I-Connect007

FEBRUARY 2022

DESIGN007 MAGAZINE

CHANGE: RESISTANCE IS FUTILE

DESIGN007
MAGAZINE

Change: Resistance is Futile

It seems like there's never a dull moment in the design segment. That's what makes this career exciting—and occasionally stressful. There's a lot of change going on in PCB design—some positive, some negative. In this issue we focus on some of the changes coming in PCB design, and how to react when you're thrust into a new, confusing situation, such as moving to a new company, switching to a new EDA tool, or dealing with new, cutting-edge technology.

FEATURE ARTICLES & INTERVIEW

22 **Keep the Change? No, Embrace It**
by Tamara Jovanovic

36 **Design Milestones: More Than Signposts Along the Road**
Interview with Happy Holden, Kelly Dack, and Bob Tise

46 **Changes on the Horizon: Is Resistance Futile?**
by Stephen V. Chavez

FEATURE COLUMNS

12 **PCB Design Challenges: Change is Good**
by Barry Olney

60 **SAP—Changing the Way You Look at PCB Design**
by Tara Dunn

DESIGN007 MAGAZINE

ARTICLE

26 **DFM 101: Plating Methods**
by Anaya Vardya

SHORTS

11 **Real Time with... IPC APEX EXPO 2022: Design Competition Group Interview**

35 **All Systems Go! Bridging the Gap Between Design and Analysis**

67 **Real Time with... IPC APEX EXPO 2022: Disruptive Technologies Growing Business**

DEPARTMENTS

95 **Career Opportunities**
110 **Educational Resource Center**
111 **Advertiser Index & Masthead**

SPECIAL GALLERY SECTION

72 **IPC APEX EXPO 2022 STEM Event**

COLUMNS

10 **Resistance is Futile. Or Is It?**
by Andy Shaughnessy

18 **Livin' in a PCB Stakeholder's Paradise**
by Kelly Dack

32 **Selecting Prepreg for Millimeter-wave PCB Applications**
by John Coonrod

52 **Design for Test, Part 2**
by Vern Solberg

56 **How to Select the Best Resin Category for Your Application**
by Beth Turner

64 **Managing Risk With Model-based Engineering**
by David Wiens

68 **The PCB Design Secret Sauce for RF Applications**
by Matt Stevenson

HIGHLIGHTS

30 **PCB007**
50 **MilAero007**
92 **Top Ten Editor's Picks**

FLEX007

This Month in Flex

The flexible circuit segment is constantly evolving. What constitutes a state-of-the-art flex or rigid-flex circuit today? This month, we bring you a chapter from the new Siemens eBook, *The Printed Circuit Designer's Guide to Stackups, the Design Within the Design,* written by Bill Hargin and published by I-Connect007. This chapter provides a working outline for designing a cutting-edge rigid-flex stackup. We also bring you a variety of video interviews with flexible circuit technologists from IPC APEX EXPO 2022.

FLEX007 ARTICLES

76 **Book Excerpt: Chapter 7 Rigid-Flex Materials**
by Bill Hargin

86 **Kris Moyer Discusses New IPC Design Role**
Interview with Kris Moyer

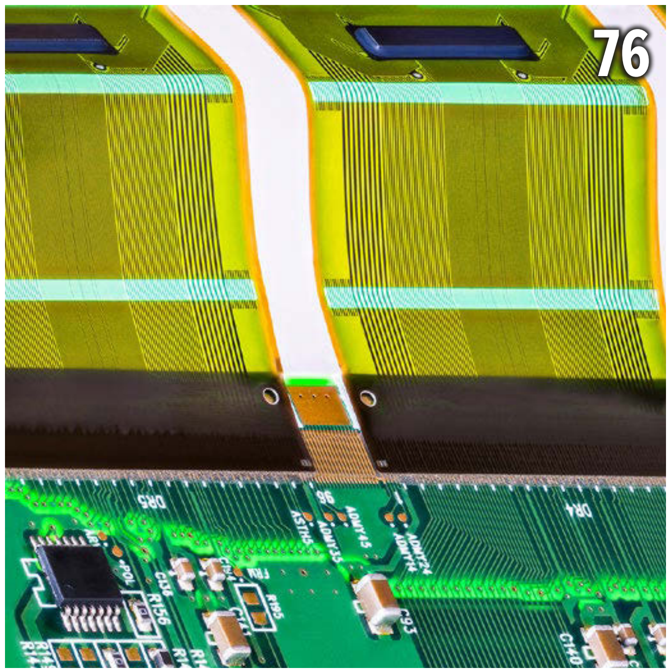
HIGHLIGHTS

84 **Flex007**

FLEX007 VIDEO SHORTS

81 **Real Time with... IPC APEX EXPO 2022: High-Tech Rigid-Flex Boards and More**

90 **Real Time with... IPC APEX EXPO 2022: Flexible Circuit Technologies: Rigid-Flex Specialists**



Resistance is Futile. Or Is It?

The Shaughnessy Report

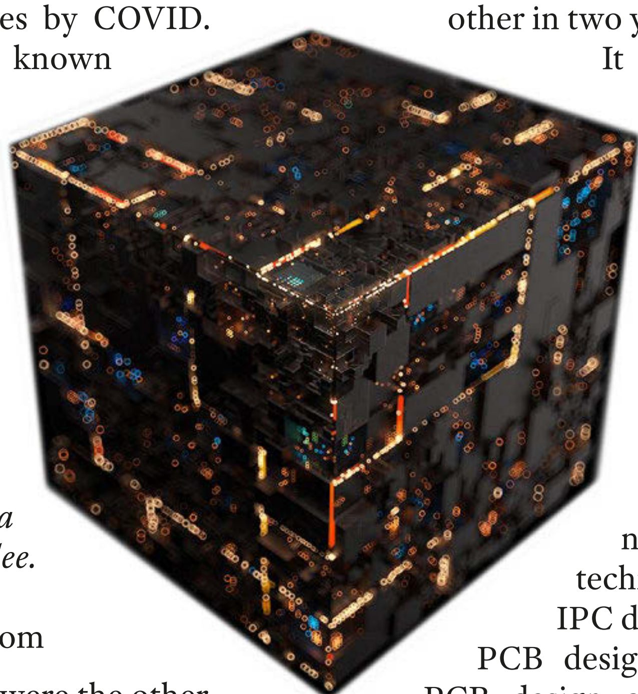
by Andy Shaughnessy, I-CONNECT007

Admit it. You thought that there was a chance that IPC APEX EXPO 2022 would be cancelled at the last minute, or held virtually, didn't you?

A lot of people did. After all, we're still not out of the pandemic and California has been hit harder than most states by COVID. And California has been known to change their COVID regulations faster than Stevie Nicks changes outfits on stage. I figured the state might pull some kind of shenanigans at the last minute:

Each trade show attendee must be accompanied by a registered nurse supplied by the state of California at a cost of $10,000 per attendee. Now, have a great show!

—Governor Gavin Newsom



But I was wrong, and so were the other Negative Nellies in the industry. The show went better than anyone expected. The technical conference and Professional Development classes and committee meetings were full of attendees. Even on the final day of the expo, there was still decent traffic in the aisles. International attendance was down overall, but Technical Editor Pete Starkey, Ventec's Alun Morgan, and Elmatica's Jan Pedersen and Didrik Bech were able to make it from Europe to San Diego with no problem.

Best of all, we were all able to get together in person, and the whole show had the vibe of a homecoming. For many in this industry, IPC APEX EXPO 2020 was our last live trade show before the pandemic, and we hadn't seen each other in two years.

It was especially heartening for us middle-aged folks to see so many young people at the show. I saw a lot of attendees who were in their 20s and 30s. And the IPC's STEM students from a local high school were all over the show floor, asking about the technology and talking to technology icons.

IPC did quite a bit to highlight PCB designers at the show. The PCB design competition got started months ago, with seven designers competing in different heats, and the three finalists battled it out at IPC APEX EXPO, albeit over Zoom from their homes in the US, the UK, and Poland. The three finalists had just a few hours to design a board using Altium Designer, which none of them were familiar with, but they all managed to get up to speed quickly and do well enough to complete the design.

Let's hear it for first-place winner Rafal Przestawski, as well as runners-up Nick Wallis and Elliot Wakefield. Elliot isn't even a designer—he's a tinkerer and a maker who has designed his own remote-controlled lawnmower, but he'd never designed a board before. These young people will surprise you every time.

Speaking of surprises, it seems like there's never a dull moment in the design segment. There's a lot of change going on in PCB design—some positive, some negative. In this issue of *Design007 Magazine* we focus on some of the changes coming in PCB design, and how to react when you're thrust into a new, confusing situation, such as moving to a new company, switching to a new EDA tool, or dealing with new, cutting-edge technology.

