals.

The DDR2 keep-out region is defined for this purpose and is shown in Figure 2. The 1.8V power plane should cover this entire region and non-DDR2 signals should be kept out of this region.

The main requirement for DDRx timing is that the clock must be slightly longer than the address, control and command traces, and that the strobe must be longer than the data and data masks. When looking at the total length of signals in the EDA tool, one can easily be mistaken in calculating the total distance and hence delay required—totally annihilating the timing.

Figure 3 illustrates the relationship between an address signal (A0) routed in a T-section topology to the address pins on the two DDR2 loads. The total length in the EDA tool is given as 0.850 + 0.575 + 0.575 = 2000 mils. So, one would assume that the clock must be at least 2000 mils to be longer than the address trace A0. This, of course, is incorrect, as we only need to worry about the longest point-to-point length.

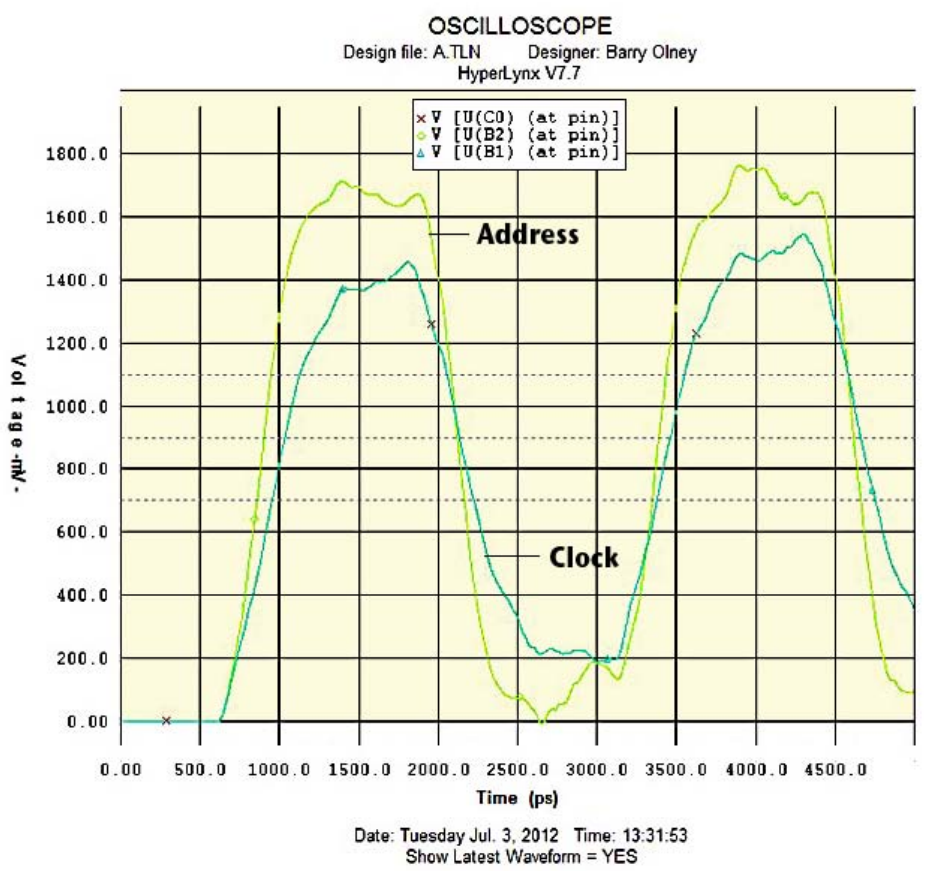


Figure 4: Relationship between the clock and address signals.

If we now look at the flight time from the iMX53 processor to A0 pin of the first memory chip, the total flight time to the pin is 250ps, which is identical to the total flight time from the processor to the clock pin of each memory chip. So, the total length should be 0.850 + 0.575 = 1425 mils.

The same applies for the data to strobe—which acts as the clock for the data signals. The strobe must be longer than any of the data or data mask signals for each lane.

Figure 4 illustrates the relationship between the clock and address signals. If a designer was to inadvertently use 2000 mils for the clock length then the clock would be delayed by an additional 100ps, putting it way out of spec and causing a system-timing failure.

Having simulated the pre-layout timing, we can now put values into the placement of the DDR2 memory chips with respect to the processor. For Figure 1, X = 850 mils and Y offset = 575 mils. These are the maximum values and should be reduced to allow for the length tuning of traces. It is best to reduce the X dimension to, say, 650 mils and keep the Y offset as large as possible to provide space for vias at the centre of the T section.

Termination

Placement of series terminators—if required—can also a debacle. We are told that series terminators should always be placed within 200 mils of the source. Since there is no termination at the far end of the trace, the series terminator has to absorb the entire reflection from the load.

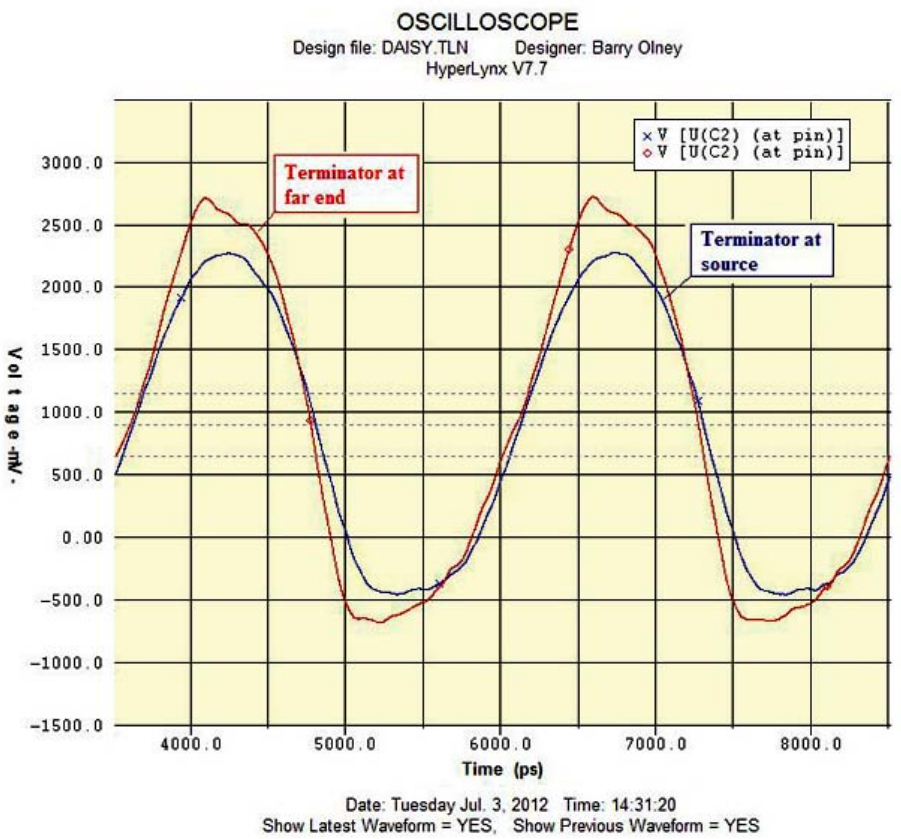


Figure 5: Difference between the series terminator at the source or load.

Figure 5 contrasts the placement of a series terminator close to the source—as typically done—and at the far end near the load—which is not recommended. It seems that either way the series terminator does its job—absorbing the reflection before it can propagate along the trace a second or third time. The only noticeable effect is that the far-end termination is more distorted whereas the near-end termination is a perfectly clean waveform.

There may also be a requirement to use parallel termination—address VTT pull-up. These are normally placed on the end of a daisy chain to pull the signal up to VTT which is the reference voltage for DDR2. This has to be at the end of the line.

Design Re-use

If the same processor and memory chips are used on consecutive designs, creating a library of matching re-use blocks for schematic and PCB makes the best use of existing design elements for future designs. Simply add a sub-circuit block to the schematic, transfer to the PCB database, and load a predefined layout block including component placement, tracks, copper and text. Whether you use it for multiple channel designs, critical digital circuitry, RF circuit blocks, or just to replicate a commonly used layout pattern, design reuse will save time and ensure repeatability of design: a proven, tested, working solution to just drop into place.

Decoupling Capacitor Planning and Placement

The number, value, dielectric material and placement of decoupling capacitors are also critical for the proper DDR2 interface operation.

Manufacturer's recommendations for decoupling capacitors are generally overrated in that they tend to overestimate the number of decaps required. This may be fine for short production batches, but a detailed power distribution network analysis is required to determine the exact number and value for mass production in order to reduce costs and optimize the PDN.

Figure 6 depicts the ICD PDN Planner analysis of the DDR2 1.8V supply (download from www.icd.com.au). The 30 x 100nF capacitors should be placed as close as possible to the DDR2 chip supply pins using thick (20 mils) traces to reduce inductance. A good capacitor dielectric choice is X7R with a footprint of 0402 or smaller. 4 x 22 uF Tantalums reduce the AC impedance around 10M, while the 100nF capacitors and tight interplane coupling (0.004" in this case) keep the impedance as low as possible at the fundamental clock frequency of 400 MHz.

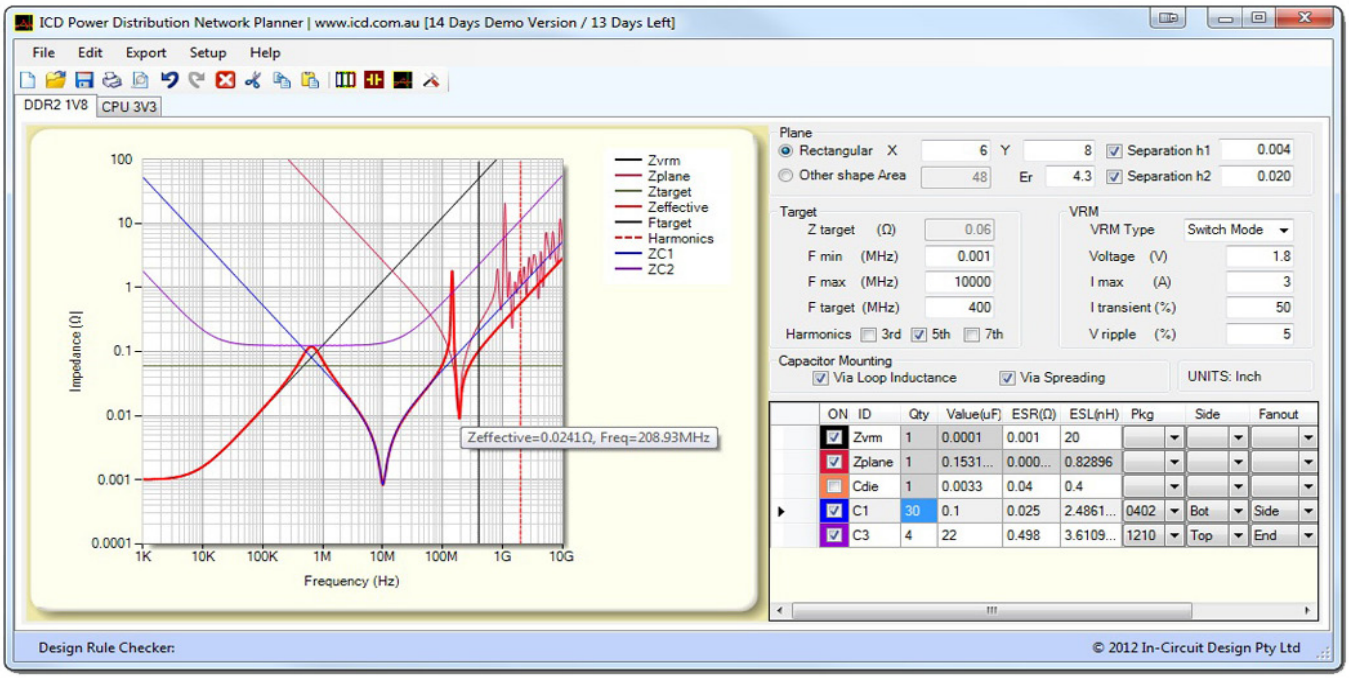


Figure 6: Illustrates the PDN requirements of DDR2 1.8V supply.

Points to Remember:

1. Mechanical constraints should be dealt with prior to component placement.
2. Consideration should be given to thermal conduction and convection, as well as radiation.
3. Controlling the maximum placement of devices limits maximum trace delay to about the longest Manhattan distance of the signals contained in the clock domain.
4. The main requirements of DDRx timing are that the clock must be longer than the address, control and command signals and the strobe must be longer than the data and data masks.
5. A cleaner waveform is delivered when a series terminator is placed close to the source.
6. Design re-use will save you time and ensure repeatability of design.
7. The quantity, capacitance, dielectric material, placement and mounting of decoupling capacitors are critical for the proper DDR2 interface operation. Detailed PDN analysis is required if you want to save on component costs.
8. For each DDR2 speed grade—with frequencies ranging from 200 to 400 MHz—interplane capacitance may be required to maintain a low-impedance supply of power. A software tool, like the ICD PDN Planner, should be employed to analyze whether this design objective has been met. **PCB**

References

1. Advanced Design for SMT—Barry Olney
2. Beyond Design - Intro to Board-Level Simulation and the PCB Design Process—Barry Olney
3. PCB Design Techniques for DDR, DDR2 & DDR3, Part 1 & 2—Barry Olney
4. Beyond Design - Power Distribution Network Planning—Barry Olney

Barry Olney is managing director of In-Circuit Design Pty Ltd. (ICD), Australia, a PCB Design Service Bureau and board level simulation specialist. Among others, ICD was awarded "Top 2005 Asian Distributor Marketing," and "Top 2005 Worldwide Distributor Marketing" by Mentor Graphics, Board System Division. For more information, contact Olney at +61 4123 14441 or by e-mail: b.olney@icd.com.au.

Circuit Flexibility

(and How to Achieve it)

by Joe Fjelstad

***SUMMARY:** The multiple definitions of flexible should be kept in mind when one sets about designing a circuit where flexibility is required. Touching on all of the definitions benefits anyone tasked with satisfying the design objectives of the product in their charge.*

In the world of electronic interconnections, the word flexible has been used in conjunction with the word circuit for nearly six decades. This is almost as long as the pairing of the words printed and circuit have been co-joined. And just as we have come to discover over the years (and especially in the last five or so) that there are layers of meaning in the term printed, with the advent of, and interest in printed electronics, so also are there layers of meaning to the term flexible.

The most common interpretation of the word flexible, as applied to the flex circuits that the industry currently makes, is something capable of being bent repeatedly without breaking. However, there are other definitions of flexible that are worthy of consideration as well when using the term, for their ability to unlock new thinking patterns relative to what is flexible. For example, there is the definition of

flexible where it means something amenable to modification, or simply adaptable, if you will. Additionally, the definition of flexible where the word can be construed to mean something which is easily disposed to yield, that is, something that is pliable. Where the most commonly used definition is viscerally understandable when one envisions a flexible circuit, the second two definitions of flexible are more abstract and could serve as a springboard to new ways of thinking about circuit flexibility, as opposed to the flexibility of the circuit. The logic may seem a little obtuse at this moment, so some examples are offered hereafter to help explain.

To begin, consider the fact that most flexible circuits are not actually used in applications where repeated bending is required; that is, most flexible circuits are designed to be used in static applications. The only bending and flexing required is during installation. This type of application really fits the second definition above in that the circuit is amenable to adaptation to the construct in which it is used. In such applications it is of great benefit to make sure that the design has incorporated some extra length in appropriate areas to facilitate assembly and accommodate any unexpected or unanticipated obstacles that might show up only during the installation process. These extra lengths of material are commonly called service loops. These features provide additional flexibility to that which is already flexible, but it is flexibility of the second type.

The third definition of flexible, something which is disposed to yield or is pliable, is one that comes to mind when considering the emerging subset of flex circuit products: stretchable circuits. Stretchable circuits are an interesting branch of flex circuit technology and one which is getting increased attention from designers of products for a range of different markets, from consumer to medical. Flexible Circuit Technology, 4th Edition has a chapter on the topic of stretchable circuits and in the interest of brevity and space, those interested in more detail are invited to look there for that information.

"The third definition of flexible, something which is disposed to yield or is pliable, is one that comes to mind when considering the emerging subset of flex circuit products: stretchable circuits."

The definitions of flexible, as broken down and examined here, apply to circuits which are physically manifest as constructions of copper and flexible polymer. However, there is another type of circuit that one should not lose sight of which can embody all of the objectives and definitions of flexible that have just been discussed, and that is the wireless circuit. No doubt, all readers have found themselves frustrated at some point trying to untangle the wires and cables used to access and control the various external devices used with our computers. To address the problem, increasing numbers of product developers are incorporating devices capable of communicating wirelessly using any or all of the various protocols which have been defined and developed to date. These include all of the various 802.11 protocols along with Bluetooth and Zigby. Clearly, at its core, wireless interconnection is the most flexible of all. It is at once dynamic, static, adaptable and extremely pliable, and if one is seeking flexibility of design, it should not be kept out of consideration.

In summary, the multiple definitions of flexible should be kept in mind when one sets about designing a circuit where flexibility is required. Touching on all of the definitions should be of benefit to anyone tasked with satisfying the design objectives of the product in their charge. **PCB**

Verdant Electronics Founder and President Joseph (Joe) Fjelstad is a four-decade veteran of the electronics industry and an international authority and innovator in the field of electronic interconnection and packaging technologies. Fjelstad has more than 250 U.S. and international patents issued or pending and is the author of *Flexible Circuit Technology.*

Most-Read Market Highlights from PCB007 This Month

Smartphone Shipments to Jump 31.6% in 2012
According to TrendForce, in 2012, smartphone shipments are projected to reach 606 million units, a 31.6% increase compared to last year; tablet shipments are forecast at 94 million units, a 69% increase. Desktops and notebooks are expected to see less than 5% yearly shipment growth, to 146 million and 206 million, respectively. Undeniably, desktops and notebooks are no longer in the spotlight; smartphones and tablets have taken center stage in the technology industry.

Automotive IC Market to Grow 8% to $19.6 Billion in 2012
IC Insights forecasts the automotive IC market will grow 8% to $19.6 billion in 2012, up from $18.2 billion in 2011. Furthermore, IC Insights forecasts the automotive IC market will grow to $27.3 billion in 2015, which represents average annual growth of 11% from 2011-2015.

Semiconductor Capital Spending Trend Update
Though only six companies are expected to increase their capital spending this year, the total semiconductor capital spending forecast figure for 2012 was raised to $63.3 billion from $60.7 billion and total 2012 semiconductor industry capital expenditures are now forecast to decline only 3% this year, as compared to the previous expectations of an 8% decline.

IDC: Global Chip Revenues Reach $315B in 2012
Semiconductor revenues worldwide will grow 4.6% in 2012 to $315 billion, according to the mid-year 2012 update of the Semiconductor Applications Forecaster (SAF) from International Data Corporation (IDC). The SAF also forecasts that semiconductor revenues will grow 6.2% to $335 billion in 2013 and grow at a compound annual growth rate (CAGR) of 4.8% from 2011-2016, reaching $380 billion in 2016.

Global Mobile Phone Shipments Hit 362M Units in Q2
Alex Spektor, Associate Director at Strategy Analytics, said, "Ongoing macroeconomic challenges in mature markets like North America and Western Europe, tighter operator upgrade policies, and shifting consumer tastes were among the key reasons why global mobile phone shipments grew just 1% annually to reach 362.0 million units in Q2 2012. Fuelled by surging demand for its popular Galaxy models, Samsung was the star performer, shipping 93 million mobile phones worldwide and capturing a record 26% market share to solidify its first-place lead."

Consumer Tech Spending to Reach $2.1T in 2012
Consumers will spend $2.1 trillion worldwide on digital information and entertainment products and services in 2012, according to Gartner, Inc. This amounts to a $114 billion global increase compared with 2011, and spending will continue to grow at a faster rate than in the past, at around $130 billion a year, to reach $2.7 trillion by the end of 2016.

Global Tablet Shipments up in Q2; Apple iPad Lands 68% Share
According to the latest research from Strategy Analytics, global tablet shipments reached 25 million units in the second quarter of 2012. Apple rose to 68% global market share, its highest level for almost two years. Microsoft tablets remain niche, but attention is turning to the upcoming Windows 8 launches.

Wireless Drives Growth of Semiconductor Market in 2012
Semiconductor revenue in the wireless communications category for 2012 is projected to reach $72.6 billion, up 10.4% from $65.8 billion in 2011, according to an IHS iSuppli Global Semiconductor Manufacturing & Supply Market Tracker report from information and analytics provider IHS.

Consumers More Confident About Overall Economy
Consumer confidence about the overall economy increased in July, according to the latest CEA Index released today by the Consumer Electronics Association (CEA)®. Consumer confidence in technology fell from last month, but it was the second-highest recorded level for July since the creation of the CEA Index.

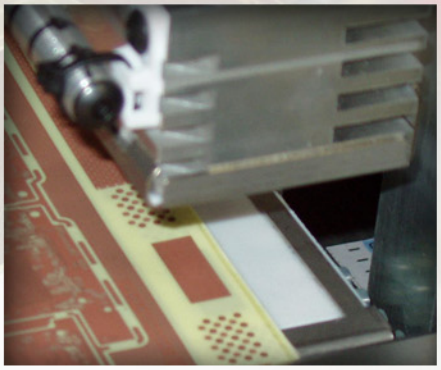
Price Drop of Electronic Components Spur Buying Opportunity
Pricing for widely used components including capacitors, crystals, filters, magnetics, oscillators and PCBs all are set to decline at above-average rates in the third and fourth quarters. "Electronic component pricing typically is stronger in the second half of the year, as suppliers strategically boost prices to take advantage of strong demand during the back-to-school and holiday build seasons," said Rick Pierson, principal analyst, semiconductor pricing, for IHS.

THE MOST IMPORTANT POINTS IN MULTILAYER ALIGNMENT ARE

ACCURACY
ACCURACY
ACCURACY

DIRECT OPTICAL REGISTRATION

34 layer panel with thin core and subs in the final stack-up



The only system on the market that will cut the layer to layer alignment tolerance in half of the best pin process available—

—and we have the customer data to prove it!

Designed, Patented and Made in U.S.A.

DIS, Inc.
85 Hoffman Lane, Suite Q
Islandia, NY 11749

DIS inc

For more information call:

(631) 851-0102

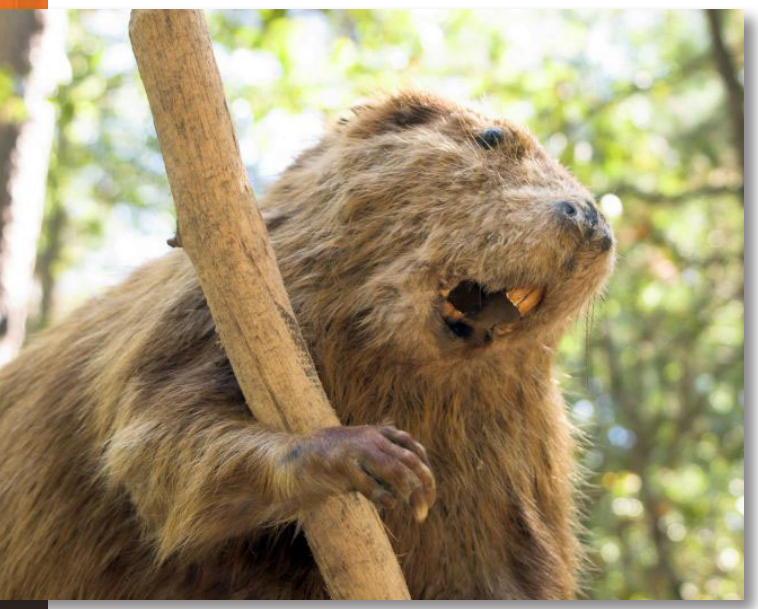
www.distechnology.com

Little Things Have the Power to Break Big DAMs

by Gray McQuarrie
GRAYROCK & ASSOCIATES

SUMMARY: Does it really make sense to roll the dice with an important customer order, just because of the inherent unreliability of your metal finishing operation? If so, it may be time for you to change the game—today.

In my book, *Change Your DAM Thinking,* I describe how Glenn Canyon Dam was nearly destroyed by microscopic cavitation bubbles. A simplistic explanation of what the engineers had done wrong was twofold. First, they looked at the likelihood of an average event, such as a very large water flow coming into Lake Powell, and decided that it wouldn't happen in their lifetime. They were wrong to focus their attention on an *average* event and should have focused on the contingency planning surrounding the *unlikely* event. Second, they thought that really large structures of an immense scale were strong and invulnerable to tiny things, like microscopic cavitation bubbles. Again, they were wrong. So wrong, in fact, that they had a crisis on their hands in less than two years after Lake Powell had reached its anticipated ideal water mark (it took about 20 years for the water to build behind the dam).



How far do any of us have to think back to an unlikely event, the thing that wasn't supposed to happen, that nearly wiped us out? Think of the global economic crisis. How often have we underestimated the impact of a seemingly inconsequential event such as an insignificant late order? If that one late order were placed by a customer who was counting on you to supply product for a major product launch, or that one late order happened when you were trying to get qualified by a major OEM, then your average on-time delivery for the plant is absolutely meaningless. If we are going to be able to identify the true game-changers for our business and take advantage of them, we have to change our DAM thinking and appreciate the power of small things that can change our lives for the better.

A nightmarish DAM problem for our industry is metal finishing, be it ENIG, solder, tin, silver, HASL, OSP, etc… In fact, a great article explaining these coatings and illustrating the scenario is Marissa Oskarsen's "What is Your PCB IQ? – PCB Final Finishes."

Our ability to perform metal finishing reliably has become challenging due to RoHS compliancy regulations. When we use a lead-free solder, we are forced to go as high as 260˚C, which means we need a high-performance T_g laminate that is more expensive then the standard laminate that use to work. If we use silver, we have to worry about creep corrosion. If we use LF HASL, we have to worry even more about wetting. If we use immersion tin, we have to worry about whiskers. If we use ENIG, we have to worry about a bunch of BGAs falling off the board and getting hit with valued-added claims because of a black-pad issue we couldn't measure, predict, or know about until it was too late. And if we use ENEPIG, well, we discover that the more layers we put down, the more strange and unexpected things that can happen, as described in this article by Hitachi Chemical.

PE4

PRECISON ECONOMICAL POST ETCH PUNCH

- **4 camera system**
- **New software**
- **Faster cycle time**
- **Improved visuals**
- **Fixed die technology**

The Pinlam PE4 maximizes accuracy and improves yield of multilayer production. Designed for simplicity, easy to use and maintain, set up is simplified, making the Pinlam PE4 much faster and easier.

C.A.PICARD®
INTERNATIONAL

Contact **Gary Weidner**
+1 269 962 2231
gweidner@capicard.com

What is a better answer to this insanity? The use of the very small: the nanometer-scale coating.

A nanometer is one billionth of a meter. The diameter of a golf ball is 42.67 million nanometers. The smallest thing the human eye can detect is 0.1 mm or 100,000 nanometers. A single skin cell is 30,000 nm and humans can't see it without a microscope. An E. coli bacterium is about 600 nm long and we can see this with a high-power light microscope. Visible light microscopes are limited by the wave-length of light, which is about 500 nm. An HIV virus is 130 nm, a hepatitis virus is 45 nm, a ribosome is 30 nm, and hemoglobin is 6.5 nm; humans need sophisticated electron microscopes to see things this small. Equipment exists that can coat printed circuit boards with a 60 nm protective layer. Why on earth would we want to do that? How could something so thin and tiny do anything for us? (If you are thinking like that, then you will suffer the same career-damaging fate as the design engineers at Glenn Canyon Dam.)

