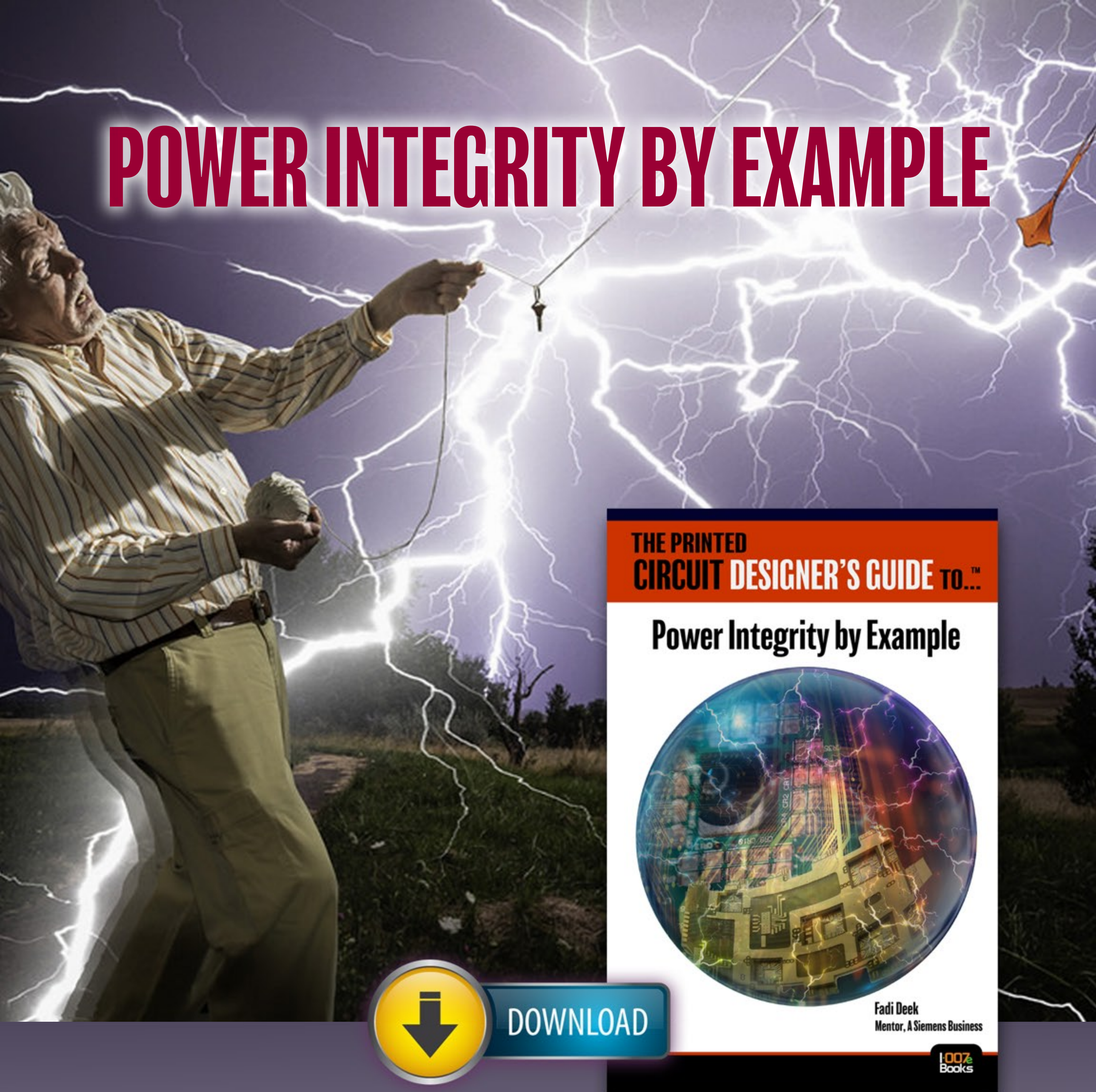
I-Connect007

JULY 2018

PCB007 MAGAZINE

The Fine Features of

Solder Mask

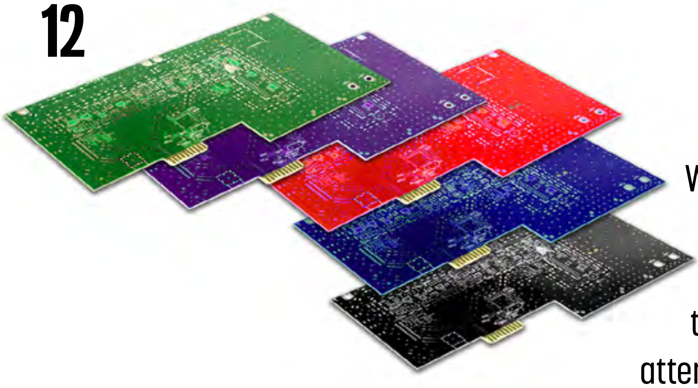


I-007e Books

TOPICS COVERED:

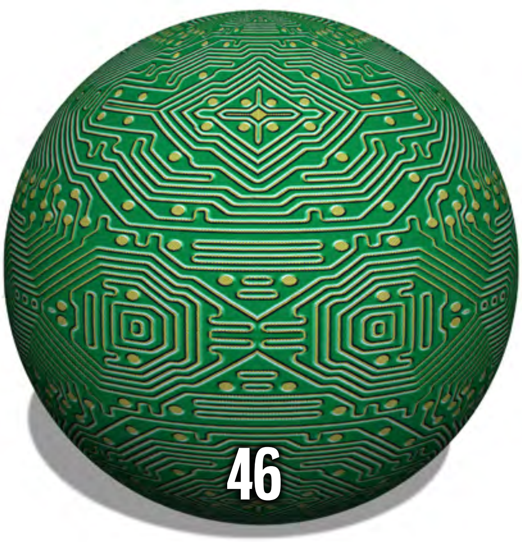
- Bare Cavity
- Decoupling Capacitor and VRM Effects
- Transparent Cavity
- Cavity Noise and Via-to-Via Coupling
- Reducing Via-to-Via Coupling Using Capacitors

PCB007
MAGAZINE



Solder Mask

With all the elements that can now affect the solder mask—finer features, higher-temperature solders, final finishes, direct imaging, inkjet—it's high time to give this final fabrication process a little more attention. This month, our contributors explain how and why solder mask must do its job well—precisely, in fact.



pluritec

Celebrating 50 Years of Service!

The Pluritec brands that provide you with tomorrow's technology today !

pluritec

X-ray Optimizer &
Automated flash/Bevel/Corner
rounding panel prep

Drills & Routers
24/7 automation

MORE INFO

ECOspray SYSTEM

Multi-color solder mask
coaters with 15 minute
fast tack oven

MORE INFO

IS International Supplies

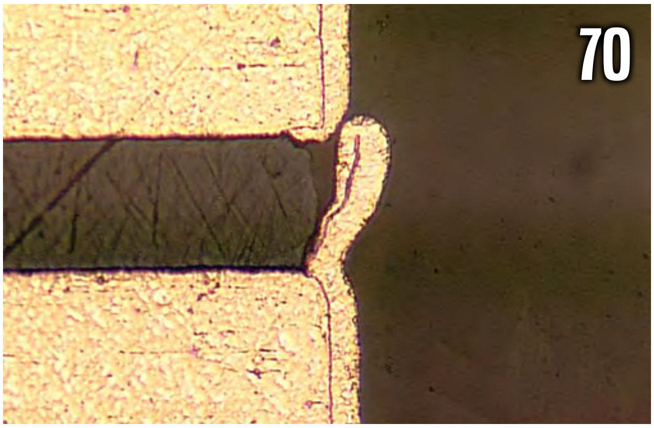
Wet Process
developers - etchers - strippers
Cobra - MEC Etch
Shadow - ChemClean

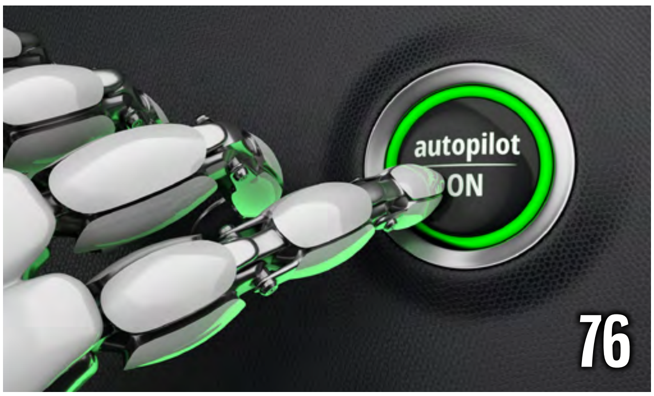
Mechanical prep
Debur - Scubbers - Pumice/Oxide
PCB brush & Chemical washing

MORE INFO

www.pluritec.com | info@pluritec.com

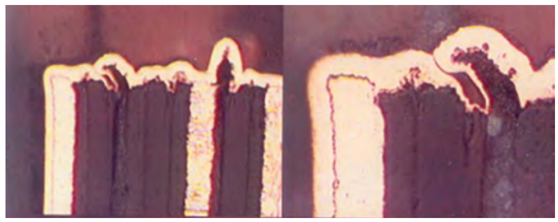
PCB007 MAGAZINE





SHORTS:

COLUMNS:



HIGHLIGHTS:

DEPARTMENTS:

More Than a Word: Solder Mask

Patty's Perspective

by Patty Goldman, I-CONNECT007

Do you spend time, as I do, musing on the language of PCBs? We have developed our own lexicon to convey as much by picture, as by word, what exactly we mean. I'm talking about words like micro-etch (hyphenated or not) and stripper (a whole new meaning), metalization and metalize (one "l" or two?); terms like mouse bites, measles, voids, vias, pits, etc. And that's not mentioning the myriad acronyms, from PCB right through to ENIPIG (we don't stop at just three letters, oh no).

Which brings me to solder mask (one word or two?). We have chosen to make it two words in this issue, and plan to keep it that way, regardless what some of our contributors say. But, why do we say "mask" at all? What exactly does that mean? I know some say solder resist, which is more apropos, as this material does indeed more or less repel or resist the solder. The mask part pertains to the underlying circuitry that is masked or protected from the solder. But, as one of our experts this month has pointed out, sometimes there will be no solder, in which case the solder mask is protecting against nickel-gold or immersion tin or one of the other final finishes. We can go on and on…

But with all the things that can now affect the solder mask—finer features, higher-temperature solders, final finishes, direct imaging, inkjet—we felt it was high time to give it a little more attention. As one of the last processes a PCB sees, you want it to do its job well and precisely; now is not the time to be scrapping your expensive multilayer—or any board for that matter. With that, let's get to our lineup this month and start learning.

We first wanted to learn what was new in the solder mask area, so we met with Electra Polymers' Shaun Tibbals and Antony Earl. We discussed the difficulties of clearing small holes of solder mask and the problems with plugging holes—opposite requirements that are often specified. The conversation ranged further into direct imaging of fine features and the projected use of inkjet for solder mask—there are advantages and disadvantages.

One area of concern with solder mask that is covered (ooh, a pun) by Atotech's Rick Nichols

et al., is the effect of a solder mask and its residues on the subsequent final finish process. The focus here is on identifying possible markers that could be detrimental to both the final finish chemistry and the solderability of the board during assembly.

While it was good to talk with the chemistry people, we wanted to hear from the equipment folks, too. Circuit Automation has been making automated screen printing equipment for a very long time, so we tapped Tom Meeker, Larry Lindland and Yuki Kojima. As is often the case, we ended up learning about much more than just screen-printing equipment, including how different formulations require different set-up parameters, the difficulty of plugging a variety of hole sizes on one board, the importance of a uniform coating, handling flex—and the list goes on.

Lackwerke Peters' Sven Kramer wrote a great article on the thermal capabilities of solder masks. Not only must they survive high-temperature solders and rather aggressive final finishes, but they are then exposed to considerable thermal stress and high humidity conditions in the field—think automotive applications—all while reliability demands have become more stringent. Kramer presents a primer on solder mask formulation, aging test data with illustrations, and then a fine discussion on white LPI solder masks.

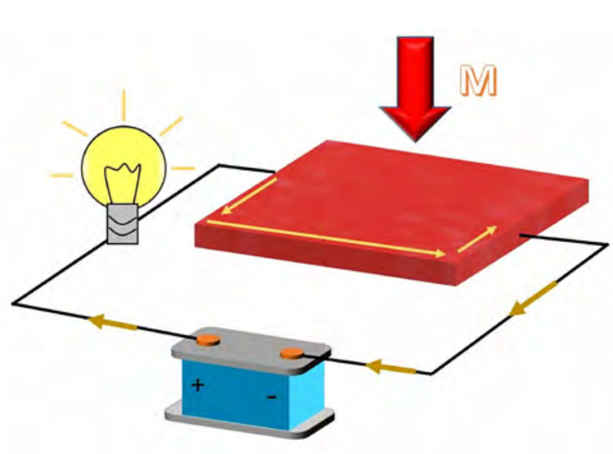
As you can see, there are many areas and much to learn and think about with solder masks, but let's move on to our columnists. This month, Mike Carano, RBP Chemical Technology, discusses flexible metalization, specifically possible problems associated with adhesion of electroless copper to both substrate and copper layers. Needless to say, the process is significantly different from that used for FR-4 and other common substrate materials.

Bringing up the rear is IPC's John Mitchell and his column about automotive electronics. After reviewing some of the major changes that have occurred in the past few years, he discusses the automotive addendums that are under development through IPC and asks for your input and expertise to help with that.

Well, it's July and 2018 is already half gone. As always, we ask, how did that happen? But of course, we are not nearly done with providing interesting and informative coverage on a wide range of topics in the upcoming issues. Next month the topic is reliability and then we plan to focus on mSAP and SLP in September. I sure hope you are a subscriber by now! **PCB007**

Patricia Goldman is managing editor of *PCB007 Magazine*. To contact Goldman, click here.

Rutgers-led Research Could Lead to More Efficient Electronics



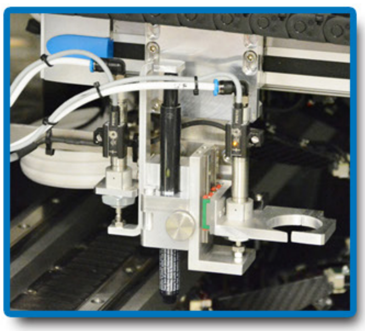
A Rutgers-led team of physicists has demonstrated a way to conduct electricity between transistors without energy loss, allowing low-power electronics and, potentially, quantum computing that would be far faster than today's computers.

Their findings, which involved using a special mix of materials with magnetic and insulator properties, are published online in Nature Physics.

"This material, although it's much diluted in terms of magnetic properties, can still behave like a magnet and conducts electricity at low temperature without energy loss," said Weida Wu, senior author of the study and associate professor in the Department of Physics and Astronomy at Rutgers University-New Brunswick.

Unrivaled Test Speed with Full Automation

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



Highlights:

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

Watch video

Get more info

atg®

LUTHER & MAELZER®

an Xcerra company

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

Solder Mask:
You've Come a Long Way, Baby!

Feature by the I-Connect007 Editorial Team

For this issue, our editing team met with Electra Polymers' Shaun Tibbals and Antony Earl to discuss what's new with solder masks, including direct imaging and inkjet printing of solder mask, and what PCB manufacturers and OEMs need to know.

Patty Goldman: Welcome, gentlemen. We're interested in finding out what our readers need to know with regard to solder masks, including the challenges and problems, especially the rapid changes our industry is experiencing. Please introduce yourselves and tell us about Electra Polymers.

Shaun Tibbals: Electra Polymers was founded in 1984 specifically to manufacture coatings for printed circuit boards. In 1984, the main products were screen-defined two-pack epoxy solder mask and UV solder mask, and then we went on to formulate photoimageable materials in the mid to late '80s. We've continued to make solder masks for the printed circuit board industry, but as the years have gone by, we've also diversified into other markets—still within the electronics business, but more focused on the semiconductor end. We now manufacture and supply wafer-level packaging resists to the semiconductor business. There's quite a crossover in terms of technologies and close to 30% of our business is now in that sector.

I am the sales and marketing director for Electra Polymers. I've held several positions within the company ranging from technical and quality positions to sales positions.

Antony Earl: I'm the technical support and quality manager. Like Shaun, I've been here for quite a long while and held a variety of positions, from quality control testing to field and internal technical support and formulations. Currently, I look after the technical support activities and the ISO quality and environmental systems for the company.

Goldman: Thanks, so let's get into some details. What do you see coming up that's new in the

area of solder masks? What are the challenges facing circuit board manufacturers?

Tibbals: I would say the two main challenges are 1) smaller features, and 2) final properties of the boards and therefore solder mask.

Circuit boards are getting smaller, and the features are getting tighter. So, what does that mean for a solder mask? Well, it really means two main things. Via holes are getting smaller and more densely packed and overall registration is getting tighter. So, from a solder mask perspective, you need to be able to resolve small features, but you also need to be able to remove the solder mask from very small holes. Those two things really pull in opposite directions, because the more you try to clear small holes, the more potential problems you face with undercut of small features on the board surface due to the prolonged developing stage.


Shaun Tibbals

Barry Matties: How do you overcome that challenge?

Tibbals: Washing smaller holes clean is just a fact of life when dealing with photoimageable solder masks. Now, there are new solder mask technologies coming along which eliminate ink in the holes, and we will come to these shortly, but with current application methods you've got two choices: You either have to plug them or you've got to wash them completely clean. The issue for the circuit board manufacturer is being in no man's land. Being halfway in between is not desirable; the concern for PCB manufacturers and end users is trapped chemistry in those partially filled holes and the subsequent impact on reliability.

If we decide to completely plug them, then there are offline processes well established for that. For via processing, there is machinery to plug those holes, but that's generally done with a dedicated hole plug material. Alternatively, a hole plugging stage could be used as part of the solder mask application process. However, to wash them clean is a perennial challenge. You will get variations in the amount of ink deposited in holes depending on the application method. The screen printing process would tend to put quite a large deposit in the holes and a spray application process would generally put less ink in the holes. But, once you get down to very small hole sizes, then irrespective of your application method, it starts to become a challenge. And that is something which our customers are aware of and a problem they have to deal with using harsher developing conditions—and that starts to impact the ability to resolve smaller and smaller features.

In terms of registration, tight registration and resolving smaller features on the circuit board are being dealt with by moving towards direct imaging, which takes away the issue associated with registration on the circuit board. Up to now, customers could just manufacture smaller size panels, in which case registration becomes slightly easier. In reality, customers need to be able to produce circuit boards on large panel sizes to make the whole thing economical and efficient. And if they're going to achieve that, trying to do it by exposing with an indirect, thermally unstable phototool becomes a challenge. Some phototool registration capability can be clawed back by making solder dams smaller but that does not help mask defined pads or via registration.

So customers have reached a point where tight registration just can't be done without yields dropping off, which is clearly a problem. The good thing is that direct imaging equipment has come along to solve this problem. Direct imaging has obviously been around for quite some time now, but in more recent years, the machinery has moved on in the sense that it's now multi wavelength, multi-head, and what that does is give a broader UV spectrum;

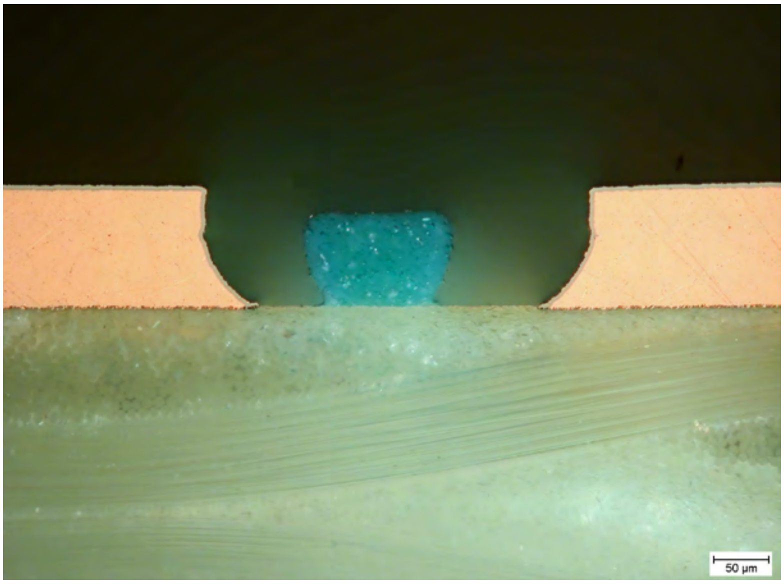


Figure 1: Profile of a solder dam using direct image photoimageable solder mask.

it gives more power, so the throughput per machine increases. So now, customers can put in a direct imaging machine and still get a reasonable throughput from the equipment. But, when you look at the throughput per capital equipment cost, it's still a very expensive investment. Instead of buying a conventional exposure unit, they're having to spend five to 10 times the cost to put in a direct imaging piece of equipment. Then the key thing is to get maximum throughput with that piece of equipment for their capital equipment cost. In many cases companies just don't have the capital to be able to invest in multiple direct image units so it is our job as solder mask suppliers to eke as much as we can out of each machine.

So although the technology exists, the capital equipment cost remains a limitation—which can be frustrating because the solution to the registration problems and the feature size resolution problems is there, but it's not necessarily readily available because of the capital equipment costs.

Matties: Are there any equipment suppliers you know of who are doing a pay-per-print model with that expensive equipment?

Tibbals: Not that I'm aware of. It's a bit of a problem because I'm not quite sure how that would work. The number of prints being done from each machine…I don't know what the payback would look like. It's an interesting idea, but I don't think any of the capital equipment people are offering it at the moment. The good thing is there are several equipment manufacturers on the market, and the competition has driven technology advancement as well as applied pressure on pricing for the machinery.

Earl: As Shaun has said, there are a variety of equipment suppliers available to the market now, and the material we've developed for direct imaging is tuned to be working towards more of those multi-wavelength systems that then bring a broader spectrum to make the imaging of the solder mask more efficient, reducing the process times and making the productivity from the expensive bit of equipment greater.

Tibbals: So that's our job, really, as a solder mask manufacturer—to help our customers get maximum throughput out of the machines on the market. The equipment guys have increased the power and the wavelengths, and that's good, but we've still got to optimize the material to reduce the exposure time as much as possible. And by doing that, we are enabling our customers to go direct image with solder mask, and that, as we said, is solving the registration and feature size problem for the sake of those manufacturers, providing they can invest in the equipment.

Matties: This sounds like ultimately, they'll have no choice as the demand for density increases.

Tibbals: Yes, you're absolutely right. We first saw that with dry film. Direct image dry film was introduced quite a few years ago. It gradually took off, and now you see a large number of board manufacturers using direct image exclusively for their dry film processes because it is absolutely necessary. There was no way around it. Solder mask is going the same way in order to achieve good yields and meet the technology demands.

Matties: Are there any equipment suppliers coming to you looking for a recipe for inkjet printing?

Tibbals: Yes, this has been a large part of our R&D focus. We see the next stage for solder mask development as being the inkjet process. Inkjet technology is still under development, currently there is no fully commercialized inkjet solution on the market for volume production but there are several installations you can probably class as beta sites, so we are almost on the cusp.