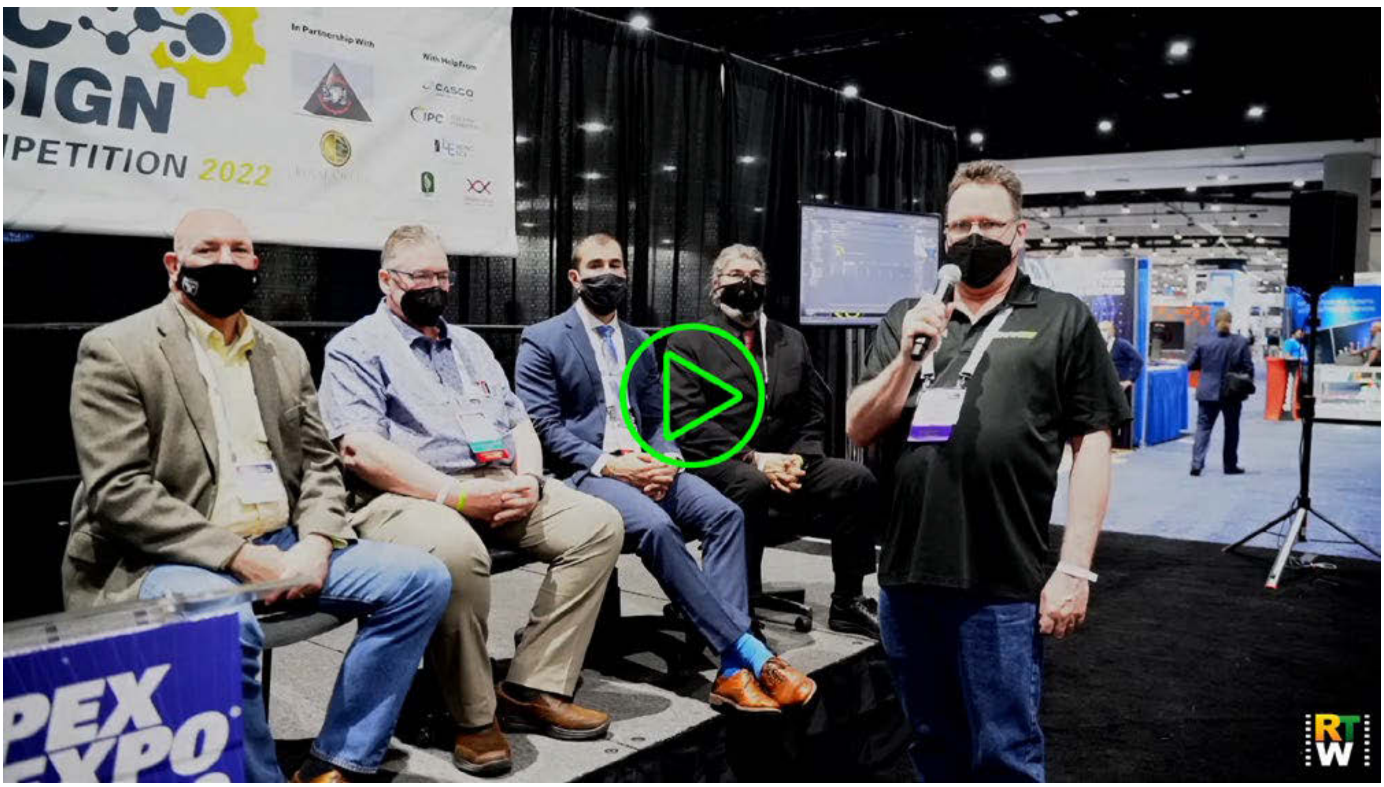
Our contributors discuss some of the changes they see happening in the industry, especially those changes that seem to be happening to them, not with them. As Stephen Chavez says in his article, when you're undergoing an unwelcome change in your department or company, it's easy to imagine the voice of the *Star Trek* entity the Borg saying, "Resistance is futile." Do you resist or become assimilated and join the hive mind?

Now, the show season is under way. DesignCon is around the corner. Are you back to traveling again? I hope to see you on the road. **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 20 years. He can be reached by clicking here.

Real Time with... IPC APEX EXPO 2022: Design Competition Group Interview

In this interview, Editor Andy Shaughnessy speaks with the designers who worked with IPC to create and judge the IPC Design Competition. Kris Moyer, Kevin Kusiak, Russ Steiner, and Steve Roy discuss their involvement in the setting up the PCB design, and what they looked for as judges of the contest.



PCB Design Challenges: Change is Good

Beyond Design

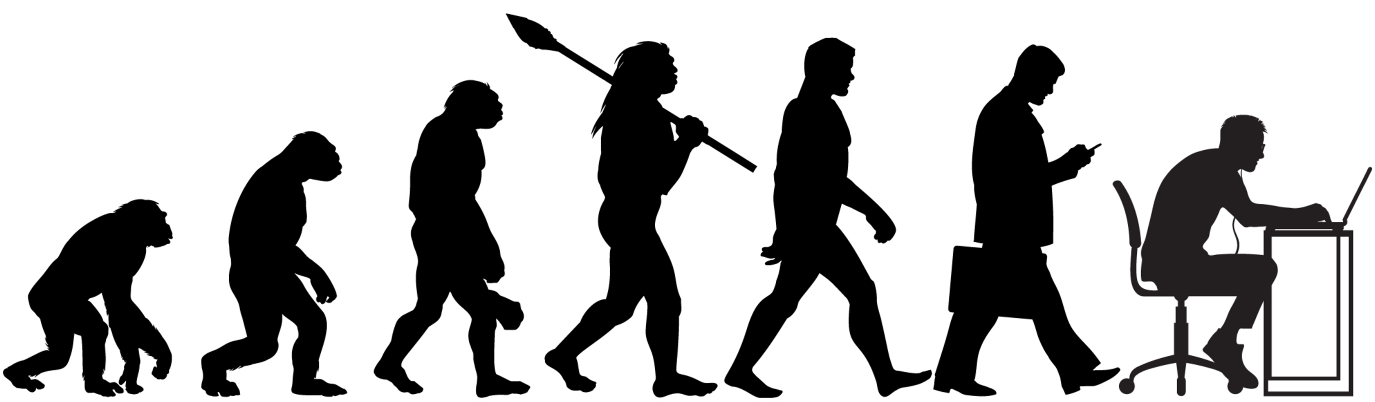
Feature Column by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

In 2022, PCB designers are faced with two big challenges: demands for increased performance and a condensed product footprint. So, what's new? I recall some 50-odd years ago the challenges for the electronics professional were much the same. I had just become comfortable with valves and then came diodes, transistors, and LEDs. After mastering germanium and then silicon diodes/transistors, along came op-amps. Now, these devices were very mysterious. Basically, they have several transistors acting as pre-built functional blocks packaged into an 8-pin DIL package. They had many uses from amplifiers to comparators and issued in the start of digital electronics. And who could forget the 555 timer with its millions of applications?

Next were digital logic gates. Now, these multiple gate devices could be run at an astonishing 10 MHz which seemed incredibly fast at the time. You could combine them to perform almost any digital function. Analog-to-digital (ADC) and digital-to-analog (DAC) converters allowed us to mix the analog world with the digital.

Then in 1974, Intel released the 8080 8-bit microprocessor followed closely by the 8086 16-bit chip. In the early 1980s, I was working in the Microprocessor Research Lab at the University of Western Australia. We were building portable computers into desks on wheels for the Masters' and PhD candidates using these processors. These machines were a major step-up from the departmental PDP-1140. The Z80 CPU, which was an extension of the 8080, became very popular with enthusiasts. I bought a UK-developed MicroBee Z80 kit which had 16K of RAM expandable up to 64K (if I would ever have a use for such a huge amount of memory).

I recall that some colleagues totally rejected computers and never took the time to step up to the new emerging technology and were left behind. Whereas those of us who embraced it were whisked along with the trending computer upsurge.



Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

Use manufacturing data to generate a 3D facsimile of the finished product.

Verify

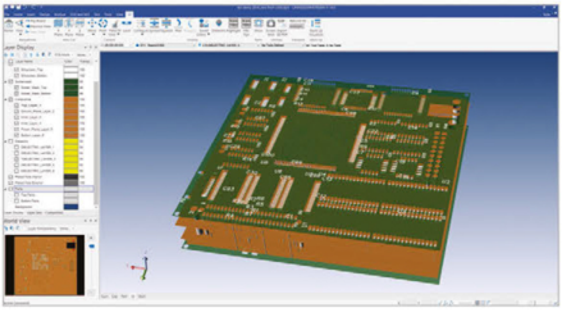
Ensure that manufacturing data is accurate for PCB construciton.

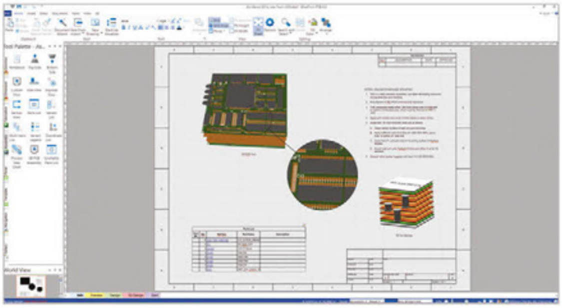
Panelize

Minimize costs in both PCB fabrication and assembly through maxiumum panel usage.