"A great example of how this seemingly insignificant, invisibly-thin coating completely changes the game is by virtually eliminating creep corrosion!"

A great example of how this seemingly insignificant, invisibly-thin coating completely changes the game is by virtually eliminating creep corrosion! It's a problem with almost any type of metal finish, but it is most apparent with the use of immersion silver. Silver has great solderability properties, but it is affected by tarnishing and is very susceptible to creep corrosion, a serious problem because it induces shorts and opens well after final assembly, leading to the potential of value add claims from your customer.

Possibilities also arise for bonding copper wire to copper pads on the board. This would eliminate several things: the cost of the gold pad and the gold wire; the worry about scrap caused by incorrect gold thickness; and/or a scratch on a wire bond pad, due to a handling mishap. Moreover, a copper-to-copper bond reduces impedance-matching effects, rendering cleaner signals, and possibly reducing power requirements because of the improved efficiency. This results in a lower-cost product that is easier to make, and has better final performance. Companies like Amkor seem to be working in this direction, and an IEEE article from about fine-pitch wire bonding on copper pad can be found here.

Certain nano-coatings available today, like those from Semblant, don't require a masking, because you can wire bond or solder right through it. When we engineer by adding steps, by adding layers of metal, by adding overall complexity, we reduce the reliability, quality, and performance of a product. These added complex steps, which are challenging to control and manage at the end of the process, make our business very unpredictable. We can have a total loss of a job happen at just the wrong time, which can do financial damage many times greater than the cost of remaking the lot. I have been witness to this very event where a medium-sized shop lost an opportunity to be a supplier to a very large medical device OEM, because the roll of the dice on their metal finishing turned up deuces, meaning they "crapped out," to use a gambling term.

So what is the problem with using this new technology? It isn't free! This means we have to get it past our accounting department, which is fixated on ROI. Sure, we may have budgeted capital expenditures, but accounting is the gatekeeper. In order to get past the gatekeeper, the new technology has to generate enough money so that it matches or exceeds the ROI target. The first thing to look at is the cost of gold, which is expensive. In the last five years, gold has been as low as $650 an ounce, and as high as $1,800 an ounce, approximately. Consider an 18" x 24" panel with 1 um thickness (0.0001 cm) for gold, where there are 80 circuits on a panel with 20 gold pads per circuit with dimensions of 60 mils (0.1524 cm) by 20 mils (0.0508 cm). Using the density of gold at 19.32 g/cm^3, we would have 0.012 grams of gold over the board, or 0.000423 of an ounce. At the time I wrote this article, the price of gold was at $1,617/oz., which amounts to $1.36 per board.

If you are building 1,000 boards a week for one year, $70,720 a year could be eliminated if you didn't need a gold finish. Many of us have CFOs who want a much stronger case than this.

Where would more monetary value come from that could help us build a stronger ROI justification for this new technology? My *DAM Scrap* column (August 2012) might offer a hint. Scrap kills capacity in the form of restarts. For example, a 20% scrap rate will reduce your available capacity by 25%, because 5% of restarts will be restarted at least one more time. Moreover, in an era where we need premium quick-turn jobs, the scrap rate can have a devastating effect on profits. An improvement of as much as 10% in scrap can have the same economic impact as having your shop located in China, with one tenth of the labor rate. When you add up all of the scrap costs caused by metal finishing it can be as high as 5%, 10% or more. Scrap isn't just a cost, but it is also a constraint on your plant's capacity. Significant improvements to scrap need to be understood as a strategic issue with regard to how best to deploy the new capacity for maximizing profit. For example, if there is an 11% improvement in capacity from using a nanometer-scale technology process and you can generate 5% more quick-turn jobs with a high average price per panel, the ROI can be very high.

Here is something else that doesn't show up on the ROI computation, but it speaks to that one time event that can have devastating consequences to your business, all because of poor timing. Figure 1 shows a discrete event model for a simple (supplier plant/customer plant) supply chain.

We will be discussing this model in greater detail in next month's column. For now, a customer decides on the reorder point (ROP) and economic order quantity (EOQ) that will minimize costs and keep its plant building the product they need to sell. Figure 2 shows the inventory fluctuation that you would see for

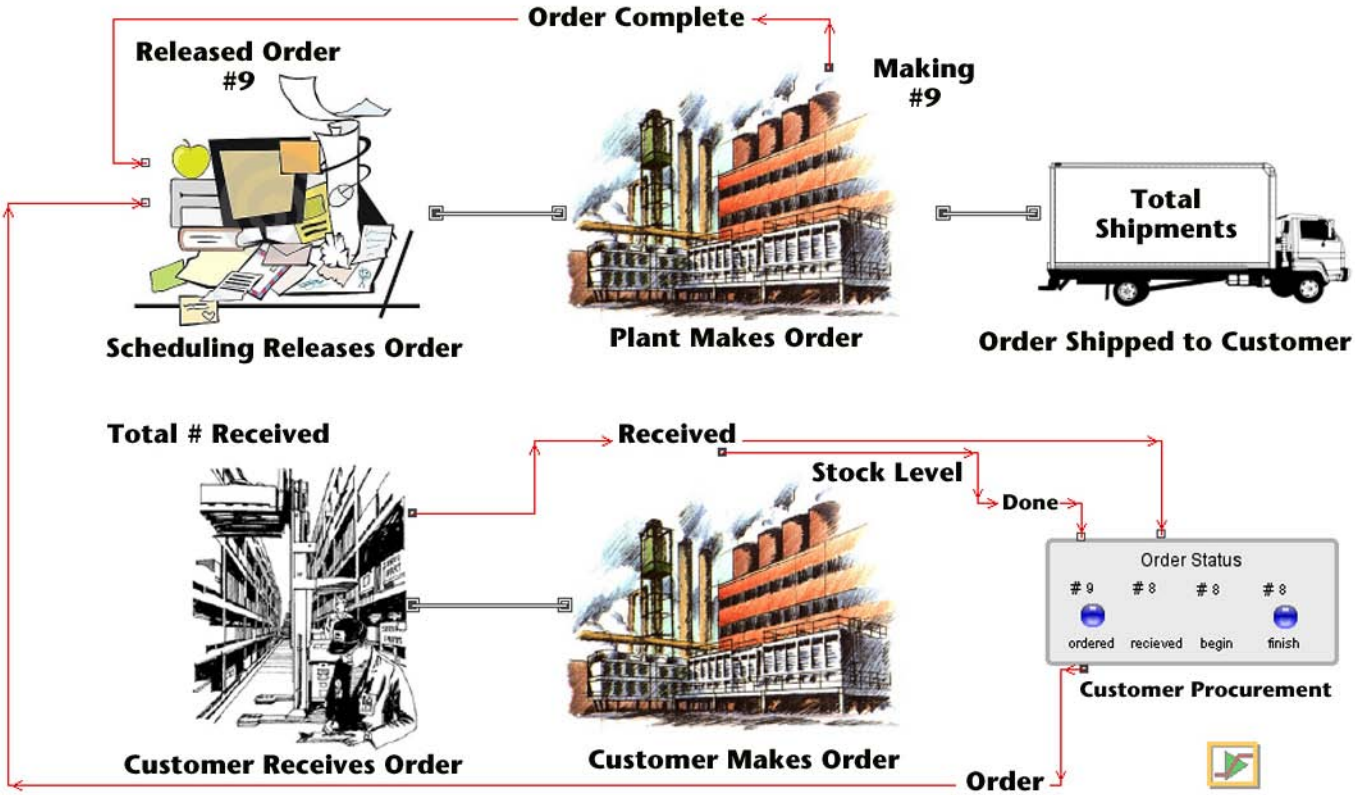


Figure 1: The discrete event model for the simple supplier/customer supply chain.

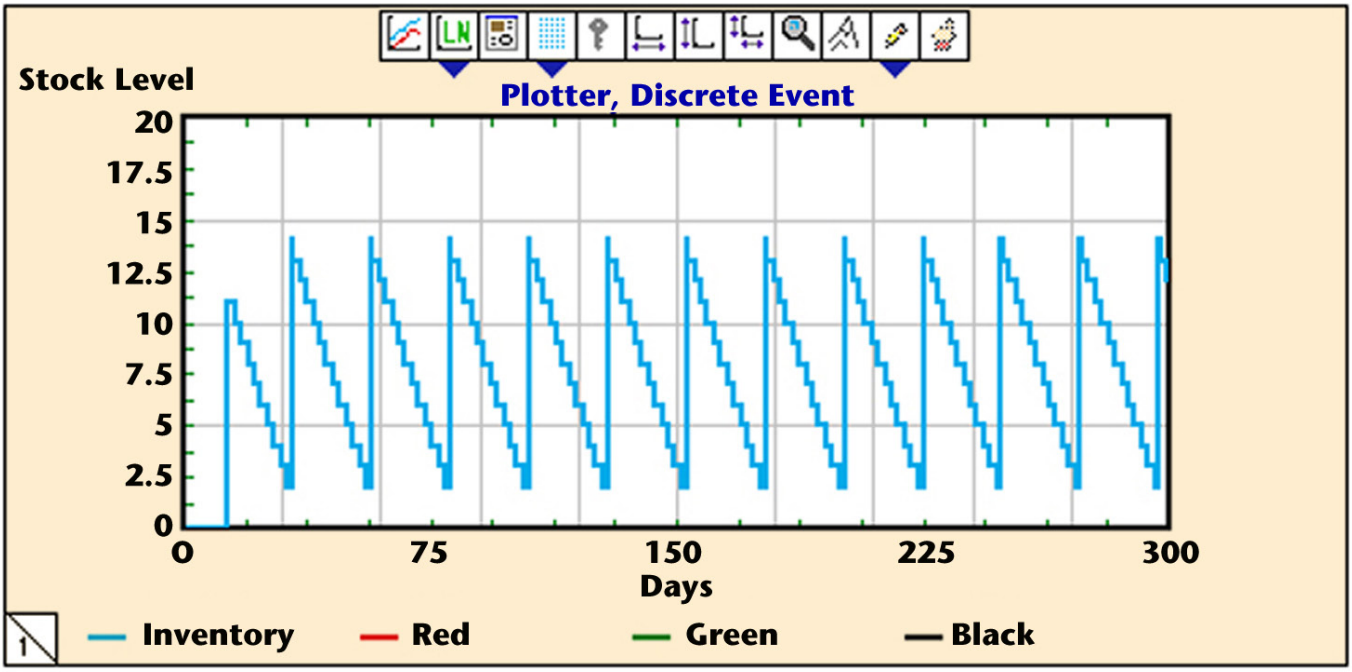


Figure 2: The typical saw pattern using the optimal settings for the reorder point (ROP) and the economic order quantity (EOQ) derived from the discrete event model.

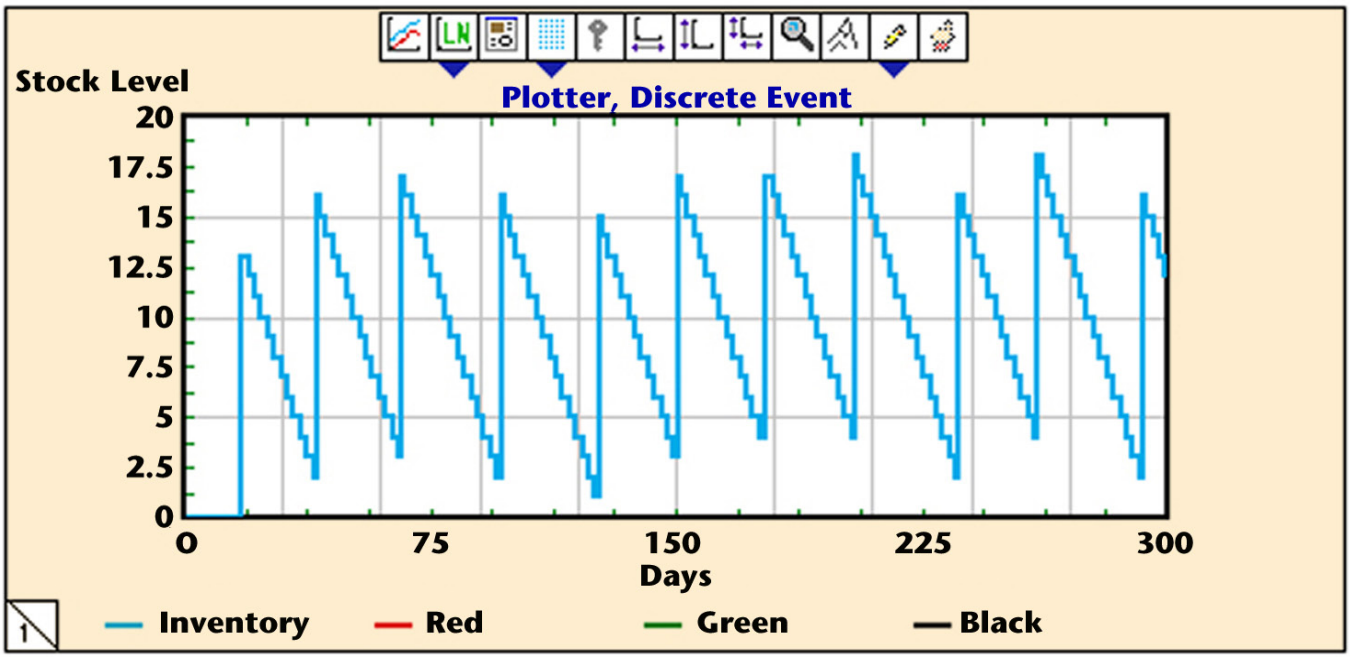


Figure 3: A somewhat unstable saw pattern due to the scrap happening in the supplier plant, resulting in a higher ROP and EOQ, as well as a higher average inventory and maximum inventory.

optimized values that were determined for no scrap in the supplier plant. In order to get these results the ROP was 8 and the EOQ was 12.

For a 10% scrap rate at the supplier, where boards are scrapped out at the end of the process, as you would find with gold or any metal finishing operation, you have to restart at the beginning. This inefficiency causes ROP to be

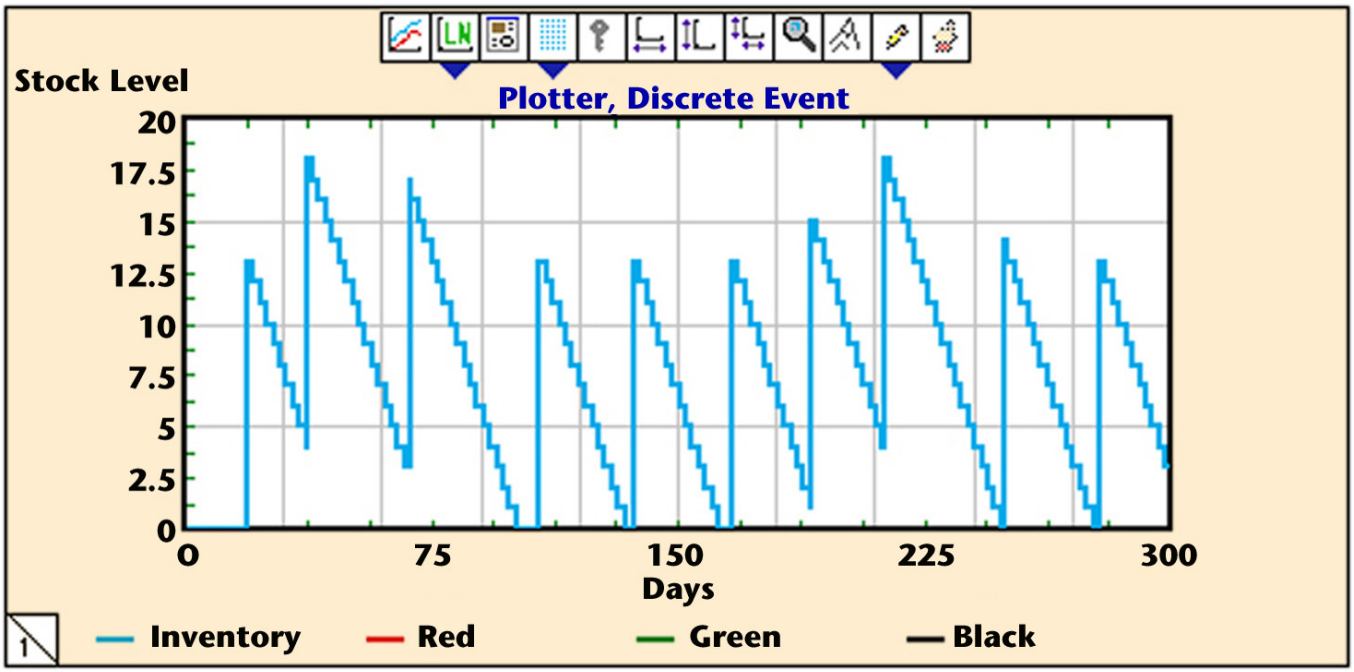


Figure 4: Even with the optimum ROP and EOQ derived in the model, you can have worst-case scenarios that cause your customer to be down for days. What is the cost to your business when these worst-case scenarios happen?

11 and the EOQ to be 14. Figure 3 shows a typical safe result where the maximum inventory was found to be 18 and the days down was 0. The maximum inventory for the no scrap case was 14.

This may not seem like a great concern, until you consider what the worst-case scenario might be. Figure 4 shows the worst case from 500 runs at the optimum settings for 10% scrap.

For this run, there were a total of 16 days of insufficient inventory, where the customer was shut down! One of these periods of shutdown, which happened just after 75 days, was for seven days straight. True, this was an unlikely outcome, one out of 500 (we will get in a deeper dive with this model in the next article about risk), but the important point is that it did happen! Just like the larger than expected water flow into Lake Powell, which was a low-probability event, it did happen. What are you going to tell your customer when their process has been shut down because of issues with restarts and scrap in your factory? That you filled out a SCAR and reprimanded the responsible operator and it will never happen again? A lie? Will you say that this is a random occurrence and it will happen again and you don't know when? The truth?

It is a no-win situation for everyone. A possible win/win would be to tell your customer that you are fundamentally changing your metal finishing process, which will drastically reduce the probability of this happening again and provide a huge improvement in your ability to supply your customer consistently. **PCB**

Gray McQuarrie is president of Grayrock & Associates, a team of experts dedicated to building collaborative team environments that make companies maximally effective. McQuarrie is the primary inventor of the patent, *Compensation Model and Registration Simulation Apparatus for Manufacturing PCBs.* He has worked for AlliedSignal, Shipley, Photocircuits, Monsanto and others. Contact McQuarrie at gray@grayrock.net.

National Healthcare Policy Drives Flex Innovation to Meet Demand for Home Medical Electronics

by Dick Pirozzolo

SUMMARY: While John Q. Public is thinking about healthcare reform during this presidential election year, PCB makers are focusing on technology that will improve home-based monitoring and treatment.

The Patient Protection and Affordable Care Act and presidential election-year rhetoric on Medicare and Medicaid reform has just about everyone thinking about the cost of healthcare.

While breakthroughs such as an HIV vaccination or early detection of the onset of dementia would have a huge cost impact, healthcare policy makers and the electronics industry—flexible PCB makers included—are looking elsewhere. Technology that will keep people out of the hospital by improving home-based monitoring and treatment looks most promising.

Ezekiel J. Emanuel, M.D., who chairs the Department of Medical Ethics and Health Policy at the University of Pennsylvania and is an advisor for health policy to the White House Office of Management and Budget, is an advocate for encouraging healthcare innovations that may be less gee-whiz but actually lower cost and improve patient outcomes. In a New York Times opinion piece titled On Falling for Fake Innovation, Emanuel writes, "By the end of the decade, we will see many … true innovations—small and large, high-tech and non-tech. Diabetics' smartphone applications could transmit their glucose levels to doctors' offices; wireless home monitoring systems could be installed for patients with congestive heart failure… There will also be many innovations in electronic health records. These aren't flashy robots, gleaming scanners or new pills, but there is already evidence that they will save money and improve care." Emanuel is a champion of electronically managed prescription medicine adherence systems as yet another innovation to be encouraged under the Affordable Healthcare Act now being implemented.

The drive toward home healthcare technology solutions foretells opportunity for flex and rigid-flex designers and manufacturers who are equipped to meet reliability standards in an environment where sophisticated electronics are being placed in the hands of older and infirm individuals. James Mahoney, applications engineer at Mass Design in Nashua, New Hampshire, points out, "Home healthcare electronic devices increasingly depend on flex circuitry to cram more functionality into smaller spaces while delivering nearly 100% reliability. And they have to withstand the rigors of 'round-the-clock operation and be easy to use, too. These applications are so demanding that instead of the term

high reliability, the PCB industry should adopt the term high-stress circuits to reflect the unprecedented user and environmental demands of home healthcare applications."

Systemic Electronic Solutions

In addition to medical management and monitoring equipment placed in the home, another big opportunity for the electronics industry is to create a foolproof integrated approach to prescription medicine administration and adherence that seamlessly links the doctors, hospital pharmacy, nurses, independent pharmacy, caregivers, family and patients together with accurate, up-to-date information.

A coordinated and properly linked communication system will improve the capability of home-based treatment markedly over the coming years. Home dialysis, unheard of a decade ago, is being improved at the patient end, with insulin pumps that keep glucose levels stable for diabetics. Those suffering from Parkinson's disease could use monitoring systems that might be incorporated into smartphone applications.

In short, patients will be going home from the hospital with electronic devices in much the same way they used to go home with a sheaf of prescriptions.

David Bear, M.D., founder and chairman of MedSentry, Inc., in Westborough, Massachusetts, and an observer of home healthcare technology trends, explains, "With Parkinson's disease, for example, medication has to be precisely administered based on patient activity. Too high of a serum level of medication and patients can develop tremors; too low of a level and the patients become frighteningly immobile. In the past, doctors had to ask their patients about their daily activities, resorting to memories, to determine the correct dosage. Today, these patients can carry an accelerometer to create a detailed, statistically reliable record of their movements. Similarly, diabetes patients can check their glucose level with a light beam focused on their blood vessels. These are not the big medical breakthroughs that make the evening news, but modest electronic applications that have an impact on better medical outcomes and lower cost."

In addition to doctors relying on patients' faulty memory, technology eliminates another human failing—lying. Studies show that patients lie to their doctor about half the time regarding drinking, medication, unprotected sex and so forth.

"Whenever patients say they have a couple of cocktails after work, doctors routinely double that number," Bear quips.

MedSentry Prescription Adherence System, about to be evaluated at Mass General Hospital, stores prescription medicines and monitors adherence through digital visualization. The device sits on a table, can be moved to a different home or apartment, and checks in via the Internet, communicating with a central monitoring facility using Wi-Fi or 4G wireless cards.

> ***"Studies show that patients lie to their doctor about half the time regarding drinking, medication, unprotected sex and so forth."***

Says Bear, "Homecare has nowhere to go but up. People are living longer. Aging at home, with an independence program, is growing in popularity. In addition to prescription medicine adherence, technology is emerging that allows us to collect electrocardiography information, blood pressure data and weight. The MD can readjust medications accordingly, with devices like MedSentry confirming the new regimen. This creates a closed data loop coming from the home and returning as a medication adjustment that is verified by home monitoring. The system cuts down on follow-up doctor visits, trips to the emergency room and, most importantly, return hospital stays where older people are subjected to stress and resistant bacteria."

Prescription medicine adherence technology is typical of the design challenges flex and rigid-flex designers and manufacturers will be facing as the type and complexity of home healthcare systems grow. MedSentry, for example, is built around a panel computer with a user-friendly, touchscreen. It has 28 individual

cups with LED illumination containing enough medication for up to four doses a day for seven days. Eight video cameras peer into the cups and send high-resolution images of the pills to those responsible for patient care. When the cups are filled, the central monitoring station also checks for accuracy. Medication reminders can be transmitted by vibration, light, sound or any combination of the three.

Mahoney points out, "These devices benefit markedly from flex. They are crammed with communication electronics, LEDs and video cameras. They pack a lot of capability into a small space. They have to be tough, easy-to-use and virtually drop proof as well as compact and attractively designed."

Reverse Technology Transfer

Mahoney observes that the trend toward more home-based medical electronics is bringing about an unexpected benefit, reverse technology transfer. "We think of home medical electronics as less-costly consumer versions of highly sophisticated, expensive equipment found in hospitals and physicians offices. Technology is going the other way too. Medical devices designed for home use are contributing to lowering the cost and improving technology for professional equipment as well."

Palomar Medical Technologies of Burlington, Massachusetts, designs, produces and sells cosmetic lasers and intense pulsed light (IPL) systems to improve skin appearance for men and women, and has now developed a low-price, non-ablative fractional laser for home use. This product is now being marketed to professionals. Explains Joseph P. Caruso, Palomar, CEO, "We have leveraged many of our consumer product technologies and manufacturing processes to substantially reduce cost and enable us to offer entry level professional systems at a very attractive price."

Figure 1: MedSentry—a prescription adherence system, stores prescription medicines and monitors adherence through digital visualization.

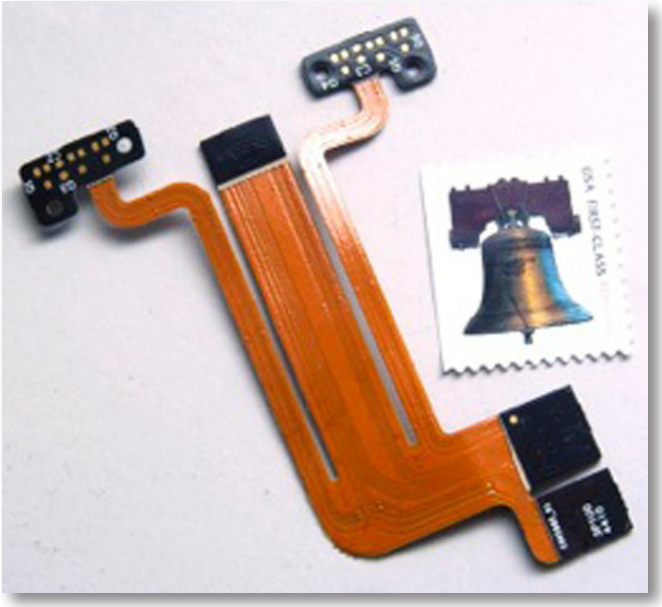


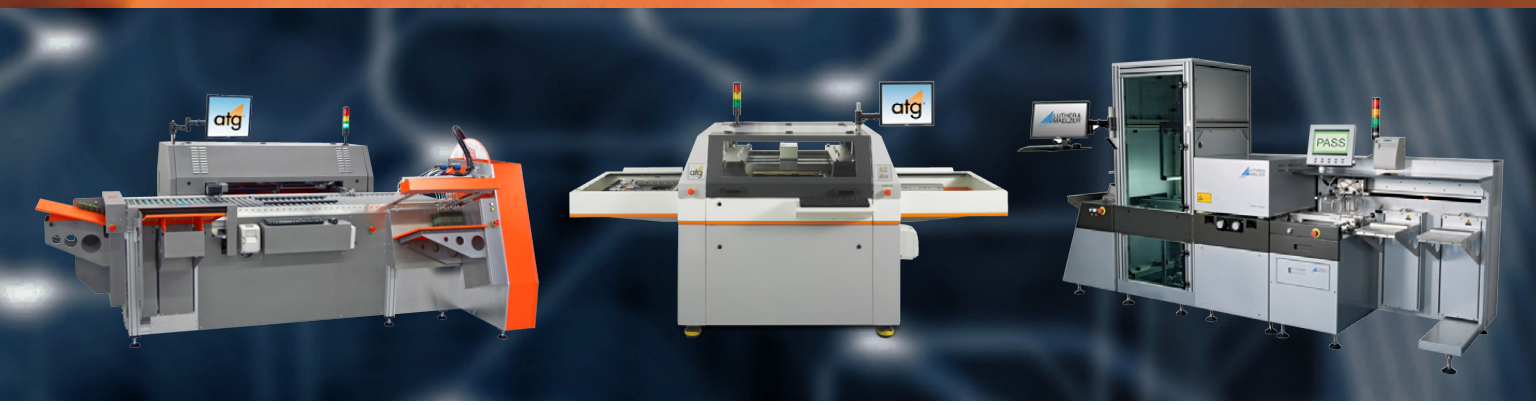
Figure 2: Home healthcare delivery and monitoring systems will become more prevalent, pushing the demand for increasingly smaller devices, benefiting from flex and rigid-flex PCBs.