Antony Earl

Matties: Speed is a big issue.

Tibbals: Yes, it is. First, we should take a step back and appreciate just how much of a challenge inkjet solder mask is. You're starting from scratch. It's completely different from a photoimageable formulation. For a start the viscosity is close to 1000 times lower; it's 100% solid; it must have the required jetting characteristics and the particle size is now measured in nanometers not microns. The challenge is to be able to deliver a solder mask material which has all the required features and still meets or exceeds the essential requirements of end performance and end resistance. This is not a straightforward exercise. This is a project we've already been working on for several years, and I'm happy to report good success.

And there are now several equipment manufacturers who are in the latter stages of development and bringing machines to market. As we all know, there have been false dawns for inkjet application of solder mask but machine manufacturers have come on tremendously in terms of the equipment development. Some of that has been enabled by the ability to process data faster and some from inkjet head technology improvements in the resolution and frequency they can deliver. When you look at how the technology has moved on, it's now at the stage where it is becoming a viable solution.

If you look at inkjet solder mask and relate it back to the problems and challenges we first talked about—miniaturization, smaller feature sizes, registration, etc.—we can see that it has all the features to offer an excellent solution. Inkjet is a digital process; by its very nature it will give us the registration that we require and won't put any ink in the holes. So, with one process, we start to address the two fundamental limitations of traditional photoimageable solder mask. We then throw in all the other benefits: 100% solids, we have no VOC emissions so it's a cleaner process; eliminating three major stages of the current solder mask process and all the energy that's consumed and the wastewater produced, plus the reduction in capital equipment, maintenance and footprint; the list goes on.

For us, inkjet is very much the future for solder mask. It's not necessarily going to be the universal solution next year, but it is coming and will gradually percolate its way through the solder mask production areas. I think it will start off probably with the smaller volume production and will gradually grow in the same way that many years ago photoimageable solder mask took a while to grow, and dry film resist took a while to grow. But, there are so many benefits associated to it that I think there will be a big pull to bring it through.

Matties: In terms of the solder mask process, curing is a large consumer of time. Will inkjet reduce the curing time? And by not using inkjet, the more traditional route, are you doing anything to have something that could be cured say, in half the time? Or, even less?

Tibbals: That's not an area we've studied. The curing time is between 30 and 60 minutes at 300°F. Currently, that would remain about the same for inkjet. One of the reasons why the time is longer is because ovens take time to ramp up to temperature, particularly if it's a conveyorized process; there's a ramp-up stage and a cool-down stage. So, we have to make sure that we've got adequate time at temperature. It is also critical that we ensure the solder mask meets the demands that are required of it now, but also are required of it in the future. And to achieve that, we need to make sure we get the correct crosslinking to take place. If we try and speed that up, the danger is that the solder mask will not hold up well to subsequent chemistry and its field requirements. Trying to reduce the curing time by 15 or 30 minutes becomes a false economy to a certain extent.

Matties: In a meeting earlier, they were reporting times of an hour and 45 minutes from tack dry to final for curing solder masks in a quick-turn environment.

Tibbals: If you look at the solder mask process, then the tack dry time alone could easily be 60 minutes in an oven. With inkjet, we would eliminate that processing time because after jetting, the boards go straight in to final cure.

Matties: You eliminate 50% of the cycle time right there.

Tibbals: Oh, you've eliminated tack dry, you've eliminated exposure and developing. Overall, theoretically you're eliminating at least an hour from the process. But in reality you save a lot more time. With current photoimageable solder mask boards you take them out of the oven but they have to cool down before you can expose them. This means they've got to stabilize in the exposure room. If you're doing conventional exposure, then you've got to make sure the film is ready and then wait for existing jobs to be completed to avoid unnecessary set-up changes. After that, they would need to be developed and probably inspected. So what theoretically could be done in the space of maybe an hour, an hour and a half, generally takes longer than that, even if they're being pushed through for a quick-turn job.

Whereas with inkjet, you jet the two sides, and you go straight to final bake. Then depending on your ramp-up and your final cure oven, one hour to one and a half hours later, you've got a finished solder mask process. Some of the inkjet machine suppliers are talking about solder mask and legend being done in the same process. This allows even more cycle time to be taken out of the process.

Matties: You'd think there would be a lot more demand for this technology to be online sooner than later, with the benefits you're describing.

Figure 2: Left, Clive Landells with Chris Wall: screen print solder mask R&D in the 1980s; right, Clive Landells: inkjet solder mask R&D in 2018. A lot more has changed than the hair colour!

Tibbals: Absolutely. We're basically pushing against an open door with this technology but customers need to be satisfied with the functionality and reliability and be sure that it's ready before they invest. This will be the first step change in solder mask technology in over 30 years. The last step change in technology was when we went from two-pack epoxy to photoimageable. You could argue that we've been through significant iterations of photoimageable. Originally, we were solvent developing and we went from solvent developing to aqueous developing, and then we went from conventional imaging to direct imaging. However, in many ways these are all variations of the same theme. They're different flavors of ice cream. Whereas now, by going to inkjet, it's a completely different change in approach and methodology.

With that comes some reservation. Obviously, the material has to meet all of the required standards and it must be fully accepted and approved by the end user. The PCB manufacturer must then invest in the capital equipment.

Matties: Sounds like a few bridges need to be built.

Tibbals: Exactly, but the drivers for change are there. I think over the coming years we'll see this process being gradually rolled out. Like anything, it will reach a point of critical mass, it will be accepted as mainstream and people will move quickly.

Goldman: One interesting benefit of inkjet is that you don't have to put down as much because there are many areas of a board that really don't need solder mask on them, that get solder mask put on them because they get blanket-covered and it's left on. That's part of the education of OEMs and those that do the specifying—there may be big areas that don't need anything at all. There could be a huge savings in material and time.

Earl: You're right there. The application of the photoimageable solder masks does put the solder mask all over the surface. With the inkjet and the data handling of that process, you can apply different thicknesses of material in different areas of the panel. So, you could be putting the solder mask thicker in critical areas, and thinner in non-critical areas, such as over laminate areas, etc.

Goldman: Or not at all, right?

Earl: Or not at all. We would hate to see that happen, for obvious reasons [laughs], but that's going to be one of the acceptance criteria of inkjet solder mask: the appearance. It is going to be different cosmetically to the eye, how it looks on the panel and how the panel appears under magnification. Everybody is used to very straight lines and straight features now, from photoimageable and direct image solder masks, and inkjet solder mask is going to look a little different.

Tibbals: I think it comes back to this acceptance. Again, you've got this difference: The PCB manufacturer who may see the benefits and would want to move, but we also need to get the end user buy-in as well, because as Antony said, it will look different. Both to the eye and particularly under magnification, it will look very different. Doesn't mean it won't fit the purpose; it just looks different and it deals with things in a different way.

Ultimately, you'll probably find that people will stop putting solder mask on certain areas of the board. And there could be huge material reductions from that perspective, and as such significant cost and environmental savings. I think that could be the driver to catch the larger OEMs' eyes. From a capital investment perspective, not putting ink down where it isn't needed could possibly double output from an inkjet machine which may make justification and return on investment faster. It's just with any new technology, it's got to find its feet and people will start to exploit it to the maximum benefit possible.

Goldman: Is dry film solder mask gone? Do some people still use it?

Tibbals: I think there are still a few people who

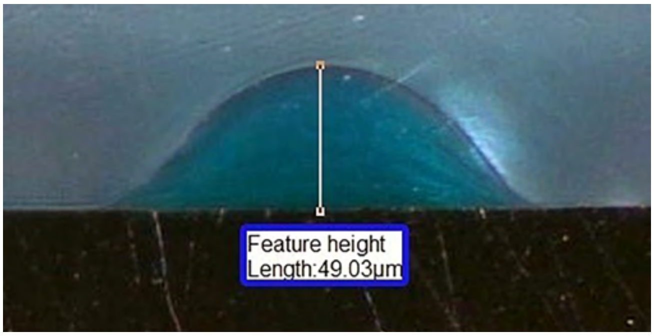


Figure 3: Solder dam profile (50 microns high) of inkjet solder mask.

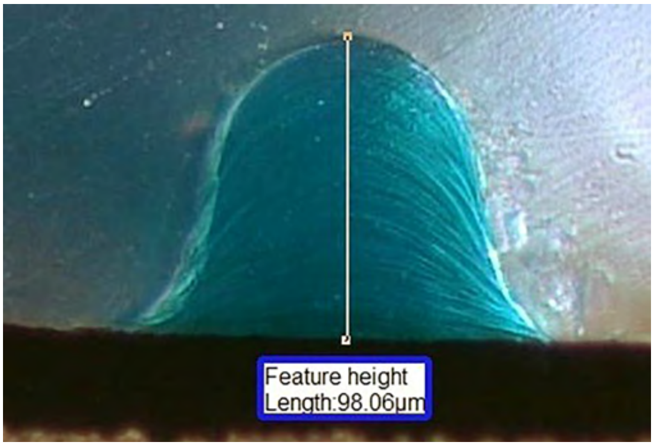


Figure 4: Solder dam profile (100 microns high) of inkjet solder mask.

use it for certain part numbers where they can't approve anything else, but it's a legacy material. There are probably a few military applications where it can't be changed.

Goldman: Are OEMs still so picky on the final look of solder mask? I mean matte versus shiny, semi-matte, colors?

Tibbals: Very much so, which always amazes me, particularly when it's buried in a box and no one sees it! In all seriousness, we fully appreciate there are some technical reasons why OEMs may favor gloss over matte or specify particular colours. Inspecting gloss boards is hard on the eye. Inspecting matte boards is generally much easier. Likewise, sometimes digital alignment cameras can get tricked by different finishes on the board and fluxes or conformal coatings interact differently with glosses or mattes. From a functional perspective, I think there's a fairly well-agreed understanding that matte solder masks are less prone to solder-balling during assembly compared to a high gloss. So clearly there are some technical reasons why the finish may be important.

The same is true for colour. White solder mask is functional because it's used on lighting boards for good reflection and black solder mask is functional because the OEM may want to avoid reflection or hide track work. So these colours are formulated to meet the particular refection requirements stipulated by the OEM. Although not functional, some end users use certain colors for different revisions of PCB or to identify prototypes. Over the years we have made pretty much every color under the sun. We probably get half a dozen inquiries a year where people will give us a Pantone number and they'll want the solder mask made to that because it matches their logo. Commercially, it makes little sense, but we do it because we keep our customer happy, they keep their customer happy, and the world goes 'round.

Green is the historical norm. We have dark green and light green. Light green is better for a solder mask because it makes it faster exposing, but people don't often like light green because if they have some stains or marks on the copper circuitry it shows through, whereas a dark green hides a multitude of things on the board. So the standard tends to be a dark green. Traditionally this was always a dark green matte or semi-matte in the U.S. and within Europe. But boards coming in from Asia tended to be a lot glossier and because companies are now trading boards as much as they're manufacturing, they want some level of consistency in what they sell. As a result, it's now becoming quite common for European and U.S. manufacturers to want to use a gloss as well.

Goldman: What are the major methods of coating, and is there very much curtain coating going on?

Earl: The majority of application is by screen, whether that's manual, semi-automatic, or with the double-sided coaters. Spray is another alternative. We see less and less curtain coat-

ing nowadays. The technology itself seems to have reached its limitations, really, with track heights and feature sizes. There are advantages and disadvantages. As we said before, no actual application method is the be all and end all.

Tibbals: The other area we have to consider is the future demands of solder mask regardless of application method. It doesn't really matter whether its screen, spray, curtain-coat, conventional image, direct image, or inkjet, there are increased demands being put on a solder mask's final properties. If we look at the requirements of a solder mask, we see those becoming more demanding in two or three key areas.

Thermal storage and thermal cycling requirements of a solder mask are increasing. Traditionally, thermal storage may have been done at 140°C or 150°C, but we're now seeing demands up to 170°C or even higher. And, we have traditionally seen cycling being done from -40° to +140°C; more recently this went to +150°C, and we are now seeing those temperatures again increase to 160–170°C.

This is all down to reliability. One of the drivers for it is automotive, because of the increase in hybrid and electric vehicles. These are putting more demands on the circuit boards due to the high level of electronics within the vehicle and a consideration that a device may be in a prolonged "on" state, unlike an internal combustion engine where the vehicle is turned off. What we're seeing is the automotive manufacturers and automotive component manufacturers put in increasing reliability demands on circuit boards and in turn the solder mask which is on those circuits.

When we test materials now, we're not just testing them to the established test procedures. We're looking to test them to beyond where we believe those tests are going in the future. If we want to bring an inkjet solder mask to the market, there's no point in us testing it and formulating it so it passes thermal storage conditions 'A', when in two years' time it's going to need to meet thermal storage conditions 'B'.

So there are the thermal conditions and demands, and there is the dielectric properties of the material. With the implementation of 5G technology and the Internet of Things, connectivity is going to be the major factor going forward. To facilitate that, we need to have higher and higher frequency boards and therefore the dielectric properties of the board itself needs to meet those requirements. And the solder mask starts to become an important factor in that. When we're formulating materials, we have to be aware that we're meeting these future dielectric demands, whether that be dielectric constants, dissipation factors, these are things we have to take into account when we're formulating materials. That's our job as a solder mask manufacturer, to make sure that we're delivering materials that meet these future demands, not just next year, but years in to the future as circuit board technology starts to move on.

Happy Holden: Getting back to Patty's question, over the years, solder masks have taken on a lot of roles other than resisting solder because of rugged durability and epoxies and things. I wonder if in order to get things like inkjet printing, people have to go back to saying, "This is only suitable to resist solder. It doesn't do all these other characteristics of solder masks you've been getting for free."

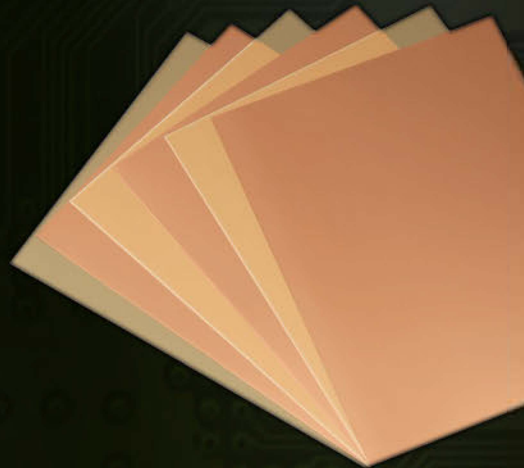
Tibbals: I think the problem is that Pandora's Box is open and no one ever wants to go backwards in terms of capability, so I think from our perspective it would be a very hard sell to get somebody to accept that a solder mask is only a mask that withstands solder. Resisting solder is not the challenge, in many ways that's the easy part. They key is formulating a universally acceptable product that withstands electroless nickel gold or immersion tin, gives the required moisture and insulation resistance values, has good dielectric strength properties, and all of these other things that don't necessarily form part of one particular published standard but are all day-to-day things which the solder mask has to do.

Solder, whether leaded or lead-free is actually quite a small part of the solder mask's job now, it's quite possible that solder mask goes through its whole life and never sees any solder, unless the solder paste spreads on top of it a little bit. It

Panasonic

Ultra-low Transmission Loss Multi-layer Circuit Board Materials for ICT Infrastructure Equipment

MEGTRON7



Laminate **R-5785(N)** | H-VLP2 Cu and Low Dk
Prepreg **R-5680(N)** | Glass-Cloth Specifications

Features

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

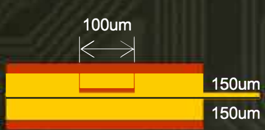
	Dk	Df
R-5785(N) + H-VLP2 Cu	3.4	0.002 @ 12GHz
R-5775	3.6	0.004 @ 12GHz

Applications

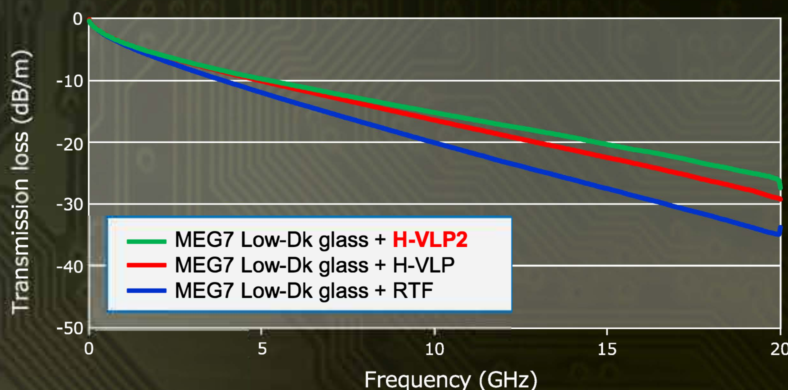
- High-end servers, High-end routers, Supercomputers, and other ICT infrastructure equipment, Antenna (Base station, Automotive millimeter-wave radar), etc.

Trasmission Loss

- Evaluation Sample



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



Contact us

More information

may not see any hot air leveling solder or wave solder, but it's still called a solder mask.

Holden: That's what I mean. The solder mask does a lot more than just restrict solder now. Like you said, it's morphed into having all these other properties.

Matties: One final question from me. What advice would you give a fabricator regarding solder mask, if there's any open-ended advice advice that you might offer?

Tibbals: My two would be equipment maintenance and process control.

Earl: I second that. Pre-clean for solder mask is a key process to give you the best adhesion capability that you can get from your solder mask. And as Shaun said, process control and equipment maintenance—control those processes, maintain those processes, and they help to look after themselves.

Matties: Those were pretty quick answers. Is this a common problem out there?

Tibbals: It would be inappropriate for us to walk in to a customer and say, "This piece of equipment is no good, and you need to change this." Even though we may think that, in reality that customer may not be able to invest in new equipment for each of those processes. Our job then is to make sure that they get the best possible out of that equipment. And it is amazing what you can get out of a process provided you maintain a piece of equipment, you control the parameters, and have good monitoring systems. So that's really number one for us.

If somebody were coming to us with a blank check and a clean sheet of paper and said, "How do I want to lay out my department and what equipment should I put in?" that would be a different answer. We could sit down and work with a customer within his budget to design the ideal solder mask department. But, I'd never known that happen. It's a case of making sure we get the most out of what they've got. A lot can be achieved with training, maintenance, and process control.

Matties: It sounds like process discipline.

Tibbals: It is. As I answer this question, other factors come to mind. For years now, solder mask dams have been getting smaller. And that's because you need the registration tolerance. The solder dam itself doesn't need to be smaller; you just need to make it smaller so you can allow for movement within the board, film and exposure process. If somebody's gone to direct image, and they've taken a part number which had a 2-mil dam on it because it was being exposed through a piece of film, and now they're doing it with a direct imager, that dam no longer needs to be a 2-mil dam. That can go back to being a 4-mil dam, or possibly even a 5-mil dam because the direct image is going to put it exactly where you want it. It's not going to encroach on the pad.

For years now, solder mask dams have been getting smaller.

So what does that do? That gives you more process window for your exposure time. It gives you more process window for your developing conditions. Again, it's trying to encourage customers to re-think, when they put in a new process, what does that enable them to adjust and adapt which might have an indirect effect of widening your process elsewhere? And those are conversations we have with our customers on a regular basis. Sometimes they can't do that because the end user won't allow them.

Matties: Well, this has been very informative, you guys. Thank you so much.

Goldman: Yes, thank you, we appreciate your time.

Tibbals: Thank you.

Earl: Thank you. PCB007

PCB007 Supplier Highlights

Gardien Japan Enhances Test Capabilities with Quad Density Tester ▶

Gardien has announced the addition of the Kaima GUQMM Quad Density Tester, at its Tokyo Service Centre. No machines bigger than this Quad Density Tester are currently operational in the Japanese market.

Panasonic Develops Glass Composite Material that Improves Reliability ▶

Panasonic's new glass composite circuit board material improves the mounting reliability of electronic circuit board parts, thereby contributing to the long-term stable operation of automotive and industrial equipment.

One World, One Industry: Meet IPC's Vice President of Global Government Relations, Chris Mitchell ▶

In this new role, Chris will work closely with IPC's Government Relations Steering Committee and its government relations team based in Washington, D.C. Chris will represent IPC and the electronics manufacturing industry before key policymakers globally.

EPTE Newsletter: JPCA SHOW 2018 ▶

One of the largest exhibitions for the printed circuit industry was held June 6–8 at Tokyo Big Sight in Japan. The three-day show is held in Tokyo every June and attracts a worldwide audience.

It's Only Common Sense: Keeping Up with the New World Order ▶

During a normal commercial flight, a pilot only touches the controls of the plane for less than three minutes. The rest of the time, the computer called "autopilot" is actually flying the plane—including landing the plane. Pilots have to undergo periodic training so they will not forget how to fly. How many phone numbers, including your own, do you know by heart? Or do you use smartphone settings to remember them for you?

Panasonic Develops Ultra-Low-Loss PCB Material for Packages and Modules ▶

Panasonic's ultra-low transmission loss circuit board material for semiconductor packages and modules allows the stable operation of semiconductor devices that process enormous data at high speed.

Matrix Materials Support Ultra-Long Flexible Circuits Applications ▶

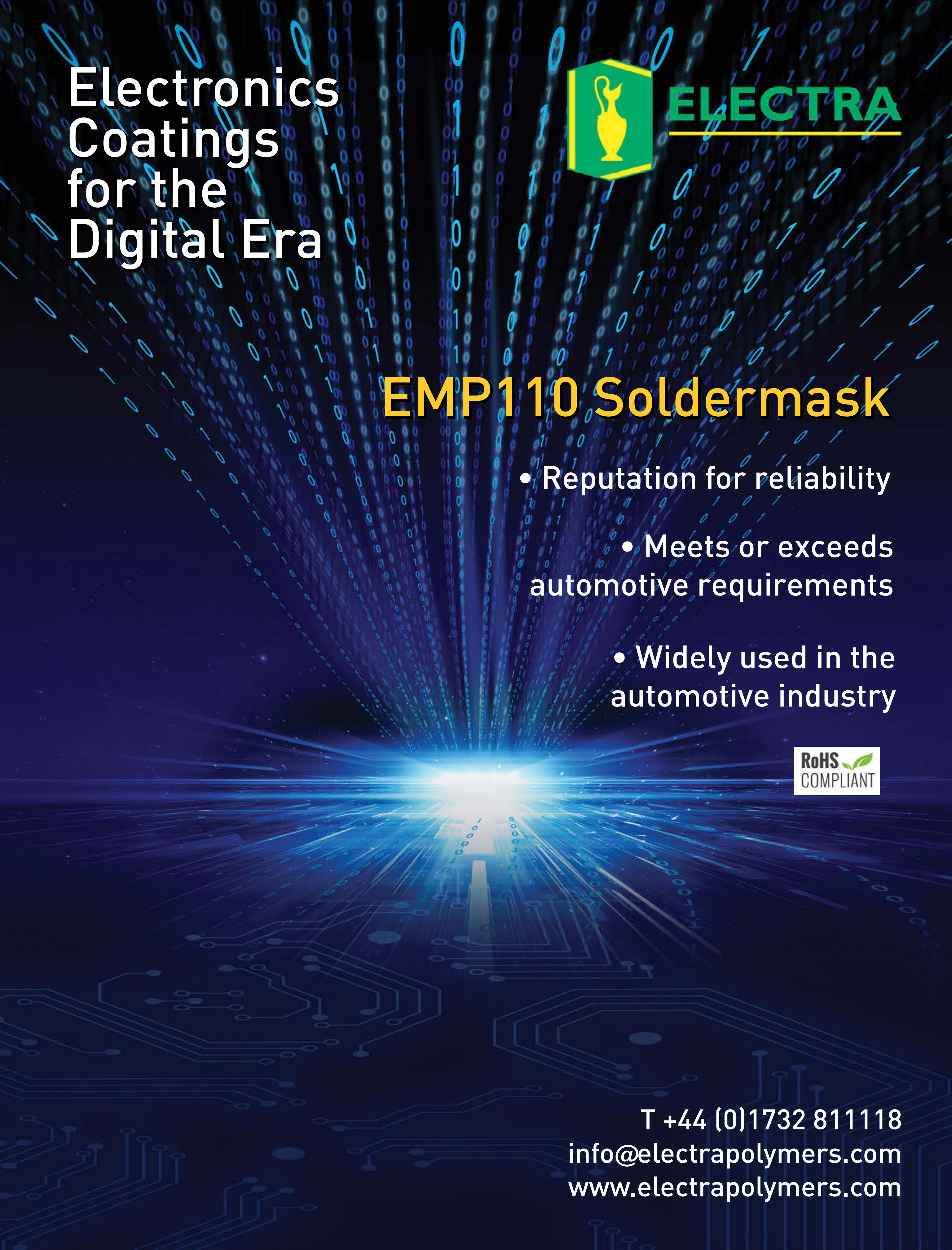
The market for long flexible printed circuits continues to grow rapidly. The performance and high reliability of flexible circuits over traditional hard wiring and cabling continues to expand into new applications in medical equipment, automotive, aerospace and defense applications.