Document

Quickly and easily specify the PCBs form, fit and function.





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

DownStream Technologies, offers an integrated manufacturing data preparation solution for PCB post processing including manufacturing data visualization, stack-up definition, PCB panel visualization and support for 3D PCB documentation. This environment allows PCB designers and engineers to visualize, verify and document a printed circuit board before manufacturing.

DownStream Technologies

800-535-3226 | downstreamtech.com

NOW WITH 3D

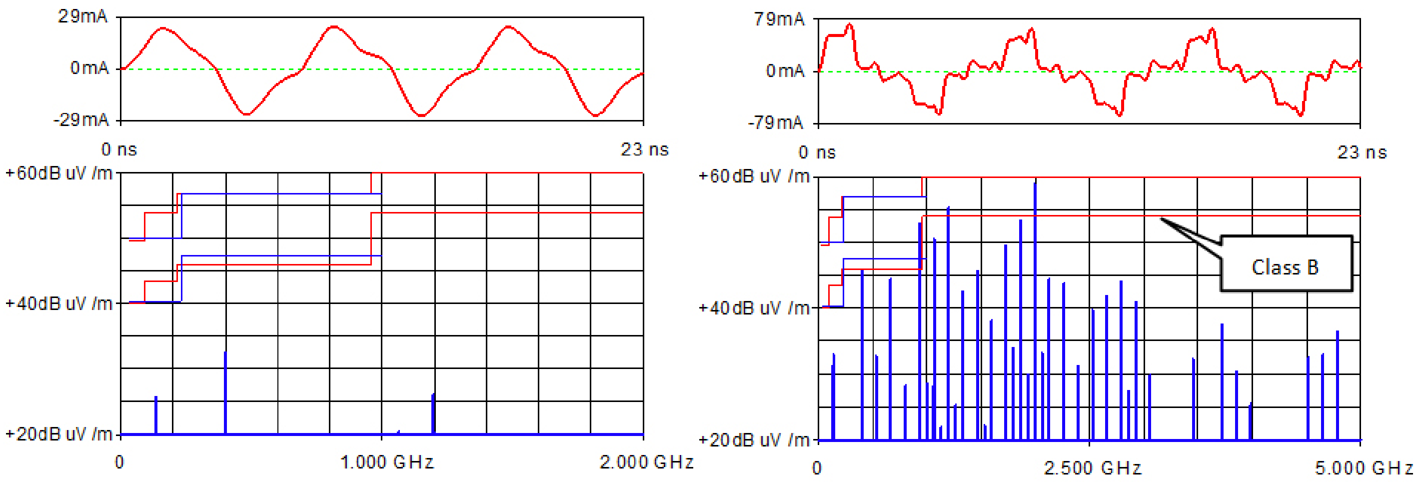


Figure 1: Radiated emissions from the 10 ns edge rate (left) and 1 ns (right).

By 1986, surface mount technology (SMT) had gained popularity and was accounting for 10% of the market. The need to increase functionality and to reduce the product size and time-to-market forced product designers to adopt this new technology. In 1990, I took on the challenge, attended the many trending SMT conferences and training courses, and changed our product development to the process.

Programmable devices enabled us to use the same hardware with different strains of software allowing the creation of multiple products from the one PCB design. FPGAs reduced the need for application-specific ICs. The list goes on, but there was always constant change.

As system performance increases, the PCB designer's challenges become more complex. The impact of lower core voltages, high frequencies, and faster edge rates has forced us into the high-speed digital domain. Power consumption has become a primary factor for FPGA selection. Whether the concern is absolute power consumption, usable performance, battery life, thermal challenges, or reliability, power consumption is at the center of it all. To reduce power consumption, IC manufacturers have moved to lower core voltages and higher operating frequencies which, of course, mean faster edge rates. The enhancements in driver edge rates have a significant impact on signal quality, timing, crosstalk, and EMC.

The faster edge rate for the same frequency and same length trace creates ringing in an unterminated transmission line. This also has a direct impact on radiated emissions. Figure 1 shows the massive increase in emissions from 10 ns to 1 ns. When dealing with sub-nanosecond rise times, the emissions can easily exceed the FCC/CISPR Class B limits for an unterminated transmission line.

As signal rise times increase, consideration should be given to the propagation time and reflections of a routed trace. If the propagation time and reflection from source to load are longer than the edge transition time, an electrically long trace will exist. If the transmission line is short, reflections still occur but will be overwhelmed by the rising or falling edge and may not pose a problem. But even if the trace is short, termination may still be required if the load is capacitive or highly inductive, to prevent ringing. Note that series terminators are the most effective for high-speed design.

Power distribution network (PDN) planning is another relatively new technology that has become an essential, interrelated component of signal integrity analysis. However, main-

stream PCB developers have yet to adopt PDN analysis as a common design process. But now that the technology is proven and the uptake costs have decreased dramatically, there is no reason why all designers should not take advantage of the technology to improve the reliability and performance of their products. The goal of robust PDN planning is to design a stable power source for all the required onboard power supplies (Figure 2).

The trend in lower DC voltages also requires tighter voltage noise tolerances and higher currents. Market demands are forcing product designers to create PDNs with greater density, higher power efficiencies and lower costs, making the process even more challenging.

Soon there will be major advances in the following fields:

1. Renewable energy technologies to reduce our carbon footprint and climate change. Fusion reactors. Electric and hydrogen vehicles and battery technology.
2. Quantum computers. They will significantly shorten product development cycles and reduce the costs for R&D. New data encryption technologies.
3. Artificial intelligence. Advances in self-driving vehicles. Virtual spaces bringing us closer together.
4. A new era in medicines. CISPRA gene editing and mRNA vaccines and their many uses.
5. Communications and streaming services. 5G networks are already in use. And constellations of Starlink satellites will bring low latency internet access globally and to lower earth orbit.
6. Space tourist, space exploration, and asteroid mining.
7. 3D printed construction. A cost-effective way to build houses, factories, and space structures. Not forgetting built-up multilayer PCB construction.

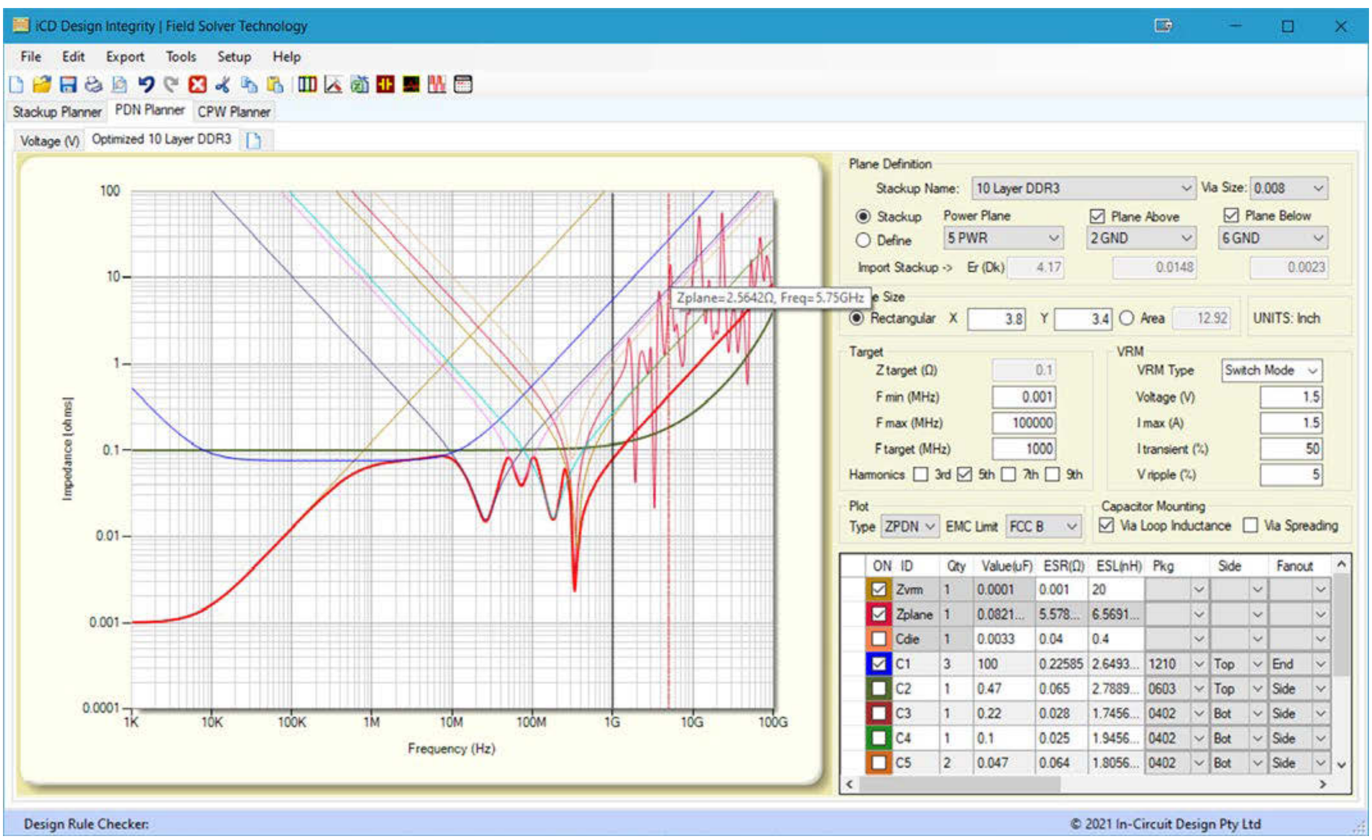


Figure 2: AC impedance profile of a DDR3 supply. (Source: iCD PDN Planner)

I'm sure there will be many more. All these technologies require new or adaptive ways of producing electronic products. Smaller size, flexible design with improved performance all adds up to advances in PCB design techniques. Designers can expect to see more high-speed serial links and faster buses and memory devices requiring signal and power integrity skills. With the advancements in automotive technologies, one would expect a merger of microwave/RF techniques in the PCB design realm.