With many devices, especially those that are hand held, Mahoney notes, "Compressing more circuitry into a smaller space by using flex not only offers the advantage of decreasing space, it also reduces the risk of uneven performance and increases circuit speeds.

"In addition to speed, the need to be thin is also a challenge. Where we used to have surface mounted components encased in ceramic material, we are now seeing components placed in recessed pockets to save space and mounted directly in contact with metal to dissipate heat," states Mahoney.

Flexibility is also inching toward stretchability. Last year, *The Economist* predicted that coming generations of circuits will stretch to mimic, say, human skin in a prosthetic hand.

Your Single Source for Your Electrical Test Needs



A5a
Fully automatic flying probe test system

S2
High speed Flying probe test system

LM800 Picomat
Grid test with Sniper optical board alignment

atg Luther & Maelzer is a leading supplier of electrical testing solutions for the pcb industry. We offer solutions for every application ranging from main stream technology all the way to HDI products such as smart phones and tablet PC technology.
Our atg flying probe tester products are renowned for speed, accuracy and automation while the Luther & Maelzer grid tester product range is known world wide for high throughput, repeatability and flexibility.

atg

LUTHER & MAELZER

www.atg-lm.com
Phone: +49 (9342) 291-0
E-mail: sales@ atg-lm.com

Even the hard components that sit on tiny islands in these circuits may one day be flexible.[1]

Says Mahoney, "True stretchability in flex circuitry is still in its infancy. Stretching a circuit board, like an elastic band, changes the electrical characteristics of the vias by making them thinner and longer. We've turned to mechanical means to add stretchability more simply. Imagine a hairpin or spring, with the ends being pulled apart. It's a mechanical approach to adding stretchability without actually stretching the material. In principle we do the same thing with flex."

Of course, cost is a factor. Mahoney points out, "I run into cost every day when I meet with OEM manufacturers. The manufacturing cost has to be weighed in several ways with reliability being the byword. Consider the cost of downtime. In a hospital, where there may be multiple devices performing the same function, downtime is not as costly or as critical as when home health equipment fails in the middle of the night."

On the outcome end of the equation, the cost savings derived from reliable home-based technology—let alone reduction in human suffering—are enormous. Says Bear, "Congestive heart failure is the most expensive illness we treat. Physicians prescribe as many as eight different medications, some of which require critically precise, constantly adjusted dosages. Patients get confused over changes, shifts to generic drugs and so forth. They typically get the dosages right only about 50% of the time. Largely because of adherence errors, about 27% of these individuals wind up back in the hospital in the first month following discharge."

Monitoring weight and blood pressure at home and having a nurse call to check up on the patient has been shown to cut hospitalization by around 40%. "With broader implementation of technology we can do much better. Our goal is to maintain adherence at greater than 95% and reduce hospitalizations stemming from poor prescription medicine adherence by more than 45% at a lower cost than intensive home health monitoring," Bear explains.

Adds Mahoney, "Increasingly, the most reliable solutions in these high stress home healthcare environments are flex and rigid-flex PCBs that can meet the high-stress demands placed on them by their users."

When it comes to the future of flex circuit technology in home healthcare monitoring and treatment, these are the six factors to watch, says Mahoney:

1. Recessed components that will squeeze more capability into thinner boards, while flex enhances heat dissipation in smaller devices to increase reliability.
2. Smaller and faster circuits in healthcare devices relying on RF communication with electronics operating at faster speeds.
3. Stretchable PCBs with wearable circuitry—including prosthetic devices with circuitry that acts as a subcutaneous layer around an extremity, possibly adding sensitivity to the fingertips of a prosthetic hand, for example.
4. Flex will play a bigger role in a vastly improved systemic approach to prescription medicine adherence—electronically linking the hospital and patient using smaller, more widely deployed, handheld devices.
5. Expect reverse technology transfer. Home healthcare devices will pave the way for making their professional counterparts less costly and more universally available, helping to improve healthcare delivery.
6. Smartphone sales will increase and lower healthcare costs. Home medical devices will increasingly be incorporated into flex-dependent smartphones that can take your pulse, read glucose levels and measure activity. This will increase sales of smartphones among an older population, as it may be cheaper to equip patients with a smartphone loaded with the medical monitoring applications than to manufacture specialized devices in lower volume. **PCB**

References

1. The Economist

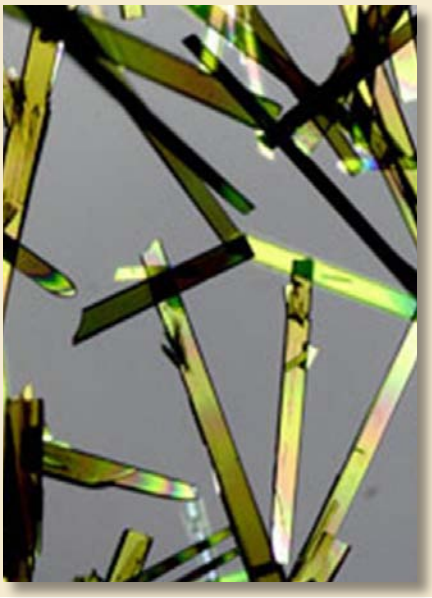
Dick Pirozzolo is a freelance journalist who writes about trends in technology. He lives in Wellesley, Massachusetts, and is a member of the Society of Professional Journalists and Foreign Press Association of New York. For more information, e-mail dick@pirozzolo.com.

Breakthrough in Ferroelectric Materials

A new class of organic materials developed at Northwestern University boasts a very attractive but elusive property: ferroelectricity. The crystalline materials also have a great memory, which could be very useful in computer and cellphone memory applications, including cloud computing. The study was published Aug. 23 by the journal Nature.

A team of organic chemists discovered they could create very long crystals with desirable properties using just two small organic molecules that are extremely attracted to each other, causing them to self assemble into an ordered network that is needed for a material to be ferroelectric.

The starting compounds are simple and inexpensive, making the lightweight materials scalable and very promising for technology applications. In contrast, conventional ferroelectric materials—special varieties of polymers and ceramics—are complex and expensive to produce. In addition to computer memory, the materials could potentially improve sensing devices, solar energy systems and nanoelectronics.



"This work will serve as a guide for designing these materials and using ferroelectricity in new ways," said Samuel I. Stupp, Board of Trustees Professor of Chemistry, Materials Science and Engineering, and Medicine and senior author of the paper. "Our molecular design enables us to invent a nearly infinite library of ferroelectric materials."

The new ferroelectric materials could be developed into non-volatile memory. With this type of memory, if the power is turned off, the information is retained. If the cloud and electronic devices operated on non-volatile memory, $6 billion in electricity costs would be saved in the U.S. annually, the researchers said.

Click here for more information.

Uyemura Investing and Growing in PCB Industry

by Real Time with...IPC APEX EXPO 2012

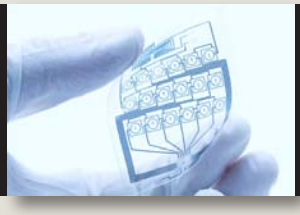
VIDEO INTERVIEW

The director of operations for Uyemura, Don Walsh, talks about the company's efforts around R&D and product development for the PCB industry.

CLICK TO VIEW

realtimewith.com

Most-Read Flex007 News Highlights This Month



Printed, Flexible, & Organic: Lowering Healthcare Costs

Emerging technologies in printed, flexible, and organic electronics are poised to impact the $300 billion global medical device market as governments, insurers, and patients seek to bring down healthcare costs, according to Lux Research.

Global Flexible Electronics Market to Hit $25.9B by 2018

Flexible electronics is an emerging field of science and manufacturing technology, and enables planting of electronic devices onto conformable plastic substrates. The flexible electronics sector, similar to large area or macro electronics, organic electronics, plastic electronics, and printed electronics verticals, is driven by the global demand for lighter and smaller electronic products that consume less power.

Career Posts Record Revenues for July

Flexible printed circuit board maker Career Technology saw its consolidated revenues gain 14.2% sequentially and 12.3% on year to top an all-time high of NT$1.145 billion (US$38.22 million) in July.

June Flex Shipments Rose, Bookings Fell Year-on-Year

Flexible circuit shipments in June 2012 were up 2.1%, and bookings were down 8.4% compared to June 2011. Year to date, flexible circuit shipments decreased 7.1% and bookings decreased 3.4%. Compared to the previous month, flexible circuit shipments increased 29.7% and flex bookings were up 14.9%. The North American flexible circuit book-to-bill ratio remained high at 1.16.

Matrix Debuts Coverlay, Bondply for FELIOS Flex

Matrix Electronics announces acrylic adhesive coverlay and bondply to complement its Panasonic FELIOS flexible laminate line. These new offerings are IPC specification certified, and cover the most popular adhesive/film combinations.

DKN Research Develops Ultra-Thin, Copper-Free Flex Circuits

DKN Research, a leading engineering firm specializing in microelectronics and packaging technology, now carries ultra-thin, copper-free flexible circuits with reliable via holes. The firm developed a series of processing technologies to generate thin nickel traces on both sides of thin polyimide film, specifically targeting those scientific and medical applications that require the elimination of copper metals from electronic devices.

AT&S' Q1 Performance Paves Way to Positive Year

AT&S reaffirms its positive outlook for current financial year following modest first-quarter performance. The introduction of new models, product launches, and declining sales of some mobile devices customers all had an impact on AT&S Group's performance in the first quarter. Revenues were up by 14% and the EBITDA margin improved slightly.

Molex Satisfied with FY2012 Results Amid Revenue Drop

"We are encouraged with our margin performance in the June quarter even though revenue was below our expectations. More importantly, bookings have now increased sequentially in Q3 and Q4 and new product design activity remains extremely strong," commented Martin P. Slark, Chief Executive Officer.

Aspocomp Benefits from Acquisition; Sales up 14%

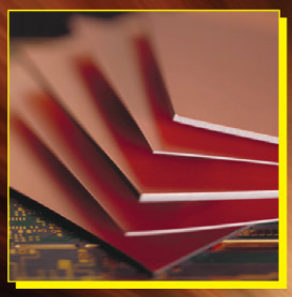
"Aspocomp posted better performance in the second quarter of 2012 than in the first in spite of the uncertain market conditions. Net sales grew to EUR 13.1 million following the acquisition of the Teuva plant. However, due to unstable demand, capacity utilization was not at a desirable level. Operating profit amounted to EUR 1.2 million, slightly over nine percent of net sales," said CEO Sami Holopainen.

Ibiden Reports Slightly Higher 1Q FY2012 Sales

PCB maker Ibiden Co. Ltd has reported 1Q FY 2012 net sales of 69.591 billion yen ($889.5 million at $1:78.2340 yen), up by 1.9% from the same period last year. Operating income, meanwhile, was down by 6.2% to 3.89 billion yen ($49.76 million) from last year, while ordinary income declined by 4% to 3.335 billion yen ($42.6 million).

Tactical Advantage.

When Reliability Counts.
Mission-critical applications have relied on Rogers' microwave materials for years. The superior electrical and mechanical characteristics of Rogers' military-grade laminates provide the stable, consistent performance over time and temperature that's so critical for aerospace and defense applications.

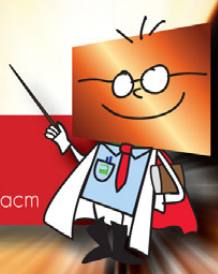


The rock solid performance of Rogers' microwave materials deliver the high performance and high-reliability demanded by the most mission-critical applications. Can you afford to use anything less?

To learn more about Rogers' aerospace and defense microwave material solutions, visit us at:
www.rogerscorp.com/military

Photos used by permission from Northrop Grumman Corporation.

Read Our New ROG Blog
www.rogerscorp.com/acm



	Features	Benefits
RT/duroid® 6202PR	Reduced planar resistor variation	Lower manufacturing costs due to decreased tuning
	Low thermal coefficient of dielectric constant	Stable electrical performance versus temperature
	Low coefficient of thermal expansion	High reliability in complex multilayer designs
RT/duroid® 5880LZ	Dielectric constant of 1.96	Lowest dielectric constant microwave PCB material
	Low z-axis coefficient of thermal expansion	Plated through hole capable
	Light weight	Advantage for airborne applications
RT/duroid® 6035HTC	Highest thermal conductivity (1.44 W/mk) for 3.5Dk printed circuit board laminates	Excellent power handling capability
	Low loss 0.0013	Excellent high frequency performance
	Low profile, thermally stable copper options	Lower insertion loss, and performance reliability in high temperature applications

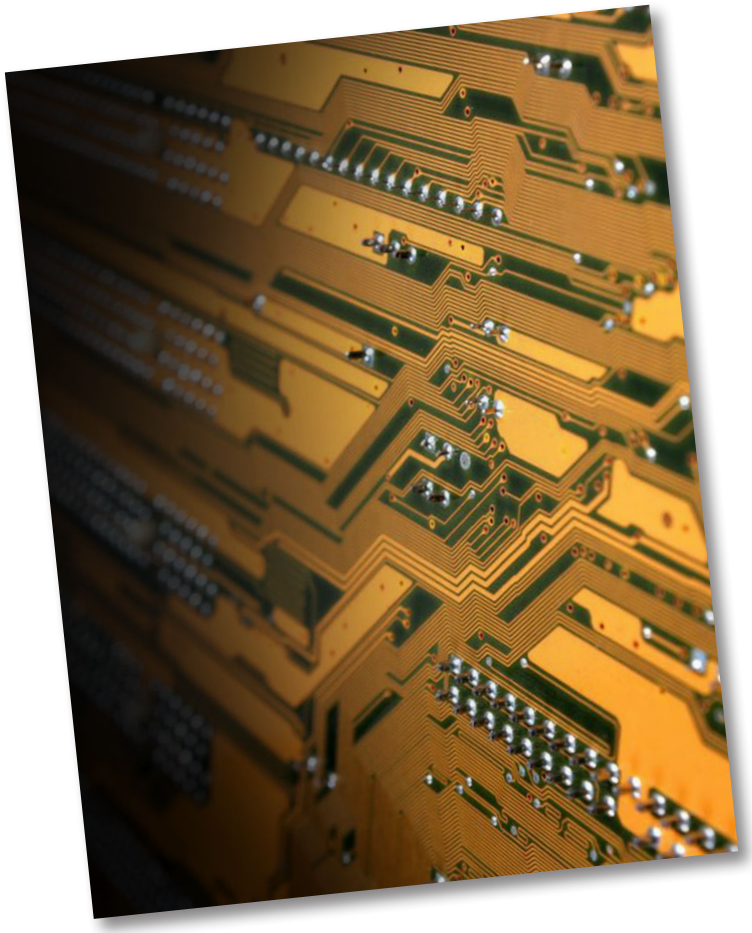
The world runs better with Rogers.®

ROGERS CORPORATION
Advanced Circuit Materials

USA - AZ, tel. +1 480-961-1382 • EUROPE - BELGIUM, tel. +32 9 235 3611 • www.rogerscorp.com/military

Copper Surface Preparation for Dry Film Lamination, Part E

by Karl Dietz
KARL DIETZ CONSULTING, LLC



SUMMARY: *In this last installment of a five-part series covering copper surface preparation for dry film lamination, surface preparation processes are examined, including brushing, jet and brush pumice, and aluminum oxide. Also: chemical surface preparation processes, with emphasis on trends in surface preparation and their root causes.*

To recap, this five-part series on copper surface preparation for dry film lamination has addressed the following:

A. The issue of copper topography and dry film conformation
B. The issue of characterizing the copper topography and establishing topography specifications
C. The issue of dry film adhesion to the copper surface, understanding the chemical interactions at the copper and resist surfaces
D. The issue of characterizing the copper surface chemistry and establishing criteria for acceptable surfaces based on empirical adhesion tests, and lastly
E. A summary of copper surface preparation processes, their advantages and limitations, as well as trends in this technology

In this final installment, a look at the surface preparation processes that are in use, with emphasis on changes and trends in surface preparation and their root causes are examined.

A Brief Overview of Surface Preparation Processes

One can group the different techniques as follows:

- Mechanical brushing
- Pumice (and aluminum oxide) cleaning, both brushing and jetting
- Chemical cleaning, which may consist of an alkaline or acid cleaner, or microetch, or combinations or sequences of such chemistries, including electrochemical cleaning

Brush Cleaning

Brush scrubbing is a common surface preparation technique. Being an abrasive process, brushing both structures the surface and removes surface contaminants at the same time. Brushing machines are typically conveyorized modules with rotating pairs of abrasive brushes to clean the top and the bottom side of a panel in one pass. Water sprays cool the brushes and

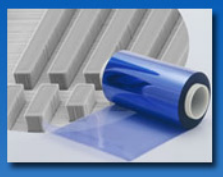
TECHNICA, U.S.A.

Supplying a Full Line of Photosensitive Process Materials in Partnership with Hitachi-Chemical

Hitachi Chemical
Working on Wonders

Photosensitive Dry Film "PHOTEC"

H-Series



H-7000

H-6200

H-N650

H-7000 Series are aqueous type photosensitive films used for high-density PCBs.

- *Excellent tenting performance is helpful in improving yield*
- *High photosensitivity can contribute to improving through-put performance*
- *Excellent adhesion is effective in decreasing the defect of circuit*

H-6230/38 are newly developed for tenting, etching and pattern plating process.

- *High photosensitivity could improve through-put performance*
- *Excellent color contrast after exposure*
- *Short foot of resist profile*

H-N650 is the fully aqueous type photosensitive film that can be used for plating in circuit formation of high-density PCBs. It's an excellent resist for electric acid plating of all metals common to the PCB industry including nickel and acid cold.

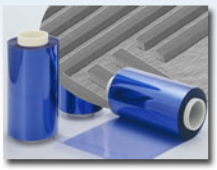
RY-3600 Series

MORE INFO

RY-3600 series are aqueous type photosensitive films used for high-density semiconductor packages such as BGA, CSP, etc.

- *Excellent adhesion and resolution*
- *Suitable for semi-additive process*
- *Shorter resist foot for finer patterns*

SL Series



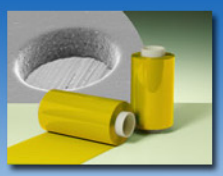
MORE INFO

SL series are fully aqueous applicable photosensitive films used for UV laser direct imaging (UV-LDI) systems.

- *High photosensitivity (8~16mJ/cm2)*
- *High contrast enables 1 minute confirmation*
- *High resolution and excellent tenting property*

Photosensitive Coverlay Film "RAYTEC"

FRSeries



MORE INFO

FR series is a photosensitive coverlay film that has excellent flexibility and low warpage developed for ultra-high-density FPCs such as cellphones, digital cameras, and medical equipment.

- *Flame resistant (UL94 VTM-0) with no halogenous or antimony compounds*
- *Excellent flexibility & low warpage performance reduces springback issues*

***Contact Us** for more information about Hitachi Chemical's line of products.*

800-909-8697 **www.technica.com**

carry the removed surface particles to filters. The two dominant brush constructions are (nylon) bristle brushes and compressed pad brushes. Bristle brushes have individual nylon bristle arranged like spokes on a wheel around a core. The abrasive material (grit), usually silicon carbide, is impregnated in the nylon bristles.

Pumice Cleaning

Pumice cleaning involves the use of pumice powder. It is applied as an aqueous slurry by brushes or by jetting. The word pumice is sometimes loosely used as a generic term for a variety of abrasive materials, whether they are pumice chemically or not. Pumice is a complex silicate of varying composition and structure depending on where it is mined. The most commonly used pumice is mined on the small island of Lipari off the shore of Sicily, Italy. The pumice is ground and used as is (unfused) or heated (fused) to yield harder particles. Pumice mined in the U.S. is normally Navajo pumice and is typically fused. Pumice comes in many grades, the most common being 3F and 4F, with an average particle size of about 60 microns.

Chemical Cleaners and Microetchants

Alkaline cleaners contain, as the name suggests, alkali. In addition, they may contain wetting agents (surfactants) and antifoam. They are best suited to remove organics and some inorganic salts, but they don't remove copper oxide and chromates. Acid cleaners are usually formulated with sulfuric acid, occasionally with HCl or phosphoric acid, or a combination of mineral acids. They, too, can contain wetting agents and antifoams. They are suitable to remove copper oxides and some or all of the chromate, and they can remove organics, but not as effectively as alkaline cleaners. Some acid cleaners may show a slight microetch. Microetchants are acidic and the active ingredient is either persulfate, monopersulfate, or hydrogen peroxide. It is advisable to remove potential etch retardants such as organic impurities before the microetching. Microetchants remove copper oxides and chromates (at a sufficient etch depth). In addition to cleaning, microetchants change the topography of the copper. They typically yield a desirable micro-roughness. One-step cleaners have become very popular in recent years. They are acidic cleaners that remove copper oxides, some or all chromate, and they do a reasonable job removing organics. Some cleaners are formulated to also microetch and provide an antitarnish.

Types and Functions of Chemical Cleaners and Microetchants

- Alkaline cleaners
  - remove organics, some inorganic salts
  - don't remove copper oxides, chromates

- Acid cleaners
  - remove copper oxides, some chromate
  - can remove organics (less effective than alkaline cleaners)
  - may microetch slightly

- Microetchants (acidic; persulfate; monopersulfate; peroxide)
  - etch retardant, e.g., organics, to be removed first
  - remove copper oxides
  - remove chromates (at sufficient etch depth)

- One-step cleaners
  - acidic cleaners, remove copper oxides some chromate
  - may remove some organics, but not very effectively
  - may also microetch and contain anti-tarnish

Considerations in the Selection of a Surface Preparation Process, Limitations, and Technology Trends

There are, of course, economic considerations in the selection of a surface preparation method such as the cost of consumables, the cost of maintenance and waste treatment, and floor space requirements. With regard to technical considerations, one trend I have noticed over the last twenty-odd years is a move away from mechanical preparation methods such as brushing or pumice cleaning to chemical cleaning/microetching, whereby the chemical clean-

Innerlayer (Total Thickness)	Applicable Surface Preparation Methods			
	500 Grit Brush Scrub	Brush Pumice	Jet Pumice	Chemical Clean
> 8 mil (0.2 mm)	Yes	Yes	Yes	Yes
6-8 mil (0.15-0.2 mm)	No	Yes	No	Yes
< 6 mil (0.15 mm)	No	No	No	Yes

Figure 1: Guideline for selecting a surface preparation method to minimize laminate distortion.

ing processes have been streamlined by the use of a single cleaner/microetch chemistries ("one-step cleaner").

There are several drivers supporting this one-step trend. Innerlayer copper-clad laminate, as well as multilayers, have become much thinner, which makes size distortion of the layer due to the introduction of stress by mechanical preparation methods more of a problem. Basically, the brushing or jetting stresses the copper-clad laminate, a stress that is released or partially released after etching, causing a distortion of the circuitry pattern, which can lead to misregistration. Chemical cleaning essentially eliminates this problem. Figure 1 offers a guideline on which method to choose based on laminate thickness.

Another reason to move away from pumice or aluminum oxide cleaning is the fact that drilled through-hole diameters have become smaller, allowing for the a greater danger of having a pumice particle stuck in the hole, which would interfere with through-hole cleaning and metallization.

Mechanical surface preparation has historically been used primarily to provide the desired surface roughness for proper dry film adhesion. With the advent of reverse treated foil there is no need to roughen such a surface. The only surface preparation that is needed on such a surface is a mild acid cleaner and/or microetch. The microetch actually makes the surface a little smoother, which is desirable since the reverse-treated foil has a surface roughness that is on the upper limit of what is desirable.

Other Selection Criteria and Trends

Copper foils have become thinner on average, and the consideration of not removing a large amount of copper during the surface preparation may come about so that enough copper is conserved to warrant the required current-carrying capability. For example, alkaline cleaners don't remover copper (nor copper oxides) while acid cleaners may remove a small amount of copper, and microetchants will remove more or less copper depending on the chemistry and process conditions. Brush pumice will remove copper while jet-pumicing won't.

An extreme situation where copper removal is a prime concern is the semi-additive process (SAP). In SAP processing, only a very thin layer of electroless copper is deposited on the prepared dielectric layer. It is so thin that no copper surface preparation can be practiced. Therefore, the topography of the copper mirrors the topography of the dielectric, which in turn is a function of the dielectric composition, e.g., the size of the silica filler particles, the domain size of the dissimilar resin components, and the desmear chemistry and processing conditions. **PCB**

Karl Dietz is president of Karl Dietz Consulting LLC, offering consulting services and tutorials in the field of circuit board and substrate fabrication technology. Dietz can be reached by e-mail at karldietz@earthlink.net or phone (001) 919 870 6230.

Conversations with…
Printed Circuits Inc.

By Marcy LaRont
I-CONNECT007

SUMMARY: Marcy LaRont sat down with PCi President Ken Tannehill and National Sales Manager Bob Burns to discuss the company's long history, the PCB industry and what we all have to look forward to in the upcoming years.

Printed Circuits Inc. (PCi) had its beginnings in the 1950s, at the very start of PCB manufacturing in the United States, serving the military and aerospace sectors. But things really took flight in 1977, when Ken Tannehill restarted his father's company with the help of three colleagues, $3,500 in capital, and a friend with surplus used equipment. With 55 years in the PCB industry under his belt, Tannehill is proud that PCi is thriving as one of the original U.S. rigid-flex PCB manufacturers.

Coming up on PCi's 35th anniversary, he is still excited by this industry, and there are a few things that he wants you to know. Marcy LaRont sat down with Tannehill and National Sales Manager Bob Burns to discuss the company's long history, the PCB industry and what we all have to look forward to.

Marcy LaRont: Let's start at the beginning. Printed Circuits Inc. goes way back. How did it all start?

Ken Tannehill: In 1950, my father started a PCB company called Precision Fabrication Products Inc. My dad was an aerospace engineer for Honeywell Aerospace in Minneapolis. He heard about circuit boards and put a few guys together and started his company in a garage, not far from the University of Minnesota. He worked for Honeywell during the day. Eventually, he pitched his little circuit board company to them and Honeywell took interest. My dad's small company started building some of their boards. At this point, my dad still had his day job as a Honeywell employee and was building PCBs as a side business.

But in 1951, Photo Circuits had begun PCB fabrication, and in 1954 the Bureau of Engraving had started up PCB operations. Thus, all of a sudden, Honeywell had options. They gave my dad an ultimatum. Either he needed to commit to his circuit board company and Honeywell would continue to use it, or he needed to stick with Honeywell. He decided to leave and commit to his PCB business. He moved out of the garage shop and built a 40,000-square-foot facility in Bloomington, Minnesota, and he changed the company name to Printed Circuits Inc. (PCi).

ML: So, you grew up in the industry?

KT: I definitely grew up in the industry. I started in my dad's garage shop, doing all sorts of odd jobs when I was about 10. So, I've been in the business for more than 50 years.

ML: Your dad founded his PCB shop in the '50s, naming it Printed Circuits Inc. in 1960. But you restarted it in 1977. What happened in those 17 years between?