Trouble in Your Tank: Looking at PTH Voids ▶

One of the areas often overlooked as a cause of plating voids is a discontinuity in the electroless copper deposit. However, what if the panels were void-free after electroless copper only to void later in the manufacturing process?

EPTE Newsletter: The Digital Camera Industry is Disappearing ▶

Digital camera manufacturers continue to decrease their manufacturing space due to the steady decline in camera sales. Olympus, one of the major camera manufacturers in Japan, closed their compact camera manufacturing plant in China. Olympus plans to focus on more profitable segments in their portfolio.


Electronics Coatings for the Digital Era
ELECTRA
EMP110 Soldermask
• Reputation for reliability
• Meets or exceeds automotive requirements
• Widely used in the automotive industry
RoHS COMPLIANT
T +44 (0)1732 811118
info@electrapolymers.com
www.electrapolymers.com

Countering Solder Mask Residue Production Concerns

Feature by Rick Nichols, Sandra Heinemann, Gustavo Ramos, Dr. Lars Nothdurft, Dr. Britta Schafsteller and Hubertus Mertens
ATOTECH DEUTSCHLAND GMBH, BERLIN, GERMANY

Abstract

The symbiotic relationship between solder masks and selective finishes is not new.

The solder mask application is one of the key considerations to ensure a successful application of a selective finish. The selective finish is the final chemical step of the PCB manufacturing process; this is when the panels are at their most valuable and are unfortunately not re-workable. Imperfections are not tolerated, even if they are wholly cosmetic. Quality issues often manifest themselves in the form of a 'ping pong' conversation between the fabricators, the solder mask suppliers and the selective finish suppliers. Without tangible evidence these discussions are difficult to resolve, and the selective finish process is usually regarded as responsible.

Solder masks identified as 'critical' in the field have been tested using state-of-the-art technology to assess whether performance markers could be found. This article will focus on the chemical characteristics and use them to predict or identify potential issues before they occur rather than specifically name 'critical' solder masks. It is also the intention of this article to address the potential of a solder mask to react to common yield improvement practices such as UV bumping. It is hoped that this awareness will help fabricators to ensure maximum yields by asking the right questions.

Critical solder masks impact all selective finishes. In this article, immersion tin will be used to highlight the relationship between critical solder masks and some of the issues seen in the field. The article will conclude with a novel approach to identify redeposited volatiles post reflow.

Background

The function of a solder mask is to provide protection to the active circuit and definition for the selective finish. These fundamental attributes include photosensitivity for lithographic imaging and sufficient production

environment resistance to protect the active circuit. These functions are simple in principle, but the resultant implications can be far reaching. The impacts that will be focused on in this article will be surface imperfections and where relevant, immersion tin will be used as an example.

The hazards of a critical solder mask are primarily two-fold (excluding developing residues): Leach-out into process chemistries and solderability issues during multiple assembly steps.

Solder masks were originally designed to facilitate hot air solder leveling (HASL). The need for co-planarity has seen the introduction of chemical selective finishes such as ENIG, ENEPIG and immersion tin to name a few. This change in application has obvious implications attached. Whereas HASL is a hot solder shock for 10 seconds (approximately 260°C for eutectic solders and 280°C for lead-free solders), the wet chemical selective finishes introduce longer dwell times in harsh chemical environments, often at high temperatures. In an ideal world, the selection of the solder mask would be optimally coupled with the selective finish employed. In the real world, however, the cost factor is prevalent as a selection tool. To counter this notion, an attempt will be made in this article to raise awareness of the components within a solder mask that render it potentially 'critical' to production.

It is equally perplexing that in this day of high-tech six sigma process control, the quality of the solder mask process is still judged by weight gain, Stouffer wedges, breakpoint tests and the temperature of the baking ovens. Whilst all these controls ensure a good coating, they provide no information on the status of curing or the potential for volatile release during selective finish processing or assembly.

It must be stressed at this stage that solder masks fulfill a fundamental part of circuit manufacture and that, in the main, this is achieved very successfully. The virtual eradication of solder mask residues after developing is testament to the advancement in the imaging arena. For this article we will assume that resist residues have been totally eradicated and focus on the impact of leaching. Leaching of the solder mask into the chemistry has been shown to have lifetime and quality impacts. This is true of all the mainstream selective finishes.

This article will identify solder mask constituents that can be critical to production using state-of-the-art analysis equipment and demonstrate that traditional industry fixes such as UV bumping, if adopted, have little impact on eradicating the potential risks to yield. The solder masks selected as 'critical' are based on field experience.

The Identification of Potentially Process-Critical Compounds in Solder Masks

Methodology

For this investigation eight samples were selected based on field experience and four samples are taken from 'critical' customer panels (identified as A, B, C and D). These are described in Table 1.

Sample ID	Description	'Critical' Y/N
1	FR-4 (Reference)	N/A
2	Light Green, Gloss	N
3	Dark Green, Gloss	Y
4	Black, Matte	N
5	Dark Blue, Gloss	N
6	Black, Matte	Y
7	Grey/Black, Matte	N
8	Green, Matte	Y
A	Customer	Y
B	Customer	Y
C	Customer	Y
D	Customer	Y

Table 1: A description of the test population.

The analysis methods employed, as listed in Figure 1 are:

- Elemental analysis (CHNS): Quantification method for the chemical composition of carbon, hydrogen, nitrogen and sulphur
- Fourier Transform infrared spectroscopy (FTIR): A fingerprint/identification of functional groups

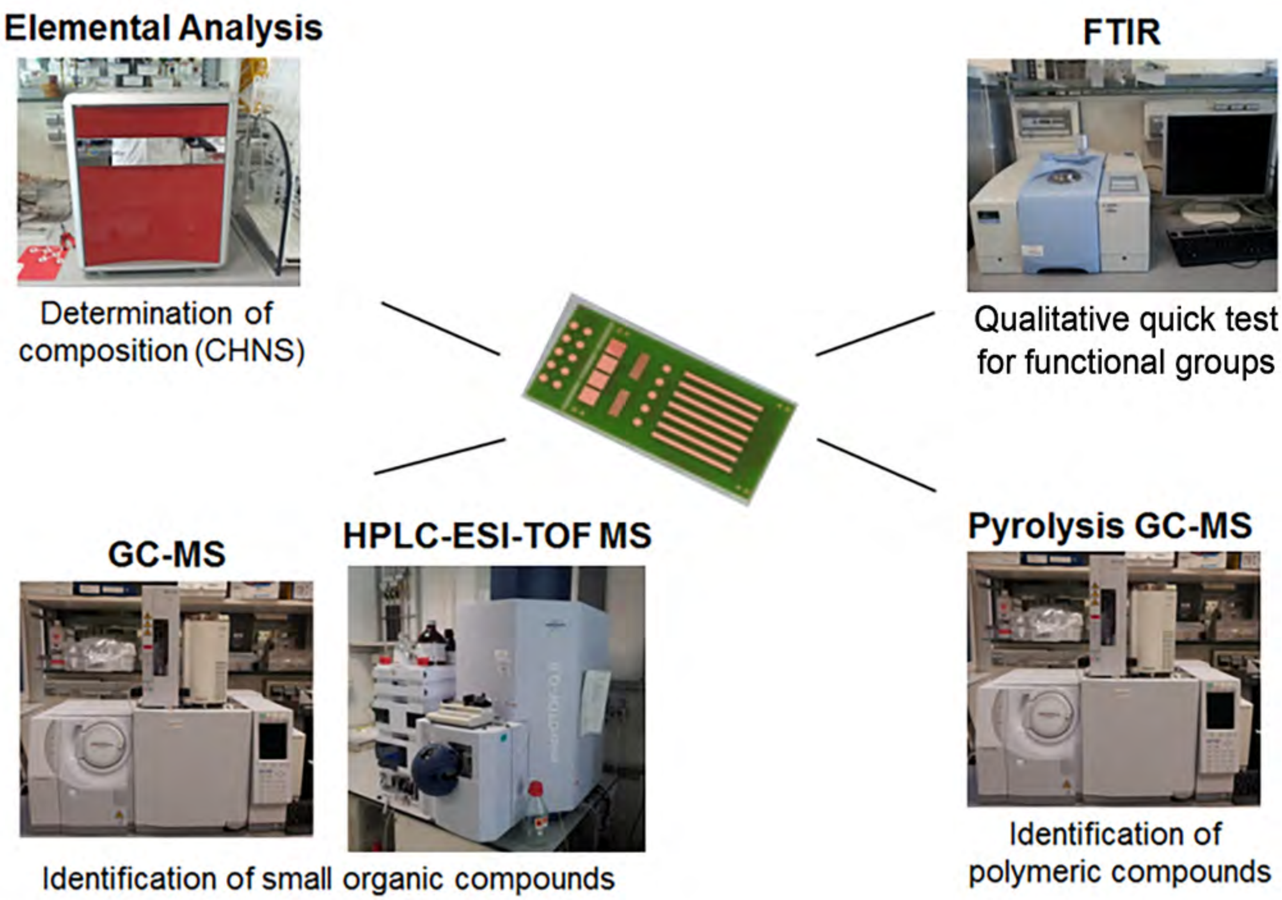


Figure 1: Demonstration of the state-of-the-art analysis applied.

- Leaching experiments: Extracts were analyzed by GC-MS and HPLC-MS—identification of leach-out products.
- Pyrolysis: GC-MS—identification of high molecular weight compounds

Elemental Analysis

This is a destructive technique that identifies the basic chemical characteristics by quantitively isolating the elements carbon, hydrogen, nitrogen and sulphur (CHNS). This method examines the properties of the solder mask but can only be used as an early indicator. Although it is apparent that there is a discrepancy between the critical and non-critical solder masks, this method cannot determine the impact of this discrepancy on the plating processes.

Figure 2 indicates that the presence of sulphur is the key difference between the critical and non-critical solder masks. This is better demonstrated in Figure 3.

Whilst there is most likely a rational explanation for the presence of sulphur, the question that should be considered is whether weight percentages over 3.0 are required and whether an alternative exists. This is beyond the expertise of our facility and as such will not be further extrapolated upon.

Fourier Transform Infrared Spectroscopy (FTIR)

FTIR is a non-destructive technique that could be applied onsite. It is a quick test requiring no preparation resulting in a wave number according to the molecules' ability to absorb infrared light. This method can be used to identify the functional groups. Figure 4 demonstrates that at wave numbers of between 1075 and 1077 cm^{-1} a signal can be observed for the critical sample. From the elemental analysis it has already been established that sample ID 3 has a sulphur weight percentage of 3.7%. The wave number at 1075 cm^{-1} probably indicates the presence of sulphur. This method

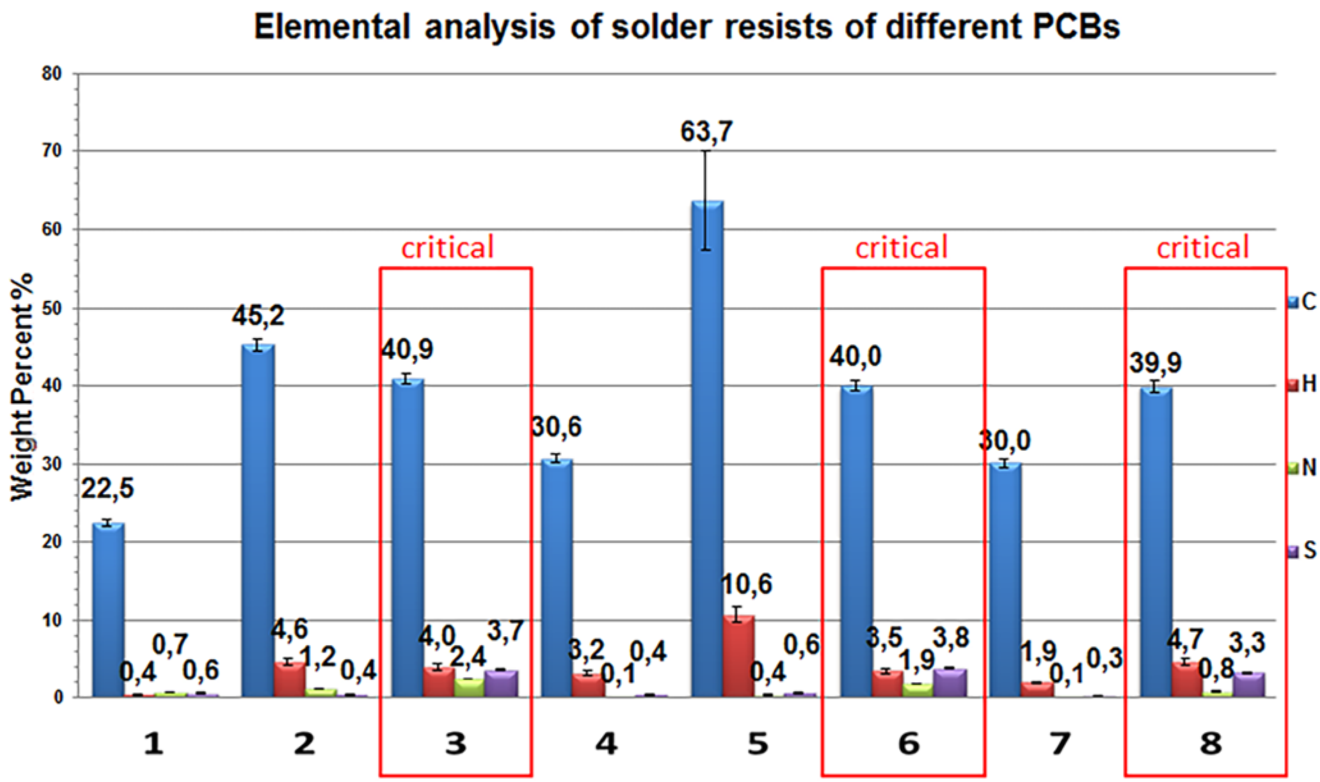


Figure 2: The elemental analysis of the test population.

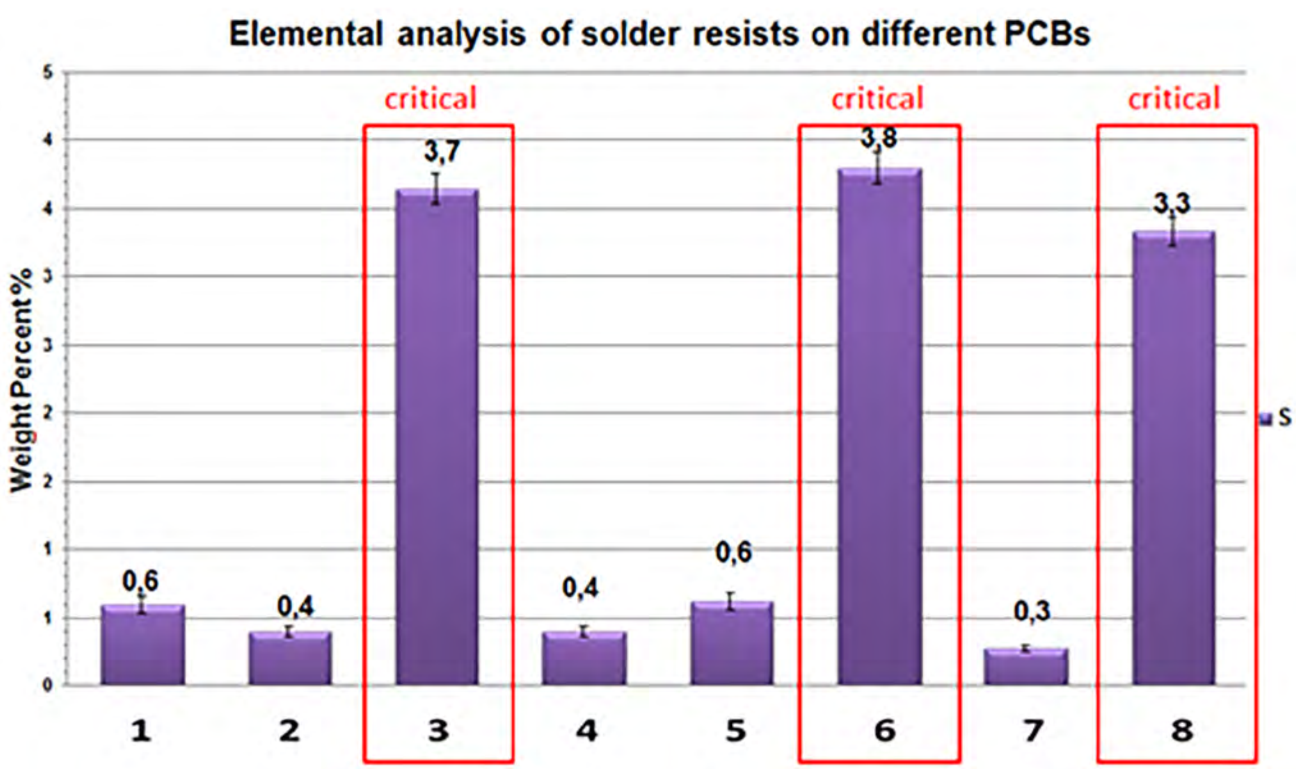


Figure 3: The elemental analysis of the test population focusing on the sulphur content.


PASSPORT

also indicates a discrepancy between the critical and non-critical solder mask. Due to the complexity of the solder mask chemistry, this method may be used as a fingerprinting technique to direct further analysis.

Figure 4 compares a cross-section of the population whilst Figure 5 focuses on the comparison of the critical population members. It can be demonstrated that the critical solder masks exhibit similar absorption signals at the almost

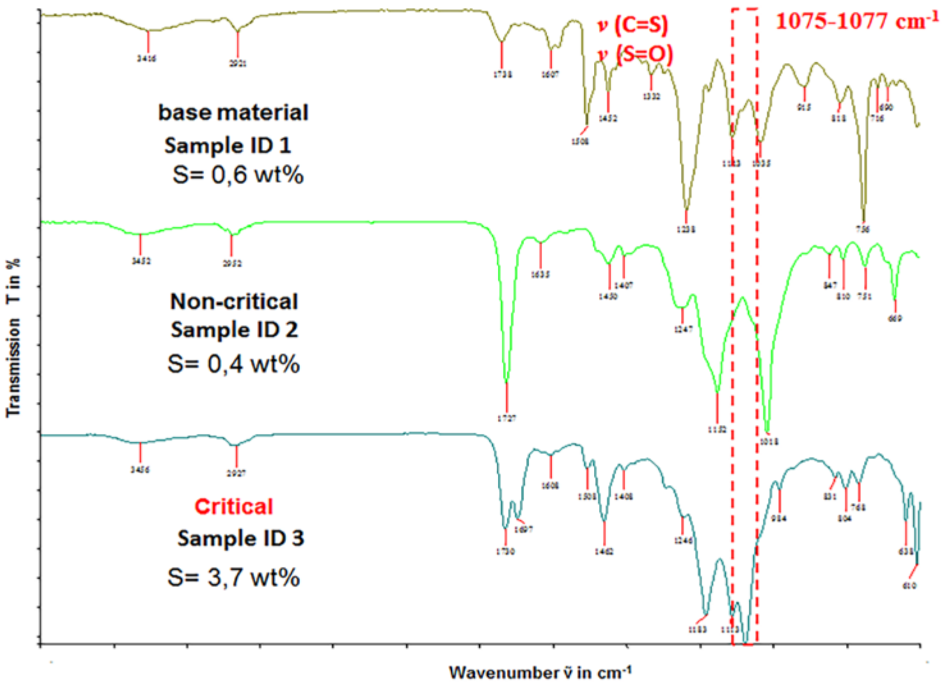


Figure 4: Comparison of critical solder mask to non-critical and reference.

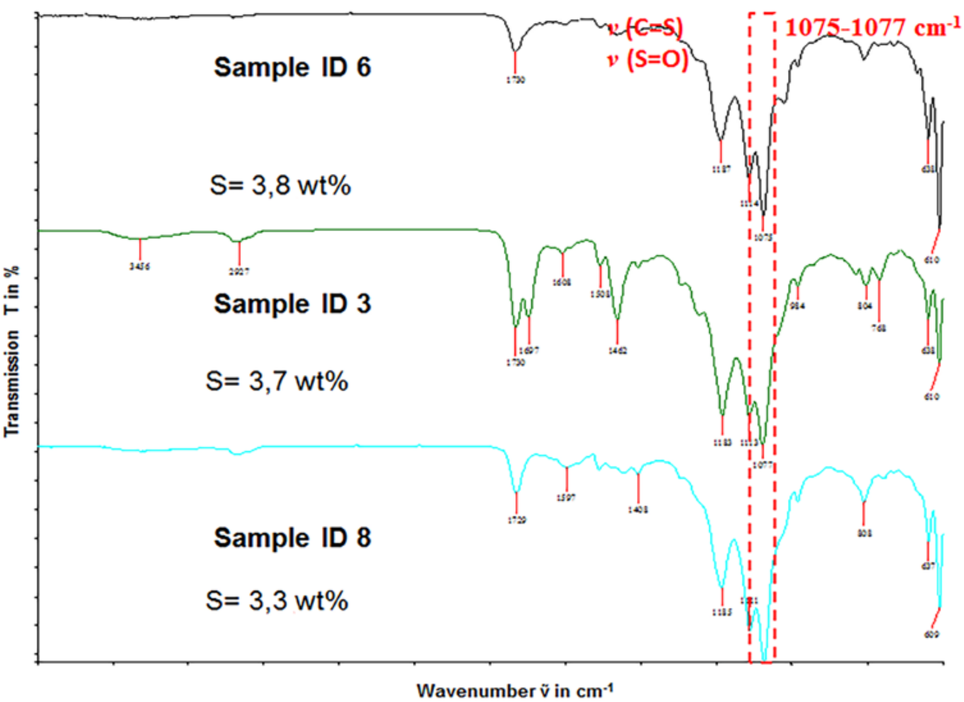


Figure 5: Similarities of all critical solder masks.