There has also been a change in the demographics of the PCB design team over the past decade. Boards have been chronically designed by a PCB designer who is a specialist in the field. However, with fewer designers entering the workforce than retiring, the response has been to involve more degreed engineers with PCB design. The engineers will not be specialists and will only design one or two boards a year, but they bring advantages too. They are much more familiar with the constraints, and why they are important. They may also have a much greater understanding of signal integrity, power distribution network design, signal propagation, and thermal characteristics. Future designs will be more complex and require these skills.

We feel comfortable and secure in doing things the way we have always done them. But constant change is part of being in the field of electronics. So why not embrace change and learn new technologies and master new skills? I have always said that change is good. It brings new challenges but also rewarding opportunities. Look for opportunities and act on them.

Key Points

- As system performance increases, the PCB designer's challenges become more complex.
- The impact of lower core voltages, high frequencies, and faster edge rates has forced us into the high-speed digital domain.
- If the propagation time and reflection from source to load are longer than the edge transition time, an electrically long trace will exist.
- Power distribution network (PDN) planning is another relatively new technology that has become an essential, interrelated component of signal integrity analysis.
- The trend in lower DC voltages also requires tighter voltage noise tolerances and higher currents.
- Smaller size, flexible design with improved performance all adds up to advances in PCB design techniques.
- With fewer designers entering the workforce than retiring, the response has been to involve more degreed engineers with PCB design.
- Constant change is part of being in the field of electronics. **DESIGN007**

Resources

1. Beyond Design: Signal Integrity Part 1 of 3, by Barry Olney, *The PCB Design Magazine,* October 2014.

Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au. To read past columns or contact Olney, click here.

Livin' in a PCB Stakeholder's Paradise

Target Condition

by Kelly Dack, CIT, CID+

A few months ago, I was offered a unique opportunity to serve as the AltiumLive Connect 2022 show host for the virtual event during the last full week of January.

It started with an idea I had during a meeting with Altium last October, to create a musical promo for the annual summit. I love to write songs and I've written a few which have been used commercially. But for a technical electronics trade show, I knew that I would have to reach far outside my normal styles of composition.

I thought of my time working for Prototron Circuits in Seattle where I had the chance to have lunch with Anthony Ray, aka, Sir Mix-a-lot, one of their celebrity customers. (Hopefully, that's another great story for another time.) I proposed, "Perhaps I could rap about it?"

I reviewed some notes and interviews about the theme and goals for the AltiumLive summit and the IPC APEX EXPO. I listened to an inspiring interview of Altium's Chief Ecosystem Officer and head of Altium Nexar, Ted Pawela, and IPC's Vice President of Standards and Technology, Dave Bergman[1].

They spoke of the Altium Summit's theme of "Connect" and IPC APEX EXPO's theme of "Digital Transcendence." Both organizations have been working hard to aim their messaging and educational programs toward PCB designers. They have executed their own dedicated missions to get the wave of new PCB engineers prepared to understand PCB design

and how the profession will be transforming soon as the trend for automated manufacturing moves forward. So, it was beautiful for me to hear that these two organizations decided to work together to reach all our PCB design and manufacturing stakeholders by working as partners and, in essence, coalescing the two show experiences.

The two show themes struck me that same evening.

To understand digital transcendence in context of the entire electronics PCB industry, PCB designers must learn to initiate design in a way that will digitally connect with all the other PCB industry stakeholders concurrently. Through the standardization of data formatting, PCB layouts of the future will be accessible and automatically modified by other project stakeholders. Commodity stakeholders will be connected with manufacturing engineering stakeholders, and design and manufacturing engineering will be connected with test engineering, for example. All will be able to work concurrently from live data within a project ecosystem to make required modifications without having to engage the original PCB engineers and designers who initiated the design.

In turn, designers will be freed up to move forward to initiate new designs while incorporating real-time feedback from the previous designs. The process of PCB design layout is envisioned to transition from being run by a single layout stakeholder with a traditionally limited scope of peripheral stakeholder capability, to one of data-driven transparency for all stakeholders concurrently. Design for manufacturing will transition to a paradigm of designing with manufacturing—and sales, supplier management, purchasing, and other key stakeholders who fill equally important roles during project development and on through production phases.

So, considering the challenge of creating a musical promo inspired by my evening realization, would I be able to find my inner "gangstah?" Could I rap about what to expect and experience at these two shows which would serve to shine as an example of international industry collaboration during the week of January 24?

Of course! I wrote down the main verse of my rap:

"So, let's all connect, c'mon people, let's reflect on the meaning of digital transcendence. Catchin' how design and manufacturing combine."

I gave it my best shot. I had fun recording the segments and the soundtrack. A couple of weeks before the show, the video was approved and "dropped" onto social media January 13.

As many of you know, editorial deadlines require me to submit this column well in advance of the publication date. Truth be told, I am writing to you a week earlier than the events I am describing will have occurred. I am filled with excitement and anticipation, but I really don't know how it's all going to play out. Let me try this. Here are some of my goals to accomplish in San Diego and be sure to read my review in *Real Time with... IPC APEX EXPO Show & Tell Magazine.*

If all goes well, and it usually does, I'll be contributing interviews and perspectives on my experience encountering the sights and especially the people who came together to make this unique event happen in this way this year. Deal?

Seek out this month's *Show & Tell Magazine* edition. I consider this a superb, custom boutique publication which does an outstanding job of highlighting the vibe of the exhibitors and attendees at IPC APEX EXPO. There is where you can read how I did in meeting my targets, along with all the I-Connect007 contributing editors' articles, interviews, and coverage of what is billed to be the largest electronics show in North America focused on printed board design and fabrication, electronics assembly, and test.

Did I meet the target conditions listed below?

- Attend some of the APEX professional development courses
- Meet, interview, and even sing a quick tech-song with APEX keynote speaker and CBS tech correspondent David Pogue
- Interview and thank the IPC education committee and the participants of the PCB design layout competition for all their hard work and raising the need for designer education
- Crash the Women in Electronics event, meet Jackie Mattox, and hear inspiring reactions to all that happens there
- Walk the APEX show floor, ask, and get answers about where CFX is going
- Meet automated assembly equipment stakeholders
- Connect via live stream with all the AltiumLive speakers, thank them, and ask them lots of questions from our livestream audience
- Connect and make good friends with at least 20 PCB stakeholders I have never met before on AltiumLive virtual networking
- Better understand and become an evangelist for reinventing the business of electronics with Nexar and Altium 365
- Get together with I-Connect007's Andy Shaughnessy, Nolan Johnson, and as many PCB industry folks as possible and hold spontaneous guitar jams
- Forget my NOOM regimen for a week and gorge on sushi

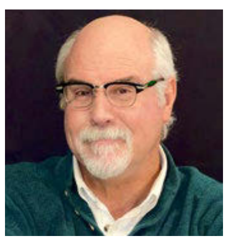
I have read the interview with columnist Dan Beaulieu about making the most of a trade show[2]. His points have stuck with me for a few years now. I think I am ready.

I have high expectations for San Diego. The trends in new technology and paradigms for product development have not slowed down. Those involved in PCB design will need to push through the present perceptions regarding DFM and design changes. More than ever, design stakeholders will not only be expected, but required, to work with and within a data-driven, tangible, collaborative, concurrent product development ecosystem.