KT: I graduated from the University of Minnesota in 1972. My dad, in the meantime, had retired and sold PCi in 1969. After graduation, I went to work for a local board shop, in sales. Five years later, that company was going through a management change, so along with

my contacts and three of my colleagues, we decided to go out on our own. To start the business, we had $3,500 and, luckily, a friend who had lots of used equipment.

ML: PCi was not always a flex and rigid-flex fabricator?

KT: No, for the first 20 years we did standard, rigid PCB manufacturing focused primarily on mil/aerospace. But, we built every type of circuit board imaginable, including a 36-layer board, which, at that time, was unheard of.

ML: Who were your first customers?

KT: Our first customer was the University of Minnesota. Our second customer was Honeywell. The relationship with Honeywell still mattered.

ML: When did you first start building flex?

KT: We did our first rigid-flex board in 1982 for Teledyne Inc. Our company was the first to qualify for the MIL-P-50884 rigid-flex specification, before it was even a requirement.

ML: And now you are an exclusively rigid-flex manufacturer. When did this change happen?

KT: Well, as time progressed and the early '90s descended upon us, I had concerns about the Asia factor, watching standard technology multilayer product transitioning overseas. I knew price was most certainly going to be a factor going forward. Thus, in 1997, I made a decision to focus on rigid-flex only. I felt it was important for us to have a market niche, to play in an arena that the overseas market was not interested in, or was not capable of participating in at that time. Rigid-flex was and still is engineering-intensive, custom fabrication requiring lots of hand-on attention, and at lower volumes.

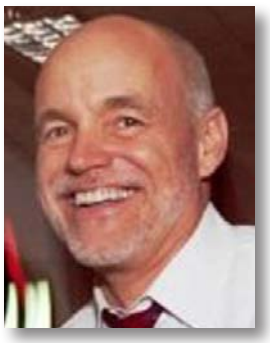
ML: What was your product mix at the time you made this announcement, and how did your customer base respond?

KT: When we made this change, our product mix was about 50/50, rigid PCBs to rigid-flex PCBs, so it was a bold move. We announced that we would honor all contracts we currently had in-house, but we would not take new business that was not rigid-flex work after a certain date. There were several customers that left us, vowing they would never design anything in a rigid-flex format. It was a rough go for about three years. But I am happy to say that today, whether direct or through subcontractors, every single one of those customers is back doing business with us.

ML: So, in 1997, you basically reinvented your company.

KT: It was like starting all over again. Customers left. We had sales voids that we needed to fill. The biggest challenge was that we needed to sell the rigid-flex concept. Engineers and designers didn't really understand rigid-flex technology and how it could benefit them in terms of utilization and packaging, and ultimately, cost. We had to get into companies at those deep, early development levels. Our belief was, and is, that one flex-rigid board can replace three to five regular multilayer circuit boards in a package, thereby creating a significant cost saving for the project overall. But getting customers past only seeing the difference in raw materials was tough. They would do a direct, cost-to-cost materials comparison and say no to the technology without going any further. That was the education process we undertook with OEMs.

Bob Burns, National Sales Manager: Absolutely—still today, when you see an assembly that contains five rigid boards in it, we know that we are in the hunt. We know we can compete with their current full assembly costs. And we have the additional selling point of increasing the reliability of the final device, by eliminating the flex connectors that have a relatively high failure



rate. The rigid-flex boards are so much more reliable than having multiple, rigid multilayer boards throughout the assembly with flex connectors that fail in the field.

ML: So how has rigid-flex developed and changed in the past 15 years?

BB: The manufacturing field has obviously gotten much stronger since we made the decision to focus on rigid-flex in 1997. But still, there are only a few truly viable suppliers that are really good at the technology. There are about 40 stated U.S. suppliers of flex and rigid-flex PCBs, but I believe that we have only about a dozen real experts in that field.

ML: That's a bold statement.

BB: Yes, maybe it is. But the companies that are really good are the ones with the experience of doing it for many years, who truly understand the failure mechanisms.

KT: Even though flex and rigid-flex PCB manufacturing is 3-4% of all PCB manufacturing, the greater PCB arena sees flex and rigid-flex as a potential growth area. So, as things have become increasingly competitive internationally, we've seen PCB fabricators move to expand into the arena. The profit margins are higher precisely because the process is technology and engineering intensive, which adds operating costs. At PCi, we have proprietary processes that we hold very dear—that we are careful not to share with anyone.

BB: Yes, within the PCB manufacturing process, there are about 20 things you have to get right for our type of manufacturing. If you don't get it completely right, you will have random opens—the most common failure mechanism in rigid-flex. We don't drill the same way or plate the same way as those building standard technology PCBs. And if you, as a manufacturer, think it's all the same, then I guarantee you shouldn't be doing it.

ML: Please tell us more about rigid-flex technology, specifically.

KT: Let's compare flex to rigid-flex. A poor man's flex is a double-sided board with a stiffener attached to it. It is relatively simple, and we are now seeing lots of flex being built in Asia. Rigid-flex, on another hand, allows for much more complex multilayer technology. And the rigid multilayer board part of the rigid-flex construction acts as the stiffener for the flex. You've got an entirely different technological ability.

> ***"PCi's customers can have a fully approved UL rigid-flex board without the additional time and expense required for UL certification. Our competitors cannot say that."***

ML: Throughout the past 35 years, what would you say are some of the most significant milestones PCi has achieved?

KT: Getting the mil/aerospace arena to accept rigid-flex PCB technology as viable technology was huge. Today, I can say confidently that nearly every single piece of manufactured equipment in both the military and aerospace sectors contains many rigid-flex PCBs.

BB: And today, the medical sector has certainly seen the benefits of rigid-flex. Now, our big challenge is getting the commercial sector to see the benefits. We are educating customers all over again. Really, we have never stopped.

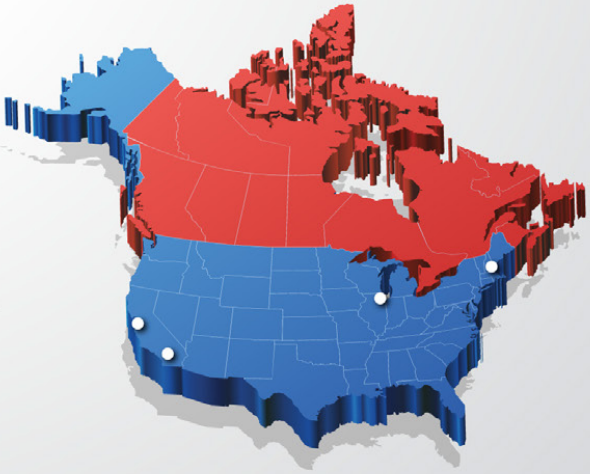
KT: Another significant achievement of which we are very proud is that today, we have the most extensive UL 94 V-0 qualification in the world. [UL94 V-0 certifies that the end-device will not burn or sustain a flame]. That is a huge benefit. PCi's customers can have a fully approved UL rigid-flex board without the additional time and expense required for UL certification. Our competitors cannot say that.

Global leaders in testing.
Industry pioneers in printing.

MicroCraft's history of Service and Support for Probe and Grid Testing, Fixturing and Jetprint is unmatched in the industry.

Don't forget we also offer Testing and Fixturing Services, offshore data prep and specialized Inkjet printing services.

We have multiple 24-hour locations to serve you. To learn more or get a quote, please contact us today at sales@usamicrocraft.com.



MicroCraft

www.usamicrocraft.com

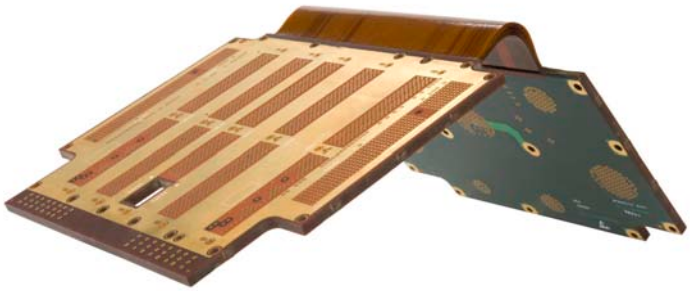


Figure 1a and 1b: Bookbinder rigid flex boards provide maximum reliability on high layer counts with tight bend radii.

BB: UL is an area where we have spent a lot of time, energy and money. And we have been trying to work with UL to educate customers and manufacturers about common misinterpretations of their qualifications, especially misrepresentations of meeting those qualifications by companies within the industry. Sometimes it's not intentional. But there is a lot of confusion and, we feel, a need for attention to these issues.

ML: Bob, I know a little bit about your efforts, and I know some of it is a little sensitive to talk about. But can you possibly explain the core issue in general terms?

BB: It's a complex issue. But what it really comes down to is that many fabricators are simply building boards with materials that are UL-certified for 94 V-O. This, however, does not necessarily represent a UL certification of the board being manufactured. This being the case, many of these finished boards are not meeting the mandated product liability insurance requirements by the end-customer. OEMs should be very concerned. They are often receiving UL-certified PCBs that have not been tested and approved by UL. The argument is out there that there is an interpretation "grey area" in regard to UL 94 V-O certification. We don't agree, nor does UL. It's pretty black and white.

ML: That is interesting and potentially controversial stuff, Bob. Unfortunately, cost is always a significant factor for companies when it comes to compliance and certifications. But obviously, customers have a lot to lose if there really is a compliance issue. I wish you luck in your efforts…

Coming back to business, Ken, what technologies are most meaningful or important to PCi right now, and into the future?

KT: We are seeing a lot of controlled impedance and buried capacitance requirements on rigid-flex product. We see this in over 30% of our work now, which is a significant increase when you consider that it was almost non-existent some years back. And impedance can be tricky. We have a tremendous amount of experience in this regard, and we believe it presents a significant opportunity for us going forward.

We also build boards with buried capacitors and resistors, and we've been told that increases reliability significantly. We work with a company who makes ALB [antilock brakes] product, for instance. They feel that buried capacitance gives them a measurable reliability performance benefit. It is a big deal. This also presents opportunity for us. It's an up-and-coming technology, but again, it's an education process. Industry technologists have to become comfortable with it. There are thousands of applications that could benefit from this technology.

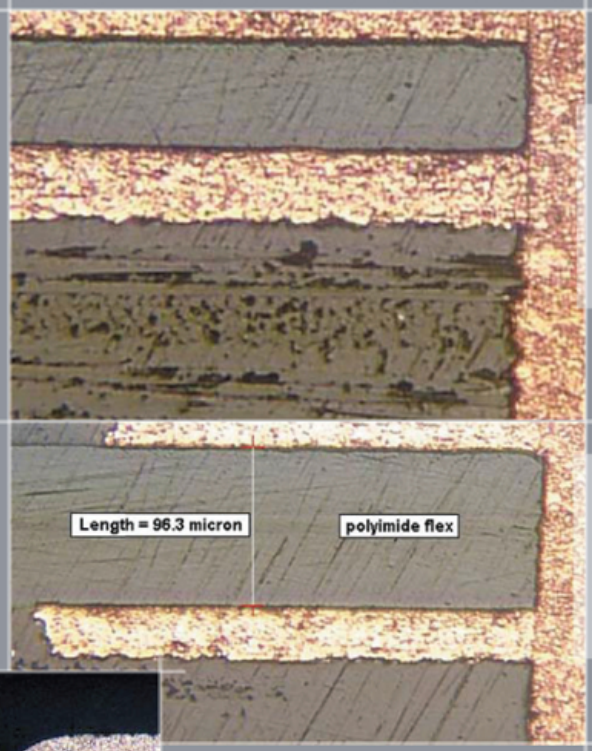
ML: And this is one of my favorite questions—what do you find to be the biggest challenges doing business today?

OM Group

Electronic Chemicals, LLC

Direct Copper to Copper Bond

Fixer Technology...
Shadow®



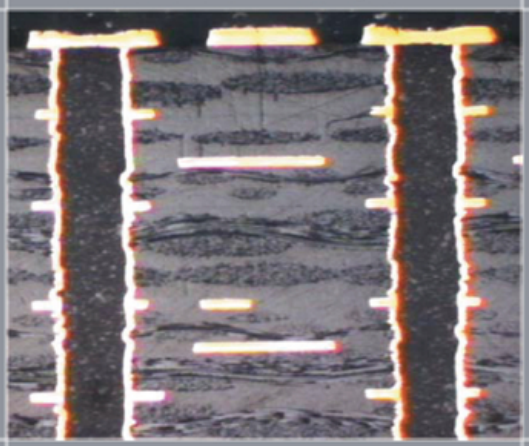
Rigid flex fabrication processed with SHADOW.®

Polyimide flex processed with SHADOW.®

Microvia Interconnect processed with SHADOW.®

SHADOW® Interconnect copper to copper bond.

The etched specimen shows complete copper to copper bond and no line of demarcation. Electroless copper and most other direct metalization systems leave several layers of conditioners, catalyst, and other materials on the interconnect, greatly compromising the integrity of the interconnect.



Sequential build PWB processed with SHADOW.®

Conveyorized Process with Flood Technology

OM Group
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: 908-222-5885

KT: I'm glad you like this question. I have a long answer: A significant concern I have is finding qualified and talented people to work in our industry. Believe it or not, when I started out in PCBs, this was a glamorous business. You'd talk about things you were doing for the military and defense, and it was exciting. It used to be that once you were part of this industry, you didn't leave until retirement. Now, I have this feeling that many folks—at least the smart and talented ones—are leaving, or trying to leave the industry altogether. I think they feel PCB manufacturing is on the downswing for the future, and they are pursuing other things. That's a big concern of mine, and should be for everyone with vested interest in our industry.

ML: I totally agree. A lot of our truly talented engineering minds are retired or retiring, and our engineering prowess is what keeps us in the game. I think we need people like you to go to local community colleges and universities and see if you can excite some of the young ones and bring them over. I'd support your efforts, for sure! What else concerns you?

KT: The blind focus solely on cost. Of course, customers want a lower cost, but it's a trickle-down issue. When cost is the primary deciding factor, quality suffers. A lot of time and money is lost in buying sub-par PCBs and the parts for those boards—all decisions based purely on cost. By the time the customer gets desperate enough to buy from someone who will give them the quality they needed in the first place, there is no lead time left. It's rush, rush, rush. Multiple-week lead times can't be compressed to two days. It makes it challenging to be successful with and for the customer.

ML: There really is something to the old adage, "You get what you pay for." I have a friend who says, "Cheaper for a reason." What else?

KT: Another big concern I have is regarding chemical bans. I follow the activity on bans in California and the Chemicals of Concern list of 3,000 chemicals, and I really wonder what the fate of PCB manufacturing will be. Are we going to have to completely reinvent the wheel, if that is even possible? We've all been mandated to be lead-free for a while now, but the government can't completely get the lead out because of tin whiskers and reliability concerns for military and defense products. No one seems to want to recognize that we are a chemical industry. Everything we do is based on chemicals or on materials with chemicals in them or used to make them. The concern is that this this type of thing will hobble the U.S. while other countries

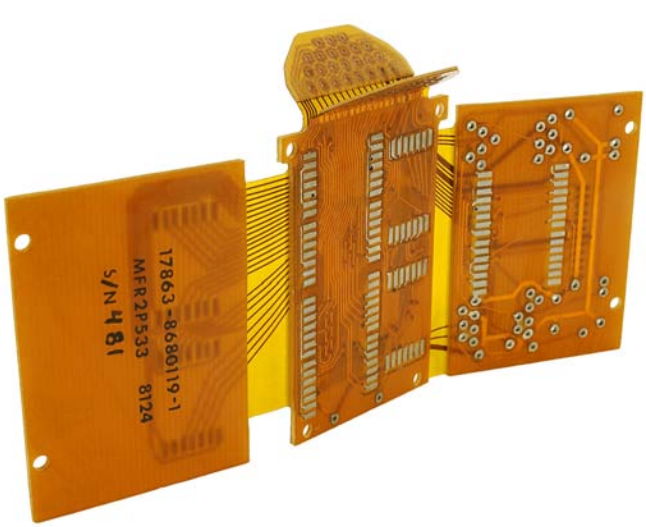


Figure 2a and 2b: PCi's first rigid-flex, built in 1982 for Teledyne for the Apache helicopter.

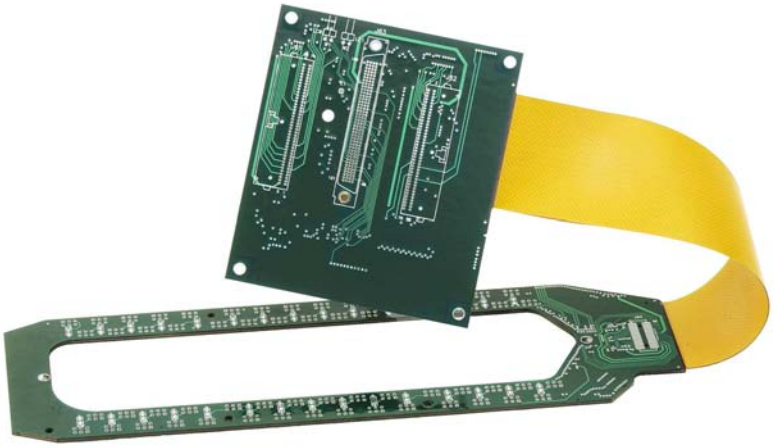


Figure 3a and 3b: Designers use rigid-flex in continuous flex applications—they can provide hundreds of thousands of bend cycles without failure.

won't have those concerns, or at least not until it's too late for us here. We already struggle with that issue economically in regard to trade imbalances. And once the business goes, it's gone. It really scares me.

And finally, the declining defense budget is a concern for my business, specifically. Thus, we hope to diversify into more work for the commercial sector, and we will work to educate that sector on the benefits and cost savings that rigid-flex will provide them.

ML: All great and important issues. Thank you. So, after all these years Ken, what makes you proud to be involved in this industry? What do you enjoy most about your work?

KT: The product application—our boards go into the strongest defense mechanism in the entire world. Our boards go into the most cutting-edge medical technology. It makes me proud to be part of this business, and proud to be an American.

ML: What about growth going forward?

KT: In the past two years, we've spent $4-5 million on new equipment, just as most everyone else was pulling in their spending and trying to get by. We continue to invest in the best equipment, and we got some good deals because we were willing to spend when others would not. We will continue to do so.

As far as sales and revenue, we set the bar really high in the last year. With the outlook for the rest of 2012 and 2013, we will feel successful if we can maintain the level we achieved last year, with maybe some small growth.

ML: Is there a final message you would like to leave with our readers? Some parting words of wisdom?

KT: Yes. My message is to those who are leaving the industry, or are thinking of doing so. I want them to know that this industry can still be exciting. And if you aren't happy where you are in this business, then I encourage you to find the right company. But don't leave. I fervently believe there are a lot of exciting things to come in technology and manufacturing. Stick around and be a part of it. **PCB**

Marcy McAllister LaRont has worked in the PCB industry for more than twenty years, and has held positions in EH&S, purchasing, production control, customer service and sales. LaRont resides in Hong Kong and may be reached at marcy@iconnect007.com.

Most-Read PCBDesign007 News Highlights This Month

Mentor Graphics: All-Time Record Revenues in Q2

"Revenues and earnings were an all-time record for a second quarter, and bookings were at the second highest level for any second quarter in company history," said Walden C. Rhines, chairman and CEO.

Avoid Signal Integrity Problems

To help engineers avoid signal integrity problems on their PCB designs, Signal Consulting Group has launched a new software tool to validate IC vendors' IBIS models.

Mentor Extends Support for 42 Embedded Development Boards

Mentor Graphics Corporation has added and improved support for 42 embedded development boards for the Nucleus RTOS, Mentor Embedded Linux, and Sourcery Codebench embedded development products. Expanded support for MIPS, SuperH, and ARM-based architectures include the following vendors: Atmel, Freescale, Logic PD, MIPS, Mistral, Renesas, STMicroelectronics and Texas Instruments.

Zuken Launches Latest PCB/IC Design Software

Zuken, a leader in PCB systems and IC package design, has released the newest version of CR-5000, its advanced PCB and IC package design software. CR-5000 version 14 sees a number of significant productivity enhancements.

Cadence Posts YOY Revenue Increase for Third Quarter

Cadence reported second quarter 2012 revenue of $326 million, compared to revenue of $283 million reported for the same period in 2011. Cadence recognized net income of $36 million compared to net income of $27 million for the year-ago period.

Early Bird Rate Still Available for Zuken Innovation World

Zuken Innovation World Americas will be held October 15-17 in Newport Beach, California. Attendees will find customers, partners, and plenty of Zuken experts ready to share their knowledge and experience.

Low-Cost Micron Copper Ink for Electronic Use

Applied Nanotech Holdings, Inc. introduced a new copper ink material based on micron copper particles for low-cost direct printing of electronic circuits for mobile devices, solar cells, display devices, and more.

The new microcopper ink material joins Applied Nanotech's nanocopper ink, which won an R&D 100 award from R&D Magazine in 2010. The copper ink is specially designed to replace expensive silver-based conductors for rapidly developing applications in the printed electronics industry. Microcopper ink has an optimal combination of high conductivity and ease of processing that allows for direct replacement of silver inks and pastes at significantly lower cost.

"Microcopper inks fill a need for lower-cost conductive materials used for applications such as smart cards, RFID antennas, touchscreens and sensors in smart phones, just to name a few," said Dr. James Novak, Director of the Nanoelectronics Division. "Furthermore, the microcopper ink retains all key advantages of nanomaterial-based copper ink, such as low-temperature sintering and ease of application."

Microcopper ink material is easily applied on various substrates by a number of additive print processes such as screen, flexographic and gravure printing, as well as some advanced dispensing techniques. This eliminates the cost and waste stream compared to other approaches, including lithography, used presently in the manufacturing of printed electronics circuits such as PCBs, for example. The microcopper ink can also be used as a patterned seed layer for electroplating and metal finishing.

For more information, click here.

Wherever technology takes you

We deliver

VENTEC designs, develops, manufactures and sells high quality copper clad laminates and pre preg bonding materials for the fabrication of a wide variety of PCB applications.

We deliver consistently high quality products utilizing world class equipment from Japan and Taiwan, in coordination with specifically designed plant layouts and production flows.

.

But it's not all about the products. As well as providing flexible production capacity and competitive cost manufacturing, our team is commited to delivering the the very best service and support to our customers.

To learn more about how we can deliver for you, please visit our website.

ventec

Ventec Europe
www.ventec-europe.com

Ventec USA
www.ventec-usa.com

Ventec International Group
www.venteclaminates.com

Will "Old" PCB Technology Save American Manufacturing?

by Steve Williams

SUMMARY: 3D printing is a hot topic right now, especially with equipment prices dropping and reports of this incredible technology entering the consumer marketplace. With its origin in the additive process of PCB manufacturing, will this technology save American manufacturing?

Full Additive PCB Fabrication

Many of us pre-1980s technology, old-school board guys remember actually running the additive PCB process back in the day, well before the subtractive process became the preferred methodology. The additive process was quite simple and had far fewer process steps than the complicated travelers seen in today's PCB shops. There was neither a need for copper-clad raw laminate, nor current selective electroplating technologies. A negative image was defined on a resin substrate, and metal was selectively added, or built-up, to form the circuitry and interconnects.

Old is New Again

An article in the business section of my local newspaper caught my eye the other day. The piece focused on a Pennsylvania company called ExOne that was in the 3D printing business. My first thought was 3D labels or PoP (point-of-purchase) displays, but what really grabbed my attention was that this company was 3D printing large, stainless steel boat propellers! There has long been talk of utilizing this technology for PCB manufacturing, but it has never seemed to get out of the science-project phase.

If you walk into any sheet metal fabrication, machining, or die cast operation, you will be immersed in the raw environment of the trade: clanging machines, ground-shaking punch press thumping, grinding metal, and weld spark fireworks. You will also be treated to the earthy smell of oil, grease, molten metal, and let's not forget sweat, created by hard, physical labor. Contrast that to the atmosphere in a 3D printing operation, which could easily be compared to a quiet, unrushed office environment.

America as an LCC

What a concept: America as a low-cost country! The biggest driver of low-cost countries is labor cost; and, a 3D operation requires only a few talented employees to run dozens of machines and hundreds of products. The equipment runs virtually on autopilot; a single person can manage a shop full of machines, literally, with the most important task being to keep each machine fully stocked with raw material.

The UK's leading PCB equipment supplier

PRODUCTS

High Speed Moving Probe testers

MICROCRAFT

High Accuracy legend Printers

MICROCRAFT

Affordable Universal Testers

MASON

Ink Jet Peinter

CircaPrint
a printing legend

Economical Drill and Rout Machines

KEJIE

Wet Process Lines

U.C.E.

Laser Processing systems

PROCESS PHOTONICS

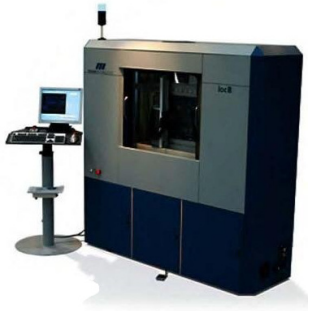
Fully Automatic Laser Plotter

WIDE

SERVICES

The UK Leading bare Board Test Service

Mania Machine Service/Support



Spindle Repair Service

Excellon Athorised Service Provider

Long Barn, Ham Barn Farm, Farnham Road, Liss, Hants GU33LG
Tel: +44 (0)1420 537020
Fax: +44 (0)1420 538950
Web: www.vikingtest.com
email: sales@vikingtest.com

Viking

3D printing is definitely a low-cost business model, needing only several employees and dozens of machines to match the production of traditional manufacturing operations employing hundred, if not thousands, of managers, engineers, operators and staff. 3D printing has been in limited use since the '80s in a variety of industries, but not in any widespread production environment. The demand for this technology is there; according to Terry Wohlers, president of the consulting firm Wohlers Associates, part production made up nearly a quarter of the $1.7 billion in sales of the 3D printing industry, and he is projecting this to rise to 80% of the industry's forecasted $6.9 billion in revenue by 2019.

The Process

The machines are very similar in look and operation to document printers, and run automatically. A single operator can design the program for new parts, load and unload the machines, and occasionally add the raw material powder that feeds the machines. Using a metal part as an example, a computer-aided design, programmed into multiple cross-sections, is entered into the machine's computer telling the printer precisely how to shape the part.

A roller puts down a layer of gray metal powder. A cartridge then quietly glides across the surface, much like an old dot matrix print head, depositing a chemical to bind together certain chemical particles, and forms multiple images of the part. It looks like a rough sketch of the part, but it's actually a 1/4000-inch layer of it. The completed layer drops down and a new gray layer of powder is applied. 3D printers have evolved to make a variety of objects using a laser or extruder that travel along an X, Y and Z path to build-up an object in three dimensions, layer by layer, sometimes only microns thick at a time, which can result in hundreds or thousands of layers to make a finished part. Other 3D printers follow different processes, depending on the material. For example, many plastic parts are formed by squirting successive layers of melted plastic through a precision nozzle.

Figure 1: 3D printed metal gear.

Think of the Possibilities

Self-Sustaining Equipment

The ability to generate replacement parts before they are needed. 3D printing is truly a lean process, since exactly the amount of raw material is being consumed that is required. There is no waste!

Bioprinting

Researchers at Wake Forest University developed this technology for U.S. troops serving in Afghanistan, which uses 3D technology to print new skin directly onto a burn wound. Within an hour, a precise topographical 3D image of a burn victim's wound is scanned, and then the 3D printer prints new layers of cells directly onto the wound using the patient's actual skin as the raw material. As far-fetched as it sounds, this technology could lead to the holy grail of medical science: the production of organs and body parts.