Excellon
Manufacturers of precision Micro-Machining Solutions for over 50 Years
COBRA II
Hybrid laser system
689
6-station precision drilling and routing system
HS-1L DS
Single-station, dual spindle high-speed system
136L
High accuracy and speed
154L
Large panel applications
HS-2L
2-station, high-speed
129
Single-station precision system
Move into the future with the latest innovations in Fabrication Technology from the Industry Leader!
www.excellon.com

the same wavenumbers. This method has the potential to be a performance indicator.

Leaching Experiments

This is a simulation of the potential of a solder mask to leach into the plating chemistry. Acetonitrile, (ACN), is used as a solvent. The samples are exposed to the solvent to extract the potential leachants. The resulting extracts were analyzed by GC-MS and HPLC-ESI-TOF MS. In this work, only the results of the HPLC-ESI-TOF MS are shown. Therefore, the compounds of the extracts were separated by liquid chromatography and analyzed by mass spectrometry. The resulting high-resolution mass spectra were used for the calculation of sum formula, which allows a prediction of structures which can be compared with reference substances. Hence, it is possible to identify additives like photoinitiators, cross-linkers, flame retardants and other plastic additives. Figures 6 and 7 show the comparison of the signals of the photoinitiators.

Sample	Sulphur (EA)
A	6.4%
B	3.3%
C	2.8%
D	2.7%

Table 2: Elemental analysis of critical customer samples A, B, C and D.

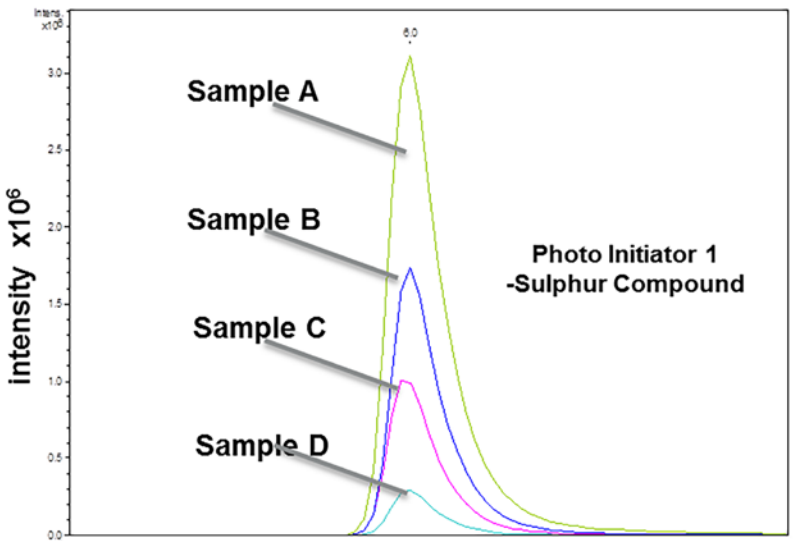


Figure 6: Extracted ion chromatogram of photoinitiator 1.

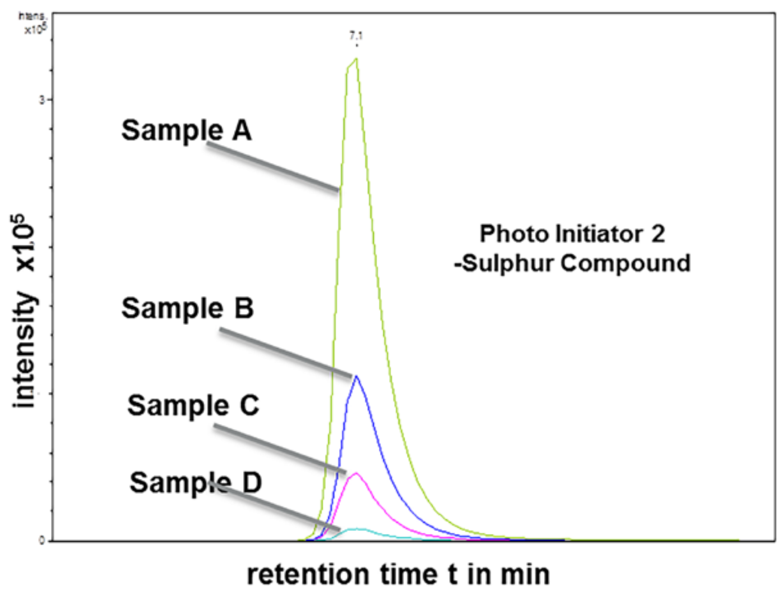


Figure 7: Extracted ion chromatogram of photoinitiator 2.

For a better comparison, the extracted ion chromatograms are used. Therefore, the desired mass was extracted from the whole mass spectrometric data. In comparison the samples show differences in their signal intensities of photoinitiators. It should be noted that the identified photo-initiators contain sulphur. Table 2 demonstrates the correlation between sulphur in the elemental analysis and the response of the mass signals. In general, samples with a high sulphur content show intense signals of leaching products. This is especially true for the sulphur-containing photoinitiators and is demonstrated in Figures 6 and 7.

Figures 6 and 7 along with Table 2 show the correlation between sulphur in the elemental analysis and the response of the extracted ion chromatogram.

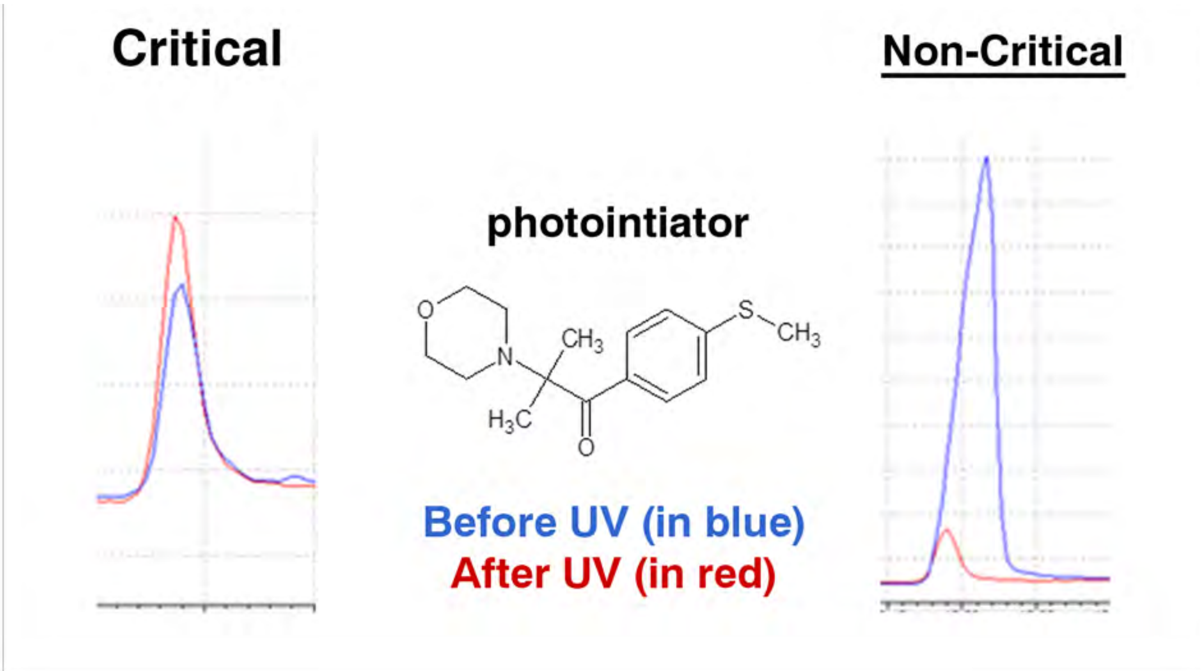


Figure 8: GC-MS results for critical and non-critical solder masks before and after UV bumping.

Pyrolysis and GC-MS

A sample of less than 200 µg is scraped from the material to be examined. This is then exposed to temperatures of 550°C. The impact of this is to fragment the molecules. The fragmented molecules are then separated by gas chromatography and subsequently identified by mass spectrometry. This method can be used to identify the polymeric compounds of solder mask. Due to the sensitive nature of the industry, response charts will not be included in this article as they may infringe upon confidential know-how. The takeaway message, however, is that, whilst the signal responses for critical solder masks are similar, they differ from those of non-critical solder masks.

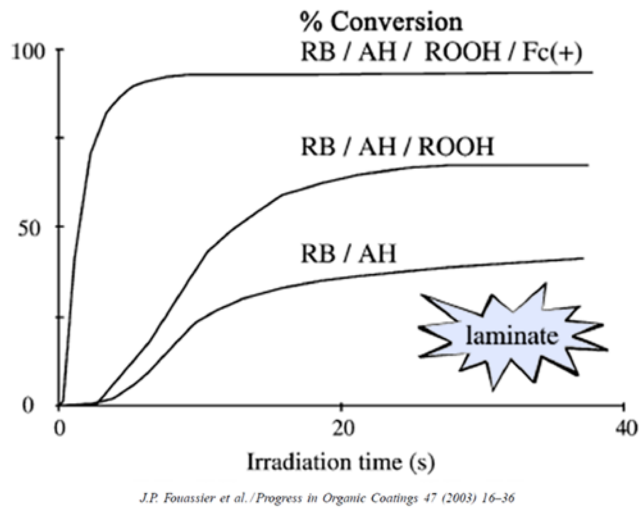


Figure 9: A representation of polymerization[1].

The Impact of Traditional Yield-Hiking Tricks on Critical Solder Masks

A typical trick within the industry to increase yields is to use ultraviolet (UV) bumping. This is a pre-selective finish treatment aimed to completely harden or polymerize the solder mask and prevent release of volatiles during the selective finish process or reflow process.

Figure 8 demonstrates that the signal is the same for the critical solder mask before and after UV bumping. This is an indication that the solder mask is not polymerized or that there is too much photoinitiator in the solder mask matrix.

Figure 9 demonstrates that percent conversion (polymerization) is related to irradiation time. The labels on the curves refer to three different monomer systems. The best situation is full polymerization in a short irradiation time(s). The figure also demonstrates that

Solder Mask Type	Post Cure @ (150°C minimum)	UV Energy (mj/cm²)
A, B, C & D	60	0
		1500
		2000
		2500
		3000
A, B, C & D	90	0
		1500
		2000
		2500
		3000

Table 3: Test matrix.

it is possible that full polymerization can never be achieved. This is usually the case if there is too much photoinitiator in the matrix. This excess of photoinitiators has the potential to result in outgassing volatiles during reflow.

Relating Theory to Reality

In practice, critical solder mask types can be potentially responsible for soldering issues. To demonstrate this, a test was performed at a PCB facility. Four solder mask types were used in the test and the post solder mask bake was varied in terms of time, and the UV energy for the solder bump was varied (refer to the test matrix in Table 3).

The outputs from Table 3 are an evaluation of the solderability in the form of a solder gap test and a measure of the ionic contamination which can be correlated to the volatiles within the solder mask.

Figure 10 is verification that the ionic contamination is a viable representation of solder mask polymerization. The evidence for the solder mask being the dominant cause of ionic contamination is proven by the result for the sample without solder mask, (the column marked as 'none'). As a point of interest, solder mask C also displays low values suggesting complete polymerization.

To evaluate solderability, a test vehicle was selected that simulates wettability. The test vehicle is a tool generated by the chemical supplier responsible for the data generated for this paper. The coupon is shown in Figure 11.

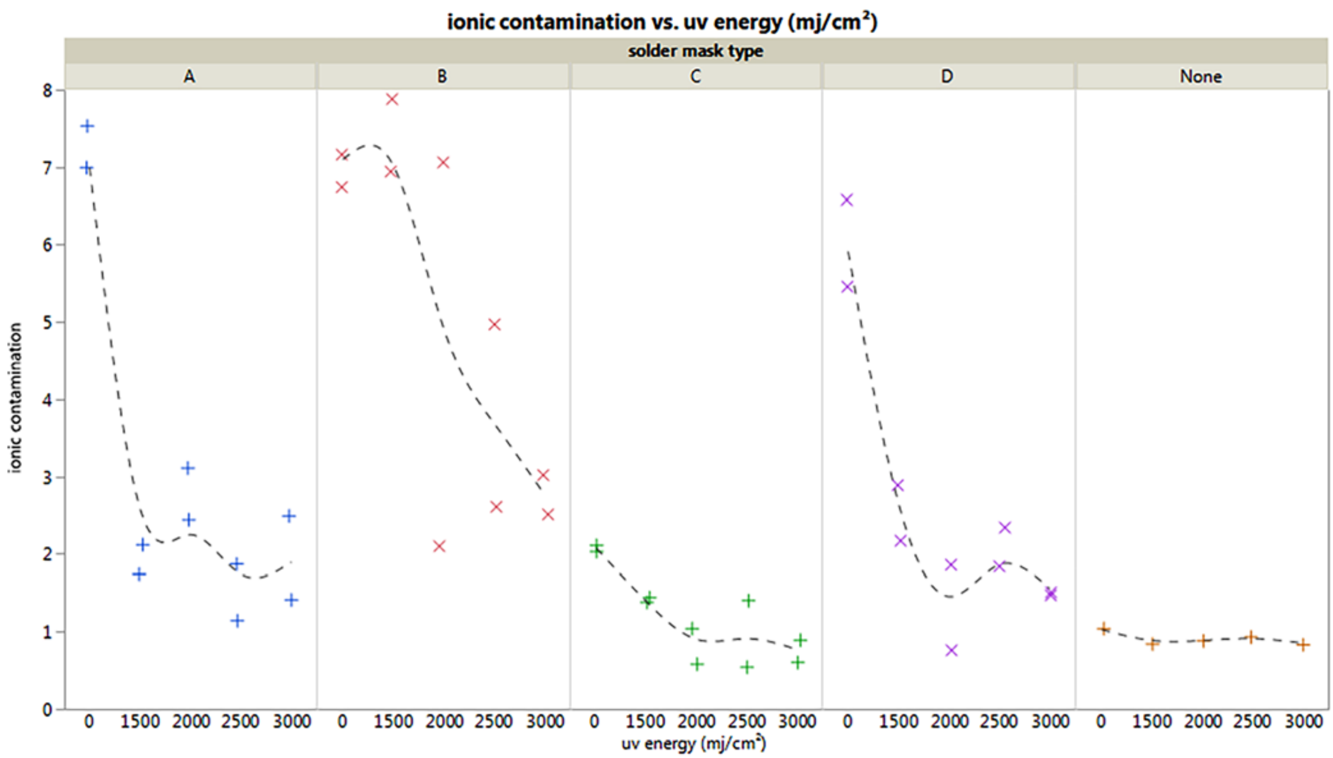


Figure 10: The relationship between solder mask, UV energy, and ionic contamination.

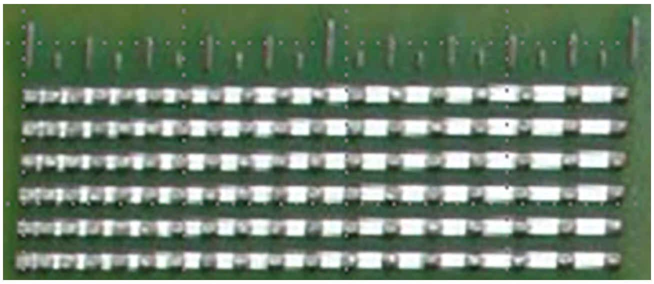


Figure 11: Solder spread test vehicle.

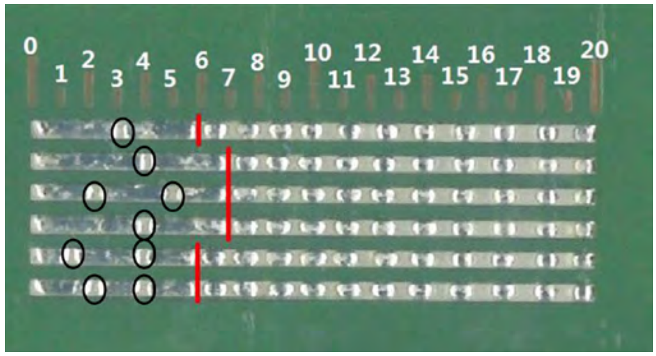


Figure 12: Solder spread test vehicle after reflow.

The mechanism of the test is to print solder paste in a pattern whereby the gap between solder paste deposits increases. When the samples are reflowed, the solder paste will spread to represent solderability.

The test vehicle is a tool that can provide a numerical value. The value is calculated by summing up the maximum spread values, (red line) and subtracting the number of breaks, (black rings) divided by 2. Therefore, the result for Figure 13 is (6 + 7 + 7 + 7 + 6 + 6) – 9/2 = 35.

These values were used to evaluate the impact of simple techniques to enhance production yields.

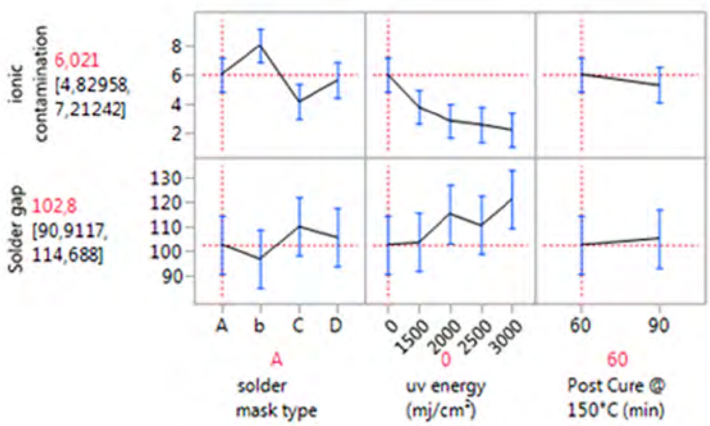
The results show that the ionic contamination is statistically related to the solder mask type and UV energy. It must be stressed that the UV energy is not a setting recommendation, as the measuring techniques and equipment can give significantly varied results, but just to indicate more energy or less energy.

In this test, more UV energy reduces ionic contamination and the impact of the UV energy is also statistically significant regarding the solder gap test. In other words, a UV treatment prior to the immersion tin plating process significantly improves ionic contamination and soldering behavior.

Conclusions

The primary finding of this paper was that sulphur (>3%) can be found in solder masks that were identified as 'critical' and could therefore be used as a marker to highlight them.

The samples tested were supplied from the field and do not represent any supplier specifically. As previously mentioned, it is the intention of this paper to help educate users to ask the correct questions about the solder mask they are using. Whilst UV treatment prior to plating is effective, it can be seen that selecting a suitable solder mask can also impact yield. In the customer-based test, solder mask C had the lowest ionic contamination results and the best solderability results.



Response ionic contamination

Effect Tests

Source	Nparm	DF	Sum of Squares	F Ratio	Prob > F
solder mask type	3	3	77,103860	16,9456	<,0001*
uv energy (mj/cm²)	4	4	76,433315	12,5987	<,0001*
Post Cure @ 150°C (min)	1	1	5,476000	3,6105	0,0668

Response Solder gap

Effect Tests

Source	Nparm	DF	Sum of Squares	F Ratio	Prob > F
solder mask type	3	3	935,0750	2,0640	0,1253
uv energy (mj/cm²)	4	4	1936,3500	3,2057	0,0259*
Post Cure @ 150°C (min)	1	1	70,2250	0,4650	0,5003

Figure 13: The results and significance for ionic contamination and solderability.


UCE
WET PROCESS LINE
Etchers
Scrubbers
Developers
PTH
Oxide Replacement
THIN CORE HANDLING • PC CONTROL • RELIABLE • LOW COST
CLICK TO VIEW OUR UCE WET PROCESS LINE
The leading European PCB Equipment and Parts Supplier
www.vikingtest.com
sales@vikingtest.com
Viking

Further study would be necessary to identify the true mechanisms involved. Although the ionic contamination has been demonstrated to be inversely related to the application of UV energy, it is unclear whether the source of the ionic contamination can be attributed to the solder mask or to the plating solution. It would be acceptable to hypothesize that the increase of SM polymerization simply reduces the absorption of chemistry. Ionic contamination can also be measured for the sample without solder mask shown in Figure 10. **PCB007**

Reference

1. J.P.Fouassier, X.Allonas, D. Burget Photopolymerization reactions under visible lights: principle, mechanisms and examples of applications.

Rick Nichols is product manager for Global Product Team-Selective Finishing (GPT SF)

Sandra Heinemann is assistant product manager, GPT SF

Gustavo Ramos is product director, GPT SF

Dr. Lars Nothdurft is scientist in Organic Analytics, GPT SF (not pictured)

Dr. Britta Schafsteller is R&D manager-Development, GPT SF

Hubertus Mertens is technical manager for Chemistry and Equipment, GPT SF.

Rick Nichols | Sandra Heinemann | Gustavo Ramos | Dr. Britta Schafsteller | Hubertus Mertens

Army's Robotics Alliance Rallies Researchers

As part of a strategy to develop and deliver new robotics capabilities to future Soldiers, Army researchers have partnered with world-renowned experts in industry and academia. The University of Pennsylvania hosted a series of meetings in Philadelphia June 5-7 for principal investigators and researchers from the Army's Robotics Collaborative Technology Alliance, or RCTA.

The group formed in 2009 to bring together' government, industrial and academic institutions to address research and development required to enable the deployment of future military unmanned ground vehicle systems ranging in size from man-portables to ground combat vehicles.

"We are coming together to tell each other what we've done over the last year," said Dr. Stuart Young, a division chief in the U.S. Army Research Laboratory at Adelphi, Maryland, and the RCTA's collaborative alliance manager. Young said the laboratory is focused on transitioning new capabilities to industry partners, so they can continue to mature them.

"Since this is a basic and applied research program, we'll transition it to them, so they can get it into an experimental prototype in development," he said. "Certainly, the problem that we are working on is very hard. It is difficult to operate robots in the wild, anywhere in the world, but that's the kind of problem the Army has to solve."

The Army's vision is to make unmanned systems an integral part of small unit teams.

"We're trying to go from tools to teammates, so you can work side-by-side with them," Young said.

MilAero007 Highlights

AWE: Introducing the Augmented World Expo ▶

Augmented World Expo (AWE USA) is perhaps the largest event for professionals focused on providing the science-fiction-like superpowers through augmented, virtual, mixed reality and associated wearable technology.

Senate Defense Bill Military Electronics Provision Reflects IPC's Priorities ▶

The U.S. defense industrial base—and the electronics industry in particular—would benefit from a provision in a newly released U.S. Senate bill. IPC welcomes the U.S. Senate's version of the National Defense Authorization Act (NDAA).

Catching up with Fineline–Global N. American CEO Eran Navick ▶

After entering the North American marketplace just six months ago, printed circuit board provider Fineline-Global is making its mark. A value-added PCB supplier, Fineline prides itself on being any able to meet any challenge for any company in any part of the world.