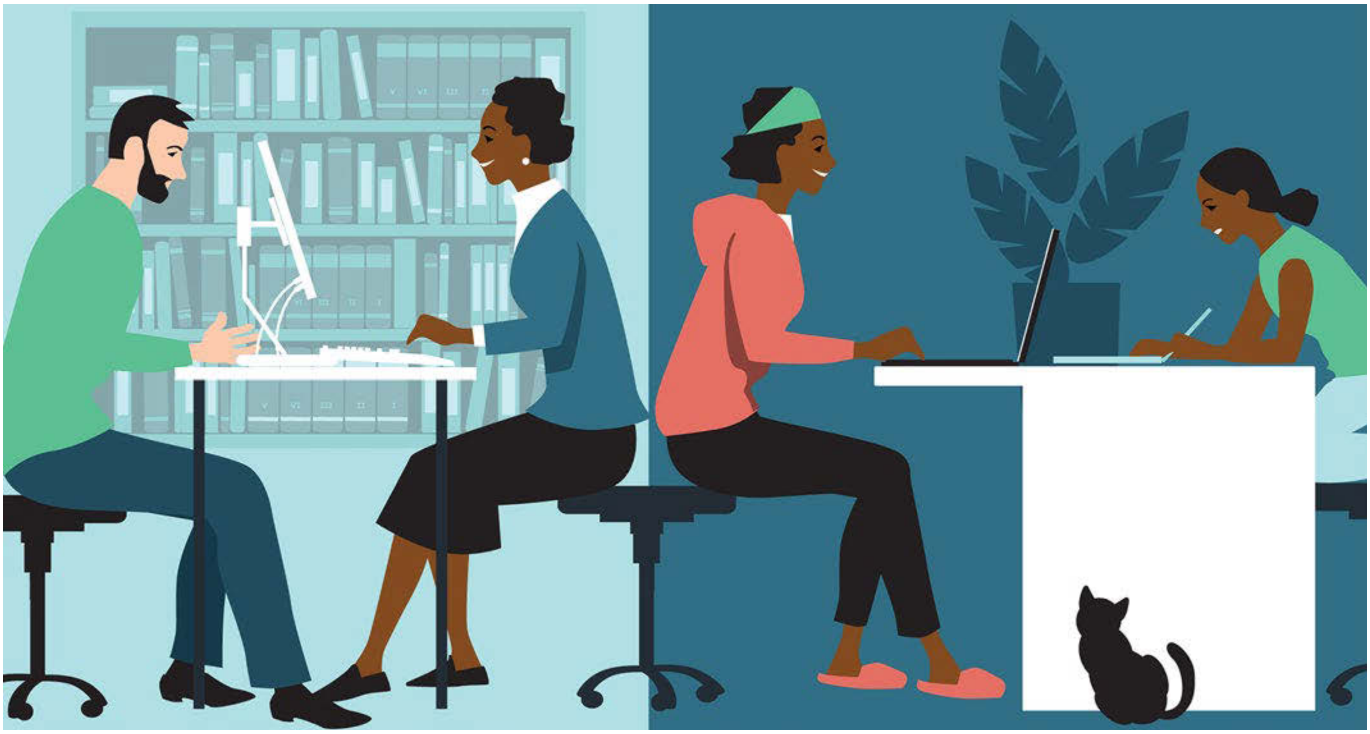
See you next month, or maybe sooner! **DESIGN007**

References

1. AltiumLive Co-locates with IPC APEX EXPO, youtube.com, Aug. 24.
2. "Dan Beaulieu on Making the Most of a Trade Show," *PCB007 Magazine,* December 2018.



Kelly Dack, CIT, CID+, provides DFX-centered PCB design and manufacturing liaison expertise for a dynamic EMS provider in the Pacific Northwest while also serving as an IPC design certification instructor (CID) for EPTAC. To read past columns or contact Dack, click here.



Keep the Change? No, Embrace It

Feature Article by Tamara Jovanovic
HAPPIEST BABY

In a world where technology is so quickly evolving, we really shouldn't be surprised that companies and professional events have gone virtual. Change can be daunting, but as we all master navigating life in this "new normal" (don't you hate that term?), we realize that not much of our professional lives has changed. You still have to go to work every day, but now you just do so from the comfort of your home, and you don't interact with people in person.

Obviously, the global pandemic had a massive impact on work culture. Even though the situation with the virus is slowly improving, a lot of companies are still maintaining a flexible workplace. Thanks to modern technology, a lot of companies were able to continue working seamlessly despite all the hurdles that the pandemic has created.

As employees continue to be efficient at working remotely, many employers don't see a need for expensive office space anymore. Many employers allow their workers to continue working from home and use the office as needed. I believe that this hybrid work environment is what the future holds for most companies. This change in work dynamic allows people to have more freedom with their personal lives while still getting their jobs done, which promotes a healthy work-life balance.

Of course, human beings are social creatures who depend on each other to thrive and survive. As a hardware engineer, I do lot of hands-on work with the product and prototypes, and this requires me to be in the office most of the time. As part of a team, I often work with other people and depend on my team members for various kinds of tasks. Most of my team also works from the office, and I personally don't have problems with communication, but I can see a potential for miscommunication.

If you're working remotely, sometimes it can be hard to get in touch with a team member. When you need information from another person to continue your own work and that person is unavailable, this could delay the project you're working on.

In the PCB design world, a variety of factors dictate the difficulty of the project and how much time is required for the design to be complete. I work closely with the mechanical engineers at my company while working on a design to make sure that what I am doing on the electrical side will not interfere with the mechanical design and constraints. Once the testing and proof of concept is complete and I have a board outline, I can work on schematics and layout on my own until completion, which is when I circle back to the mechanicals to double-check the final design and make any tweaks if necessary. Regardless of what is happening in the outside world, this process for designers in consumer electronics is unlikely to change, because it's usually a collaborative effort to make all the pieces of the puzzle come together.

Yes, during the pandemic I went back into the hallowed halls of academia, hot on the trail of my MSEE.

Back to School

Yes, during the pandemic I went back into the hallowed halls of academia, hot on the trail of my MSEE. As you may know, universities have transitioned to virtual classes and lectures. Online classes work great for me, since I have a career, but I could see how this could be a real problem for undergrads looking to enjoy college life and explore networking opportunities. But I think things will change soon. As the pandemic situation slowly improves, some schools have gone back to in-person lectures, and I believe most universities will eventually return to in-person classes.

The pandemic has allowed technology to work its way into more of our everyday lives. In the baby device segment, especially, we're seeing families become more accustomed to relying on technology than on other human beings. Back in the day, parents with newborn babies used to have their families move in and help them out. Or they would hire a babysitter or a night nurse.

But during the pandemic, you couldn't risk having a stranger come into your home. This is where new technology, such as our smart bassinet (shameless plug here!) that keeps babies on their back and rocks them to sleep has proven to be a great help for new parents. The average consumer now uses a variety of electronic devices each day, and that's a giant positive for PCB designers. There are many silver linings in this pandemic cloud.

We've all gone through what seems like a lifetime's worth of change in the past few years, and with this change, there is always fear of the unknown, which is totally understandable. However, if we learned anything over the last few years, it's that change can bring many positive outcomes in school, the workplace, or just life in general. While change can be scary, we live in the world where adaptability is a prerequisite to surviving and keeping up with the world. This is why I've found that it's best to always be open to learning new things and accept and embrace change when it happens. **DESIGN007**

Tamara Jovanovic is an electrical engineer at Happiest Baby, a Los Angeles-based company that designs and manufactures smart baby beds.

DFM 101: Plating Methods

Article by Anaya Vardya
CEO-AMERICAN STANDARD CIRCUITS

Introduction

One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. This article is the latest in a series that will discuss these cost drivers (from the PCB manufacturer's perspective) and the design decisions that will impact product reliability.

PCB Plating Methods

There are two methods of plating copper on PCBs—pattern plating and panel plating. The panel plating method eliminates most of the copper plating distribution issues, but because it adds copper thickness to the base layer, it makes maintaining fine line definition and consistency difficult. Base copper is measured in ounces of copper per square foot of surface area.

Pattern Plate

This standard process has major advantages in that only the base copper is required to be etched. This process yields finer, better-defined lines (traces). One possible disadvantage is the variations in trace height due to surface density (Figure 1).

Panel Plate

This plating fabrication method eliminates most of the copper plating distribution issues but now the excess surface copper (after the circuit pattern is defined) must be etched along with the base foil. This makes maintaining fine line definition and consistency difficult.