Transportation

What if a large ocean liner, train, or airplane had a 3D printer on board that could make replacement parts on the spot and avoid breakdowns, or more importantly, catastrophes? A vehi-

cle's computers may one day contain a database of 3D CAD images of every part from nuts to bolts to engine parts. If any of these should fail, a 3D printer could be able to manufacture immediate, OEM specification replacement parts on the spot.

Cloud Manufacturing

3D printing would allow the ultimate virtual manufacturing scenario: By investing in a single machine and a small amount of raw material, one could become a manufacturer of a diverse portfolio of products virtually (pun intended) overnight. With very little capital expense, one could receive CAD data electronically in the morning, 3D print almost any form of prototype parts overnight, and ship the order the next day.

Reshoring?

This minimalist factory exemplifies the latest chapter of the industrial revolution, one that could make U.S. manufacturing more competitive globally and bring more jobs back to the United States (reshoring). Will this technology really save American manufacturing? I don't know, but creative and industrious Americans have always been the hallmark of our nation, and I would put our inventiveness and entrepreneurial spirit up against any country in the world. These are the traits that revolutions are built on, and the time may be quickly approaching for a new industrial revolution. **PCB**

Steven Williams is a 35-year veteran in the electronics industry and an authority on manufacturing and management. He is currently the commodity manager for a large global EMS provider, a distinguished faculty member at several universities and author of the book Survival Is Not Mandatory: 10 Things Every CEO Should Know About Lean.

Overcoming Challenges in Laminate Development

by Real Time with...IPC APEX EXPO 2012

John Coonrod, market development engineer for Rogers Corporation, discusses his company's newest development in high-speed, high-frequency laminates. This new material is also halogen-free and compatible with lead-free soldering. John explains the many challenges that had to be met to bring this material to the marketplace.

CLICK TO VIEW

realtimewith.com

VIDEO INTERVIEW

Stevenage Circuits Discusses Flex PCB Trends, Issues and Developments

by Richard Ayes
I-CONNECT007

SUMMARY: A dedicated flex, flex-rigid, rigid and RF PCB/antenna manufacturer catering to the defense, medical implantable, aerospace, security, and communications and sensing markets, Stevenage Circuits Ltd celebrates 40 years in business this year.

Established in 1972 by John Brown and based in Stevenage, UK, Stevenage Circuits Ltd (SCL) is one of the most advanced and best equipped independent PCB manufacturers in Europe. Celebrating its 40th anniversary this year, the company is the former Stevenage Circuits Group, which has another facility—RF products manufacturer Trulon. The two companies merged as part of a major and innovative restructuring plan in the fourth quarter of 2010 to form what is now SCL.

SCL is a dedicated flex, flex-rigid, rigid and RF PCB/antenna manufacturer catering to the defense, medical implantable, aerospace, security, and communications and sensing markets. It touts comprehensive in-process handling capability, which ranges from 25µm to 12 mm thick core materials, and has the largest material and substrate portfolio of any single facility in Europe, including ceramic loaded substrates, crystal polymers, RF materials and specialist clad laminates.

Robert Brown, SCL's managing director, was approximately seven years old when his father, John, started SCL. Brown recalls that he and his three brothers—who are directors at the company, were in the factory making boards on Saturday mornings while their friends were still in bed.

"Dad instilled a strong work ethic in me and by brothers. Today, such a practice wouldn't be allowed; but we and the company, without doubt, benefited from the experience," said Brown. "Our core employee base has been with us for many years; we grew up knowing them and then working alongside their own children and family who joined the company. Having successive generations of families has provided the company with a very quick transfer of knowledge and skill. This knowledge base is essential, particularly as high-end PCB production is an important differentiator from other producers in lower-end markets. Dad had a passion to make SCL one of the most

Robert Brown, Stevenage Circuits' managing director.

John Brown, original founder of Stevenage Circuits.

advanced fabricators in Europe. He would be proud to see that this passion still remains and continues within our company and workforce."

In an interview with I-Connect007, Brown explains how the company's business is shaping up after the restructuring, and the latest trends in the markets they are covering, and how these trends are affecting their flex and flex-rigid PCB manufacturing. The following is an excerpt of that interview.

How was business last year and how is it shaping up this year?

Since our merger and integration of the two manufacturing sites into the existing Stevenage facility, including a major internal reorganisation and capex investment for clean rooms, copper-fill plating lines and further laser imaging equipment, we have been getting back on track since October 2011. This has been qualified by our current SC21 performance figures and highly positive client feedback. We are currently working towards achieving our Bronze Award status this October.

How much does your flexible PCB and rigid-flex PCB segment account for your total PCB production, and what are some of the major drivers for its growth?

In 2011, flex-rigid accounted for about 40% of our turnover, but it is really dependent on which of our clients' programs are dominant within the measured period. We have major programs covering all three segments—rigid, flex-rigid and RF—and more recently, increased enquiries for sculptured flex. We note an increased demand for smaller, lighter products that feature controlled impedance and high-reliability interconnectivity.

Which industries will drive the growth for the flexible PCB industry this year and the next?

The packaging, aerospace and defence, sensors, and the medical electronics industries will drive the flex PCB industry.

What are some of the top trends in the markets you serve, and how do these trends affect your product development and manufacturing?

We have seen three emerging trends: ultrafine features and fewer layers for the packaging, medical and sensing markets; high-layer HDI with copper-filled microvias for the defence, aerospace, and communications sectors; and even more advanced demands for signal integrity and low moisture absorption using thermoplastic flexible materials such as LCP.

Smaller and lighter designs are driving ultrafine features. These types of products necessitate the use of ultrathin copper foils and wet-resist coating technologies in conjunction with laser direct imaging, which brings qualified benefits for both the resolution of fine features and auto-scaling of the image to register the individual panels. The challenges presented when using 25µm substrates on high-frequency and multifunctional materials are considerable and heightened further when allied to embedded component technologies.

SCL has been and continues to be involved with technology development programs for products and technologies requiring ultrafine features. These involve mainly our corporate clients on both EU-funded and private programs, and this is across all sectors.

How has the flex/rigid-flex segment changed over the past three to five years?

There are fewer dedicated flex and flex-rigid PCB manufacturers now within Western Europe, but a greater number of rigid manufacturers are selling a flex-rigid capability—buyers beware.

What key technology issues are you seeing in flex development and manufacturing?

We are seeing a lot of business as a result of other manufacturers' failures. The failure mechanisms are mainly due to poor material handling and processing, and the inability of many fabricators to produce fine features consistently—imaging, plating and etching. Also we see that microvia copper filling is a growing technology that only a few manufactures like SCL have in-house; subcontracting this process appears to be causing many manufacturers major issues around quality control. In recognition of this increased demand, we have just installed a new plating line with an additional 10 copper-fill plating cells that will enable us to triple our current capacity. We are also searching for applications where we can deploy our proven stretchable circuit manufacturing capability.

What are companies doing or what should they do to address these issues?

It is all about continuous investment in equipment and staff training. Flex-rigid PCB production is fundamentally different from rigid PCB processing; unless you have dedicated thin-core handling, jigging and BB test equipment, fabricators will always experience quality issues and clients will continue to be let down. For instance, a single hair or fiber trapped underneath a flex coverlay will scrap circuits—failing the IPC Class 3 release. Therefore, we lay up our flex and flex-rigid multilayers within a Class 10,000 clean room. Also worth mentioning is that the majority of our shop floor staff are trained PCB technicians—and to IPC standards by in-house certified trainers—who, therefore, understand the whole manufacturing process from start to finish.

What about the developments in raw materials such as laminates?

Demand for multifunctional laminates is on the increase for mixed-signal PCB designs—again, this technology also reduces the size and weight of many products. In addition, we have also seen an increase in designs using multiple 25µm substrates within a single construction and LCP multilayer/buried via and microvia technology, which involves high-temperature fusion bonding techniques.

Overall, how do you see flex segment growth this year and the next?

The flex industry has consistently shown the highest growth rate since 2008, so I guess it will continue to do so. We are anticipating an additional 10% growth for the forthcoming 12-month period. **PCB**

The innovative direct-write, digital imaging solution that's the best investment you'll ever make.

APCT, Inc
when reliability and performance are critical

"When we buy a piece of equipment at APCT we do our homework. Because we are in the business of producing advanced technology printed circuit boards in compressed cycle times we have to make sure that every piece of equipment we invest in will enhance that initiative.

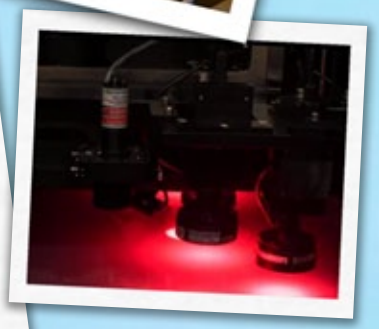
In the case of direct imaging, we spent a great deal of time evaluating a number of systems, ultimately deciding that the Maskless System was the one we were the most interested in. We invested significant resources working with their engineers to make sure that the system would do everything we needed to fortify our leadership position in quick turn manufacturing. These months of diligence have proven invaluable to both APCT and the Maskless engineering group.

After much testing, we launched the Maskless System into production. The quickest turn we previously produced was 72 hours for 18+ layer technology. With the Maskless System we were able to produce 18+ layer product in 48 hours…the cycle time reductions exceeded expectations utilizing the Maskless technology."

Steve Robinson, President of APCT, Inc.

"The Maskless Lithography Imaging line has been a complete game changer for us, the way it can change part numbers on the run and the speed of imaging has made a significant contribution to our cycle time reduction. We made the right choice in choosing Maskless."

~ Eric Schmidt, VP Operations



Come see us in Booth 109 at PCB West Wednesday, Sept. 26th

MADE IN USA

www.maskless.com

Maskless
LITHOGRAPHY
DIRECT IMAGING AT ITS FINEST
2550 Zanker Road • San Jose, CA 95131
Telephone: +1 (408) 433-1861

Most-Read Mil/Aero007 News Highlights This Month

IPC Launches Educational ITAR and Printed Boards Campaign

"IPC's educational campaign seeks to clarify the importance of applying ITAR controls to printed boards given that each printed board is specifically designed for the larger defense item of which it is a part," says IPC CEO John Mitchell. "We view this effort as a pivotal private-sector government partnership on an important policy issue that ultimately bolsters national security."

Electronic Interconnect Now Offers PowerLink PLPCB

Electronic Interconnect (EI) announces the availability of Powerlink PLPCB constructions for electronic assemblies where thermal management/heat dissipation capabilities are desired.

Hughes Circuits Earns AS9100 Revision C Certification

Hughes Circuits, Inc. (HCI) has successfully completed the recertification audit for the AS9100C, Quality Management Systems—Requirements for Aviation, Space, and Defense Organizations. The scope of this recertification includes HCI's PCB layout, fabrication, and assembly services.

Graphic PLC Wins SC21 Bronze Award

Specialist PCB manufacturer Graphic PLC has received the prestigious SC21 Bronze Award at an event hosted by ADS at the Farnborough Air Show July 12, 2012. The Supply Chain 21st Century (SC21) is a nationally recognised change programme designed to accelerate the competitiveness of the aerospace and defence industry by raising supply-chain performance.

Career Renews Mil-P-50884E Qualification

Career Technologies-USA has renewed its Department of Defense Performance Specification MIL-P-50884E, Amendment 3, for type-4 flexible circuits. "We are strongly committed to serving the United States military market. Renewing our qualification to MIL-P-50884 confirms that commitment," said Career Tech-USA's President, Armen Hayrapetian.

Teknoflex Achieves AS9100 Rev C Approval

After more than three years of operation under the AS9100 Rev B standard, Teknoflex achieved the onward transition to AS9100 Rev C in Q1 of 2012. Certification was granted by BSI after Teknoflex successfully passed the stringent audit requirements and a rigorous evaluation of its design, engineering, manufacturing, and assembly processes.

Mass Design Adds Farmer to Business Development Team

David R. Farmer has joined Mass Design, Inc. as a sales and new business development representative. He will focus on the Mass Design's medical, defense, and aerospace markets.

Deloitte: Global Aerospace & Defense Industry Sluggish in 2011

According to a recent report by Deloitte, the financial performance of the global aerospace and defense (A&D) industry generally fell in 2011. Despite the uncertainty in the defense sector, the report found that the global A&D industry as a whole grew in 2011 to $681 billion, posting a sluggish revenue gain of 2.3%, compared to 2.5% in 2010.

Market of Launchers for UAS & Targets to Hit $1.3B by 2018

The 2012 study has 325 pages and 101 tables and figures. Worldwide markets are poised to achieve significant growth as the launchers for unmanned aerial systems provide a way to automate surveillance of wide areas and implement strategic military missions that strike at terrorists without injuring civilians.

Satellite Operators Continue to See Growth

"The Ku-band market will continue to be the main growth engine for the commercial satellite market for the coming ten years," noted Patrick M. French, NSR Senior Analyst and the report's author. "The direct-to-home (DTH) TV market alone could add US $1.4 billion in net new revenues by 2021, out of US $4.3 billion expected in total for the Ku-band segment. Solid Ku-band revenue gains are also expected from the video distribution, enterprise data, commercial mobility and gov/mil verticals."

Test & Measurement

Storage

Telecom

I-Speed™

Military

Infrastructure

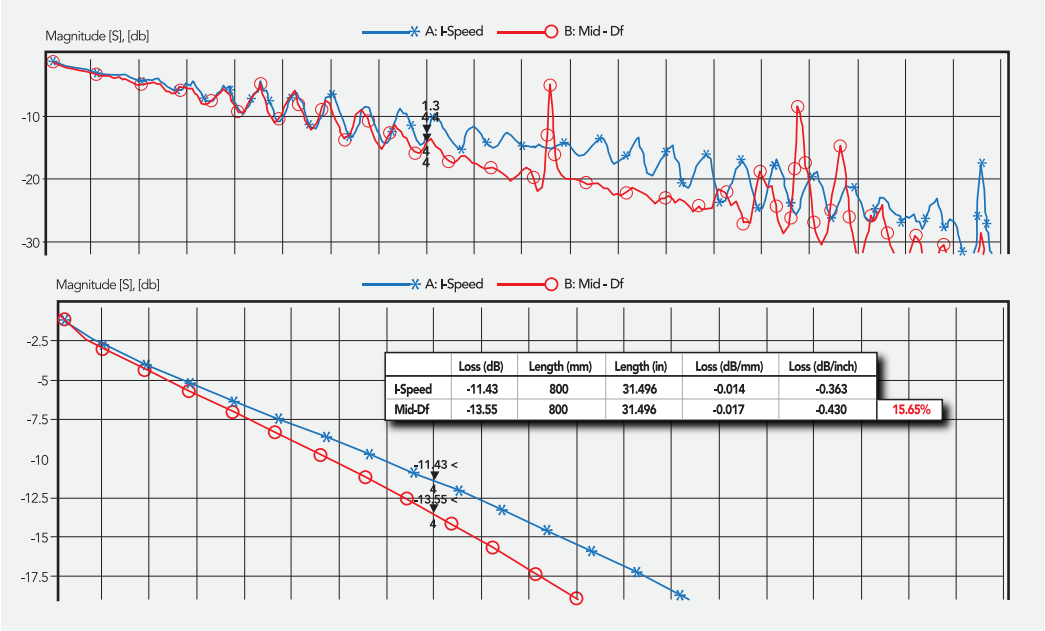
Backplane

Servers

The Next Generation High Speed Product from Isola

- Global Constructions available in all regions
- Optimized Constructions to improve lead free performance
- I-Speed delivers 15-20% lower insertion loss over competitive products through reduced copper roughness and dielectric loss
- Improved Z-axis CTE 2.70%
- I-Speed – IPC 4101 Rev. C /21 /24 /121 /124 /129
- Offer spread and square weave glass styles (1067, 1086, 3313, etc.) for laminates and prepregs
  - Minimizes micro-Dk effects of glass fabrics
  - Enables the glass to absorb resin better and enhances CAF capabilities
  - Improves yields at laser and mechanical drilling
- A low Df product with a low cost of ownership

Effective Loss @ 4 GHz on a 32 inch line



	Loss (dB)	Length (mm)	Length (in)	Loss (dB/mm)	Loss (dB/inch)	
I-Speed	-11.43	800	31.496	-0.014	-0.363	
Mid-Df	-13.55	800	31.496	-0.017	-0.430	15.65%

I-Speed delivers 15-20% lower insertion loss over competitive low Df products.

The data, while believed to be accurate and based on analytical methods considered to be reliable, is for information purposes only. Any sales of these materials will be governed by the terms and conditions of the agreement under which they are sold.

Isola Corp., 3100 West Ray Rd., Suite 301, Chandler, AZ 85226, +1-480-893-6527, www.isola-group.com

isola 100 YEAR ANNIVERSARY 1912-2012

isola

Supplying the base for innovation

Interconnect Defect (ICD): Getting to the Root Cause

by Mike Carano
OMG ELECTRONIC CHEMICALS

SUMMARY: Interconnect defect (ICD), a.k.a. innerplane separation, is one of the most difficult technical problems to troubleshoot in PCB fabrication. This month, Trouble in Your Tank presents an actual case history and analysis of the problem.

Introduction

Circuit board fabricators and end-users are always concerned about innerplane separation, or innerconnect defect, which is a separation of the electroless copper deposit from the innerlayer copper face. Of course, there are other types of separation that can be encountered. It is well known that the electrolytic copper may separate from the electroless copper, but the electroless copper remains on the interconnect, referred to as type 2 ICD. Type 1 references the electroless copper separating from the innerconnect post or innerplane. There is a third type in which the electroless copper deposit actually separates from itself. This is more like a cohesive failure within the deposit. Type 3 is very difficult to detect and requires excellent microsectioning and polishing techniques to properly detect and trouble shoot.

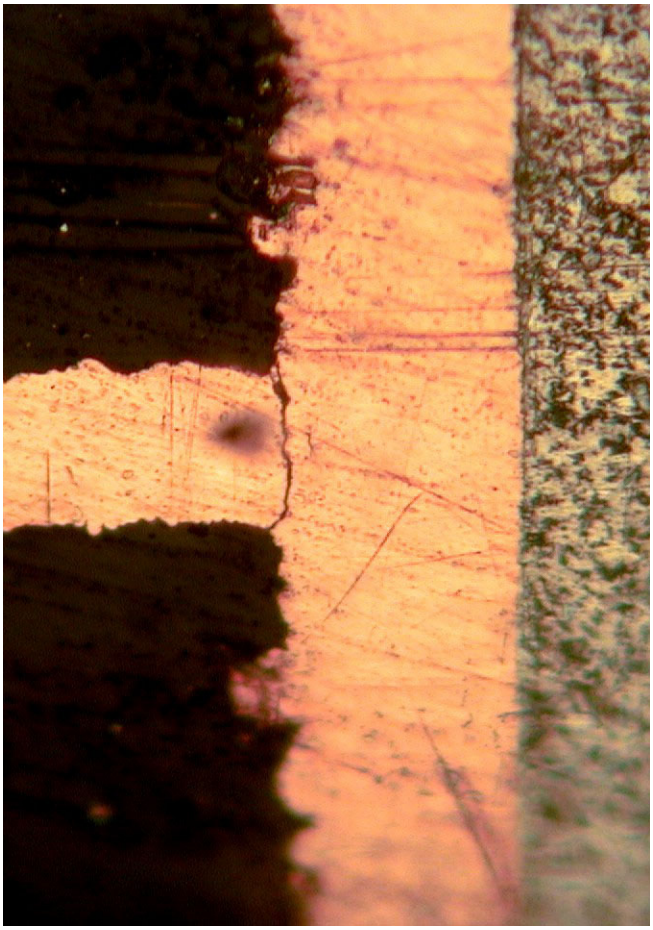


Figure 1: Typical ICD or innerplane separation.

Shown in Figure 1 is an example of an ICD.

Certainly, an ICD is not confined to just the interplane. Blind vias can also exhibit a separation of the plated copper from the capture pad. This will be investigated in a future column of Trouble in Your Tank.

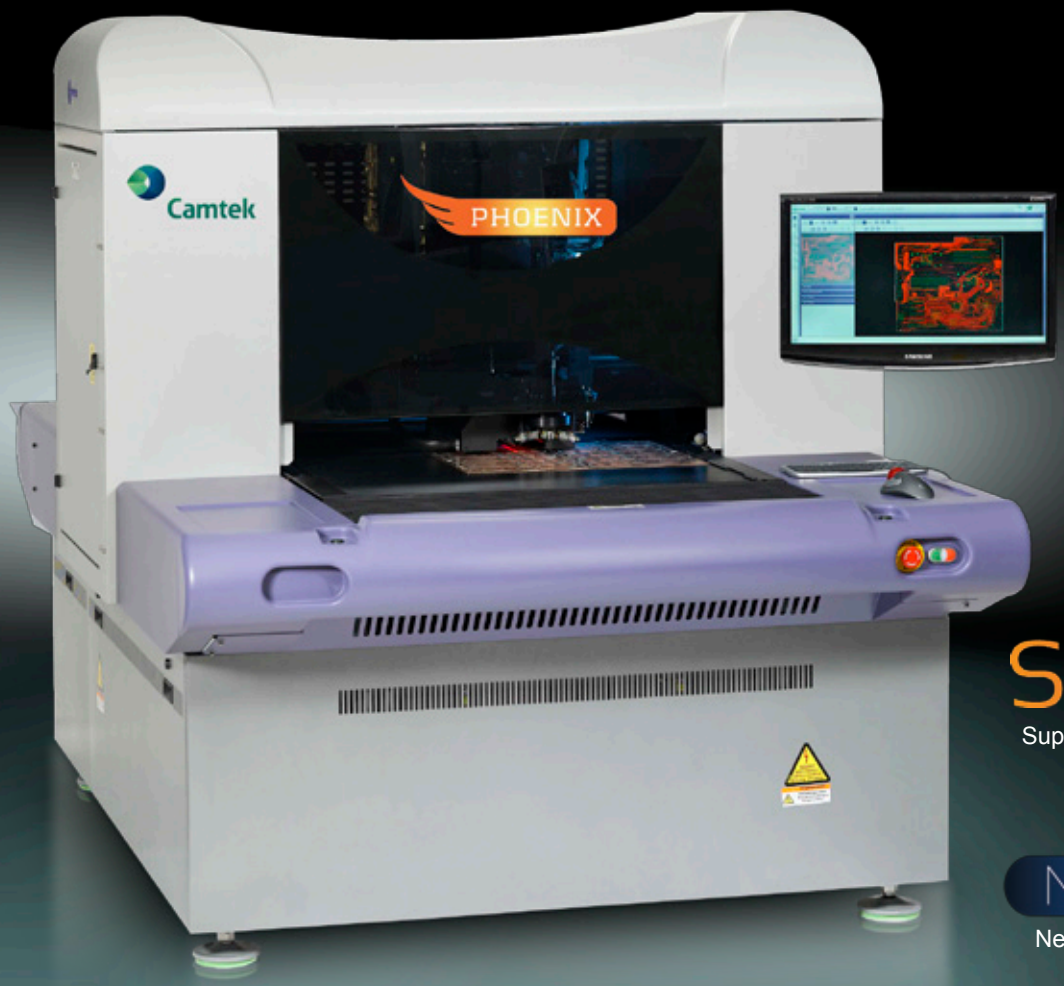
First, what can one expect from this type of defect? What does it look like under the microscope?

Well, one can expect that innerconnect defect (IP separation or ICD) generally exhibits these underlying characteristics: [1]

1. Occurs during the stress of thermal shocks such as soldering. This produces expansion in the Z-axis and places strain on the copper in the hole wall.
2. Most often seen on the first innerlayer from either side of the board. (i.e., in a 12-layer board, the defect is seen on layers 2 and 11)
3. The weakest bond breaks first.
4. Under-cured boards are most susceptible.
5. Affected by board construction, PWBs with more resin, less copper, lower T_g materials, tend to show more expansion in the Z-axis.
6. ICDs are often accompanied by hole wall pull away, but not always.

PHOENIX

The sharpest, quickest, smartest AOI ever made



Powered by
SPARK
Superior detection engine

Illuminated by
Microlight
New optical technology

A family of products designed to increase your yield:

- **Phoenix FL** - for high-end IC Substrate products of down to 10 µm line/space
- **Phoenix R2R** - for the reel-to-reel production process
- **Phoenix Flex** - for flexible PCBs of all types

Camtek
See Beyond

Visit us at **TPCA**
October 24-26
Booth number: I616

Visit us at **HKPCA**
November 28-30
Booth number: 1J39

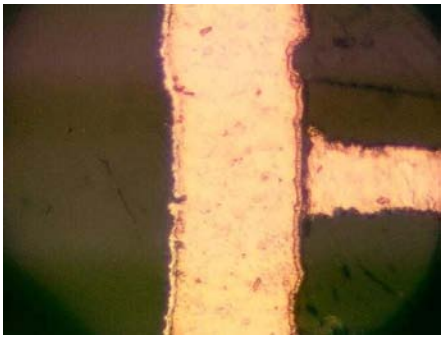


Figure 2: ICD type 1—electroless copper separates from the post, confirmed with etch.

Figure 3: ICD type 2—electrolytic copper separates from the electroless copper.

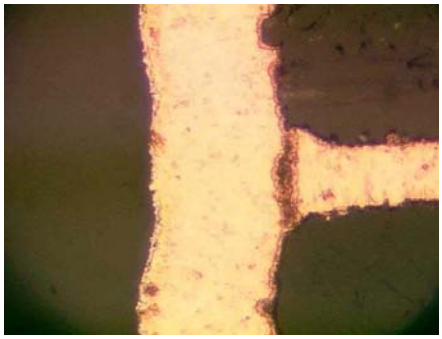


Figure 4: ICD type 3—cohesive failure of the electroless copper deposit.

Understand that innerplane separation has many causes. This is what makes this defect so difficult to pinpoint and eliminate. And to confuse the issue further, one must first determine which of the three types of IP separation it is.

Now, before we dive into this case study where you can test your troubleshooting diagnostic skills, we need to get a close look, in general terms, at what type 2 and type 3 ICD look like.

Case Study

So it begins. The fabricator believes there are some issues with ICDs but he is not certain. No other diagnostic tests were performed as the boards in question (other than having solder float coupons) did not have any IST coupons that could be tested.

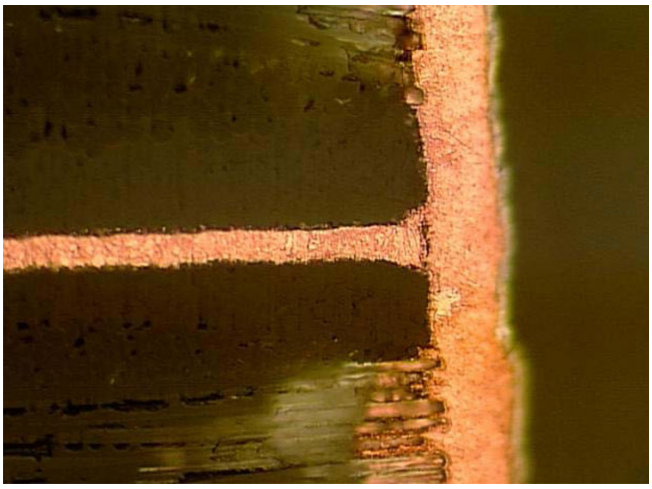


Figure 5: Note not only the ICD, but the torn-out glass fiber bundles as well.

The boards in question were approximately 230-mil thick panel, 20-mil holes, ½ oz innerlayers.

An example of the microsection is shown below in Figure 5.