Nano Dimension Now Certified U.S. DoD Vendor ▶

As a registered and approved vendor, Nano Dimension is now positioned to pursue and conduct business directly with the U.S. federal government and its many agencies, including the Department of Homeland Security and other United States Department of Defense entities.

Sanmina's Costa Mesa Facility Achieves AS9100D Certification ▶

Sanmina Corporation announced that its technology center in Costa Mesa, California, has earned AS9100D certification for defense and aerospace manufacturing.

President Trump Announces 25% Tariff Imposition on Chinese Imports ▶

President Trump announced that he will impose 25% tariffs on Chinese imports worth roughly $50 billion. The tariffs are the result of an investigation that found China's technology transfer policies had harmed U.S. companies.

NASA Technology Managers Visit TopLine at Space Tech Expo ▶

Members of the NASA Technology Licensing and Commercialization Team visited the TopLine exhibit at the recent 2018 Space Tech Expo show, where they awarded a NASA lapel pin to TopLine CEO Martin Hart.

FTG Inks Long-Term Agreements with Rockwell Collins ▶

The award incorporates a variety of technologies for use on major airframe platforms across business, regional, air transport and government systems market applications and consists of multi-year awards up to five years.

US Defense Military Satellite Sector Fueled by Heightened Funding against Escalating Cyber Warfare ▶

Modernization strategies along with counter and offensive space initiatives create significant growth opportunities that could take the market past $30 billion by 2023, finds Frost & Sullivan.

Design and Manufacturing Perspectives from DISH Technology's Les Beller ▶

Interview with Les Beller, a long-time PCB designer who is now a manufacturing engineer for DISH Technology. We discussed his company's business shift towards 5G and streaming, and the stresses that puts on a design team.

Circuit Automation on the Ever-Evolving World of Solder Mask

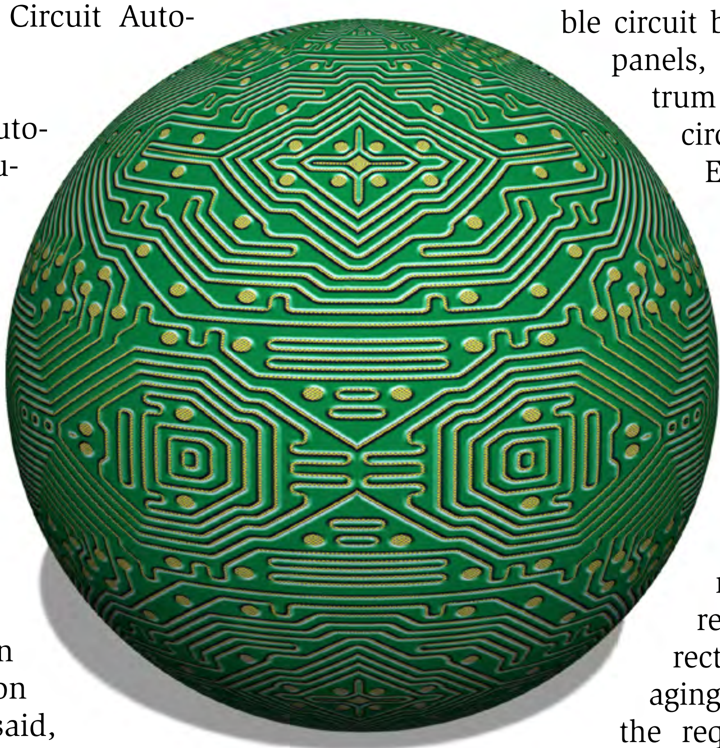
Feature by the I-Connect007 Editorial Team

In a recent conference call, our I-Connect007 editorial team was joined by Circuit Automation's Yuki Kojima, VP of engineering; Larry Lindland, sales and applications manager; and Tom Meeker, CEO, for a lively discussion about solder mask. Spoiler: It's not all about the equipment.

Patty Goldman: Gentlemen, let's start with some background on Circuit Automation.

Yuki Kojima: Circuit Automation has been manufacturing double-sided screen printing systems for the circuit board industry since around 1980. Mainly it's been for the rigid circuit board industry, but now we're getting into the flexible circuit board industry, too.

Larry Lindland: I've been with Circuit Automation since 1995. As Yuki said, we've been working with solder mask manufacturers all over the world trying to provide the best application system out there.



Goldman: Yuki, if you would, please give us an overview of what you see happening in the solder mask world. What are the current problems and where do you see things are going?

Kojima: The requirements and specifications for solder mask have been diversifying. It's been just computer boards and consumer products. But now, with HDI, smartphones, flexible circuit boards, communication panels, there is a wide spectrum of different types of circuit boards out there. Each has such unique requirements, and we have to meet different demands. For example, I think the biggest challenges right now are the exposure, or I would say imaging, after photo masking. Now, many people are getting more and more into direct imaging—laser direct imaging, UV direct imaging—and we need to meet the requirements, which are basically the mask thickness, evenness, and distribution and that sort of thing.

I also do sales and marketing in Asia where the biggest challenge I see right now is via

IT-170GRA1

IT-170GRA1 Value Proposition

- Halogen Free
- Best in class performance in its space
- Best in class CAF performance
- Very attractive price to performance ratio
- Pass MRT-6 requirements
  - -CAF > 1000 hours
  - -0.65mm pitch equivalent, no thermal issue
- Available with RTF and 2μ copper which provides additional performance on a very solid platform
- Df = 0.0075 @10GHz

Intel Purley Mid-Loss Solution – IT-170GRA1

Futures

Halogen free mid loss, Df < 0.01
Tg 180ºC by DSC, low CTE

Applications

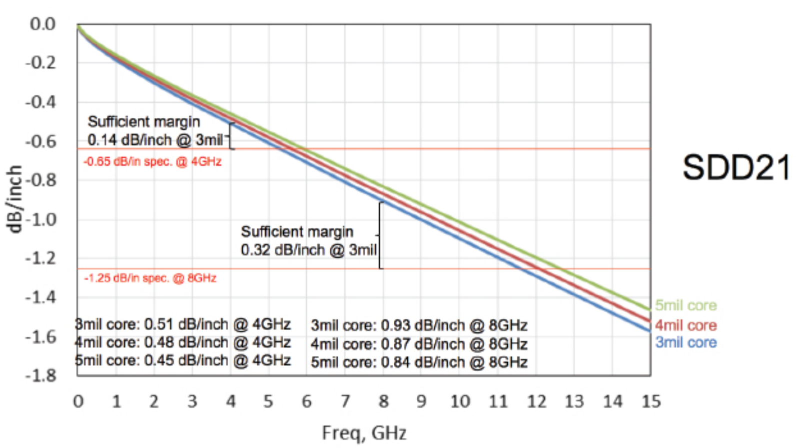
Server/Storage/Switch (10G, 40G)
Backplane, Telecom
Base station

Feature — *Purley Platform Mid-Loss Solution – Hi Tg*

General Properties

Items	Methods	IT-170GRA1
Tg (℃)	DSC	180
T-288 (w/ 1 Oz Cu, min)	TMA	60+
Td-5%(℃)	TGA 5% loss	380
CTE (%), 50-260℃	TMA	2.4
Peel strength (lb/inch)	1 oz	7.0
Water absorption	D-24/23	0.1
Dk: 2-10 GHz	Bereskin	3.96 – 3.99
Df: 2-10 GHz	Bereskin	0.0073 - 0.0075

IT-170GRA1 Insertion Loss



ITEQ

holes. A panel such as an OLED requires microvia holes to be plugged, which is a very difficult thing to do. Many companies, as you all know, want via holes cleared or plugged. Now there are microvia holes, blind via holes, and they need to be plugged with solder mask. That has been a challenge for us.

Goldman: And that's because you're basically screen coating with the vertical system, right?

Kojima: Yes, correct. Well, not just limited to our products, but in general, whether it is spray coating or dry film or whatever. It is not easy to plug microvia holes.

Goldman: Why is it so difficult to fill them? Is it because there is extra mask right there?

Kojima: Well, you can set up the process to meet some requirements. For example, there are parameters we can set so that ink can go into the hole easier. The challenge is that there are a lot of different sized via holes out there. Plugging is very diameter- and volume-specific; you can plug big holes, but then the small holes won't be plugged, and vice versa, that type of thing. Also, with microvia holes, obviously there's air inside the hole, and the air has to go someplace. Therefore, we print and capture some, but we can safely say we can plug probably up to 99% of the via holes. But there's always the 1% or 0.1% of the holes that we cannot plug. So that's a big challenge.


Yuki Kojima

Goldman: You've mainly seen that challenge in Asia at this time?

Kojima: That's correct. Mainly where they do OLED panels and such products.

Lindland: Yuki's right. One of the challenges that we have faced in production is applying some of the things like getting a very even coating, minimal mask in the holes, and doing it on a daily consistent basis. For 20 years, we've been keeping ink out of the holes, and then we hear, "Oh, by the way, now will you please fill the holes?" And we try to be right up front and say 100% is very difficult to do. As Yuki explained, the holes are different sizes and will fill at different rates. We struggle with that, not being able to give them 100%. At the same time, we've worked very hard to provide extremely even mask thickness, which enhances the next step of imaging. We used to measure solder masks in mils, but now we're measuring in microns, and that's been a big advantage as we do some of the higher-end boards.

Goldman: You are referring to thickness, right?

Lindland: Right, the finished thickness. Solder mask usually is about 25% solvent after you tack dry to get your finished thickness or when you lay it down wet, and we're measuring that in microns now.

Tom Meeker: One thing, if I could add, is that there's a wide variety of difference in the ink formulations. In setting up for one customer's ink, you get totally different results if they switch from a matte to a gloss, or something. There's a huge variation in performances that we've seen based on just the specific ink choice that the customer uses.

Goldman: I would think most customers use more than one ink. As you said, matte versus glossy, and then there are the different manufacturers and the different colors, also.

Lindland: That's true. Customers are using many different colors now.

Meeker: There are the colors, formulations and rheology. I'm working with a current customer who is routinely doing four different versions or colors, switching back and forth every 10 panels. They've got all these screens set up and different setups for going back between gloss and matte. There's a huge difference in performance between them.

Goldman: So the customers are switching between all the colors, the matte, maybe even different manufacturers, all on one machine.

Kojima: Well, some customers. Not all of them, but many do.

Goldman: I understand you have your equipment at Whelen, which is now GreenSource Fabrication. Is there anything you can tell us about that?

Lindland: We have had some success with GreenSource, and in their upcoming expansion we were fortunate enough to be one of the few American suppliers selected to supply the next round of equipment, which we'll be doing very soon. We're excited about what they're doing with automation and getting into some of the higher end boards back in North America.

Barry Matties: In terms of automation, what does that look like? Would North American manufacturers consider that?

Meeker: Almost all of our equipment in the U.S. is semi-automatic, and a high percentage of what we ship out of the U.S. is fully automated. The newest machine that we're building for GreenSource has the capability of full automation, which means you scan in a recipe and it takes a panel horizontally in and coats it, dries it, and brings it back out horizontally. The challenge with the new Whelen project is that the machine must be capable of handling all thicknesses, from flexible materials to backplane panels. So it was a big challenge to integrate a machine capable of doing flex and backplanes at the same time. This is with no operator making the changes to the recipes or anything. This is strictly recipe throughout off a barcode reader.

Goldman: And you said that was horizontal?

Figure 1: Automatic loader system.

Meeker: Well, it goes in horizontal; the panels come off a clean line, so they're horizontal. We take them horizontal, we center them, then we make them vertical, so we can do double-sided coating on them, and then we dry them vertically, because that really saves a lot of space. You can have the panels on inch-and-a-half centers, or 38-millimeter centers, and then they come back out to the next process, and they get turned back to horizontal, where they're either racked or stacked or handled.

Goldman: And everything from back panels, like you said, all the way down to extremely thin flex. Is that the biggest challenge?

Meeker: Correct.

Kojima: One of the problems with processing materials such as flexible circuits is that we can print them okay, but the drying usually would be a problem because they're so thin. And when you hang the panel inside, all of them touch each other. Circuit Automation invented this device called ASA, which is an anti-sway attachment. It attaches to each circuit board at the bottom inside the oven where the air is blowing and the panels tend to sway in the heated air. The ASAs add mass plus a physical barrier, preventing the panels from touching each other. Remember, the panels are wet on both sides, so they cannot touch. That's one of the unique features we have with the system, and it's what makes this machine capable of running from thick panels to very thin panels, like flexible circuit boards. It can go down to about a one-mil thick panel.

Goldman: I know GreenSource's lot size is one, at least theoretically. But, to change between a matte and a shiny, or a different solder mask, how automated is that process?

Kojima: It's all controlled by a server, and the only thing you need to do is enter the dimension of the panel. Of course, the screen opening needs to be adjusted according to the panel size. That's the only manual process you'll probably need to do to set up the machine.

Meeker: But changing ink is still very manual. Fortunately, at GreenSource, they're only going to use one ink. Because otherwise, to switch back and forth, you then have to change screens and squeegees. We had an automatic ink filler, but their plan is to go with one mask at a time, so we don't have the challenge of figuring out how to automatically switch all the wetted parts, which would be difficult to do.

Figure 2: Oven and unloader.

Matties: That would be difficult. From a circuit board design point of view, are there any considerations PCB designers should have regarding solder mask?

Lindland: One thing that people struggle with is hole size, aspect ratio, and dam size, in terms of clearing. The other thing that's difficult

these days is when they talk about via plugging, as Yuki said. Very few give an actual requirement such as tented, filled, plugged, percentage filled, things like that. Some parameters are not clearly specified (if at all); you get variable results on that. You know, copper heights, coverage, thickness at the knee, etc. As Tom said, the mask used has a big effect on the actual finished thickness.

Andy Shaughnessy: So the more information they can give you, the better?

Lindland: Absolutely, yes. Like I said, the hole size, aspect ratio, copper heights, trace heights...

Kojima: More information is always better, because, for instance, when you say hole plug, what does that mean? Does it mean 80% of the hole needs to be filled? How about the number of holes being plugged? Let's say the panel has 2,000 via holes. And let's say one is missing the ink inside the hole. Is that bad, or is it okay? What percent is allowable? People just do not have the answer to these questions, sometimes.

Larry Lindland

Shaughnessy: Sometimes designer don't even specify the copper, whether it's finished copper or not. They assume everybody knows what they're talking about. And sometimes they're using slang, or other terms that are kind of tribal within their company.

Kojima: That's true.

Meeker: At the same time, as an equipment supplier trying to meet specifications, we're very uncomfortable saying 100%. Because there's always some fallout for whatever reason, and what percentage is acceptable is usually never defined.

Kojima: Another criterion is how to check these holes. Like I said, some circuit boards have probably three or four thousand via holes. I mean, how do you check them?

Goldman: That's very true. And what gets me is that you're putting down a thinner mask, but you're also trying to put extra material in the holes. That seems next to impossible.

Kojima: That's correct. And yet they want us to control the amount of ink that goes inside the through-hole or, like I said, micro and blind via holes.

Lindland: The other thing, too, for the designers, is that as you're in the field making panels, and there are very few times that they'll actually specify thickness. A lot of times they'll specify a minimum mask thickness, without saying a maximum thickness. The Circuit Automation machines do not have many issues with thickness; we tend to error on too thick of coverage, which can be easily adjusted thinner, and they're rejected because the mask is too thick. And that creates tombstoning or whatever problems at assembly, yet it's never specified.

Goldman: Very frustrating. Well, I guess that mask thickness is partly defined by how fine the features are. But you don't always know that either, I suppose.

Meeker: And it's also dependent on the subsequent process; if they're doing something like deep gold they're going to want the mask thicker for more protection. A lot of the subsequent processing after ours makes a big difference in how thick or how thin they want the solder mask.

Matties: Is this commonly known by designers? If they're not specifying thickness, do they understand that there could be assembly challenges, or, as you're talking about, a gold bath issue? How do they know this?

Lindland: That's a good question, and I would say probably not—they don't know it. And the more information you could provide would help tremendously. The other thing, too, as Tom said, with our equipment and what's going on now, you're able to apply your solder mask and meet a target thickness. That allows us to do some of the higher end and more difficult panels, whereas 20 years ago, it wasn't really specified. There wasn't that real target, and there wasn't a real requirement as a solder mask equipment manufacturer, to meet that. As it's gotten tighter, we've been very effective and very successful at doing that.

Matties: I would think if you discover in your assembly process that the mask is too thick, that's an enormous amount of waste. I don't know what the outcome is with thick mask. Is the board absolutely rejected at the assembly level if the mask is too thick?

Lindland: You're right. If that is happening at the assembler, it's way too late. And that's where people have tombstoning and other problems with their components. And yet, if the designer would specify what the desired thickness is, I think everyone would be more successful.

Goldman: You know, it always comes down to communication. No matter what the subject is, it still comes back to communication, doesn't it?

Matties: I think there's communication. But in some cases, there are new standards or needs that creep up that we're not aware of until it's too late. And so it's not just communication, it's being aware of the trends and the requirements or the impact that you have on the process.

Shaughnessy: And then the manufacturer will fix this problem, and sometimes they don't even tell the designer.

Matties: Yes. That feedback loop needs to exist. We're seeing more and more demand for HDI, especially here in North America; we're seeing more people move into that. Are there any considerations for HDI as it relates to solder mask?

Lindland: It's all about the mask thickness. It really is, because with an HDI panel, you have tighter tolerances all the way around, from your copper heights to your panel thickness, hole sizes, everything. It puts tighter demands on the entire process, and for solder mask, it's a very process-oriented system where you really rely on how you clean the panel, how you apply the mask, how you tack cure it, how you expose it, how you dry it, handle it, everything, to get the results you need. And the demands have gotten tighter in the last few years, and the ability to put the mask on very evenly on a wide range and do different things is much more difficult for a lot of shops.

It's all about the mask thickness.

Matties: Is that based on the equipment that they have or is it also based on the discipline that they have?

Lindland: A lot of it is that the equipment was designed 20 years ago, and a lot of advancement hasn't been made, or they haven't made a lot of investment in new.

Meeker: Well, the equipment was originally designed to replace flatbed screen printing. And in the North American market, which is all pattern-plated, there's a lot of high and low copper plating across the entire panel. The original goal was to put the solder mask on really thick and consistently over even where you have a variation of 2 – 3-mils in traces, so you have something that was more tolerant to uneven copper plating. Throughout the rest of the world, everything is panel-plated, so it's not as big an issue as it is here with pattern plate. That was the original goal, so the earlier machines were made to put it on thick, and thick was good; the thicker the better. And now, you have the DI machines that are really bogged down; the differential in the thickness of the solder mask makes them much slower. Now the driving force is to make that solder mask as thin and as consistent as possible, because it speeds up the direct imaging step.

Matties: In terms of cycle time for the solder mask process, curing probably takes the greatest amount of time. Are there any improvements in cycle time in solder mask? What sort of advancement or reductions in cycle time are you seeing?

Meeker: There have been a lot of attempts to reduce the cycle time in solder mask by decreasing the time in the oven. A standard panel is tack-baked for about 45 minutes, and there have been a lot of attempts with infrared, which could reduce that dramatically. IR has been only marginally successful, for a lot of reasons. A lot of people came out with infrared drying systems, to try to speed up the bake. It's not been particularly successful in a lot of application sizes and thicknesses and types. You know, if you have 100,000 of the exact same panels then that makes it easier to do. But especially in the North American market where it's five of this panel type, and 10 of that one, it's really hard to adjust an infrared system to be successful; whereas, a thermal, convection-based system doesn't really care. That would be a challenge to come up with something faster in the oven.

You know, if you have 100,000 of the exact same panels then that makes it easier to do.

Matties: I was going to say, back in early days, systems like True Cure came out. They had recipes, but the variations were so many and so deep, that it wasn't a practical piece of equipment.

Meeker: Yeah. True Cure is a good example, but there were 20 of them out there. I don't think any of them are still in operation.

Matties: No, I don't think so. But today, with the IoT and the quantity of sensors and, looking at what GreenSource is doing with a lot size of one, it seems to me there may be opportunities to reexamine that and see what's possible. In a quick-turn environment, an hour or two in an oven is a lot of time.

Meeker: That is true. And especially, if you look at the two processes, it's about a 45-minute tack-dry and about an hour final cure, so a two-hour cycle time. It's much easier to achieve something for the final cure because you're not trying to hit a target window. Tack dry, you've got to go 85°C plus or minus two degrees, or something. In final bake, it's just got to get hot enough to cure it. There's a much wider process latitude for final curing in solder mask, and I see that there's some new technology on the horizon that is somewhat promising. It's a much easier application.

Matties: Is this something that the fabricators are asking for, or do they just accept the process as is?

Meeker: No, they would like it. Saving any amount of cycle time is great. Most people, especially if you look in the North American market, are in the quick turnaround business, so if you can save an hour or two in cycle time, that's a huge thing.

Matties: If a fabricator came to you and asked for the best advice you could give about solder mask, or the process of solder mask, what would that be?

Lindland: Test the available options out there, which are probably spray, flatbed, and obviously double-sided screen printing. Talk to the existing shops out there, see what they're doing, and then talk to us and we'll make the best recommendation based on what your needs are.

Meeker: Larry's right. Every system has pluses and minuses and tailoring the system for the specific application is the key. What are you trying to do and how much real estate space do you have for these equipment processes?

Lindland: The one other piece of advice I think we would offer is to consider what you're doing now, but consider what you're going to be doing five years from now, and your systems. Because we've had many people that we sell a system to, put it in, and everything works great. then Their workload or demands change, and their requirements change. So, they went in a direction they didn't originally think they needed to go in. Therefore, look what you're doing now, and get a feel for what you'll be trying to do in the future, as well.

Goldman: Do people still do a lot of curtain coating and spray coating?

Meeker: Curtain coating in the U.S. is dead. There's less than a handful. Spray coating, there's some. Spray coating is much more popular in Asia and Europe. Some of the major guys in Europe still have curtain coaters, but nobody is buying a new curtain coater, let's put it that way.


Tom Meeker

Goldman: Is that a volume thing? I'm just curious.

Meeker: Well, the curtain coaters work really well for solvent-based ink, which means solvent developable, which is obsolete. It's still heavily used, I think, in a lot of the automotive industry. So, that's why there are still a lot of curtain coaters in Europe.

Kojima: Curtain coaters are better for low tech, rigid panels, and no fine definitions. Panels like that will be okay. For thinner panels, or more intricate circuits, I don't think it's as desirable. That's why the number of installations in Asia, for example, have significantly decreased. I don't think there are that many left anymore.

Lindland: Plus, as we talked about before, many people are required to use different solder masks, and a curtain coater does not lend itself to easy mask changes.

Goldman: So not a good option all around. But if people are doing spray, is that mostly solvent-based also?

Meeker: No, most of the spray ink is water-based chemistry. From the Circuit Automation point of view, the spray is our competitor. The advantage of our coating is it's much more even across the entire panel than a spray board, because by nature, when you're rastering across the panel with a variety of spray heads, some parts get sprayed twice. Some parts get sprayed three times. Some parts get sprayed once. You have high-low valleys. When that becomes an issue, that's where we excel. In a lot of places, that doesn't matter to them, so therefore, spraying is okay.

Goldman: Perhaps it isn't very versatile; I suspect changing to a different mask chemistry could be pretty involved.