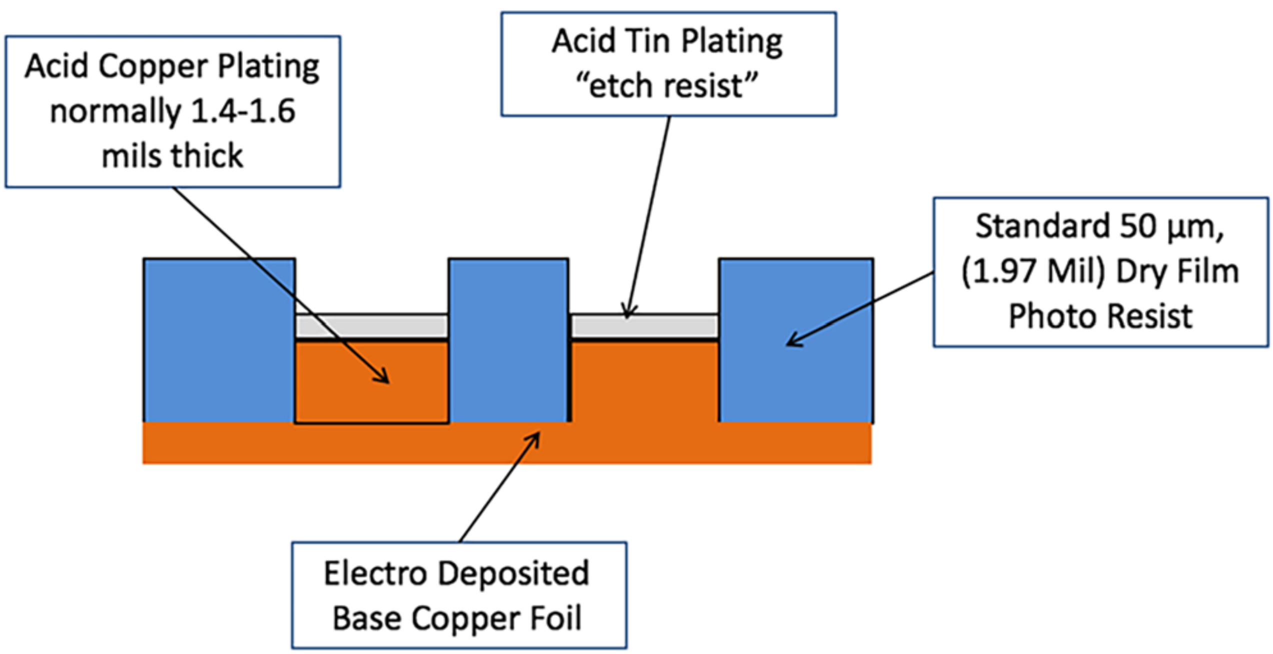


Figure 1.

Copper Weight	Typical Reduction in Line Width From Nominal	Minimum Design Line Width
0.24 ounce	0.0005”	0.003”
0.5 ounce	0.0007”	0.004”
1.0 ounce	0.0014”	0.005”
2.0 ounce	0.0028”	0.008”

Table 1.

The allowance for etching of plated designs is shown in Table 1.

Note: It is important to mention that the above guidance is for the raw copper foil weights and not any additional design features like via fill or sequential blind laminations that will add copper thickness to the foil.

Design Compensation

Because of the physics of producing printed circuits, the PCB fabricator first needs to plate the circuit pattern (pattern plate), followed by a chemical etching (subtractive) process to define the circuit line width and space. To accomplish this, the initial circuit definition will need to be compensated for line width loss due to etching to assure that the final circuit will meet the design criteria.

Surface to Hole Plating Ratio

Regardless of how much copper a fabricator begins with, additional copper on the surface will be plated (pattern plating) as the plated through-holes are metalized. The key here is that this is not a 1:1 ratio; more copper will always be deposited on the surface than in the holes due to a higher plating current density on the surface. A working guide would be to calculate that 1.2 to 1.4 times the average minimum hole thickness will be deposited on the surface. Using the industry standard requirement of 1.0 mils in the hole, the surface would receive 1.2 to 1.4 mils of plated copper on the surface, plus any base copper thickness.

Impact of Sequential Laminations and Via Fill

Sequential lamination (subassembly processing) is used to manufacture buried/blind vias, via fill, microvias, etc. These technologies further compound the final surface copper thickness considerations discussed above as each process requires additional plating and etching processes. Designers need to fully understand the impact of additional plating processes on the final thickness of copper traces and surfaces.

Designers need to fully understand the impact of additional plating processes on the final thickness of copper traces and surfaces.

Understanding the cost drivers in PCB fabrication and early engagement between the designer and the fabricator are crucial elements that lead to cost-effective design success. Following your fabricator’s DFM guidelines is the first place to start. DESIGN007

Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer’s Guide to... Fundamentals of RF/Microwave PCBs* and *Flex and Rigid-Flex Fundamentals;* and author of *Thermal Management: A Fabricator’s Perspective*. Visit I-007eBooks.com to download these and other free, educational titles. He also co-authored Fundamentals of Printed Circuit Board Technologies and provides a discussion of flex and rigid flex PCBs at *RealTime with... American Standard Circuits.*

PCB007 Highlights

CES 2022: Half Virtual, Still Valuable, and Here's Why ▸

Another CES has wrapped up, and while not fully back to its glittery self, the show still managed to create quite a buzz in the electronics world. I reported on the show from the safety of my home office, which was a bit of a letdown because I really enjoy walking the aisles of the show.

Trouble in Your Tank: Plating Anomalies and Defects, Part 2 ▸

One thing about troubleshooting PCB defects is getting to and understanding the root cause of defects. Many of these defects have can have multiple origins. And many may not manifest themselves in the process where the defect occurred.

Best Technical Papers at IPC APEX EXPO 2022 Selected ▸

The best technical conference papers of IPC APEX EXPO 2022 have been selected. Voted on by members of the IPC APEX EXPO 2022 Technical Program Committee (TPC), the paper authors will be recognized during show opening remarks on Tuesday, January 25.

Catching Up With RBP Chemical and Schlötter ▸

It's always great to see two very good companies form a mutually beneficial alliance. I was lucky enough to watch this particular strategic partnership come to fruition this year between RBP Chemical and Schlötter. I sat down with Matthias Hampel, global executive representative-PCB and electronics at Schlötter, and Ernest Litynski, president of RBP Chemical Technology, to get the inside story.

Testing Todd: Why, Why, Why—Never Stop Questioning ▸

If you have been around young children, I'm sure you have fallen into the "Why" game. I'm now a grandfather and the game started again a long time ago. "Grampa? Why is that man limping?" "Well, he has a cast on his foot." "Why, Grampa?" "Well, it looks like he was injured." "Why?" And so it goes. These brilliant young children have mastered root cause analysis and they don't even know it.

Epec Invests Over $1M in New Equipment at Netvia Location ▸

Epec Engineered Technologies announced that it has invested more than $1 million in new equipment at its NetVia Group PCB manufacturing facility located outside of Dallas, Texas.

Eltek Names Ron Freund as Chief Financial Officer ▸

Eltek Ltd., a global manufacturer and supplier of technologically advanced solutions in the field of printed circuit boards, announced that its Board of Directors has named Ron Freund as its chief financial officer.

Trackwise Designs Secures £2.4M Purchase Order from UK EV OEM Customer ▸

Trackwise Designs, a leading provider of specialist products using printed circuit technology, is pleased to confirm the receipt of the Q2 2022 purchase order of £2.4 million, in accordance with the Product Manufacture and Supply Agreement with its UK EV OEM customer.

Selecting Prepreg for Millimeter-wave PCB Applications

Lightning Speed Laminates

by **John Coonrod**, ROGERS CORPORATION

There are numerous prepreg choices in the PCB industry. In the past, the choice of prepreg was often dominated by PCB fabrication concerns. The prepregs used for high frequency or high-speed digital (HSD) address many of the same PCB fabrication concerns, but they also have other properties which are important for consistent high-frequency or HSD performance. There are also other bonding materials used as prepreg, such as a film or non-glass woven bonding materials, which are called bonding film or bondplys. Basically, these prepregs and other bonding materials are used to adhere the different layers in a multilayer PCB and can have a significant influence on many aspects of the circuit.

Over the past several years there have been more high-volume PCB programs that are using millimeter-wave (mmWave) frequencies. Most of these applications, until the past few years, are PCBs typically built as a four- or six-layer circuit with only the top two copper layers being related to the high frequency circuitry. Many of the more recent mmWave PCBs, especially those used in high resolution automotive radar, are multilayer PCBs with many high frequency circuit layers and in some cases the entire multilayer PCB is designed to have all layers to be supportive of mmWave frequencies. It is these types of circuitry where the prepreg and bondplys can be critical to good RF performance and still need to maintain all the other properties for good PCB fabrication and reliability.

A circuit using mmWave frequencies is a propagating medium for the wave, and the wavelengths at these high frequencies are very small. Generally, the small wavelength means the propagating wave will be more sensitive to anomalies in the medium than a propagating wave at a lower frequency. There are several items which are relatively normal to the circuit fabrication process, which could be considered an anomaly. Having a very small etching

defect that is normally acceptable at lower frequencies may not be acceptable at mmWave frequencies. The normal variation of conductor trapezoidal effects, differential etching, slight undercut, layer-to-layer alignment, are all things that can normally happen in the PCB fabrication process and are relatively harmless to the RF performance of the PCB when operating at lower frequencies. However, at mmWave frequencies these normal variations can be problematic.

Having a very small etching defect that is normally acceptable at lower frequencies may not be acceptable at mmWave frequencies.

Another issue that is relatively normal and related to the high frequency circuit materials, is the glass reinforcement layers which are used to give mechanical stability to the material, among other things. Depending on the formulation of the laminate or prepreg and the glass style used, the supporting glass fabric can cause small disturbances in the RF propagation medium of the PCB. Again, at lower frequencies this may not be a concern, but at mmWave frequencies the very small wavelength can easily detect and be affected by the small differences due to the glass and/or overall formulation of the laminate or prepreg.