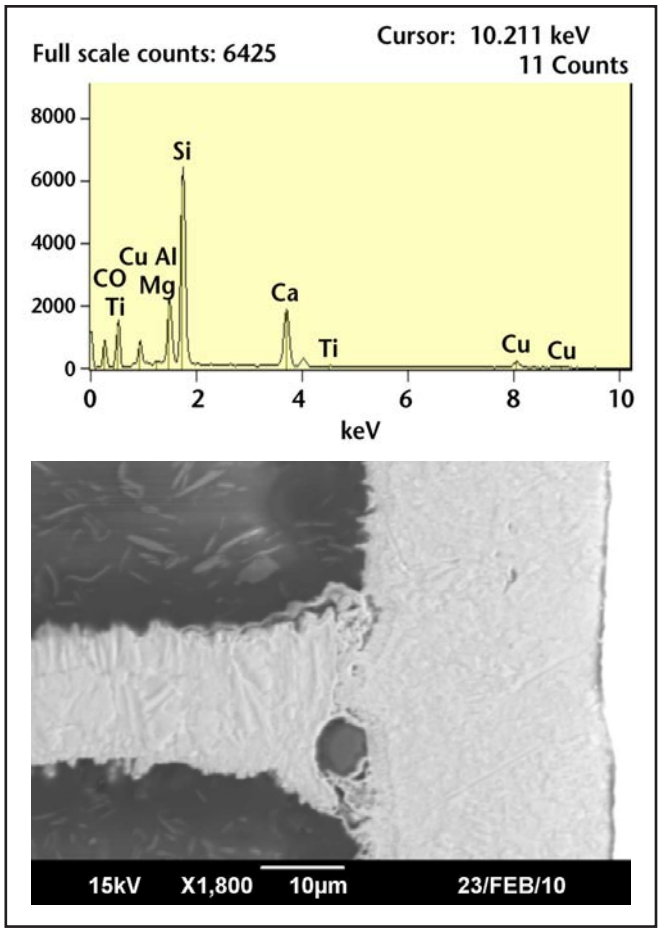


Figure 6: Actual section (bottom) and EDAX scan (top).

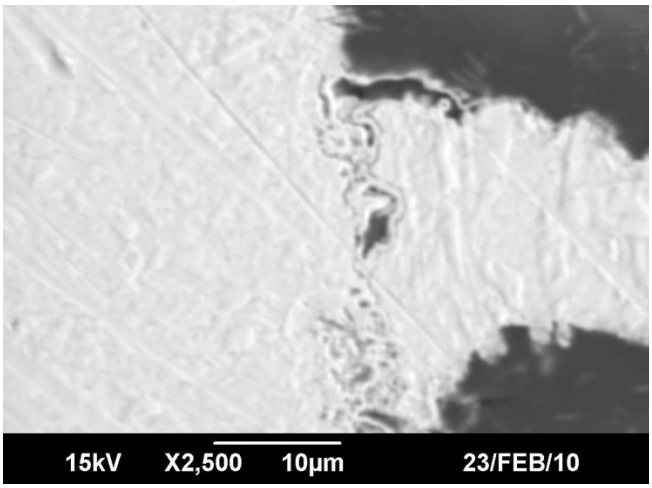


Figure 7: Another look at the separation-EDAX scan again shows a high level of silicon and aluminum in the defect area.

What does the troubleshooter conclude from this, other than an innerplane separation?

Well, to begin with, one needs to perform additional analysis. The glass bundle issue raises some concern. Let's look further.

The analysis focused on the gap or open area in the section above. The EDAX analysis gives a higher than normal level of silicon and aluminum. This is drill debris that has been embedded into the epoxy resin. Again, the clue was the glass fiber bundle issue noted on the cross section.

In Figure 7, there is evidence of nail heading. Again, this is further evidence that the defect in this case is related to the drilling operation.

Finally one more piece of evidence to consider. Look at the cross sections below:

Since the drilling operation generates significant heat as it is, it is not uncommon for drill debris to become embedded in the resin.

Summary

In this case, the cause of the ICD was clearly unrelated to the plating or desmear operation. Drilling problems as shown by the evidence here lead to the team optimizing the drilling parameters, including feeds and speeds, SFM, and being more vigilant with respect to drill-bit quality. **PCB**

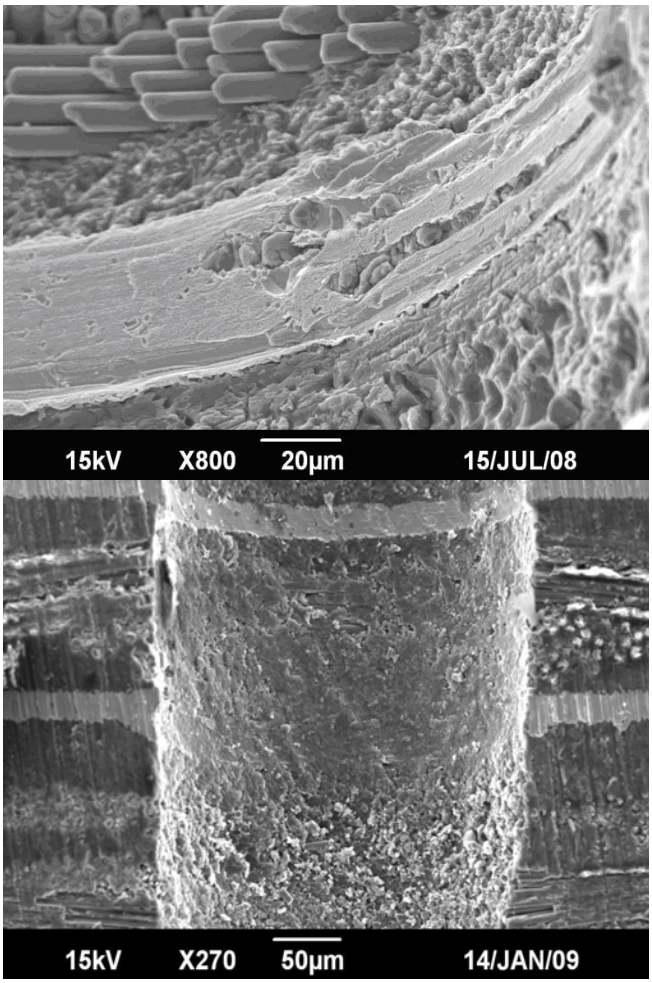


Figure 8: Top shows severe plowing of the copper innerlayer (note copper folded over on itself) and bottom shows embedded glass fibers and drill debris.

References:

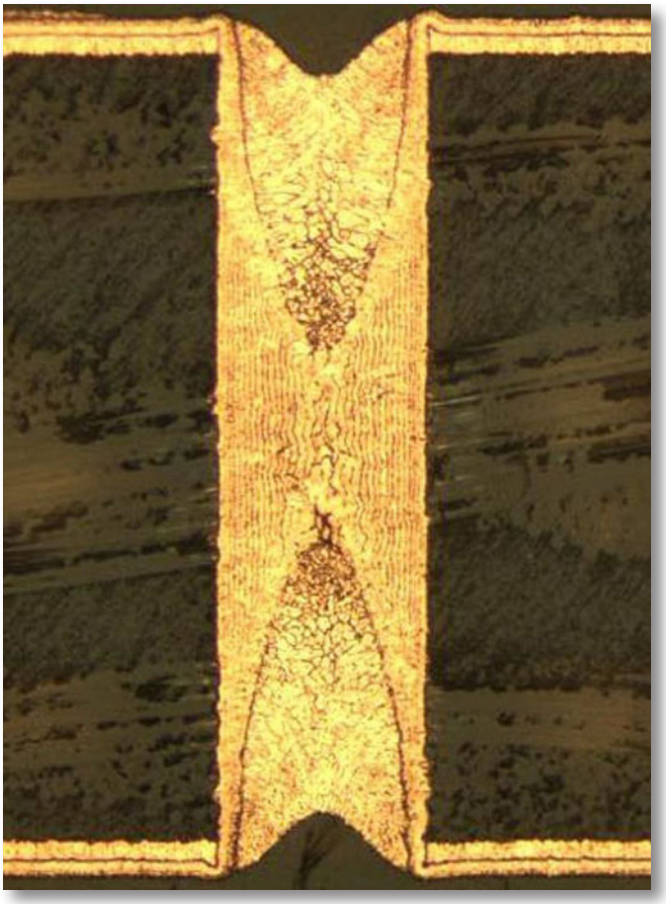
1. IPC PE-740 Process Effects Guide.

Michael Carano is with OMG Electronic Chemicals (formerly Electrochemicals), a developer and provider of processes and materials for the electronics industry supply chain. He has been involved in the PWB, general metal finishing photovoltaic industries for nearly 30 years. Carano holds nine U.S. patents in topics including plating, metallization processes and PWB fabrication techniques.

New Challenges for Higher Aspect Ratio:

Filling Through-holes and Blind Microvias with Copper by Reverse Pulse Plating

by Mike Palazzola, ATOTECH USA;
Nina Dambrowsky, Christof Erben, Stephen Kenny, and Bernd Roelfs,
ATOTECH DEUTSCHLAND GMBH



EDITOR'S NOTE: This article was originally published in the Proceedings of IPC APEX EXPO, San Diego, California, February 28-March 1, 2012.

Abstract

This paper presents systematic investigations on complete through-hole filling for cores by a Cu electroplating process as an alternative to the common paste-plugging process. This electroplating process consists of two steps, a first process to merge both centers of the through-hole walls (X-plating) followed by filling up the resulting blind microvias. Processes and manufacturing technology are described as well as current limitations and requirements. Complete filling of through-holes is achieved by reverse pulse plating (RPP). This through-hole filling technology is targeting both at HDI production and also at the packaging level.

Through-hole filling by RPP offers a viable alternative to the standard paste plugging for core processing in substrate manufacturing. Current core manufacturing requires a paste plugging process for through-holes so that subsequent build-up layers can be produced by sequential lamination, and the flat core surface is essential for stacked via and also via-in-pad technology.

This paste plugging process requires additional process steps, each of which brings its own limitations and contributes to the overall cost. Filling the core through-vias by electroplating can eliminate the plugging process and significantly reduces the number of overall process steps which will also reduce costs. Moreover, it offers certain advantages such as potentially higher reliability in accelerated aging tests and an improved thermal management as the thermal conductivity of a completely copper-filled through-via is significantly higher than a paste-plugged through-via.

Today's challenges in the so-called through-hole filling process are represented by through-holes with a high aspect ratio. Voids after X-plating occur easily for smaller through-hole diameter and higher board thickness. In addition, depending on designs, different pitches on one board increase the difficulty to achieve an acceptable plating uniformity. This paper presents systemic variations of some key parameters and concentrates on the performances of the second plating step, the blind microvias filling. The main focus is laid on recess distribution. Parameters as reverse pulse parameters, inorganic concentrations (Cu, Fe, and sulfuric acid), organic concentrations, electrolyte flow and temperature have been systematically varied and their influence on the filling performance are described.

Introduction

Through-hole filling by RPP offers a viable alternative to the standard paste plugging for core processing in substrate manufacturing. Current core manufacturing requires a paste plugging after Cu galvanization for through-holes so that subsequent build-up layers can be produced by sequential lamination, and the flat core surface is essential for stacked via and also via-in-pad technology.

Instead of plating only a conformal Cu barrel into the through-via, it is now possible to completely fill it with Cu and thus eliminate the need for the plugging process, reducing significantly the number of overall process steps and costs. Moreover, it offers certain advantages such as potentially higher reliability in accelerated aging tests and an improved thermal management as the thermal conductivity of a completely copper-filled through-via is significantly higher than a paste plugged through-via.

Today's challenges in the so-called through-hole filling process are represented by through-holes with a high aspect ratio. Voids after X-plating occur easily for smaller through-hole diameter and higher board thickness. In addition, depending on designs, different pitches on one board increase the difficulty to achieve an acceptable plating uniformity. To understand the limitations of the process we are aiming at an identification of the key parameters. As there are two steps in the through-via filling process, we divided our experimental work accordingly. The first section presents an overview over the process itself (Figure 1), the second section concentrates on the formation of the X (merging through-via walls) and the third section deals with electrochemical filling of the resulting blind vias. Most of the tests have been conducted in a single board cell (volume 400 l), i.e., each plating cell is handling exactly one 18" x 24" sample board. Different test panels have been used and are described within the experiment itself. All panels have been electroplated in the so-called panel plating mode, i.e., there were no dry film patterns on the board during plating. This panel plate technology is being used in standard production processes by applying the resist and the conductor pattern in a subsequent step.

General Description of Through-Via Filling

Table 1 describes the process flow of the electrochemical through-via filling technology.

Electrochemical Cu Through-via Filling
1. Drilling
2. Seed Layer Deposition (E'less Cu 0.2-0.5 µm)
3. Galvanic Reinforcement (2-5 µm Cu)
4. Galvanic Cu Filling -X-Plating (5-10 µm) -Via filling (10-15 µm)
5. Dry Film Application and Subsequent Structuring

Table 1: Process steps characterizing the EC Cu through-via fill technology.

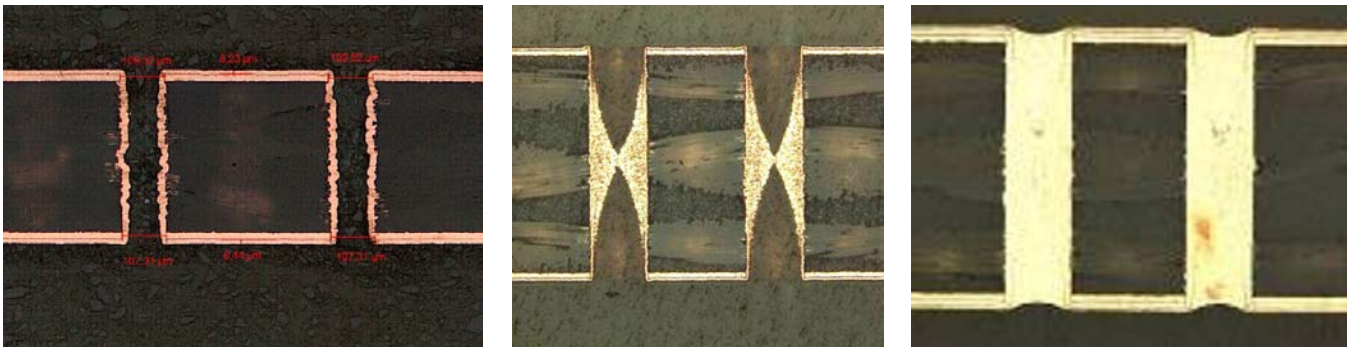


Figure 1: Plating steps to a complete fill of a through-via with Cu, starting from a conformal plated layer. The second plating step is the formation of the X, resulting in two blind vias on both sides, which are then completely filled.

The EC plating process described in Figure 1 shows the two main plating processes of step 4 above.

As an alternative electrochemical route, one could consider a slow, conformal growth of copper on the side walls to a complete fill. That process has several disadvantages such as being slow (less productivity) and having a higher risk for voids (see Figure 2).

Moreover, this conformal technique would require too much Cu on the surface (approximately ½ hole diameter), whereas the above mentioned X-formation offers the possibility to significantly reduce the Cu overburden. That is achieved by applying a combination of RPP and etching in one electrolyte. The electrolyte contains not only Cu, but also Fe2+ and Fe3+. Cu is deposited as usual on the cathode (panel) whereas the Fe2+ ions are used to carry the current at the dimensionally stable anodes (DSA) and prevent oxidation of organic additives. The oxidation product Fe3+ is being used to dissolve the consumed copper chemically in a second compartment filled with pure Cu balls/clippings and reduced again to Fe2+. But the Fe3+ cannot be used only to replenish Cu, but also to lower the amount of the Cu deposition on the panel surface.

$$Cu^{2+} + 2e^- \longrightarrow Cu \qquad E = 0{,}340 \text{ mV vs. SHE} \qquad \text{(I)}$$

$$2\,Fe^{3+} + 2e^- \longrightarrow 2\,Fe^{2+} \qquad E = 0{,}771 \text{ mV} \qquad \text{(II)}$$

Reaction (II) is electrochemically favored against (I) and thus the amount of Fe3+ reduces and determines the amount of deposited Cu. A control unit between plating and dissolving tank regulates the exchange between both compartments and can control the amount of Fe3+ within ±0.1 g/l, which is a prerequisite for a good process control as can be seen later in this report. The exchange of electrolyte is much stronger on the surface than in a via, so that this effect is mainly observed on a surface and not in vias—especially not in a blind vias.

With this technology we were able to reduce the amount of Cu overburden for a 100 µm thick core from 50 µm down to 13 µm (Figure 3).

This reduction was a prerequisite for a production scenario since the Cu thickness determines in the end the achievable lines and spaces in the circuitization process of approximately 2 x Cu overburden: i.e., 13 µm —> 26 µm l/s.

Nevertheless, the main core thickness for IC substrates is >100 µm. Most of today's products show cores of 400 µm thickness or more. These cores are extremely difficult to fill without void

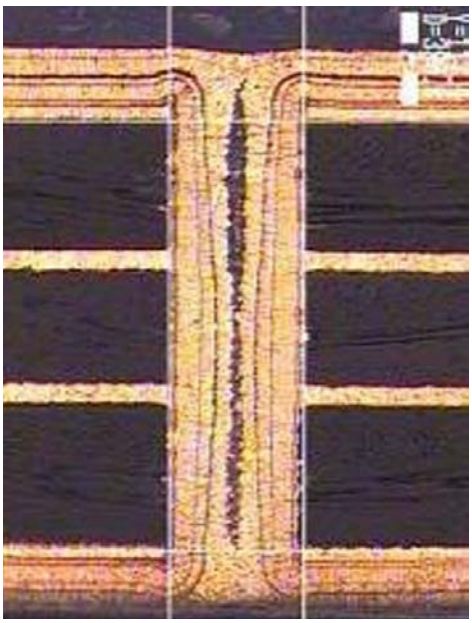


Figure 2: Conformal Cu growth to a complete fill may end up producing a long, ribbon-like void.

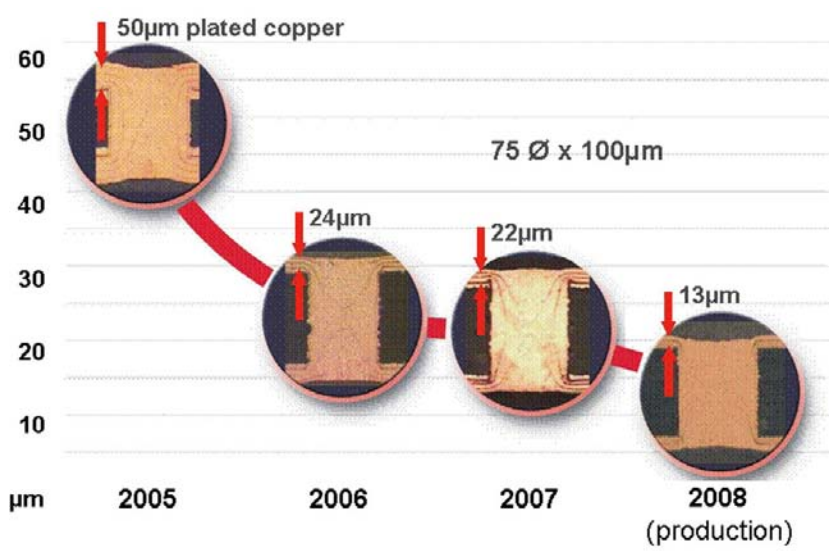
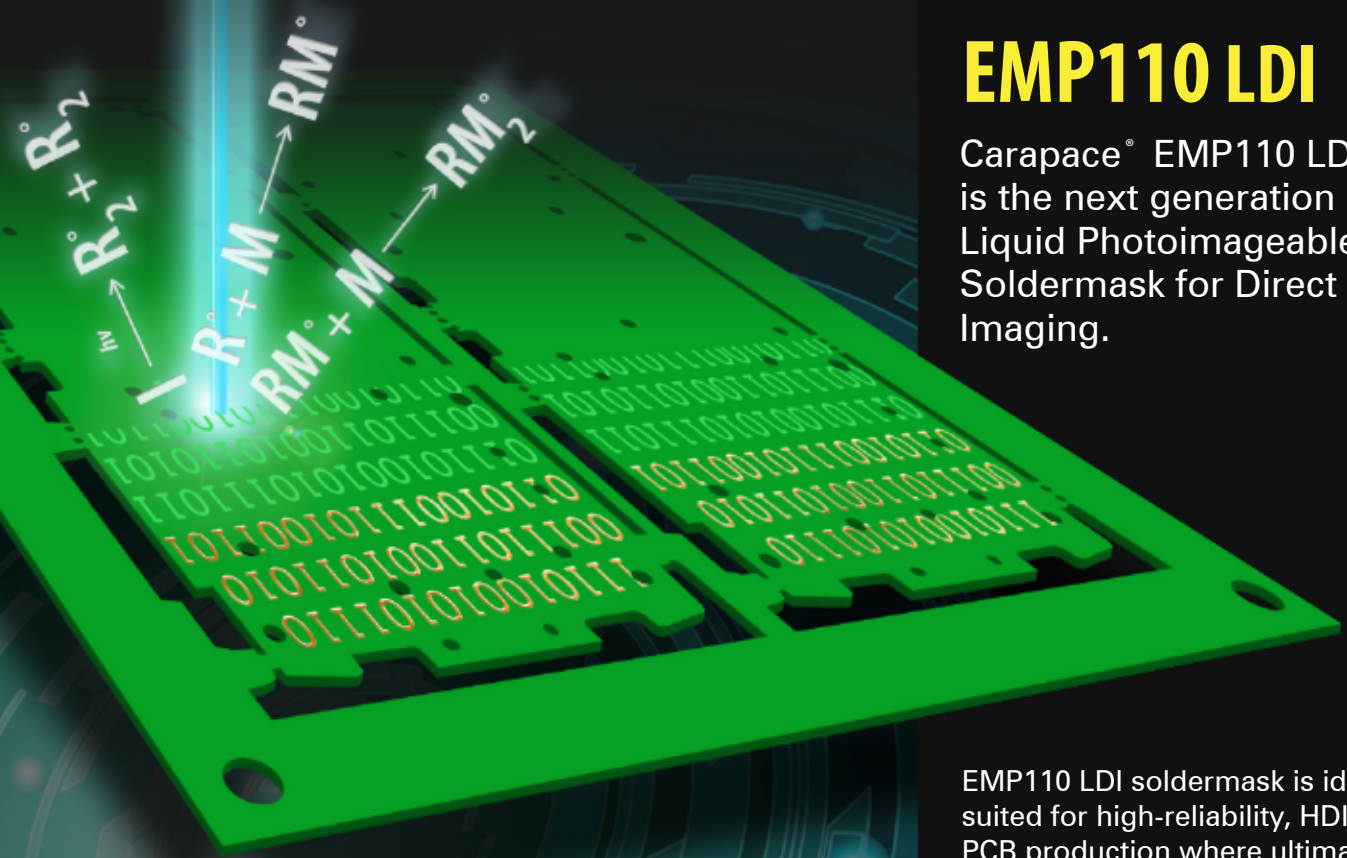


Figure 3: Development of through-via filling technology: reduction of Cu overburden for a 100 µm thick core.

Direct Image Solder Mask



EMP110 LDI

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

EMP110 LDI soldermask is ideally suited for high-reliability, HDI PCB production where ultimate resolution and registration is required.

The EMP110 LDI formulation has been engineered to deliver straight sidewalls and fine solder-dam resolution over the wide range of coating thicknesses associated with screen-printed and sprayed PCBs.

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

- **Low Exposure Energy (50-80 mJcm^{-2}) to Resolve Small Features**
- **Tack-free Surface During Exposure**
- **Fine Solder-dam Resolution (50µm, 2mil)**
- **Lead-free Compatible**
- **High Chemical Resistance to ENIG & Sn Processes**
- **RoHS Compliant and Halogen Free**
- **Non Toxic**

ELECTRA

www.electrapolymers.com

formation and require a long plating time. So it was the aim of this work to expand the technology to such cores by determining main parameters and improve each step of the process.

Experimental

We are using several RPP parameters in the following chapters and would like to explain notation.

Example: 5/40 80/2 —>
5 = Forward or cathodic current density (ASD)
40 = Reverse or anodic current density (ASD)
80 = Cathodic plating time (ms)
2 = Cathodic plating time (ms)
So 5/40 80/2 is an 80ms cathodic pulse with 5 ASD followed by a 2ms anodic pulse of 40 ASD

X- Plating

As there are a huge number of parameters, we started a first series of experiment in a small lab tool to determine which of the following parameters might have a significant influence. We used a small 20 L test cell, 400 µm thick core panels with 100 µm hole through-holes and varied in several small DOEs of three parameters. We found that the parameters in italic showed a significant influence determined by statistical software tools and set up further experiments in a production tool on 400 µm panels with through-hole of 100 µm diameter.

- Current density and pulse parameters
- Fe^{3+} concentration
- Temperature
- Cu concentration
- Cl- concentration
- Electrolyte agitation
- Fe^{2+} concentration
- Sulfuric acid concentration
- Brightener (accelerator) and leveller (inhibitor) concentration (commercially available inpulse electrolytes)

The nature and type of additive has a big influence on the plating performance, but in this specific case, chosen ratio of brightener and leveller does not have a significant influence. We also investigated the influence of the panel design on the X-formation, as it turns out that isolated and clustered through-holes (small pitch, many neighboring through-holes) may show different results as depicted in Figure 4.

Parameters for this experiment at 40°C were:

- Cu: 25 g/l
- H2SO4: 200 g/l
- Fe2+: 15.0 g/l
- Fe3+: 0.5 - 1.0 g/l
- Cl^-: 100 mg/l
- RPP: 1.5/40 – 80/4

Commercially available additives were used according to data sheet. The differently located through-holes required different plating times to form the bridge in the middle. So the minimum time to form the X is always determined by the clustered through-holes.

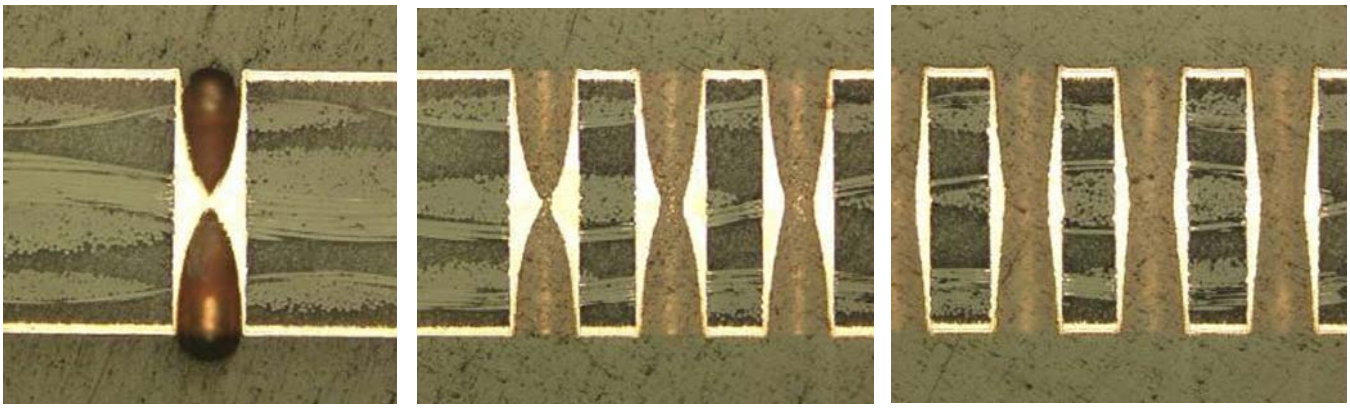


Figure 4: Difference between isolated and clustered through-holes. Left: isolated, right-clustered TH with 200 µm pitch, middle through-holes at the edge of such a cluster.