Meeker: Well, some of the small spray guys try to put four or five different colors all at once into the machines. If you're doing a really small group, one or two, that's an acceptable process. But it hasn't really lent itself to production-level volume.

Kojima: Also in Asia, due to the production requirements, they still use fully automated spray lines. And that's due to speed.

Goldman: OK, thank you so much for your time, gentlemen. We appreciate it, and so will our readers.

Meeker: Thanks, Patty. **PCB007**

Electronics Industry News and Market Highlights

NAU Scientist Contributes to Significant Advance in Silicon Photonics ▶

Northern Arizona University assistant professor Ryan Behunin collaborated with a team of physicists from Yale and the University of Texas at Austin in discovering an innovative way to manipulate light in silicon.

5G Networks Have a Paramount Role in Autonomous Vehicle Connectivity ▶

5G networks may be 10 times more efficient than 4G networks. With this new level of network capability, communications service providers (CSPs) can secure future market opportunities with manufacturers of autonomous vehicles(AVs).

IEEE Releases the International Roadmap for Devices and Systems ▶

IEEE announced the release of the 2017 edition of the International Roadmap for Devices and Systems (IRDS), projecting technology needs for evolving the semiconductor and computer industries.

The Survey Said: Industry Optimistic After Strong 2017 ▶

During recent trade shows and conferences, we spoke with a variety of fabricators and assembly providers. Every company achieved strong growth in 2017 and shared a positive outlook about the future.

Consumer Adoption, Interest in Wireless Charging Continues to Surge, New International Survey Shows ▶

Consumer adoption of wireless charging technology has increased dramatically, according to a new survey conducted by the Wireless Power Consortium (WPC).

PCB Design Software Market to Reach $4B by 2023 ▶

The PCB design software market is expected to grow at a CAGR of around 26% from 2018 to 2023 to reach $4 billion by the end of the forecast period.

Tiny Defects in Semiconductors Created 'Speed Bumps' for Electrons ▶

UCLA scientists and engineers have developed a new process for assembling semiconductor devices. The advance could lead to much more energy-efficient transistors for electronics and computer chips, diodes for solar cells and light-emitting diodes, and other semiconductor-based devices.

Worldwide Semiconductor Equipment Billings in Q1 2018 Reach Record $17.0 Billion ▶

Worldwide semiconductor manufacturing equipment billings reached a historic quarterly high of US$17.0 billion for the first quarter of 2018, surging 59% in March to an all-time monthly high of $7.8 billion.

AT&T, BT, Orange, Singtel, and Vodafone Lead the Way for Next-Generation elecom Services in Asia/Pacific ▶

AT&T, BT, Orange, Singtel, and Vodafone are identified as 'Leaders' of the next-generation telecom service providers in Asia/Pacific, but there are plenty of challengers in the market.

Thermal Capabilities of Solder Masks: How High Can We Go?

Feature by Sven Kramer
LACKWERKE PETERS GMBH & CO. KG
KEMPEN, GERMANY

Abstract

This article focuses on three different coating material groups that were formulated to operate under high thermal stress and are applied at the printed circuit board manufacturing level. While used for principally different applications, these coatings have in common that they can be key to a successful thermal management concept especially in e-mobility and lighting applications. The coatings looked at consist of:

Specialty (green transparent) liquid photoimageable solder masks (LPiSM) are compatible with long-term thermal storage/stress in excess of 150°C. Combined with the appropriate high-temperature base material, and along with a suitable copper pre-treatment, these solder resists are capable of fulfilling higher thermal demands placed on them. In this context, long-term storage tests as well as temperature cycling tests were carried out. Moreover, the effect of various copper pre-treatment methods on the adhesion of the solder masks was examined at 150, 175 and 200°C over the aging process. For this purpose, test panels were stored for 2000 hours at the respective temperatures and were submitted to a cross-cut test every 500 hours. Within this test set-up it was found that a multi-level chemical pre-treatment gives significantly better adhesion results, in particular at 175°C and 200°C, as compared to a pretreatment by brush or pumice brush. Breakdown voltage as well as tracking resistance were also studied.

For an application in LED technology, the light reflectivity and white colour stability of the printed circuit board are of major importance, especially when high-power LED are used which can generate larger amounts of heat. For this reason, a very high coverage power and an intense white colour with high reflectivity values are essential for white solder masks. These "ultra-white" and largely nonyellowing LPiSM need to be able to withstand

specific thermal loads, especially in combination with high-power LED lighting applications.

Introduction

The demands and loads placed on photoimageable solder masks rise to unknown new levels. Today's solder masks are already exposed to a considerable level of thermal stress, high humidity and/or condensation. Especially in automotive electronics, the level specified for thermal resistance and thermal cycling resistance is constantly increasing. Due to higher currents and applications mounted in the engine compartment/gear box or close to the exhaust system, common photoimageable solder masks reach their performance limits. The thermal stress resulting from higher operating temperatures triggered the development of new solder masks or further development of existing solder masks. Here the level demanded by the industry in terms of permanent high temperature loads is typically 175°C.

Photoimageable Solder Masks

While an LPiSM can consist of more than twenty different ingredients—which of course are all needed to create a coating which can be easy, fast and reproducibly be processed in the specified quality—the actual "backbone" of such a coating is made up of only three components: The resins, fillers and hardener used. These constituents ultimately determine the (di)electrical and physical/mechanical properties. Table 1 and Table 2 give an overview of a principle LPiSM formulation composition.

Aging Process of Polymers

When looking at the most relevant factors involved in the general aging of polymers the following come to mind: First, a loss in volatile components (e.g., a loss of low-molecular components that were present from the beginning); second, oxidations that are often accompanied by an increase in cross-linking and embrittlement. The continuation of molecular polymerisation often initially leads to an increase in electrical and mechanical stability, but finally ends in embrittlement. Additionally, the chemical separation of low-molecular components plays a role. This separation is a result of the aging process. These processes can act autocatalytically. Finally, also a hydrolysis of the polymer due to present moisture needs to be considered.

Raw material	Property
Resins/binders	Photoreactive and thermally cross-linking resins, decisive for adhesion, insulation/ resistance
Fillers	Adhesion, scratch resistance, hardness, flow behaviour (thixotropy), thermo-mechanical properties
Dyes/pigments	Colouring substances
Additives	Adjusting wetting, flow, defoaming, etc.
Solvents	Viscosity adjustment, control of drying

Table 1: Constituents of resin component in LPiSM.

Raw material	Property
Photoinitiators	Initiate photopolymer-isation in exposure, influence on exposure time/energy and resolution
Solvents	Viscosity adjustment, control of drying
Hardener	Cross-linking medium of the thermally curing resins, decisive for resistance and physical and chemical final properties of the coating

Table 2: Constituents of hardener component in LPiSM.

General Procedure for Long-Term Temperature Storage Testing of LPiSM

The approach for LPiSM testing consisted of the application (20 to 40 µm dry thickness) and processing of different coating materials on various (high Tg, low CTE) substrates followed by a surface finish (ENIG, IMT or HASL). Prior to the actual testing, a pre-conditioning consisting of one wave soldering or two reflow soldering processes was carried out before the actual temperature

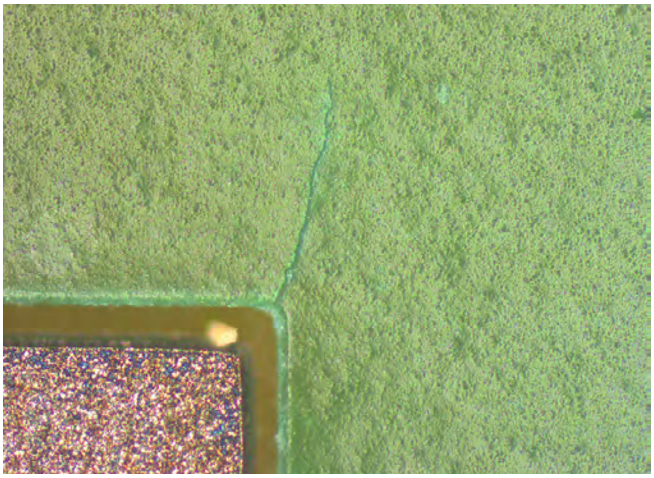


Figure 1: Corner crack of an LPiSM on laminate/colour change—relevant or not?

storage followed. The intermediate or subsequent testing procedures discussed in this paper consisted of:

- Breakdown voltage testing
- Tracking resistance (CTI)
- Adhesion using different pre-cleaning modes

and were carried out with a screen printing LPiSM with a 20 µm dry thickness.

The thermal performance of high-temperature-resistant LPiSM cannot be viewed as an isolated entity. It must always be looked at in combination with the substrate used and its treatment (i.e., cleaning, degreasing, deoxidation). Without a matching (Tg/CTE) laminate, even the best possible pre-treatment prior to solder mask coating will not lead to the desired results in terms of adhesion and inevitably lead to fissures and cracks as shown in Figure 1.

Figure 2 shows the colour change of an LPiSM before and after a thermal load of 2000 hours at 175°C and 200°C, which is acceptable to most as long as the insulation properties are within tolerances. The discolouration is caused by a certain degradation/oxidation of the pigmentation and binders. Depending on the actual coating tested, electrical properties can of course decline (e.g., insulation resistance) and must be looked at individually.

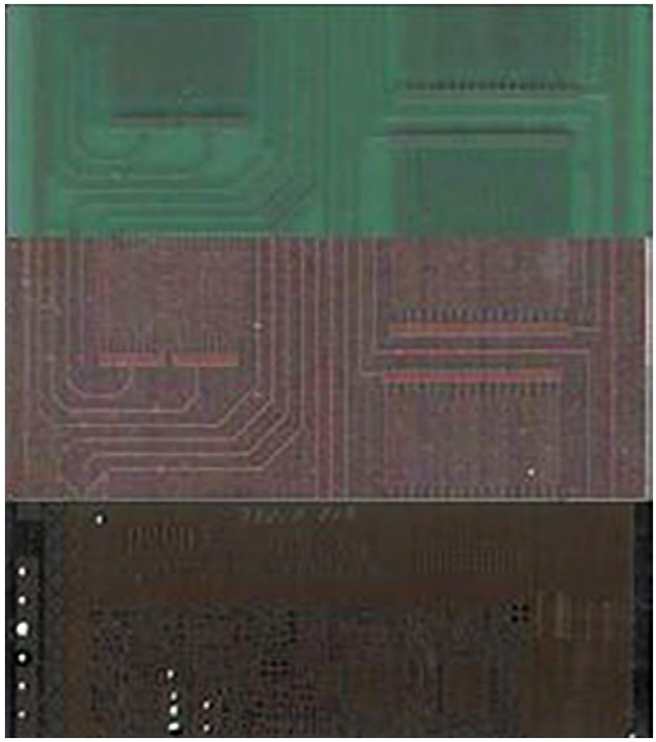


Figure 2: Visual appearance of LPiSM before and after 2000 h storage at 175°C and 200°C—influence of pre-treatment on solder mask adhesion.

Figures 3, 4 and 5 very clearly show the impact of pumice brushing, brushing and a (multi-level) chemical pre-treatment at 150°C, 175°C and 200°C for up to 2000 hours dwell time in combination with an aqueous-alkaline

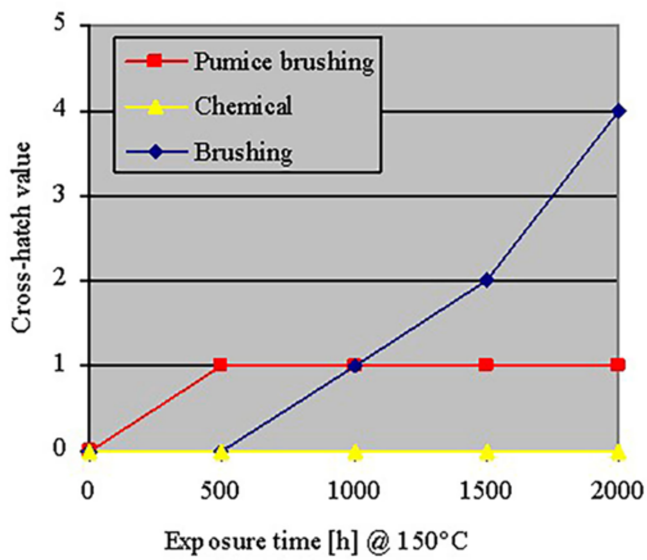


Figure 3: Adhesion of LPiSM after temperature storage at 150°C.

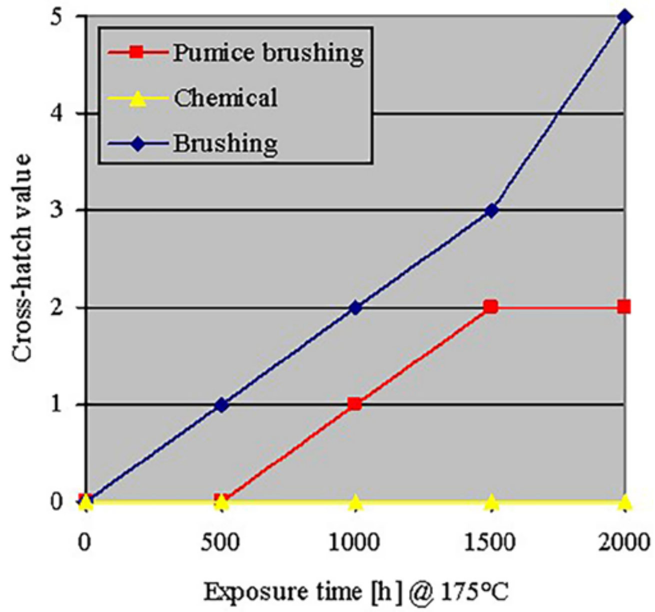


Figure 4: Adhesion of LPiSM after temperature storage at 175°C.

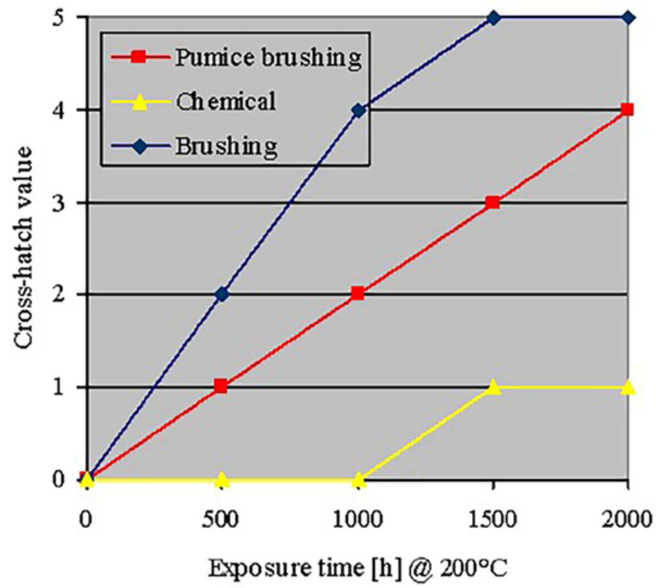


Figure 5: Adhesion of LPiSM after temperature storage at 200°C.

developable LPiSM specifically formulated for high temperature storage applications with a matching laminate. The results of the cross-hatch test according to EN ISO 2409 show that a purely mechanical treatment will not enable a long-term thermal storage/resistance even at 150°C.

The measuring of breakdown voltage on a steel plate with 20 µm dry thickness (Figure 6) during 200°C storage for a total of 20 days indicates that with increasing thermal load insulation generally decreases and begins to fluctuate after approximately 10 days. This can potentially be caused by fissures and pores or microbubbles.

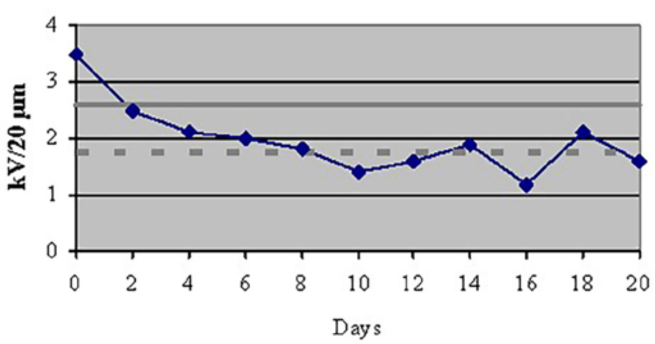


Figure 6: Breakdown voltage during temperature storage at 200°C (coated steel plates).

Figure 7 shows that the aim at least to maintain the CTI value of the laminate can be met even after a 2000 hours thermal load at 175°C with specialty coatings.

The Tg and CTE values as indicated in Table 3 seem to be largely independent of the curing temperature. An additional triple reflow soldering process at 265°C seems to lead to a post cross-linking because the corresponding Tg values increase by approximately 20°C.

The severe thermal load applied as indicated in Table 4 leads to a slight increase in the Tg

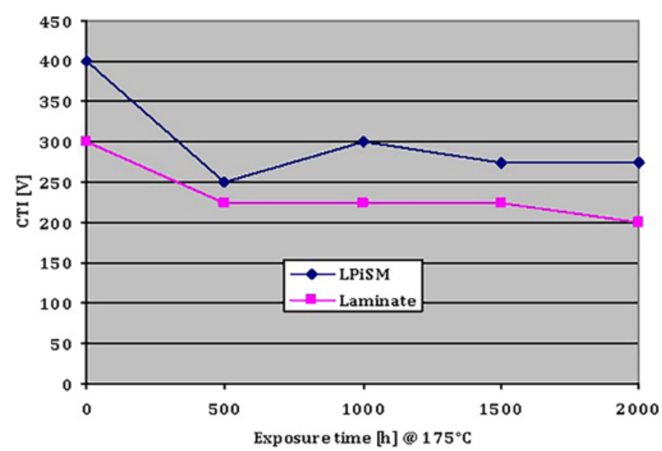


Figure 7: Tracking resistance after temperature storage at 175°C.

Conditioning of specimen	Tg [°C]	CTE < Tg [ppm]	CTE > Tg [ppm]
60 min @ 150°C	152	45	90
60 min @ 160°C	145	48	99
60 min @ 170°C	153	44	97
60 min @ 180°C	140	53	104
60 min @ 150°C + reflow	169	50	80
60 min @ 160°C + reflow	173	52	90
60 min @ 170°C + reflow	178	42	85
60 min @ 180°C + reflow	171	39	84

Table 3: Dependency of Tg on curing temperature (+ 3X reflow).

and a (more favourable) lower CTE. The mechanism is presently not understood. However, it can be summarised that the tested LPiSM shows a high stability of Tg and CTE over a wide temperature range, which obviously can be advantageous when thermal shocks/temperature cycling and long-term thermal storage are carried out.

Defining Ultra-White and Largely Non-Yellowing LPiSM

Even though the following group of coatings discussed principally also belongs to the LPiSM family, these are very different in certain aspects from the previously discussed ones. The most important and obvious difference is the fact that the colour impression is not created by using dye-stuff—which leads to a transparent coating film—but is achieved by using an insoluble white pigment (usually TiO_2) which is there largely to hide the substrate.

To describe a colour or a difference in colour from a quantitative point of view, one can illuminate the surface to be measured by means of a defined light source and measure the light that has been remitted. In mathematical calculations both the type of the light source and the colour impression of the eye is considered for measuring colour. To characterise a colour through values, the so-called CIE Lab System (to be precise, the L*, a*, b* system of the

Conditioning of specimen	Tg [°C]	CTE < Tg [ppm]	CTE > Tg [ppm]
+ 500 h @ 150°C	177	29	39
+ 500 h @ 160°C	175	31	38
+ 500 h @ 170°C	177	28	35
+ 500 h @ 180°C	180	22	29
+ 1000 h @ 150°C	173	24	34
+ 1000 h @ 160°C	173	15	21
+ 1000 h @ 170°C	176	18	26
+ 1000 h @ 180°C	182	17	22

Table 4: Dependency of Tg on curing temperature + 3x reflow + thermal aging.

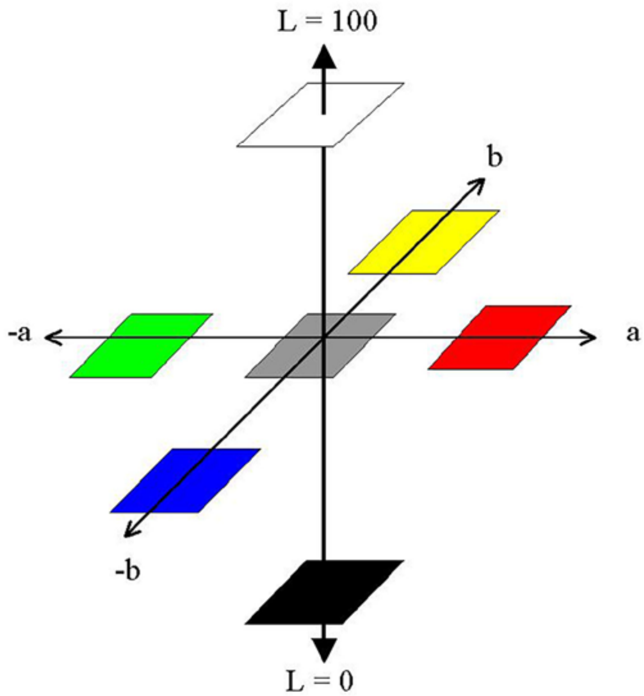


Figure 8: Scheme of the CIE L*a*b* system and its colour coordinates.

CIE–Commission Internationale de l'Eclairage, International Commission on Lumination, as shown in Figure 8) is commonly referred to. This lab system is based on the transformation of the primarily calculated colour values X, Y, Z to the coordinates L* (brightness), a* and b* (yellow-blue-value).

The major benefit of the CIE lab system is its suitability for determining colour distances. The value defining a colour distance, i.e. the similarity or identity of colours, is ΔE*, which is calculated as a spacial Pythagoras from the three values $L_1^* - L_2^*$, $a_1^* - a_2^*$ and $b_1^* - b_2^*$:

$$\Delta E^* = \sqrt{[(L_1^* - L_2^*)^2 + (a_1^* - a_2^*)^2 + (b_1^* - b_2^*)^2]}$$

In view of describing a white surface one can either use a reflectivity graph (see Figure 9 which shows various reflectivity levels depending on LPiSM type and thermal stress applied) or the brightness value L* from the CIE lab system. According to the definition, a perfectly white surface would have the coordinates L* = 100, a* = 0 and b* = 0. As an advantage, the CIE lab system restricts the values to be measured to no more than three figures and provides the option of forming differences that largely correlate with the colour perceived.

This approach is also very helpful in the case of white LPiSM for the electronics/printed circuit board field. Besides an evaluation in accordance with the CIE lab system, there is also the possibility of defining colour changes—i.e., yellowing—as it is practiced, for example, by the paper industry according to DIN 6167. Based on DIN 6167:1980-01, one speaks about yellowing if a non-desired yellowing value is recognised with the material observed, in consequence of its handling. From the primarily calculated X, Y and Z values a yellow value G is obtained. The change of the yellowing degree between an untreated (G_0) and a treated specimen (G_1) is calculated and referred to as the yellowing value, V.