The main issue with the glass is known as the glass-weave effect, fiber-weave effect, or woven-glass-weave effect. The glass styles that have been used for decades are typically open-weave glass and, as the name implies, there are openings between the glass bundles. The glass that has been typically used is E-glass and that type of glass has a Dk of about 6 and a Df near 0.004. However, the E-glass Df will significantly increase with an increase in frequency and the Dk will decrease some as well. The basic issue for the glass-weave effect is related to the opening size in the glass fabric as well as the wavelength of the propagating wave.

For example, if the average opening size in the glass fabric is 16 mils x 16 mils and operating at 3.6 GHz with a wavelength of 2.1 inches (or 2100 mils), the openings in the glass fabric are such a small percentage of the propagating wavelength, that the openings do not have a significant effect on the wave performance. Although, it is well known that a fraction of a wavelength can be significant and if an anomaly is half wavelength in size it can cause wave disturbance in the propagation of the wave. If the anomaly is one-quarter wavelength it can also cause a disturbance but usually less than the half wavelength anomaly.

Typically, around one-eighth wavelength or less and the anomaly will not influence the propagating wave. Continuing with the example and using the same glass with openings of 16 mils x 16 mils and now considering the wavelength at 77 GHz, which is 97 mils, it is obvious that the 16-mil anomaly is a larger portion of the 97-mil wavelength and can have a disturbance on the propagating wave. At this frequency, the 1/8th wavelength size is 12 mils and the 16-mil opening in the glass will likely have some influence on the wave behavior.

These mmWave prepregs will often use open-weave glass due to its lower cost than spread-glass fabric, a better performing alternative glass fabric. Spread-glass fabric has very few or no openings between the glass bundles and will behave as a more uniform propagating medium than one using open-weave glass. Additionally, it has been found that if the laminate is heavily loaded with ceramic fillers, the differences of the propagating medium between the openings and the glass bundles of the glass fabric are greatly minimized.

Due to concerns of the glass-weave effect, many of the legacy mmWave PCB designs used an unsupported ceramic-filled laminate such as RO3003G2 laminate. In this case, there is no glass fabric layer, so the glass-weave effect is not applicable. However, due to the more recent high resolution mmWave designs, which use many layers for mmWave circuitry, a laminate with supporting glass fabric is needed for improved circuit fabrication, and then laminates using spread-glass are used such as the CLTE-MW laminate. Additionally, this laminate not only has the spread-glass, but it is also heavily loaded with ceramic filler and the glass-weave effect is typically not a concern at 77 GHz.

With the higher resolution mmWave radar circuits, the prepreg must also have very good RF performance properties. There are prepregs which have no glass reinforcement, which means the glass-weave effect is not a concern. Some of these prepregs or bondplys may need special lamination conditions and it is recommended to speak with your material provider's technical support person about how to process these materials for circuit fabrication.

For certain high resolution mmWave radar circuitry, circuit materials that are glass reinforced are desired for the PCB fabricator to obtain good yields and good reliability of the PCB. These prepregs are often available with open-weave or spread-weave glass styles and it is best to use the spread-weave versions of this prepreg for mmWave circuitry. Additionally, some of these prepregs can have spread glass and be heavily loaded with ceramic filler, which is a best-case scenario for mitigating the glass-weave effect. Many of these prepregs are formulated to be friendly in the PCB fabrication process, but it is still suggested to contact your technical support engineer at the material company to ensure the processing and final performance will be as needed. **DESIGN007**

John Coonrod is technical marketing manager at Rogers Corporation. To read past columns or contact Coonrod, click here.

All Systems Go!

Bridging the Gap Between Design and Analysis

By Brad Griffin

Brad Griffin

Electronic designs are increasing in capacity, complexity, and performance. This is coupled with increasing pressure to get new products to market as quickly as possible while, at the same time, ensuring that these products are robust and will not fail in the field. The only practical way to address all these diverse requirements is to make design and verification tools and methodologies more powerful, intuitive, and easier to use.

Almost any electronic development methodology and/or workflow can be segmented into three main stages: a pre-design exploration phase, the design phase itself, and a post-design verification phase.

Historically, the engineers responsible for the exploration, design, and verification stages have used different tools. In this case, the exploration and design teams often prefer to use more simplistic tools that are easier to use. However, these tools then do not correlate 100% with the results from the tools preferred by the verification team which requires more powerful and sophisticated tools to ensure design integrity. What's more, all the tools in use across these three stages have traditionally been standalone point tools that are not well-integrated into the overall environment, thereby adding unnecessary risk and error into the workflow.

To complicate things further, communication from the verification team back to the design team is often ad-hoc, being composed of screenshots, documents, emails, etc. As a result, a typical development flow tends to involve multiple iterations between the verification team discovering problems and the design team fixing them, hopefully without introducing new issues into the mix.

To read the entire column, click here.

Design Milestones: More Than Signposts Along the Road

Feature Interview by the I-Connect007 Editorial Team

As technology has evolved over the years, the milestones in PCB design have changed as well. While we look at the past and future of the design process, it's instructive to review the way that design milestones have iterated along the way.

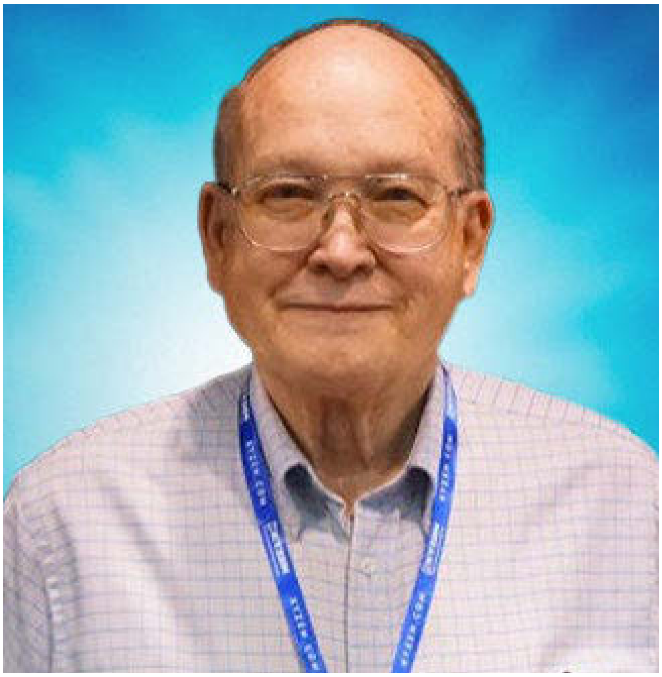
We asked Happy Holden, Kelly Dack, and Bob Tise to share their thoughts on establishing productive design process milestones, and what they mean to the PCB designers and engineers of today and tomorrow. They also discuss how designs can go awry if these "signposts" in the design cycle are not obeyed.

Barry Matties: Happy, why don't you start by telling us about your thoughts on milestones.

Happy Holden: Here's how I'm using the term milestone. At Hewlett Packard, we had to establish a standardized printed circuit design process, because PC layout was merged out of several product divisions, with a few different design philosophies because of different types of products. We were manufacturing the printed circuit boards, but they decided to merge us under the manufacturing umbrella instead of the design umbrella, because all processes in Hewlett Packard were with their own segment of the corporation, with an executive VP who happened to have a Ph.D. in chemical engineering.

We shared part of our structure with the IC manufacturing and integrated circuit design. I only accepted the job if I could increase the salary of our designers. By increasing their salary, I could now also require courses in skills improvement over time as part of their job description.

Everybody wanted to do this differently. They all had the belief that printed circuit design was not something that you can standardize. But I said, "That's one reason that you're here, and not in your product division, because we believe you can standardize it into



Happy Holden

a process, and you're all going to help us come to consensus in what that process is."

One of the key elements that came up was that during the design of a printed circuit board, the designer always had questions that another designer couldn't answer. They had to be answered by somebody in fabrication, manufacturing, or test. Because of schedules, a lot of times you couldn't find the right person to answer your question. You just did it the way you did it the last time or turned to somebody close to you and asked them and stayed on schedule.

The one thing that we endeavored to do was identify these key milestones that fundamentally define the printed circuit layout process, costs, and everything else, while also developing alternatives, including the cost of each alternative. To get past the cost, we invented a fake currency called the relative cost index, RCI, that was not dollars and had no units, but it allowed you then to compare one alternative to another, by which one had the lower percent cost and/or the higher reliability, and then move on.

I'm using the concept of milestones in that I've never seen any flow chart of the design process that recognizes that a lot of decisions must be made. The answers come from somebody else, not from the design engineer, but from the fabricator and the assembler. How do they want to do inspection and test?

Fabricators and assemblers need to collect information to help answer the questions at these milestones that make it easier for designers to find a path to complete a board, because there's an almost infinite number of ways you could design a printed circuit board.

Matties: Kelly, what are your thoughts about what Happy just described?

Kelly Dack: I appreciate Happy's experience with regard to design over the years, especially as he has worked for a large company with its own captive fabrication capabilities. When a captive designer is in touch in this way with the capabilities of a manufacturer which sits directly on the same project table, design can easily be—and must be standardized to match the documented in-house, or corporate manufacturing capabilities. Pushing designs into new technology only comes by a corporate decision to invest in new tooling and machinery to match the proposed design concept which can exist within or without common, commercially available manufacturing capabilities.