Electrolyte Flow

The difference between these clustered and isolated THs could be minimized by increasing the electrolyte flow onto the panel. The flow direction was 90°C (rectangular) to the surface. Doubling the flow rate resulted in a homogenous X-formation with little difference. The effect of Fe3+ is even stronger as can be seen from Figure 5. Here, we varied the Fe3+ concentration only from 0.5 g/l up to 1.0 g/l. The other electrolytic parameters were, Cu: 50 g/l sulfuric acid. The result is that with a higher ferric concentration a much slower X- formation is observed. Both experiments have been conducted under identical parameters. The consequence of this result is that the control of ferric ions is extremely important for handling the process. Therefore, we installed the control unit between the plating and the dissolving tank so that the ferric ion concentration can be controlled within ± 0.1 g/l.

The effect of Cu concentration on the X-formation is different to the usual blind via filling: In contrast to the latter process a lower Cu concentration turned out to be beneficial. The influence is significant both for the X-formation and for the distribution of the X between the isolated and clustered TH. An additional test run with 18 and 32 g/l Cu on very thick panels (3.2 mm with 0.3 mm hole diameter) confirmed the result. With these panels it is not possible yet to achieve through-via filling but we did not aim at a complete X-formation since differences between experiments can better be evaluated before the X is finally formed. We stopped plating after a certain time period (55 min.) and measured the thickness in the middle of the hole compared to the surface. With the 18 g/l we increased the Cu thickness in the middle of the hole by a factor of 1.9.

The same effect had the increase of temperature from 20 to 40°C.

Summarizing our findings for the optimization of X-formation results in following proposals:

- Use strong reverse pulse. A 5/50 80/4 for 15 min leads to closure of the X-formation on 400 µm cores with 100 µm diameter
- Use lower Cu concentrations of 25 g/l for regular distribution
- Use higher Cl- concentration 90 instead of 60 mg/l
- Use a strong electrolyte flow to minimize differences between clustered and isolated TH
- Use low ferric concentrations (0.5 g/l)
- Above mentioned parameters were used to plate a 400 µm/100 µm core as depicted in Figure 6. Additional parameters were 240 g/l sulfuric acid and 5 g/l Fe2+ added as ferrous sulfate

Figure 6: 400 µm thick core with 100 µm TH plated within 15 minutes. Right detail showing 9 µm plated Cu on the surface while >50 µm Cu plated in the middle of the via.

Figure 5: Effect of ferric ions on the X-formation. Top 1.0 g/l and bottom 0.5 g/l ferric, left isolated and right clustered through-vias.

Blind Via Filling

Most of the following experiments have been performed with boards having three different blind microvias with a depth of 40 µm and width of 75, 100, and 125 µm. These sample boards have been chosen because of their well-defined and reproducible via structures. Information gathered on these boards was then applied on through-vias to achieve the best possible result. All tests have been conducted with an inpulse electrolyte system (Cu/Fe) with the same commercially available additive system.

The following parameters have been investigated:

- Reverse and forward pulse time and current
- Temperature
- Electrolyte flow
- Fe3+ concentration

The Cu concentration has not been investigated in this series due to the fact that its effect is already well known: A high Cu concentration is beneficial for via filling.

Reverse and forward current: The 5/40 80/2 parameter set has been chosen as a start point. The electrolyte was set up with 50 g/l Cu, 150 g/l sulfuric acid, 63 mg/l HCl with inpulse brightener (accelerator) and leveller (inhibitor) and plated at 45°C

Increasing the reverse time has a significant effect on the filling performance, as it changes from a deposit with a recess of a few µm into one with a convex dome on top of the via. As expected, the smaller vias are easier to fill than the larger ones. Shortening the cathodic pulse while keeping the anodic pulse constant, leads to a similar result. This effect is in line with the explanation that via filling is mainly achieved by a changing ratio of accelerating and inhibiting additives in a via and surface. As described elsewhere [3], the Cu accelerating agents are depleted during a reverse cycle mainly on the surface, but not in a via. There, a net accumulation, and thus, acceleration is observed. Net accumulation of accelerator in a via compared to the surface can be achieved either by intensifying the depletion rate from the surface (stronger anodic pulse) or by increasing the frequency of the anodic pulse.

Best results are achieved with lower forward current densities as it is expected. From via filling experiments in pure DC mode it is well known that lower current density usually leads to a better filling performance. This effect is also seen under reverse pulse conditions.

Electrolyte Agitation

The influence of electrolyte agitation on the filling performance has been investigated by using frequency controlled pumps that feed the spray bars. The electrolyte is pumped through nozzles directly onto the board (90° angle). We used the same electrolyte set up as before. The frequency is then changed during the plating procedure, as it turned out that this procedure gave the best performance. At first, a thin protecting Cu layer with moderate pulse parameters is deposited and then followed by the modification of the electrolyte agitation. The higher the pump frequency, the higher the agitation rate. At the end of the cycle again a moderate pulse layer is plated to achieve a smooth surface.

In general, one can observe with lower agitation rates a better via filling performance. With increasing pump frequency, significant dimples are observed so that the best results are achieved with low agitation rates.

Temperature Effects

Since the dimple performance is depending on the adsorption/desorption ratio between accelerator and inhibitor, one would expect a certain influence of the temperature. Therefore, we varied the temperature over a range from 25-50°C under the same electrolyte setup as before and used the best pump frequency program. As it turns out, there is an optimum temperature of about 35°C, but the influence of this parameter is not as strong as both aforementioned ones (pulse and agitation).

The panel surface is also affected. The surface becomes rough below 35°C and very smooth above 40°C, so that depending on requirements a compromise has to be found between good via filling and surface appearance.

The Fe3+ influence: It can be expected from equation (I) and (II) that the overall yield for the

PRiSM

BY TAIYO

LIQUID PHOTOIMAGEABLE SOLDER MASK

PSR-4000® PRSM™

TAIYO AMERICA™

Your Ultimate Solution

Phone [775] 885-9959 • Fax [775] 885-9972 • www.taiyo-america.com

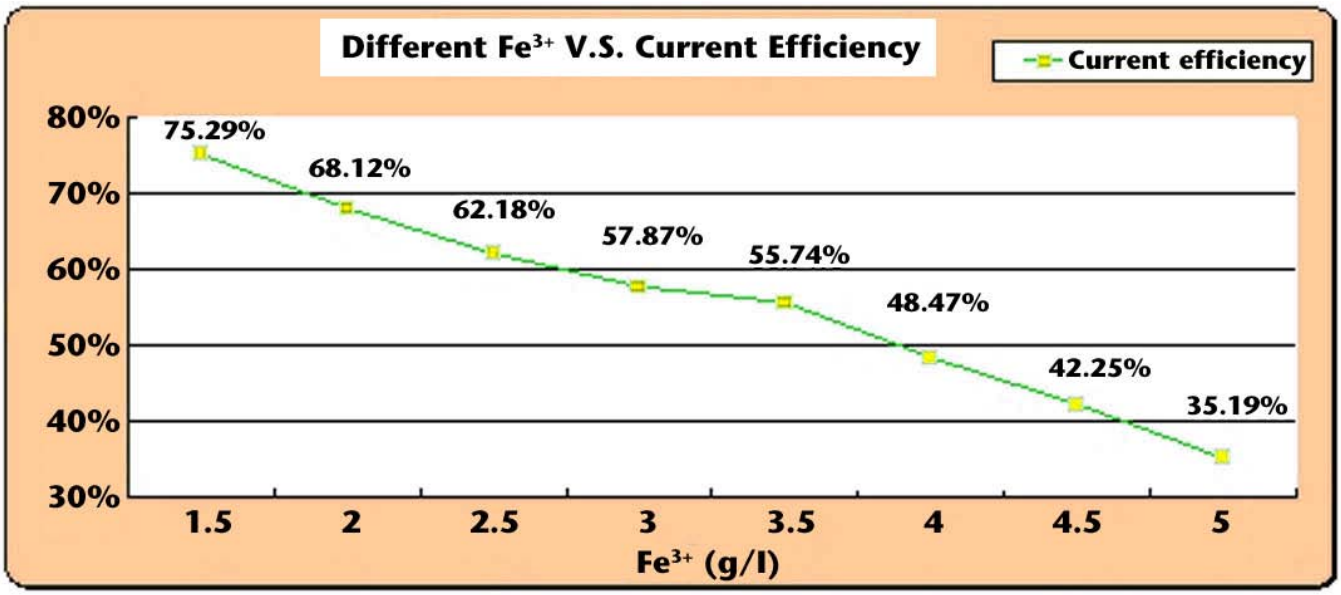


Figure 7: Yield (Cu) as a function of Fe3+ concentration.

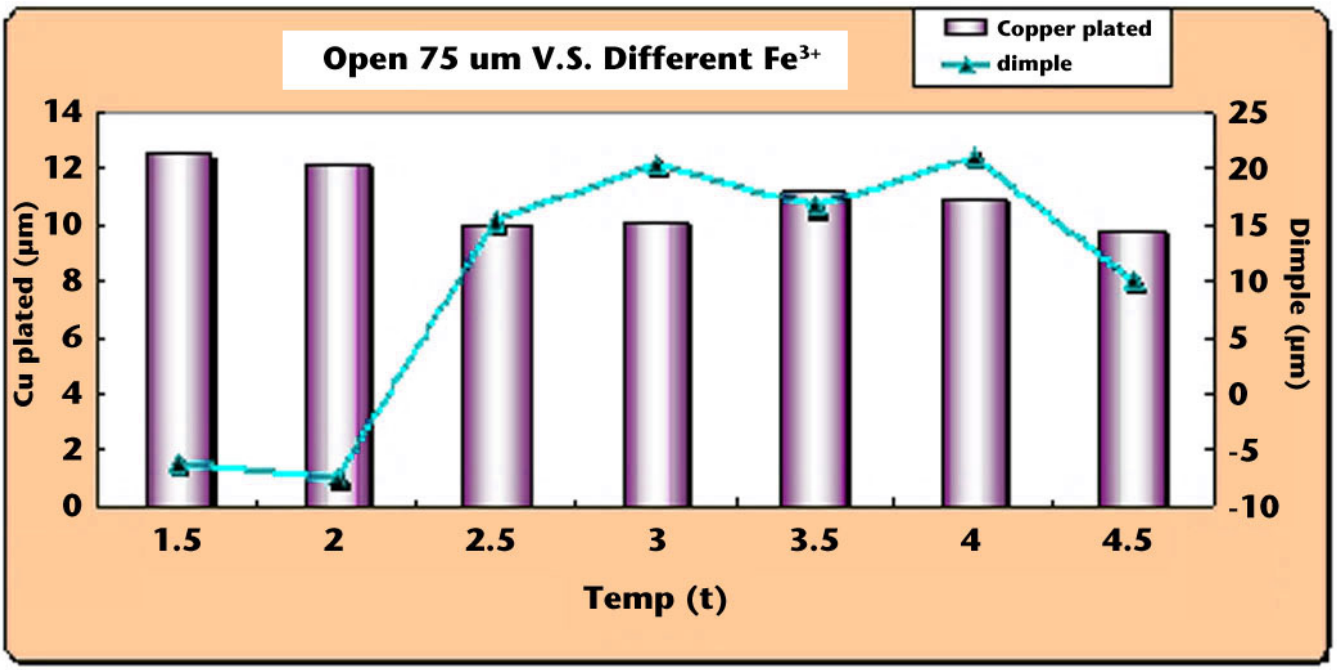


Figure 8: Plated Cu thickness (columns) and dimple (red line) as a function of Fe3+ concentration for 75 µm x 40 µm blind microvias.

copper deposition is strongly affected by Fe3+. Figure 7 depicts this very strong dependency. This current yield is measured by weight difference against the theoretically possible Cu mass of 100%. A lower current yield than 100% usually needs to be counterbalanced by additional plating time. This is not necessarily the case for the via filling application since the amount of plated Cu on the surface is not equivalent to the amount in a via. The relevant parameter is the dimple, not the amount of plated Cu on the surface. Hence, the amount of plated Cu can vary with the Fe3+ concentration to achieve the best performance. The experiments shown

Prototron Circuits

Quality printed circuit boards

Here's what one of our customers had to say about Prototron...

Monsoon Solutions Inc.

As a design group, Monsoon is often tasked with the responsibility of building initial prototypes and other small volume builds. While I have worked with many different shops across the nation, I've found Prototron to be the most consistent when it comes to shipping on time. Whenever schedule is important (and for us it almost always is) I really can count on Prototron to deliver...and every once in a while, even early! I've been sending boards to Prototron for over 10 years (literally hundreds of boards) and I am yet to see any quality issues at Prototron. They really are one of the best shops out there.

Paul Butler, CFO

Monsoon Solutions, Inc.

Bellevue, WA

...and of course the customer is always right!

in the graph 13 were conducted with aforementioned electrolyte, pulse and pump parameters at 45 °C.

As it turns out, dimples are lowest for lower ferric concentrations. Contrary to this is again the surface appearance of the boards. The surface becomes smoother with increasing ferric concentrations. Again a compromise between dimple performance and surface appearance has to be found.

Summarizing the optimization experiments to fill blind vias leads to the following results:

- Apply CD of about 3-4 ASD cathodic and apply stronger reverse parameters to decrease the dimple (on cost of surface roughness): best compromise 3-4/40 80/4 via filling is achieved within 24 minutes
- Decrease electrolyte agitation for a smaller dimple
- Apply low Fe3+ concentrations, depending on required surface roughness
- Work at 35-40°C to achieve best dimple

Summary and outlook

Aim of the above presented optimization experiments was to identify the best possible parameter sets and combine it for through-vias thicker than 100 µm, especially for 400 µm thick cores. For some parameters a compromise between dimple and surface roughness had to be chosen.

This test series showed that for a successful fill of thicker core as 400 µm, a split into different electrolyte types is necessary with very different parameter sets.

Parameter	X-formation	Via fill
Pulse parameter	Strong reverse	Strong reverse
Overall current density	3-4 ASD	3-4 ASD
Temperature	40°C	35-40°C
Cu^{2+}	Low 20-30 g/l	High > 40 g/l
Fe^{3+}	Very low 0.5g/l	> 1.5 g/l
Electrolyte agitation	Strong	Low
Time	Approx 15 min.	40-60 min.

Table 2: Process parameters for both TH fill steps.

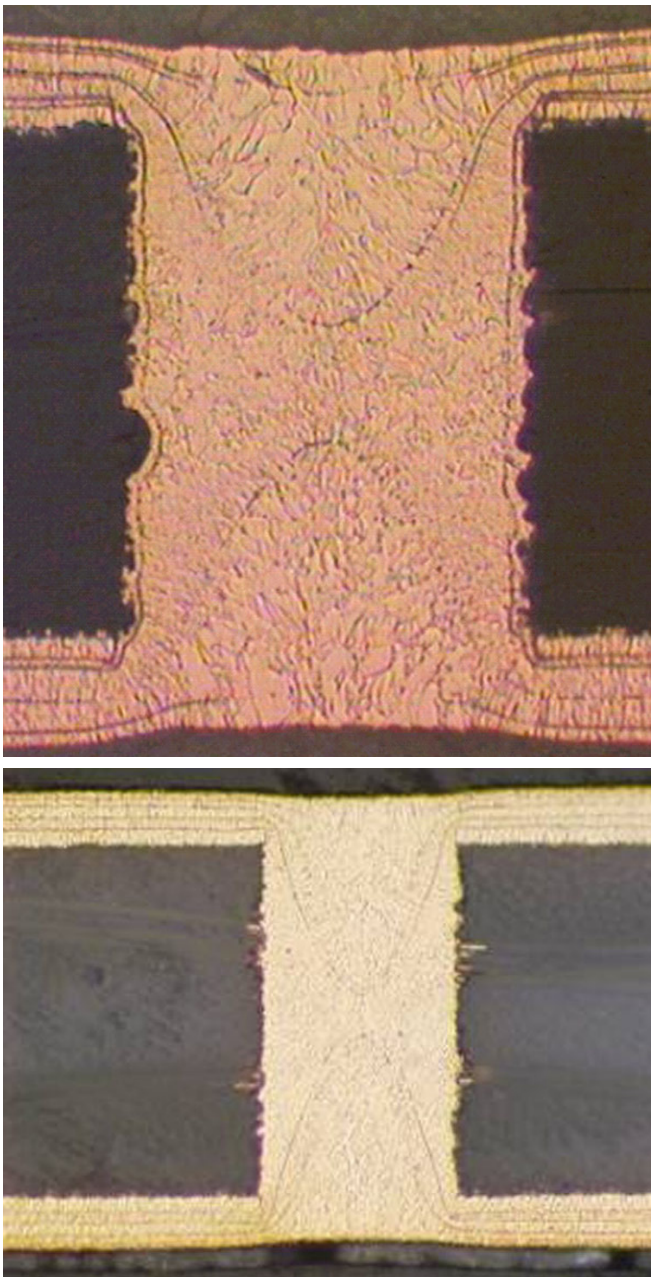


Figure 9: Filled vias: 150 and 200 µm cores that have been plated with 22 resp. 23 µm Cu to achieve less than 10 resp. 5 µm dimple.

At this moment we can successfully plate and fill through-vias for core with thickness of 300 µm or less within one hour and without occurrence of voids. Some examples are given in Figure 9.

On the other hand, 400 µm cores show some voids still. These voids are usually small (10 µm or less, see Figure 10).

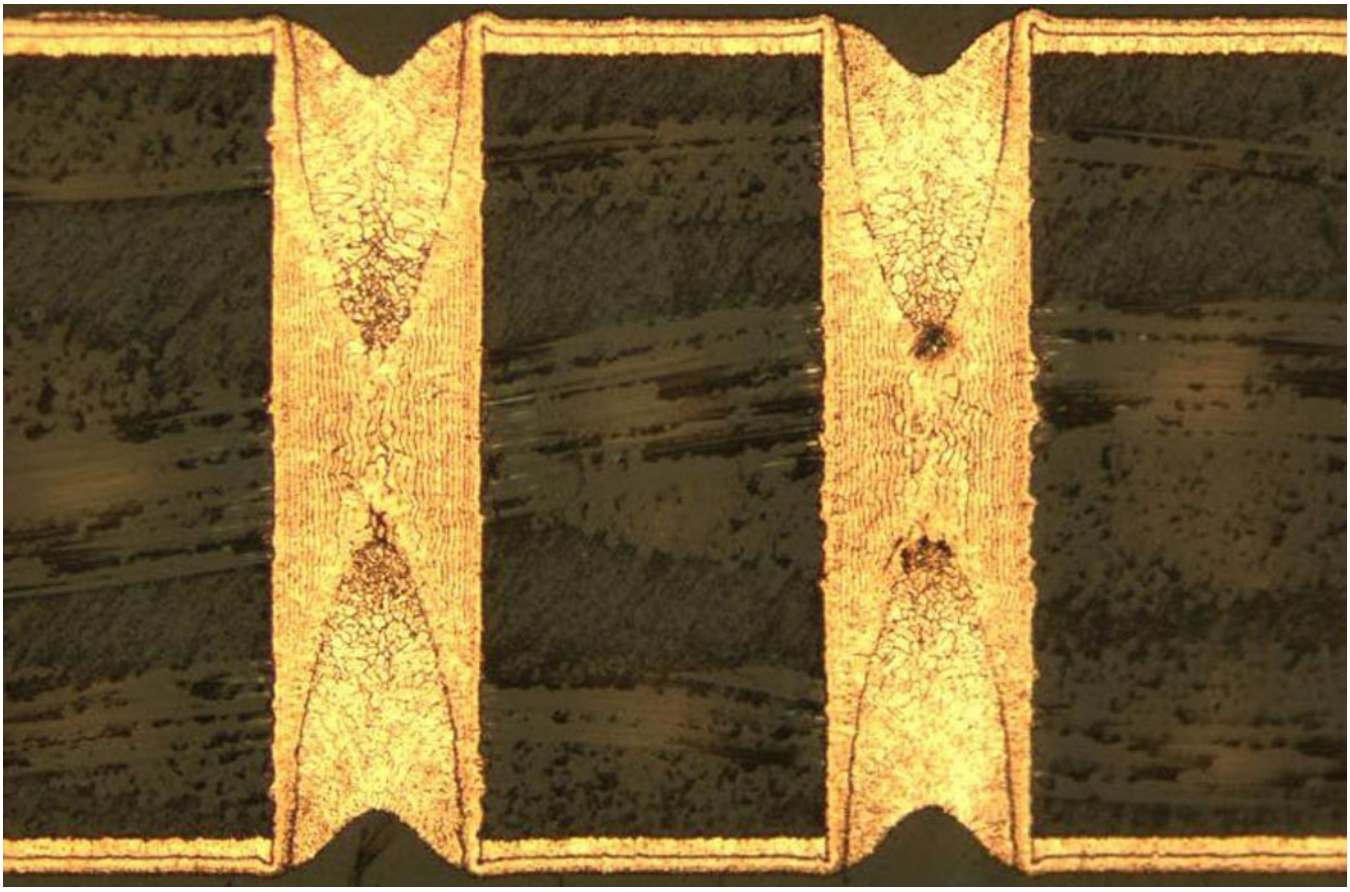


Figure 10: Completely filled TH showing small voids.

Investigations on the effect of these voids are under way and further planned. Especially, reliability measurements will be performed, which should show whether the voids will lead to cracks or stay inside the via as they seem to be perfectly encapsulated. Whether this technology can be expanded over 400 µm cores is still unclear. There is a great potential for completely filled Cu vias as thermal vias since the thermal conductivity of Cu is several magnitudes higher at every paste. But the greatest hurdle for this application is the use of very thick panels (>1mm) which require huge amounts of Cu, and thus long process times. **PCB**

References

1. M. Carano , "Via Hole Plugging Technology," CircuiTree, May 2007

2. Stephen Kenny and Bernd Roelfs; "Copper Electroplating Process for Next Generation Core Through-Via Filling" Proceedings of the Pan Pacific Microelectronics Symposium, Big Island Hawaii, Feb 10-12, 2009

3. Stephen Kenny, Bernd Roelfs," High Aspect Ratio Package Core Production with Electrolytic Deposited Copper," IMAPS 2010—Research Triangle, 43rd International Symposium on Microelectronics October 31 - November 4, 2010

Mike Palazzola has more than 30 years experience in the PCB industry, and is currently a product manager for the electroplating and surface treatment product lines at Atotech USA. He has recently presented several papers at IPC conferences. Mike would like to acknowledge his co-authors and colleagues: Nina Dambrowsky, Christof Erben, Stephen Kenny, and Bernd Roelfs from Atotech Deutschland GmbH.

Most-Read Supplier/New Product News Highlights This Month

Enthone Appoints Rick Reagan President

Cookson Performance Materials, a division of London-based Cookson Group plc, announced the appointment of Rick Reagan as president of Enthone, Inc. Reagan will lead the high-performance specialty chemicals company's global organization that operates in more than 40 countries and includes 10 manufacturing facilities and nine technical centers strategically located worldwide.

Nelco to Close China Facility

Park Electrochemical Corporation announced that its Nelco Technology (Zhuhai FTZ) Ltd. facility, located in the Free Trade Zone in Zhuhai, China, plans to commence procedures required to cease its operations and liquidate, subject to the requisite approvals of government authorities in China.

Orbotech's PCB Equipment Sales Up in Q2

In the company's Production Solutions for Electronics Industry segment, sales of equipment to the PCB industry were $48.6 million in the second quarter of 2012, compared to $48.5 million in the first quarter of 2012 and $58.7 million in the second quarter of 2011; sales of equipment to the FPD industry were $12.3 million in the second quarter of 2012, compared to $11.8 million in the first quarter of 2012, and $56.7 million in the second quarter of 2011.

ITW Takes Licence to Integrate PragmatIC's Tech

PragmatIC Printing Ltd. and ITW Foils and Thermal Films Group, a part of Illinois Tool Works Inc., have announced a licensing agreement enabling ITW to develop and sell novel printed electronics solutions incorporating PragmatIC's unique technology.

Burkle Achieves Positive 1H; Poised to Hit Sales Target

The first six months of 2012 have been positive for the machine manufacturer specialized in presses and coating lines. Robert Bürkle GmbH, with headquarters in Freudenstadt, Germany, will reach the planned sales target of 94 million euros by the end of the year.

Orbotech Receives Multiple Orders for LDI Systems

Orbotech, Ltd. announced the receipt of multiple orders from leading touch screen manufacturers in Asia Pacific, including Taiwan, China and Korea, for its Orbotech LDI™ 150 laser direct imaging system. Orbotech's LDI systems are used to enable the mass production of One Glass Solution (OGS) sensors on routed cover glass, a cutting-edge touch screen application that is lighter, thinner, less expensive and more reliable than existing (multilayer) glass-based touch sensors.

Streamline Circuits Invests in Frontline's InSight PCB Software

Greg Halvorson, president of Streamline Circuits, said, "As time and technology experts in today's fast-paced PCB industry, new tools like InSight PCB help us to make our process even faster, more efficient and precise. By automating the data review process, we can sharpen the accuracy and competitiveness of our bids while also reducing the iterations between our front-end staff, OEM engineers and sales team."

Taiyo Hires Scott McLeod as Technical Service Engineer

According to Don Monn, Eastern region sales manager at Taiyo America, "Scott brings years of PCB manufacturing expertise, as well as extensive solder mask application and troubleshooting experience. Scott is well versed in Taiyo's product line and processing requirements. He becomes an immediate asset to our U.S. technical service team."

Rogers' Printed Circuit Materials Q2 Sales Down to $40.5M

Rogers Corporation today announced second quarter 2012 net sales of $126.7 million and income from continuing operations of $0.38 per diluted share, which includes net one-time charges of $0.09 per diluted share associated primarily with the ceasing of operations in its leased facility in Bremen, Germany.

Maskless Lithography Sells D.I. Line to APCT

APCT President Steve Robinson said, "The company has poured millions of dollars into personnel and equipment including our first Maskless Lithography digital imaging system. Our engineering team has worked side by side with the OEM to further develop the technology. These months of diligence have proven invaluable to both APCT and the Maskless engineering group."

Association Connecting Electronics Industries

IPC

SAVE THE DATE!