The colour coordinates of the CIE lab system (i.e., the values L*, a* and b*), can be directly measured by means of colorimetric devices. Visible differences are indicated as follows:

Δb* = 0–1:	not visible in general
Δb* = 1–2:	low yellowing, only visible by the trained eye
Δb* = 2–3.5:	medium difference, visible by the untrained eye
Δb* = 3.5–5:	distinct difference
Δb* > 5:	strong difference

This assessment can also be transferred, in good approximation, to the interpretation of

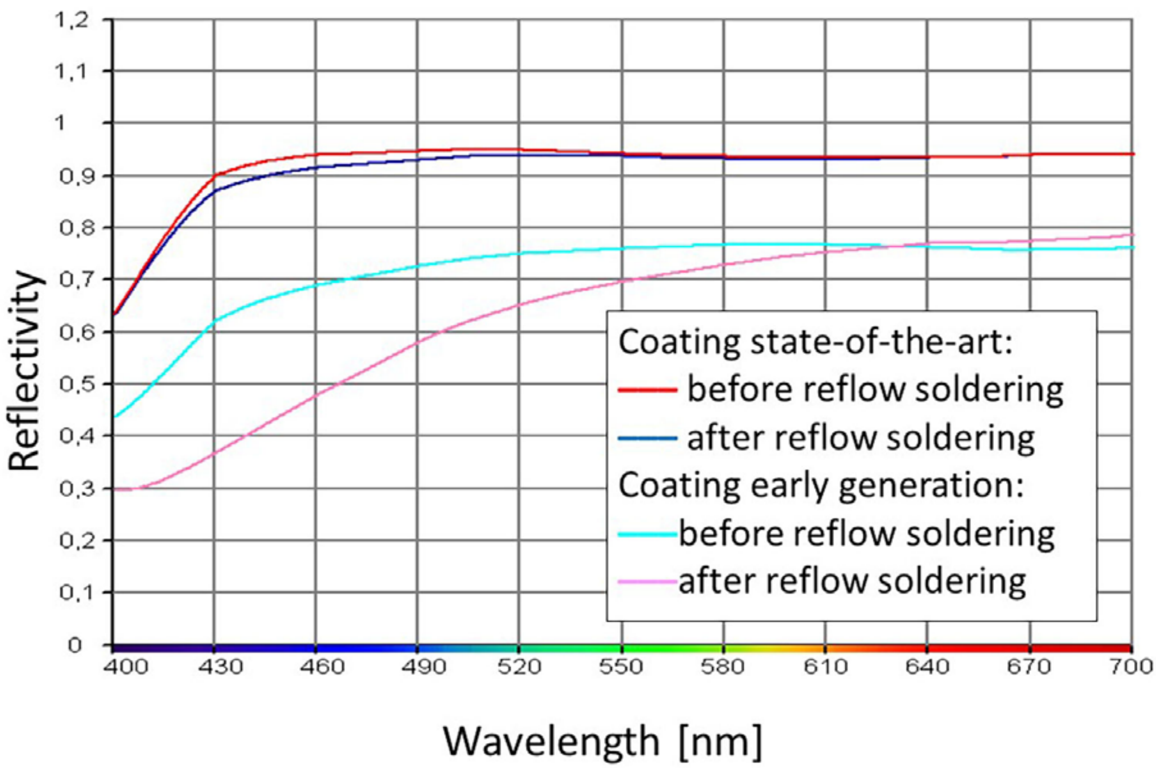


Figure 9: Reflectivity graphs of a white "classical" and new-generation LPiSM.

the pre-cited ΔE*, ΔL* and Δa* values. Given the widespread acceptance of the AE*, ΔL* and Δa* values for describing colour distances and/or colour changes, it is recommended to apply these data.

White LPiSM—Covering Power

A basic feature of coloured coating materials is the so-called covering power which indicates the performance in covering the colour, or colour difference, of the base material. As a criterion, a contrast ratio is predefined between the contrasting areas of the base. The covering power is achieved by absorption and diffusion and is a function of the colour-giving pigments. In the case of white pigments, it is only a consequence of diffusion. The covering power depends on both the layer thickness and the respective contrast with the base and has a characteristic limit. The covering power cannot be adapted to any thinness of layer. The minimum layer thickness required for a white covering coating in a black-white contrast is approx. 40 µm. Unlike thermal-curing solder resists, UV-curing coating materials—including LPiSM—must be slightly transparent to ensure a sufficient UV cross-linking down to the substrate.

There are certain consequences for processing, given the physical limits of the covering power. This way, a 10 µm layer looks less white than a layer of 20 µm, and again a 30 µm layer seems to be whiter than a 20 µm layer. Typically for a layer thickness of approximately 40 µm or higher, the white colour no longer depends on the layer thickness since maximum covering power has been reached. Within colour evaluations and measurings of a layer thickness below maximum covering power (i.e., lower than approx. 40 µm), the base material must be included in the evaluation. This means that the reflectivity graph and the L* value also depend on the layer thickness (Figure 10).

When processed, solder resists undergo several thermal loads, which may have a visible impact on the (white) colour. This is particularly valid for the soldering process which causes a shift towards the yellow colour (i.e., yellowing). A white solder resist supplied to

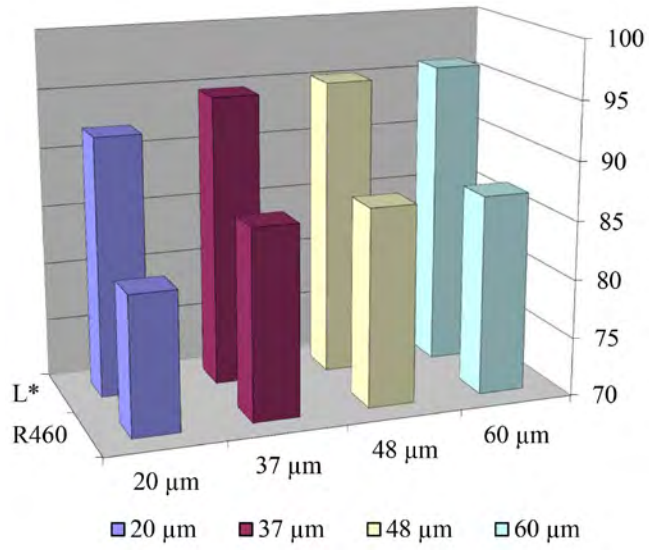


Figure 10: Reflectivity or L* value of a white solder mask depending on the layer thickness.

the assembler should undergo the least possible colour change in subsequent solder processes.

As for the underlying mechanism, there are different causes for yellowing:

- Heat
- Sunlight
- Environmental influences (e.g., humidity, chemicals)

In this context, it is primarily yellowing by heat or sunlight, or by a sunlight-simulating time-lapsed UV radiation that takes place. Light-induced yellowing is produced whenever light beams fall on polymers with a wavelength of less than 380 nm (UV radiation). So-called chromophore groups producing a discolouration are created through radically-induced photochemical reactions. Heat-induced yellowing is the formation of such chromophore groups that are produced when polymers are exposed to temperatures > 100°C for a longer period.

The colour change can be expressed by the so-called yellowing value on the one hand, or by measuring the colour distance in the CIE lab system on the other. To define this colour change, the previously described colour

Ucamco
Former Barco ETS

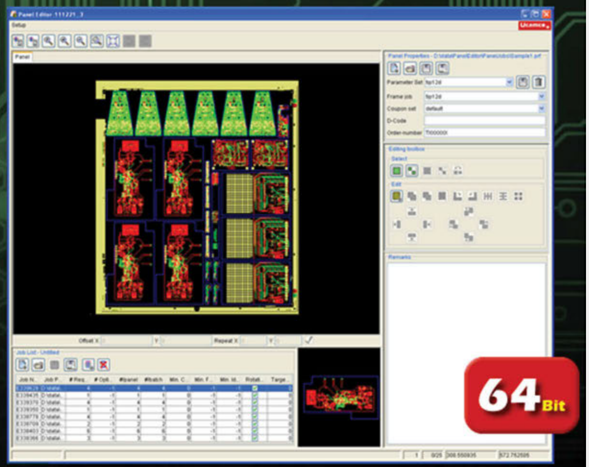
SOFTWARE | PHOTOPLOTTERS | DIRECT IMAGERS | NEWS | SERVICES | COMPANY | DOWNLOADS | CONTACT

Software

Multi Job Panelizer

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

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



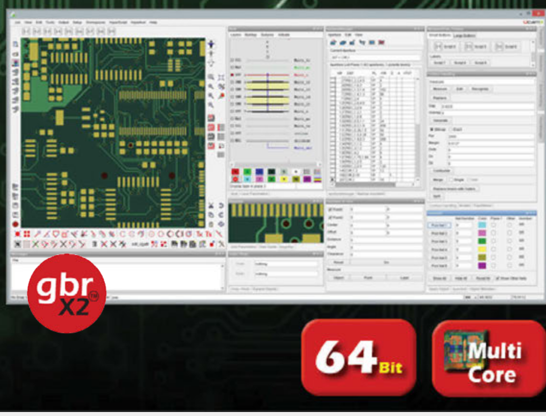
Read more

Software

UcamX

New ground-breaking CAM Software

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



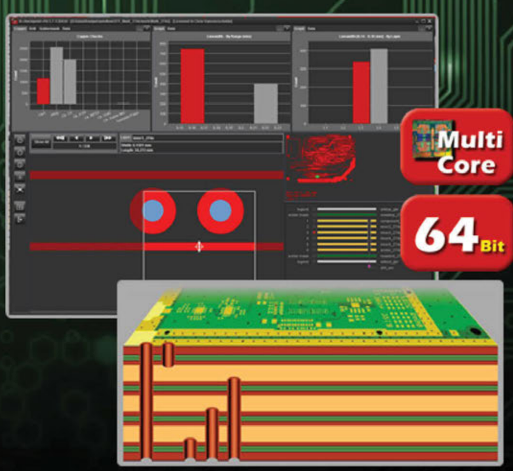
Read more

Software

Integr8tor

Automated data entry and design analysis

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



Read more

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

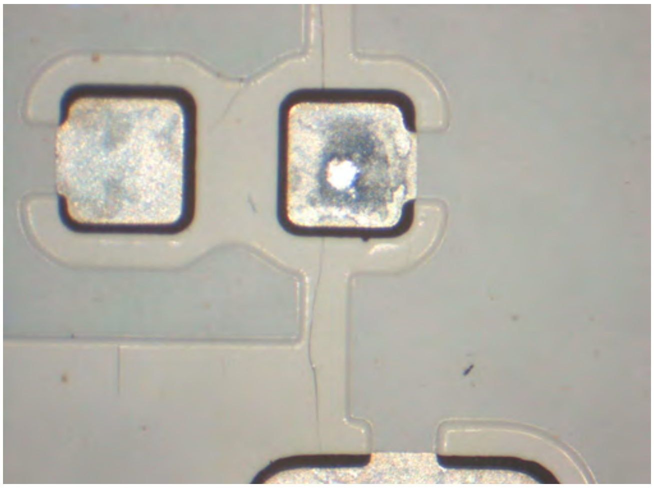


Figure 11: Fissures and cracks after 500 cycles of -40°C to +125°C.

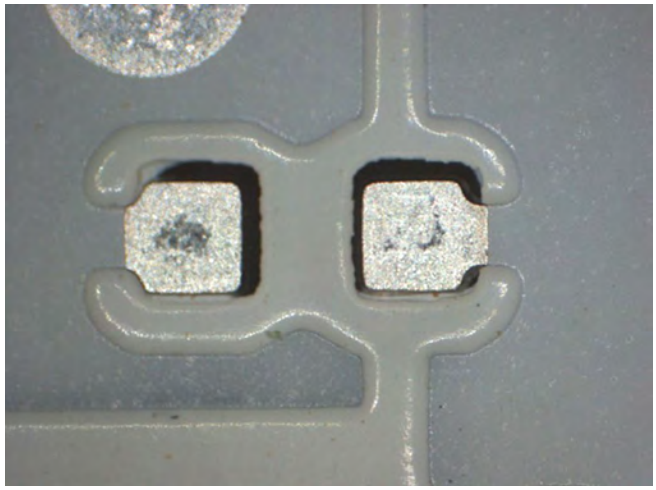


Figure 12: Coating with no fissures or cracks after 500 cycles of -40°C to +125°C.

distance ΔE* can be applied. The two values ΔL* and Δb* shall be given attention in case of purely white surfaces. While ΔL* describes the change of brightness, Δb* defines the blue-yellow shift (i.e., the yellowing effect in this case [Figure 4]). The Δ value, as low as possible, stands for the quality of the colour stability of the solder resist, with differences of less than 0.5 being hardly perceivable for the untrained eye.

Besides the fact that very low levels of discolouration are desired and reflectivity is anticipated to be in the range of 95% for the most advanced white LPiSM, also thermo-mechanical properties are considered and the coating is expected to withstand thermal cycling without displaying fissures or cracks (e.g., IMT/HASL finish, 3X reflow conditioning, 500 cycles of 40°C to +125°C). Figures 11 and 12 show an example of occurring/non-occurring fissures as a consequence of thermo-mechanical fatigue, which are of course unacceptable.

Summary

Presently, the long-term thermal resistance for specialised LPiSM is 175°C. Suitable LPiSMs have a Tg of around 170°C. Adhesion and insulation performance are on a similar level compared to standard 150°C long-term resistance coatings. Only a suitable substrate treatment prior to LPiSM coating will allow one to achieve the desired results. Of course, suitable high-Tg substrates must be used to avoid thermal mismatches and resulting cracks. 200°C long-term thermal resistance is presently not possible. While the adhesion level is good, insulation properties largely suffer after long-term exposure. The mechanism is not yet fully understood.

Thanks to the colormetric approach as used in many coating fields it is also possible in the printed circuit board/electronics industry to exactly define, compare and monitor the level of white and its stability under different conditions. When it comes to ultra-white and non-yellowing LPiSM, the latest available generation of ultra-white LPiSM typically shows a reflectivity in the range of 0.9 at dry thicknesses of only 20 µm, which means that due to the increased covering power considerable material savings compared to many existing materials can be made. **PCB007**

The original paper was presented at IPC APEX EXPO 2018 and published in the proceedings.

Sven Kramer is manager, Application Technology, with Lackwerke Peters GmbH & Co.

Flexible Metalization, Part II

Trouble in Your Tank

by Michael Carano, RBP CHEMICAL TECHNOLOGY

In previous columns and elsewhere in literature, concerns about electroless copper peeling from polyimide material have been reported. The main concern with is the creation of a void caused by peeling or blistering (Figure 1).

It is hypothesized that the blisters originate from one of, or a combination of, the potential causes listed below:

- Excessive catalyst adsorption causing a hyperdeposition of electroless copper
- Highly stressed electroless copper deposit
- Plasma residues causing poor bondability of electroless copper
- Overactive resin surface due to plasma
- Lack of surface profilometry due to plasma

Often, fabricators resort to double passing the circuits through the electroless copper process with the hope that the second pass will cover the void and prevent further blistering. Just remember, "Hope is not a strategy."

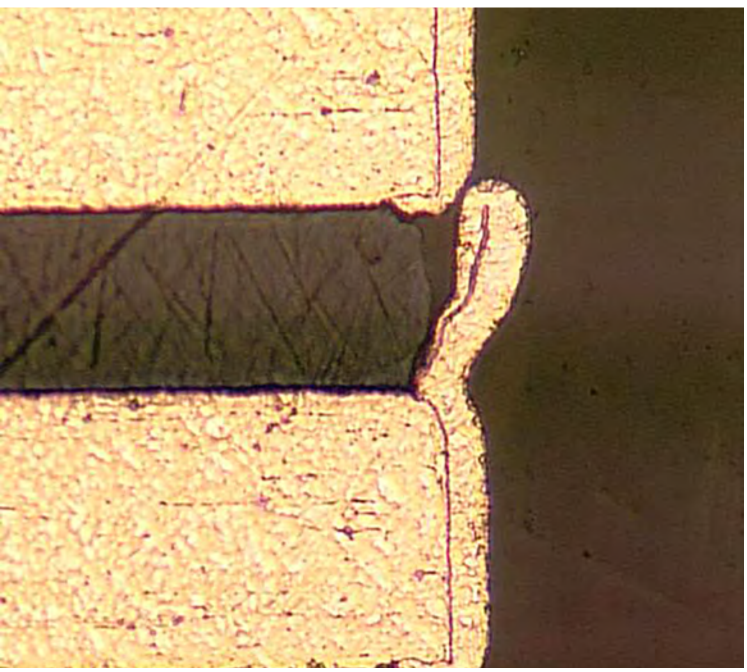


Figure 1: Flexible circuit. Plasma desmeared prior to plating.

It is recognized that plasma desmear leaves the surface in somewhat of an inert state. It is important that the fabricator work very closely with the plasma supplier to ensure an optimized process. It is also critical that residues from the plasma desmear process be removed prior to electroless copper plating.

As is often the case, adhesive-based flexible circuit constructions present another challenge related to achieving void-free metalization. Due to the difficulty in plating flexible materials (particularly when plasma-treated only), one would think that alkaline permanganate treatment could be used to enhance the plating of a void-free deposit. Unfortunately, using a full permanganate cycle can have disastrous consequences (Figure 2).

The suggestion is if one has voiding issues with certain flexible or rigid-flex builds, use a modified permanganate cycle after plasma.

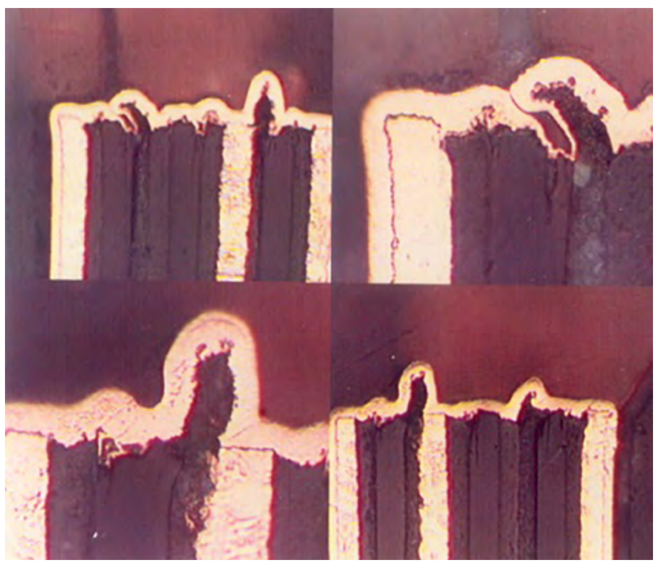


Figure 2: Excessive attack on the adhesive due to a full permanganate cycle.

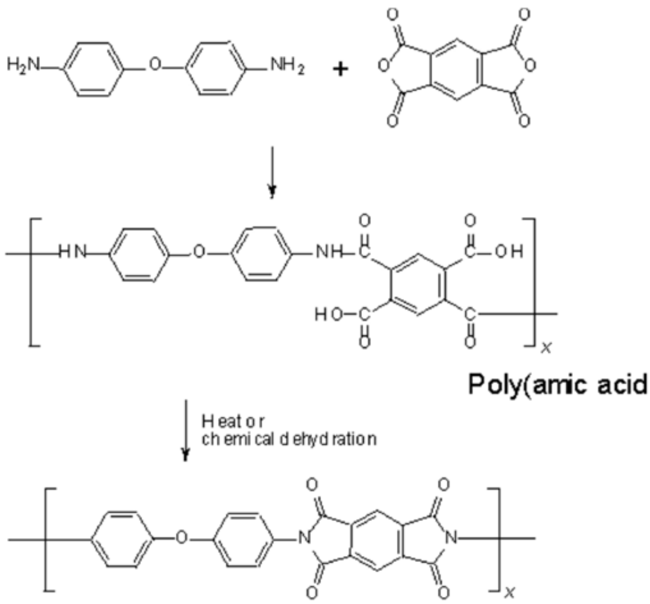


Figure 3: Synthesis of polyimide.

In this case, modified means to skip the solvent-sweller step (either NMP or butyl/caustic-based) and only immerse the parts in the permanganate solution, followed by the neutralization step.

Another critical success factor relates to the operational and chemical conditions within the electroless copper deposition process. These key aspects will now be presented in some detail. Much of what is detailed below are the results of many DOEs. These experiments yielded several important pieces of information.

After several DOEs and other tests, the best condition for eliminating the peeling electroless is to implement what is known as a low deposition rate electroless copper process. The definition of low deposition is a process that deposits 1.0 to 1.5 µin of copper per minute of dwell time in the electroless copper solution. This is achieved by making several modifications to the current process. First, the electroless copper plating solution should be reduced in temperature to 80 – 90°F with a deposition time of 20 minutes. The target deposit thickness is 20 – 25 µin in 20 minutes.

To further slow the deposition rate of the bath, the amount of palladium-based catalyst deposited on the dielectric materials must be reduced. This is accomplished by reducing the cleaner/conditioner as well as catalyst concentrations. Of course, it helps if the cleaner/conditioner chemistry is designed to reduce the thickness of the catalyst layer as it is adsorbed onto the polyimide material. Reducing the catalyst layer thickness, and in turn the amount of palladium on the material, improves adhesion and reduces the tendency of the electroless copper to blister from the substrate.

Polyimide Materials

It is a good idea to review the chemistry involved in polyimide flex materials. Polyimide materials are a condensation reaction of a dianhydride and a diamine compound in a dipolar aprotic solvent, such as DMF and DMAc. Poly(amic acid) is converted to the final polyimide product by thermal imidization. Figure 2 details the reaction showing the synthesis of polyimide. Polyimide has exceptional solvent resistance, high thermal stability, and good dielectric properties.

One of the critical aspects of plating is the ability to activate the substrate to be plated.

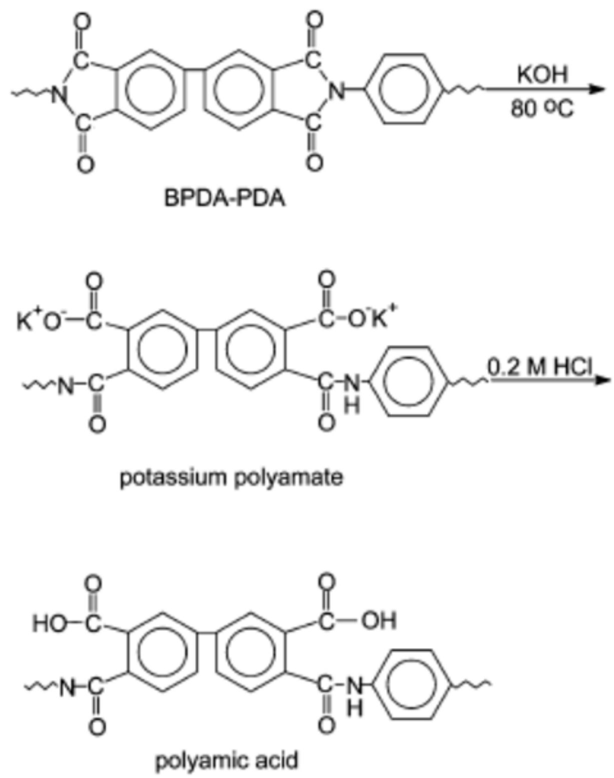


Figure 4: Treating polyimide substrates. KOH reactions of amides.

Amides, such as polyimide, are one such material. Amides and imides are the least reactive of the carboxylic acid derivatives. However, they are attacked by strong base and acid (heat). Base hydrolysis of an amide is shown in Figure 3 using 1M KOH at 80°C for 20 minutes, followed by 0.2N HCl at RT for five minutes.

It is highly recommended that this type of procedure be implemented when attempting to metalize polyimide. This is referred to as a preconditioning step.

Successful metalization of flex and rigid-flex board structures is challenging. However, if one understands the limitations of the wet processes (never assume you can apply FR-4 processing here), a void-free continuous adherent copper deposit is achievable.

A few additional caveats:

- Not all electroless copper processes perform equally
- Plasma leaves the surface in an inert state, which is not easy to obtain adhesion and coverage
- Pay attention to the pre-plating parameters and tighten the operating window as needed
- Flex and rigid-flex is "not your father's FR-4" anymore

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

Smart Stent Detects Narrowing of Arteries

For every three individuals who have had a stent implanted to keep clogged arteries open and prevent a heart attack, at least one will experience restenosis–the renewed narrowing of the artery due to plaque buildup or scarring–leading to additional complications.

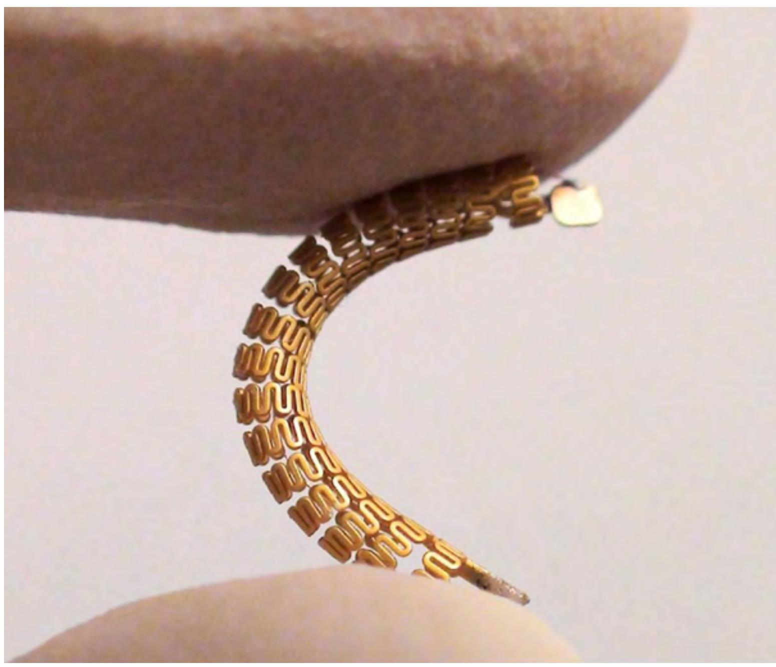
Now, a team led by UBC electrical and computer engineering professor Kenichi Takahata has developed a type of "smart stent" that monitors even subtle changes in the flow of blood through the artery, detecting the narrowing in its earliest stages and making early diagnosis and treatment possible.

"We modified a stent to function as a miniature antenna and added a special micro-sensor that we developed to continuously track blood flow. The data can then be sent wirelessly to an external reader, providing constantly updated information on the artery's condition," said Takahata.

The device uses medical-grade stainless steel and looks like most commercial stents. Researchers say it's the first angioplasty-ready smart stent–it can be implanted using current medical procedures without modifications.

Research collaborator Dr. York Hsiang, a UBC professor of surgery and a vascular surgeon at Vancouver General Hospital, noted that monitoring for restenosis is critical in managing heart disease.

"X-rays such as CT or diagnostic angiograms, which are the standard tools for diagnosis, can be impractical or inconvenient for the patient," said Hsiang. "Putting a smart stent in place of a standard one can enable physicians to monitor their patient's health more easily and offer treatment sooner."




Meet your new sales personal trainer.
Increasing sales isn't rocket science. We use proven methods to help you find the right customer, motivate your sales team, and increase your profits.
We've helped dozens of companies big and small improve their bottom line.
db MANAGEMENT GROUP
Learn how we can help YOU
207-649-0879
danbbeaulieu@aol.com

Automotive Electronics: Past / Present / Future

One World, One Industry

by John Mitchell, IPC-Association Connecting Electronics Industries

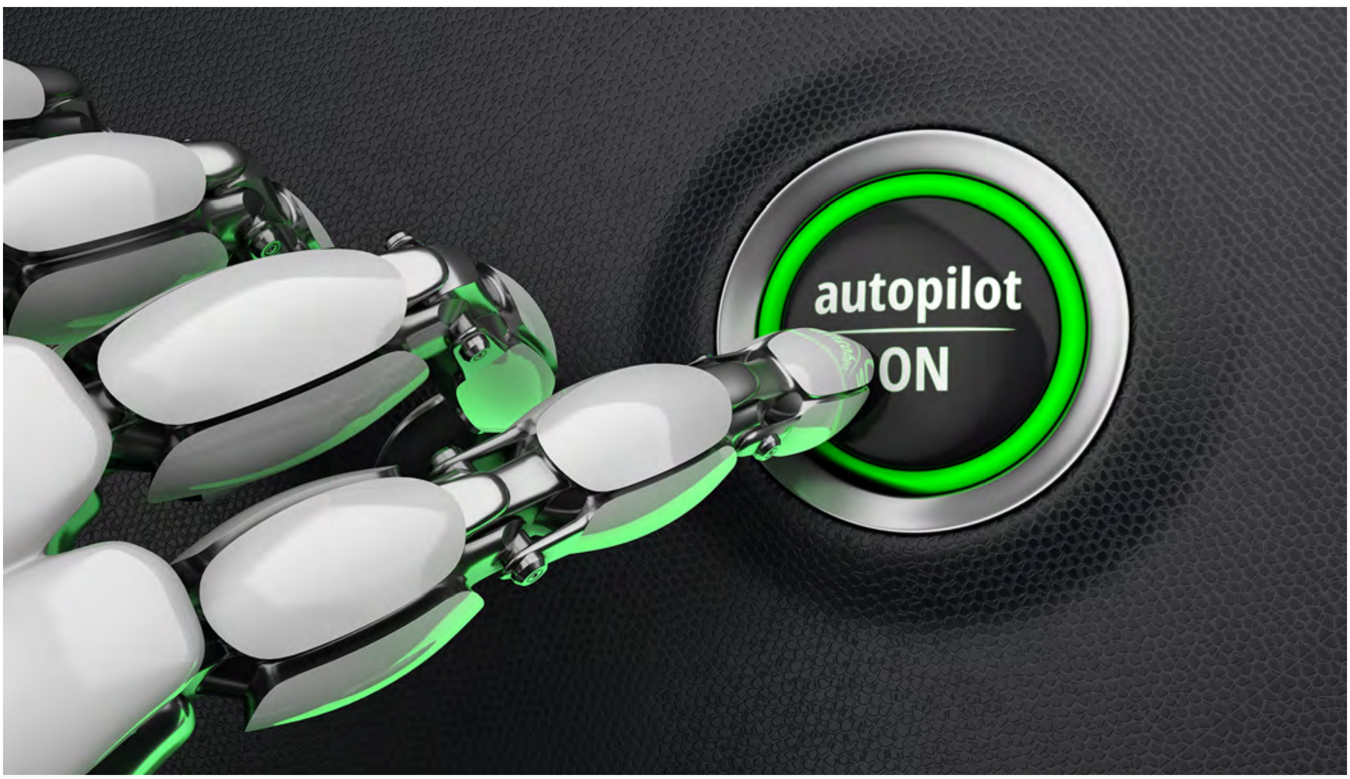
As electronics play an increasingly important role in automotive manufacturing, tremendous change and great progress have been made worldwide. We are at the crossroads of incredible technological advancements, and it's been exhilarating to watch. I am eager to see what happens next.

Did you know that your car is likely the most technologically advanced device you own? It seems weird to refer to a car as a device, but modern automobiles have more chips and circuit boards than your average home internet appliance.

Printed circuit boards play a vital role in the performance and reliability of many critical systems. Most of today's cars have between 30 and 80 separate electronic controllers.

We are seeing changes in many different areas of technology related to automotive electronics and I'd like to cover three in more detail: artificial intelligence (AI), augmented reality (AR) and additive manufacturing.

Artificial intelligence, computers and computer software capable of intelligent behavior, is moving ahead at a rapid pace. Today, we have driverless cars that can navigate traffic on their own. According to Gartner Reports, autonomous vehicles will comprise 25% of the market by 2030. It is suggested that AI will improve the auto manufacturing sector in several ways, among them: less equipment failure, more productive employees through robot-human collaboration, and fewer quality problems.



Augmented reality (AR) adds texture and information to the world we already live in, providing a deeper learning experience by integrating digital information in a real-time environment. For the automotive industry, AR will likely be instrumental in future advancements. Volvo uses Microsoft's HoloLens, a type of AR glasses, that enables production line workers to view assembly instructions digitally in real-time while working to put together parts of the vehicle.

For consumers, Volkswagen recently launched an app for its luxury brand Audi that allows car shoppers to experience an AR-based test drive without being close to the automobile.

Additive manufacturing, commonly known as 3D printing, is also something that has come a long way in the past several years. Additive manufacturing enables the production of components more quickly, and it shortens the design process by digitally changing part of the model. It allows changes to be made more quickly than traditionally.

Components that can be produced with additive manufacturing in automotive applications include exhaust and emissions, fluid handling systems, and exterior parts such as bumpers. Additive manufacturing is expected to be used in the very near future on interiors and seating, tires and suspension, electronics, framework and doors and engine components.

With all this rapid change, what can we do to move forward? IPC is involved in creating standards for the automotive industry and has found the advances in electrical systems in car manufacturing a natural growth industry for electronics manufacturing.

Two years ago, IPC developed its first automotive addendum, IPC-6012DA, Automotive Applications Addendum to IPC-6012D Qualification and Performance Specification for Rigid Printed Boards. The addendum addresses the reliability of rigid printed boards which must survive the vibration and thermal cycling environments of electronic interconnects within the automotive industry.

Under development are additional automotive addendums for the following IPC standards: IPC-A-610, Acceptability of Electronic Assemblies; IPC J-STD-001, Requirements for Soldered Electrical and Electronic Assemblies; and IPC/WHMA-A-620, Requirements and Acceptance for Cable and Wire Harness Assemblies.

We encourage you to join us in standards development activities to ensure the needs you face in this ever-changing and rapidly advancing industry are met. We welcome your input and expertise as we all embrace the future of automotive electronics. **PCB007**

John Mitchell is president and CEO of IPC-Association Connecting Electronics Industries. To read past columns or to contact Mitchell, click here.

Taiyo America Direct-Imaging Solder Mask Developments

I-Connect007's Pete Starkey and Taiyo America's regional sales manager, Don Monn, discuss the latest developments in direct imaging solder mask, specifically a direct image version of their high-performance one. With the continuous reduction in feature sizes, direct imaging has become a necessity to achieve the required registration on a panel.

Why Choose Fein-Line?

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

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

Mr. Feinberg and his associates specialize in:

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

Dan (Baer) Feinberg

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

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

Fein-Line ASSOCIATES, INC.

www.feinline.com

Recent Highlights from PCB007

1 The Augmented World Expo: Go XR or Become Extinct ▶

Now on its ninth year, the Augmented World Expo (AWE USA) is perhaps the largest event for professionals focused on providing science fiction-like abilities through XR (cross reality) and associated wearable technology. This year's event showcased over 100,000 square feet of exhibit space and featured numerous presentations and discussions covering topics ranging from the latest and greatest uses and devices for XR to the business of marketing and monetizing it.

2 The Institute of Circuit Technology Annual Symposium 2018 ▶

ICT technical director Bill Wilkie is well-known for choosing notable venues for Institute of Circuit Technology events, and his choice for this year's Annual Symposium was the National Motor Museum, located in the village of Beaulieu in the heart of the New Forest, a national park in the county of Hampshire in Southern England. The region is known for its heathland, forest trails and native ponies.

3 The Shaughnessy Report: Got Flex? ▶

Welcome to the first issue of *Flex007 Magazine*. This new quarterly magazine is dedicated to flex system designers, electrical engineers, flex PCB designers, and anyone responsible for integrating flex into their products at the OEM/ CEM level.

4 Standard of Excellence: Making Your PCB Fabricator Your PCB Partner ▶

Are your PCB needs exceeding your vendor base's capabilities? Do you feel that your suppliers are no longer able to keep up with your PCB needs? Do your designers feel limited by your vendors' limited capabilities? Are you finding an issue locating alternate sources for PCBs due to the consolidation of the larger shops in our industry? If so, you are not alone.

5 ACE's AccuWrap Simplifies Multiple Sequential Laminations with Blind Vias ▶

At DesignCon 2018, I spoke with James Hofer, general manager for Accurate Circuit Engineering, a quick-turn fabricator based in Santa Ana, California. James gave me a preview of AccuWrap, a new type of processing technology that lets designers reduce the amount of copper during sequential laminations while still meeting IPC specs, which should be of great interest to RF designers.

6 APCT Moves into Rigid-Flex with Cartel, Cirtech Acquisition ▶

When I spoke with APCT President Steve Robinson a year ago, he said he was interested in adding flex and rigid-flex capabilities, and working closely with designers and engineers. With the recent acquisition of Cartel and their subsidiary Cirtech, APCT now has a flex and rigidflex facility, along with military and aerospace certifications. At DesignCon 2018, I asked Steve to discuss these acquisitions and what they mean for APCT and their customers.

7 Candor is Breaking Boundaries with Flex-Core Boards ▶

Sunny Patel, Candor's technical sales manager, remarked about the accomplishment, "We are pushing the limits on our flex manufacturing, going for much thinner laminates and different material stiffeners."

8 It's Only Common Sense: Producing the Greatest Products Possible ▶

Laurene Powel Jobs, wife of Apple Co-Founder and CEO Steve Jobs, was once quoted describing how her husband and legendary designer Jony Ive would spend hours discussing corners. Yes, that's right—corners.

9 AT&S Offers Interconnection Tech for Next-Generation 5G Mobile Communications ▶

Significantly higher data rates and volumes together with high power densities are resulting in rising requirements for interconnection technologies, too. The new 5G mobile communications generation is undoubtedly an overarching and defining trend here, with applications ranging from infrastructure (base stations) to terminal devices (mobile phones), as well as real-time automation scenarios.

10 Experts Discussion: The Flex Technologists Speak ▶

For our first issue of the *Flex007 Magazine*, we invited a group of flexible circuit experts to discuss their work in this rapidly growing segment. Participants included Jonathan Weldon of DuPont, Mark Finstad of Flexible Circuit Technologies, and Scott McCurdy and Scott Miller of Freedom CAD. In a free-wheeling discussion with Andy Shaughnessy and Barry Matties, these technologists share their thoughts on the challenges and opportunities in flexible circuits, as well as what constitutes the cutting edge of flex right now.

Career Opportunities

Pssst!
Are You Looking for Someone?

Place your notice in our Help Wanted section.

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

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

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

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

Technical Support Engineer, Germany

We are looking for a Technical Support Engineer to join our team at our German facility in Kirchheimbolanden. The successful candidate will assist potential customers and current customers in appreciating the benefits of using–and optimizing the use of–Ventec materials in their printed circuit board manufacturing processes, and so enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. The Technical Support Engineer will provide a two-way channel of technical communication between Ventec's production facilities and UK/European customers.

Skills and abilities required for the role:

- Scientific/technical educational background.
- Experience in the PCB industry in engineering and/or manufacturing
- Good communications skills (German and English), able to write full technical reports for group or customer distribution.
- Ability to work in an organized, proactive, and enthusiastic way.
- Ability to work well both in a team as well as an individual.
- Good user knowledge of common Microsoft Office programs.
- Full driving license essential.
- Willingness to travel regularly throughout Europe and occasionally to Asia.

What we offer:

- Excellent salary & benefits commensurate with experience

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

Please forward your resume to
applytoventec@ventec-europe.com.

apply now

Mentor®
A Siemens Business

Technology Communications Writer/Content Manager
Board Systems Division

Mentor Graphics, a Siemens business, is a global technology leader in EDA software, enabling global companies to develop new and highly innovative electronic products in the increasingly complex world of chip, board, and system design.

Job Duties:
The Mentor printed circuit board (PCB) technical writer/content manager will:

- Write and produce high-quality content for various properties (blogs, product collateral, technical white papers, case studies, industry publications, etc.).
- Gather research and data, interview subject matter experts, and transform complex information into clear, concise marketing communications.
- Manage projects across multiple PCB product teams (high-speed design/analysis, advanced packaging, board design) within a deadline-driven environment.

Job Qualifications:
The ideal candidate should possess:

- Strong writing and editing skills with experience in PCB design technologies.
- Desktop publishing skills (InDesign) using project templates and knowledge of online publications and social media.
- A technical background (B.S. in electrical engineering or computer science preferred; this role works closely with the PCB division's technical marketing engineers and managers.
- Solid project planning and management skills; appreciation for adhering to deadlines; creativity for turning technical information into compelling content; teamwork and strong interpersonal communications skills; ability to be a self-starter.

apply now

Career Opportunities

ZENTECH

Zentech Manufacturing is Hiring Manufacturing Engineers!

Are you looking to excel in your career and grow professionally in a thriving business? Zentech, established in Baltimore, Maryland, in 1998, has proven to be one of the premier electronics contract manufacturers in the U.S.

Zentech is rapidly growing and seeking to add a Manufacturing Engineer. We offer an excellent benefit package including health/dental insurance and an employer-matched 401k program, and we hold the ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including: ISO:9001, AS9100, DD2345, and ISO 13485. Zentech is an IPC Trusted Source QML and ITAR registered.

U.S. citizens only need apply.

Please email resume to sales@zentech.com.

apply now

MacDermid Enthone
ELECTRONICS SOLUTIONS

Technical Service Rep, Northeast

Do you have what it takes? MacDermid Enthone Electronics Solutions is a leading supplier of specialty chemicals, providing application-specific solutions and unsurpassed technical support.

The position of the Technical Service Rep will be responsible for day-to-day support for fabricators using MacDermid Enthone's chemical products. The position requires a proactive self-starter who can work closely and independently with customers, sales group members and management to ensure that customer expectations and company interests are served.

- Thoroughly understand the overall PCB business, and specifics in wet processing areas
- Prepare action plans for identification of root cause of customer process issues
- Provide feedback to management regarding performance
- Create and conduct customer technical presentations
- Develop technical strategy for customers
- Possess the ability to calm difficult situations with customers, initiate a step by step plan, and involve other technical help quickly to find resolution

Hiring Profile
- Bachelor's Degree or 5-7 years' job-related experience
- Strong understanding of chemistry and chemical interaction within PCB manufacturing
- Excellent written and oral communication skills
- Strong track record of navigating technically through complex organizations
- Extensive experience in all aspects of customer relationship management
- Willingness to travel

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Master Instructor

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

For more information, click below.

apply now

Mentor®
A Siemens Business

PCB Manufacturing, Marketing Engineer

Use your knowledge of PCB assembly and process engineering to promote Mentor's Valor digital manufacturing solutions via industry articles, industry events, blogs, and relevant social networking sites. The Valor division is seeking a seasoned professional who has operated within the PCB manufacturing industry to be a leading voice in advocating our solutions through a variety of marketing platforms including digital, media, trade show, conferences, and forums.

The successful candidate is expected to have solid experience within the PCB assembly industry and the ability to represent the Valor solutions with authority and credibility. A solid background in PCB Process Engineering or Quality management to leverage in day-to-day activities is preferred. The candidate should be a good "storyteller" who can develop relatable content in an interesting and compelling manner, and who is comfortable in presenting in public as well as engaging in on-line forums; should have solid experience with professional social platforms such as LinkedIn.

Success will be measured quantitatively in terms of number of interactions, increase in digital engagements, measurement of sentiment, article placements, presentations delivered. Qualitatively, success will be measured by feedback from colleagues and relevant industry players.

This is an excellent opportunity for an industry professional who has a passion for marketing and public presentation.

Location flexible: Israel, UK or US

apply now

Career Opportunities

SMT Field Technician

Huntingdon Valley, PA

Manncorp, a leader in the electronics assembly industry, is looking for an additional SMT Field Technician to install and support our wide array of SMT equipment.

Duties and Responsibilities:

- Manage on-site installation of equipment and training of customers
- Troubleshoot and diagnose technical problems by phone, email or additional on-site visits, when necessary during post-installation service and support
- Assist with demonstrations of equipment to potential customers
- Build and maintain positive relationships with customers
- Produce service reports
- Cooperate with technical team and share information across the organization
- Assist with the crating and uncrating of equipment

Requirements and Qualifications:

- Three to five years of experience with SMT equipment, or equivalent technical degree
- Strong mechanical and electrical troubleshooting skills
- Proficiency in reading and verifying electrical, pneumatic, and mechanical schematics/drawings
- Organizational skills, detail orientated and capable of multitasking
- Good written and oral interpersonal skills with an ability to work under minimum supervision
- Ability to work with little supervision while traveling
- Availability for frequent travel
- Ability to arrange and schedule service trips
- USA Citizenship required

We Offer:

- Health & dental insurance
- Retirement fund matching

apply now

Zentech Manufacturing is Hiring!

Looking to excel in your career and grow professionally in a thriving business? Zentech, established in Baltimore, Maryland, in 1998, has proven to be one of the premier electronics contract manufacturers in the U.S.

Zentech is rapidly growing and seeking to add a circuit layout team leader with hands-on Mentor Xpedition experience. We offer an excellent benefit package including health/dental insurance and an employer matched 401k program, and we hold the ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including: ISO:9001, AS9100, DD2345, and ISO 13485. Zentech is an IPC Trusted Source QML and ITAR registered.

U.S. citizens only need apply.
Please email resume to
sales@zentech.com.

apply now

Events Calendar

IPC E-Textiles 2018 Workshop ▶
September 13, 2018
Des Plaines, Illinois, USA

electronica India productronica India ▶
September 26–28, 2018
Bengaluru, India

electronicAsia 2018 ▶
October 13–16, 2018
Hong Kong

SMTA International ▶
October 16–17, 2018
Rosemont, Illinois, USA

TPCA Show 2018 ▶
October 24–26, 2018
Taipei, Taiwan

electronica 2018 ▶
November 13–16, 2018
Munich, Germany

HKPCA/IPC International Printed Circuit & South China Fair ▶
December 5–7, 2018
Shenzhen, China

48th NEPCON JAPAN ▶
January 16–18, 2019
Tokyo Big Sight, Japan

IPC APEX EXPO Conference and Exhibition ▶
January 26–31, 2019
San Diego, California, USA

DesignCon 2019 ▶
January 29–31, 2019
Santa Clara, California, USA

Additional Event Calendars

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

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

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

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

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

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

CONTRIBUTING TECHNICAL EDITOR: **DAN FEINBERG**
baer@iconnect007.com

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

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © ALEX MALIKOV**

PCB007 MAGAZINE

PCB007 MAGAZINE®
is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

July 2018, Volume 8, Number 7
PCB007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to PCB007 Magazine:

AUGUST: Reliability
In the new world of IoT, autonomous vehicles and 5G.