Upcoming IPC Professional Development Events

October 3

ITAR/EAR Compliance for PCB Manufacturers Workshop — Complying with U.S. Export Controls
Irvine, CA

October 24–25

IPC HDI Conference: Advancements in Materials, Processes and Applications
Sponsored by Raytheon
Los Angeles, CA

October 30–31

It's Not Easy Being Green: Electronics and the Environment
San Francisco, CA

November 7–9

IPC Medical Industries Technical Conference: Lead-Free Reliability Readiness
Sponsored by Philips Healthcare
Boston, MA

November 13–15

IPC-SMTA High-Reliability Cleaning and Conformal Coating Conference
Chicago, IL

December 4–5

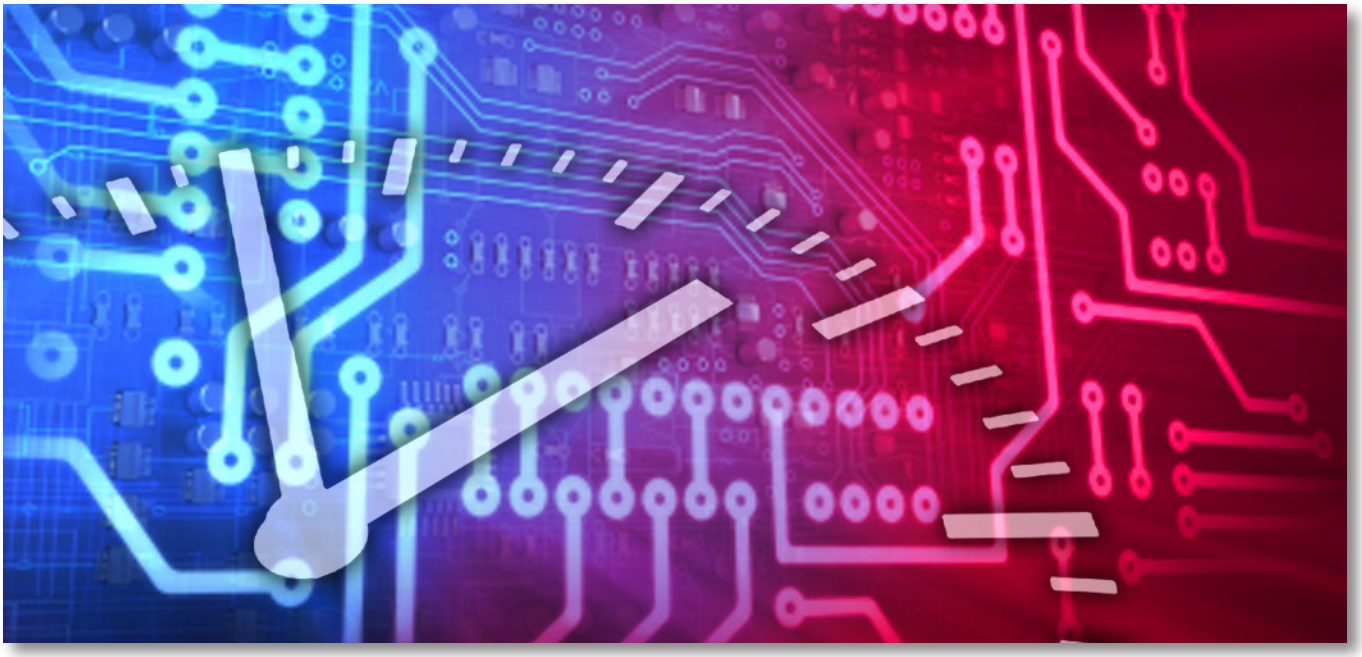
IPC Symposium on Lean for the Electronics Assembly Industry: Putting Theory to Practice
Costa Mesa, CA

December 4–6

National Conference on Electronics Design, Assembly and Reliability
Bangalore, India

More Information

Questions? Contact Susan Filz at +1 847-597-2884
www.ipc.org/events



A Time-Dependent Analysis of Heat Transfer in a PCB During a Thermal Excursion

by J. Lee Parker
JLP CONSULTANTS

EDITOR'S NOTE: This paper was originally presented at IPC APEX EXPO in February 2012 in San Diego.

SUMMARY: This study goes beyond the equilibrium condition to explore the temperature-time distribution of a PCB when a variable temperature heat source is applied to both outer surfaces.

A great deal of work already has been done to determine the equilibrium temperature of a PCB when exposed to a heat source such as the thermal environment of reflow soldering. This study will go beyond an equilibrium condition and explore the temperature-time distribution of the board when a variable temperature heat source is applied to both outer surfaces. For simplicity, the model will be a two-sided board. Obviously, the model board has two material interfaces. An interesting observation is that anywhere within the board, including the material interface, thermal energy must be conserved. There is not a similar requirement for the temperature. Consequently, at the material interfaces, we can expect the thermal properties of the board to change in a profound manner. A similar situation occurs when a fluid passes through a shock wave. This will be reflected in such board properties as the thermal stresses in the various layers and the resulting warp. This phenomenon also explains and quantifies why a thermal shock can be devastating, while a slow temperature rise to the same endpoint may well be tolerated.

The analysis will use a one-dimensional, time-dependent model (i.e., there are two independent variables). This necessitates a partial differential equation to describe the temperature variation within the board. The boundary conditions are the outer temperature of the board, which is the temperature of the heat source on both outer surfaces. The third boundary condition is at the copper epoxy interface where conservation of thermal energy is required.

Introduction

Techniques for measuring the surface temperature of a PCB have been available for some time. Measurements for assessing the temperature at various positions in the laminate are at least problematic. Inserting thermal couple wires well into a thin laminate will more than often distort the temperature as they now become a part of the thermal mass of the system—

the so called Helmholtz effect (to some extent the tool used to measure any physical quantity will disturb the measurement). In this case, the thermocouple wire will conduct heat locally, away from the laminate, cooling that area of the package. The larger the thermocouple, the greater the issue. In situations of this nature, it often is best to use analytical models to estimate the value of the desired measurement (i.e., the internal temperature history of a PCB as it goes through a heat excursion. If the temperature history distribution in the PCB package can then be estimated, the internal shear stress can be quantified. Once this is accomplished, the likelihood of forming an internal delamination can be established. Developing such a procedure and using it to determine the internal laminate history as the PCB is exposed to a temperature ramp, such as in reflow, is the objective of this analysis.

Analysis

To simplify the analysis, the investigative package will be a symmetrical, double-sided PCB. The copper thickness is defined to be 1.7 mils. The thickness of the internal laminate is 60 mils and composed of FR-4, (Figure 1). The PCB will be exposed to several temperature ramps, all ending at 600°F.

It will be assumed that the board has been preheated to a uniformed temperature of 3000°F before the temperature ramp.

According to first principals, heat conduction through a solid is governed by Fourier's law of heat conduction. The assumption is made

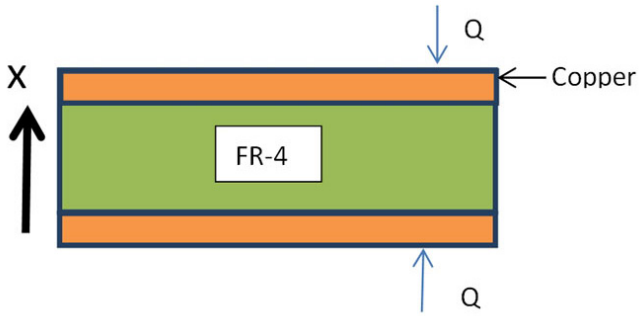


Figure 1: The symmetrical double-sided PCB for investigation features a copper thickness of 1.7 mils and an internal laminate thickness of 60 mils, and is composed of FR-4 laminate.

that the heat flow is one-dimensional. Consequently, there are two independent variables: time and position. For one-dimensional, time-dependent heat transfer, Fourier Law becomes

Equation 1:

$$\frac{\partial T}{\partial t} = k\frac{\partial^2 T}{\partial x^2}$$

A unique solution requires three boundary conditions:

- At t=0, T=3,000°F for the entire board
- At x=0, T is specified by the ramp temperature
- Along the axis of symmetry of the board,

$$\frac{\partial T}{\partial x} = 0$$

Unfortunately, since the second geometric derivative of temperature appears in (1), the first derivate must be continuous. Conservation of energy requires a consistent heat-flow rate across the interface of copper and FR-4. Since the thermal properties of the material change at the interface, conservation of energy requires a discontinuous adjustment in the temperature gradient. To avoid the issue we will introduce the following transformation[1]:

Equation 2:

$$X = k\,\beta$$

Making β the new geometric variable.

After making the transformation, Equation 1 becomes:

Equation 3:

$$\frac{\partial T}{\partial t} = \frac{1}{k^2}\frac{\partial^2 T}{\partial \beta^2}$$

A closed-form solution exists for Equation 1 modified by the above transformation, but it involves several infinite series, which defy ex-

traction. A better course of action is to use a numerical solution. For the finite difference approximation to Equation 3, the time derivative will be replaced by a forward difference and the geometric derivative by a centered difference. The result is:

Equation 4:

$$T_i^{n+1} = T_i^n + \left(\frac{1}{k^2}\frac{T_{i+1}^n - 2T_i^n + T_{i-1}^n}{\Delta\beta^2}\right)\Delta t$$

It is shown in Reference 1 that a numerical solution for Equation 4 will converge provided:

Equation 5:

$$\Delta t \le \frac{k}{2}(\Delta\beta)^2$$

Results

A numerical integration of Equations 4 and 5 will now be examined for temperature ramps of 2, 4, and 6 degrees Fahrenheit/second. Initially, the board is at a uniform temperature of 3,000°F. The temperature ramp is completed when the surface temperature of the copper reaches 600°F. The temperature ramps are shown in Figure 2.

The integration of Figures 3 and 4 also defines the temperature within the board as it passes along the temperature ramp. Figure 3 shows the board's temperature profile when the surface of the board first reaches 600°F.

As seen, the temperature gradient is large in the outer portion of the board. Then, beyond the copper/FR-4 interface, the gradient rapidly approaches zero. This is caused by the favorable heat transfer properties of copper and the adverse heat transfer properties of FR-4. As seen later, this abrupt behavior causes large shear stresses to form at the interface, which can result in copper delamination.

In order to estimate the shear stress, it is first necessary to calculate the difference in the temperatures of the copper and the substrate. For this purpose, the characteristic temperature of the PCB component is defined as the average temperature of each component. The result is presented in Figure 4, which shows the temperature difference as a function of temperature ramp. The temperature differential is obviously nonlinear. The rate of change in the temperature differential rapidly changes as the temperature ramp increases. It then becomes nearly constant at the high end.

In Reference 2, it is shown that the shear stress at the interface is:

Equation 6:

$$\tau = \frac{1}{L}\frac{E_{cu}E_L t_{cu} t_L}{E_{cu}t_{cu} + E_L t_L}(\alpha_{cu} - \alpha_L)\Delta T$$

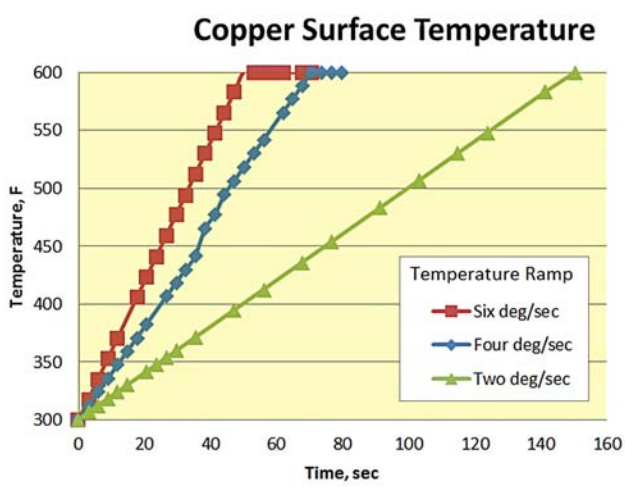


Figure 2: Copper surface temperature.

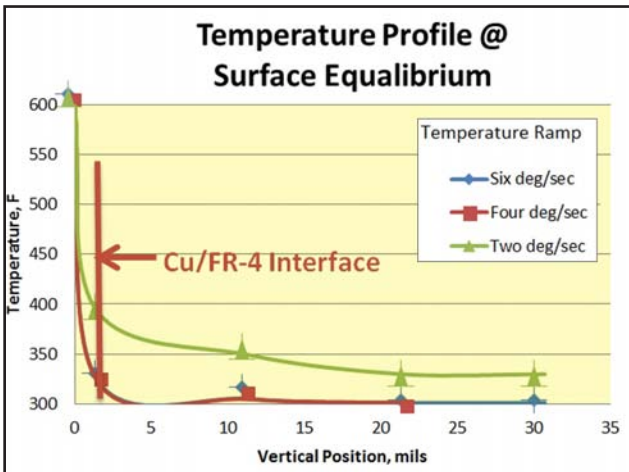
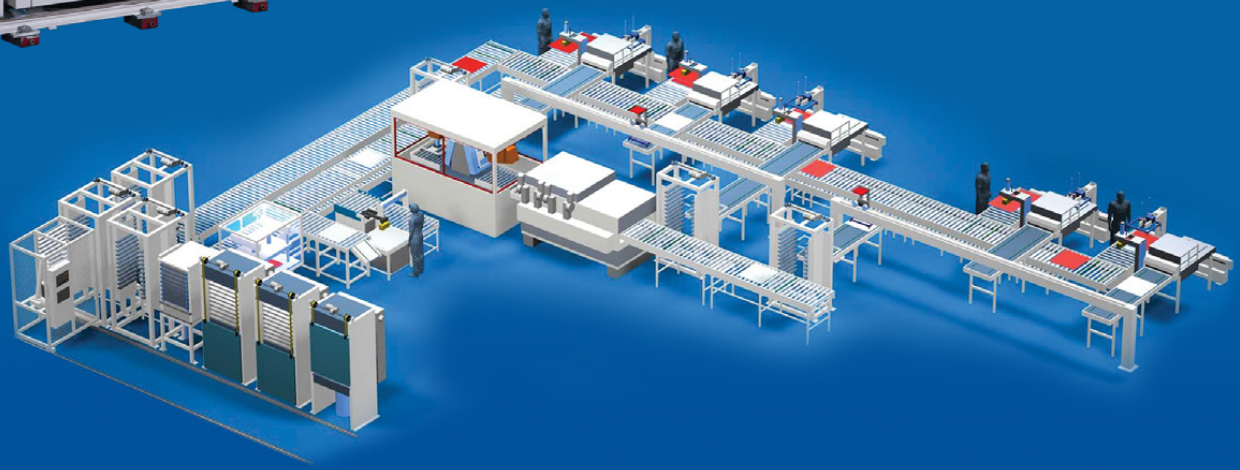


Figure 3: Temperature profile at surface equilibrium.

WorkCell² Vacuum Lamination System

... A Modular Design That GROWS With You

With the ability to expand as production needs dictate, Bürkle's modular WorkCell² provides a precision Vacuum Lamination System for every phase of business – from start-ups to high-volume production houses – we have the right system for you.



The Bürkle WorkCell² is part of Bürkle's wide range of laminating and coating systems. Bürkle is a member of the Bürkle North America alliance with Schmoll-Maschinen, Bürkle GmbH, IMPEX, LHMT and Bacher Systems. For more information contact Bürkle North America via e-mail: sales@burkleusa.com.

Alliance partner web sites:

- www.schmoll-maschinen.de
- www.buerkle-gmbh.de
- www.bacher-systems.com
- www.impex.co.at
- www.lhmt.de

BÜRKLE

NORTH AMERICA, INC.

WWW.BURKLEUSA.COM

Bürkle North America, Inc.
12802 Valley View St., Ste. 12/13
Garden Grove, CA 92845
TEL: 714-379-5090
FAX: 714-379-5092

Where α is the coefficient of thermal expansion, L is the characteristic length of the copper feature and ΔT is the temperature differential. The shear stress is presented as a function of the copper feature size and ramp rate in Figure 5. It will be noticed that the shear stress increases rapidly as the feature size diminishes. In fact, one can expect to experience shear stresses in excess of 30K psi for present day PCBs.

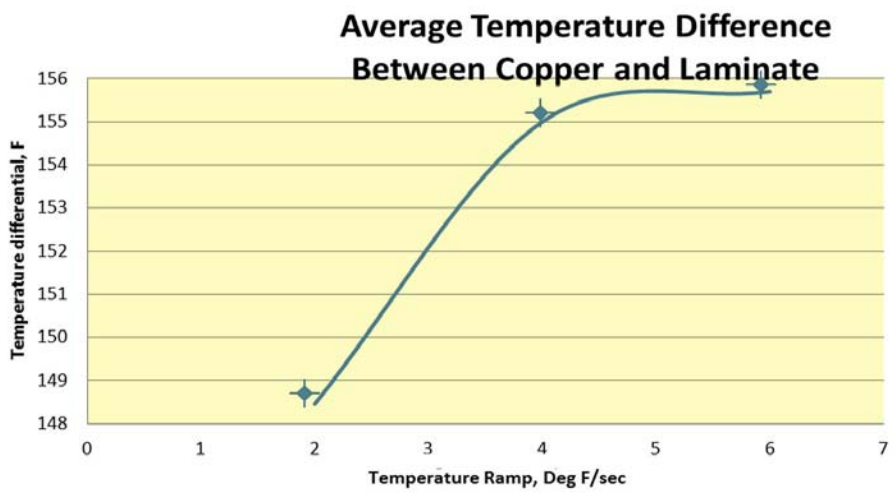


Figure 4: Average temperature difference between copper and laminate.

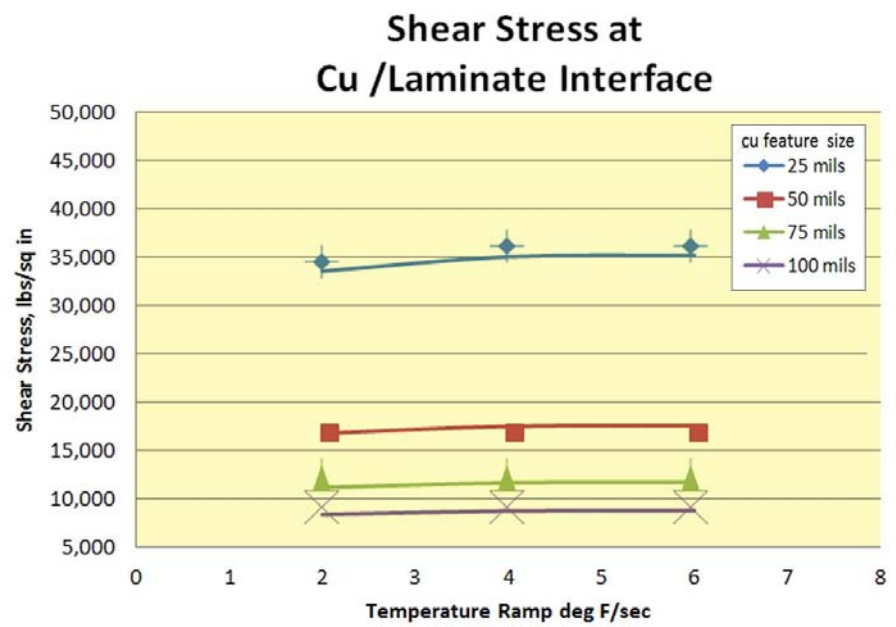


Figure 5: Shear stress at Cu/laminate interface.

Summary

The analysis above has developed a technique for numerically integrating the one-dimensional, time-dependent Fourier heat conduction equation for a PCB which is stressed by a heating ramp. This in turn will quantify the relationship between the process and design parameters and the stresses incurred during a thermal excursion. The analysis shows there are substantial temperature gradients at the interface of the copper and FR-4 components. It is furthermore shown that these temperature gradients can produce very substantial shear stress at the copper/FR-4 interface. This stress is inversely proportional to the size of the copper feature and directly proportional to the gradient of the heating ramp. **PCB**

References

1. Jakob , Max Heat Transfer, John Wiley and Sons, New York, 1962
2. J. Lee Parker, Proceedings of IPC APEX, April 2011

Dr. Lee Parker retired from Lucent Bell Laboratories as a Distinguished Member of the technical staff after a 27-year career. He has more than 30 years experience in the electronics industry and is now an industry consultant. He was a joint recipient of the Best Technical Paper Award at the 1996 IPC Expo and is a past chairman of the IPC Flexible PC Design Guide Subcommittee.

APEX EXPO IPC

CONFERENCE & EXHIBITION
February 19–21, 2013
San Diego Convention Center

IPC APEX EXPO is "THE SHOW" … to attend valuable courses and see new equipment that will help you and your company see new possibilities, improve your efficiencies and cut costs. Keep your competitive edge.

Mario Dion
Manufacturing Engineer Specialist
Trilliant Networks

INFORMATION that INSPIRES INNOVATION

Join **thousands of your colleagues** from more than **50 countries** and more than **400 exhibitors** at **IPC APEX EXPO 2013** in beautiful San Diego. Learn about **new technologies and processes,** see **new products,** work on **industry standards,** network with **industry experts** and participate in the **world's premier technical conference** for electronics manufacturing. **Register today! Find solutions** that will help you **save time and money** and **gain a competitive advantage.**

www.IPCAPEXEXPO.org

Scan for your chance to win a three-night hotel stay or an All-Access registration.

design | printed boards | electronics assembly | test and printed electronics

TOP TEN

Most-Read News Highlights from PCB007 this Month

1 IPC Launches Educational ITAR and Printed Boards Campaign

"IPC's educational campaign seeks to clarify the importance of applying ITAR controls to printed boards given that each printed board is specifically designed for the larger defense item of which it is a part," says Mitchell. "We view this effort as a pivotal private-sector government partnership on an important policy issue that ultimately bolsters national security."

2 I-Connect007 Introduces thePCBlist™

I-Connect007's Andy Shaughnessy joins Patty Goldman, project manager for I-Connect007's newest addition to the PCB industry: thePCBlist™. Goldman, a PCB industry veteran, explains the scope of this newly-launched, comprehensive directory for PCB buyers, and why it far surpasses anything the industry has seen previously.

3 Viasystems' Q2 PCB Sales Up 12% to $240.4 Million

Net sales and operating income in the company's PCB segment for the second quarter of 2012 were $240.4 million and $15.1 million, respectively, compared with PCB sales and operating income of $215.2 million and $12.4 million, respectively, for the second quarter of 2011 and compared with PCB net sales and operating income of $213.1 million and $7.0 million, respectively, for the quarter ended March 31, 2012.

4 IPC, JPCA Release Operational-Level Standard for PE Industry

To help designers and companies create reliable products using printed electronics technologies, IPC/JPCA-4921, Requirements for Printed Electronics Base Materials (Substrates), defines terms and establishes basic requirements for five material categories of substrates used today in printed electronics: Ceramic, organic, metal, glass, and other.

5 NCAB Group Expands; Acquires P. D. Circuits

"I'm extremely happy for this next major step in NCAB's development. NCAB Group and P. D. Circuits have the same business concept and customer-driven organization and perhaps, most importantly, we have the same way of thinking regarding quality, the fundament of all PCB production," said Hans Stahl, CEO at NCAB Group.

6 Würth Elektronik Closes Pforzheim Facility

Alternative job offers in other business units of the Würth Elektronik Group at different locations were made to the 29 employees. Business operations, refinement of printed circuit boards by polymer print will be relocated to other business units and corporate divisions within the Würth Elektronik GmbH & Co. KG.

7 Weak Demand, Higher Labor Costs Show in TTM's Q2 Results

"Second-quarter revenue was within our expected range as sales increased following the normal seasonality we experienced in the first quarter. However, weaker than expected demand for advanced technology PCBs as well as higher labor costs in Asia impacted margins and earnings during the quarter," said Kent Alder, president and CEO.

8 IPC: PCB Shipments, Bookings Down Year-on-Year

Rigid PCB shipments were down 7.3% in June 2012, and bookings decreased 2.0%. Year to date, rigid PCB shipments decreased 5.4%, but bookings increased 2.1%. Rigid PCB shipments were up 11.6% and rigid bookings gained 11.8%. The book-to-bill ratio for the N.A. rigid PCB industry in June 2012 continued to slip, but remained above parity at 1.01.

9 Royal Circuits Invests in Maskless Lithography System

Milan Shah, president and owner of Royal Circuits Solutions, one of the industry's fastest growing PCB fabricators, has announced the acquisition of a Maskless Lithography direct imaging system. The new unit allows the company to provide their customers with the most accurate imaging on the market today. The new system, built in San Jose, California, provides superior side-to-side registration; scaling for flexible alignment on tight tolerance boards; excellent trace and space quality; and, lines below 3 mils.

10 GSPK Circuits Invests in Multilayer Technology

GSPK Circuits has purchased two Burkle bonding presses and a cooling press recently to augment its already thriving multi-layer production process. This additional multilayer bonding capacity provides GSPK with the facility to process up to 384 multilayer panels daily. The cycle time is optimized, which in turn, generates high-production throughput, reducing lead-times.

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

EVENTS

- IPC Complete Calendar of Events
- SMTA Calendar of Events
- iNEMI Calendar
- PCB007 Online Events

J-STD-001E: "Requirement for Soldered Electrical and Electronic Assemblies"
September 10-14, 2012
Hakko's Facility, Valencia, California, USA

PCIM South America
September 11-13, 2012
Sao Paulo, Brazil

West Penn Expo and Tech Forum
September 13, 2012
Monroeville, Pennsylvania, USA

EIPC Summer Conference
September 13-14, 2012
Milan, Italy

The ECOC Exhibition
September 16-20, 2012
Amsterdam, The Netherlands

Advancement in Thermal Management 2012
September 18-19, 2012
Denver, Colorado, USA

Manufacturing in Guadalajara 2012
September 18-20, 2012
Fiesta American Grande, Guadalajara, Jalisco, Mexico

IPC Workshop: Design for Manufacture
September 19-20, 2012
Fredericksburg, Virginia, USA

MEDevice Forum San Diego
September 19-20, 2012
San Diego Convention Center, San Diego, California, USA

Northeast Shingo Prize Conference
September 25-26, 2012
Worcester, Massachusetts, USA

Modern Day Marine
September 25-27, 2012
MCB, Quantico, Virginia, USA

MEPTEC/SMTA Medical Electronics Symposium
September 26-27, 2012
Arizona State University, Tempe, Arizona, USA

Austin (CTEA) Expo and Tech Forum
September 26, 2012
Austin, Texas, USA

Printed Electronics Asia
October 2-3, 2012
Tokyo, Japan

DMS Osaka
October 3-5, 2012
Intex Osaka, Japan

21ST ANNUAL INTERNATIONAL ELECTRONICS FORUM
October 3-5, 2012
Marriott Hotel, Bratislava, Slovakia

NEPCON Vietnam 2012
October 4 -6, 2012
Saigon Exhibition & Convention Center (SECC), Ho Chi Minh City, Vietnam

Plastic Electronics Conference 2012
October 9-11, 2012
Dresden, Germany

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

PUBLISHER: **RAY RASMUSSEN**
(916) 294-7147; ray@iconnect007.com

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

EDITORIAL:
GROUP EDITORIAL DIRECTOR: **RAY RASMUSSEN**
(916) 294-7147; 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, BRYSON MATTIES, SHELLY STEIN**
INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**
COVER: **MIKE RADOGNA**

PCB007 Presents

the pcb magazine

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

©2012 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

September 2012, Volume 2, Number 9 • The PCB Magazine© is published monthly, by BR Publishing, Inc

MASTHEAD

ADVERTISER INDEX

atg Luther & Maelzer GmbH..... 43
Atotech..... 1, 2
Burkle North America..... 93
Camtek..... 73
CA Picard..... 35
DIS Inc..... 33
Electra Polymers..... 79
I-Connect007..... 3
IPC..... 89, 95
Isola..... 71
Maskless Lithography..... 69
Matrix Electronics..... 15
Mentor Graphics..... 19, 21, 23
Microcraft..... 55
National Graphic Supply..... 7
OMG Electronic Chemicals..... 57
Prototron Circuits..... 85
Rogers Corp..... 47
Semblant..... 5
Stebgo Metals..... 9
Taiyo America..... 83
Technica USA..... 49
Uyemura Intl. Corp..... 25
Ventec..... 61
Viking Test..... 63

Government Regulations—in the October Issue of *The PCB Magazine*

Arguably, a main function of government is to create rules, but what rules are necessary or desirable, and what benefits do they bring to society and industry? Environmental legislation worldwide is causing a total rethink of product design and manufacturing processes. In the October issue of *The PCB Magazine,* we explore the effects and consequences of government regulations on the PCB and electronics assembly industries. Contributors ranging from industry insiders to electronics' association members will weigh in with their perspectives on how the electronics industry is impacted by government regulations.

Not yet a subscriber? Click here to get *The PCB Magazine* delivered to your inbox every month.