Having worked my entire career only for companies which must purchase their PCBs from third-party sources, however, gives me a bit of a different perspective. At the start of every PCB design project, I myself and many other designers are systematically put in the position of not knowing where the boards we design will end up being fabricated. Even when we rely on commercially available design standards, like the IPC 222X family, we've come to always expect requests from a new supplier to allow manufacturing discrepancies or modifications which were not considered a problem for a previous supplier.

Without establishing the milestone of selecting a dedicated supplier with known and verified capabilities, "manufacturing" in the acronym DFM, is a moving target based upon unknown capability. Project schedules are always behind. We're often faced with a compression of design tasks, design scope, and design technology, to the point where we're seeing that all these tasks are becoming blurred without answering the question, "Who is going to be the board supplier?" on the front end. Due to increasing differences in supplier capability, even the term design for manufacturing is being reconsidered. We're going so fast with technology changes, and the requirements for electronics, that we're having to consider design *with* manufacturing. But until then, the traditional injustice of a designer not knowing where their boards are going to be manufactured is still real. We must push back against that.

Matties: Bob, what's your reaction to what Happy had to say?

Bob Tise: I think Happy's got a great point. I worked at HP for four or five years, and it was a pretty good experience. It's also very structured. They're big enough that they need structure. At Sunstone, I put together a set of requirements before I'd even talked to a customer about doing design. Then, if they were willing to meet them, it was pretty rigid, because in order to have a successful design, you have to have certain things. Number one, they must supply you with a completed bill of materials and the schematic involved, let you know what they need the design to do, and tell you about anything critical.

You must have the entire design thought out and defined before you ever start doing the layout. I was usually dealing with one person. A lot of times, they had a hard time understanding that. Then, once you pick it up, you must know the board size and the keypads and all the things that go into that, to define the scope of work. Again, if you don't have that, you're winging it and it's probably not going to fit, or they don't care. If they don't care, that's about a 30% hit.

Dack: From a design perspective, we must consider all the stakeholders in the process. Gone are the days when a designer sat down in a vacuum with an engineer and designed a successful board. What we always say is, "What we breadboard in the lab is easy, but go try to get that manufactured and make a million of them." It's very difficult, unless you involve all the stakeholders. We define these milestones as constraints. PCB designers must take into consideration all the constraints of all the stakeholders.

PCB designers must take into consideration all the constraints of all the stakeholders.

One possibility is the customer. The consumer is concerned about cost, but how much is this product going to cost overall? How much will this added cost affect profitability? Complexity adds cost. When we talk about a circuit board, an electronic assembly, we're bringing together hundreds, if not thousands of different parts. You've got 1,000 exponentially more complex scenarios that you must throttle, which means a designer sometimes must rob Peter to pay Paul in order to get a success story.

Holden: "How big is the printed circuit board?" is actually a milestone, because there's a tradeoff. Which size should they be? I'm focused on cost, minimizing weight in laminate, things like that. That's a milestone, because the question right after that is "How many layers—power layers, ground layers, and signal layers?"

Bob Tise

Tise: There's a whole lot of stuff that you must have defined up-front as design rules, written in stone before you ever start. DFM is not something that you necessarily have to use as a design input. It's just something that must be in place and written in stone and part of your culture, or you're going to build things that can't be built.

That starts at the library creation and design rules creation, in the partnership between the designer and the fabricator. You better know where your fab shop's at, or you're going to send them something half-built, or they're going to really charge you for it. Also, you must worry about assembly as well, so that your boards can be built to the end.

Dack: Yes. That's back to stakeholder awareness. We must remember the five Ws: who, what, when, where, why, and maybe how. Who is going to be building this? We must know their constraints. Again, that could be considered a milestone. I have an old friend from back in the '80s, when I was starting out. His unforgettable quote to me was, "Kelly, never design anything that can't be built."

I cite this quote often because it happens. It continues to happen over and over. We're working with suppliers who may use an additive process because the product can be built at their facility. But then purchasing gets a lower quote from a supplier using a common subtractive process who ends up not being able to build the board. That would certainly add to the timeline, and probably shoot all the project milestones for miles down the road.

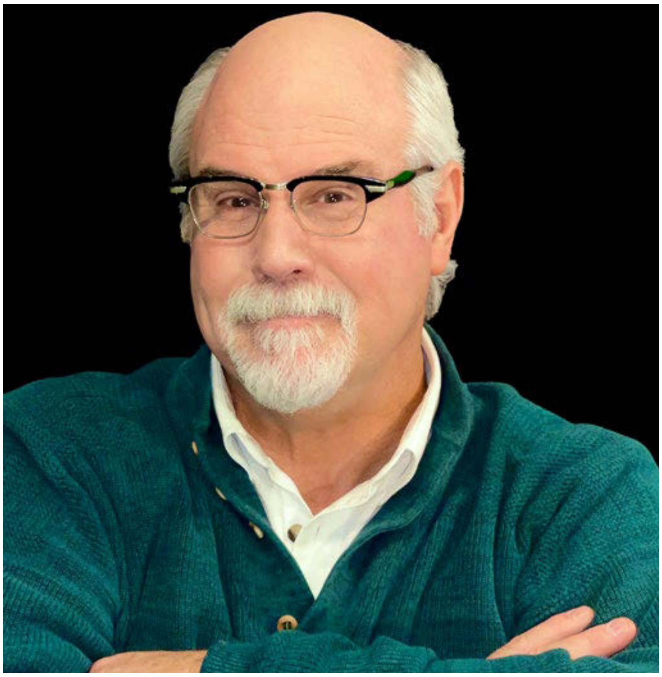
Tise: When I was at Sunstone, it had to be something that could be built. Otherwise, we'd say, "You've gone beyond the lines of what could be done in the normal circuit board manufacturing process. Maybe there's somebody out there that can build this, but it's not me."

You get down to the below a 0.3-millimeter BGA, and you're in trouble. Maybe they could use 3D printing to do the circuit board, but that gets outside of normal design constraints. Don't say, "I drew it, so you should be able to make it." You should start out saying, "What can your fab shop and your assembly shop do using nominal processes and nominal costs?" Rather than, "What's your highest level of technology?"

Dack: Because of time-to-market issues, we're forced to ask, "Can we speed up the design cycle by concurrent engineering?"

Matties: I get what you're saying, but there are still milestones in that, whether you're doing it concurrently or independently, you still need to have all the information. You still need the parameters. You still need the supply line, and you still need to know the fabricator. These are all milestones. I think that's what you're saying, it's going to come, perhaps you'll know the fabricator at the end, in some processes, and others, you may know at the beginning, but it's still a milestone.

Dack: It is. It's important to note that traditional milestones are sequential. The story I



Kelly Dack

like to tell in our IPC CID classes is that our system is kind of jacked up because milestone 10 is often put out as milestone one. It really starts with our industrial design team. The industrial designers get out their big board with pastels and they work with the customer to dream up this great looking product from the outside. It keeps getting smaller and smaller and ergonomically friendly, and suddenly, it fits in the palm of a hand or on a wrist. Then, they hand that over to the mechanical folks, who take all that information; that's milestone nine, and they put it into a nice mechanical package.

At the end of the process, one of the last milestones is to then hand it over to the electronics team, which is tasked with finding components and all the technologies to shrink this thing down into the package. That's where they must back-pedal. If they cannot find the parts that are available, or the components are taking up too much room and won't fit, they must go back and forth, like the project team missed a turn somewhere because they weren't riding the same bus.

Matties: In each of those milestones that you're describing, each one of those functions may have their own set of milestones.

Dack: That's true.

Matties: Kelly, we all know about DFM, but is there a new acronym called DWM, design with manufacturing?

Dack: Yes, DWM is floating around. We could also say DWX, design with excellence, so that it applies to assembly, test, etc.

Andy Shaughnessy: Kelly and Bob, is every process step necessarily a milestone? A lot of flow charts for PCB design showed milestones and process steps, and they used the words interchangeably. Or are milestones primarily critical points in the process?

Dack: I think you must look at every process step. We use a program here called monday.com, a communication app that organizes multi-stakeholder or team ideas, infographic displays and charts.

Tise: Yes, we use that.

Dack: What it does is let other stakeholders know how far along in the process one is toward meeting a specific milestone. Milestones equal success, I would think, in a project. You must have a measurable milestone. How far along in a measurement are you toward meeting that goal?

Tise: Right. Some of the milestones took place before I'd even start the design. A lot of times, I got the schematic on a scanned copy, and I had to input the schematic with all the parts and such. When I got that complete, I would send that out to have it reviewed by the customer, make sure that I connected all his dots the way he wanted them connected. Then, that would be a milestone. He'd sign off on that.

We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
ITAR

Then, I'd make all my library parts, double and triple check to make sure that they were reasonably close to what the data sheet said. That would be in a milestone. That's a huge task sometimes. I'd place the part on this, match them up with the schematic, place it on the board, have the customer look at it, verify that it's going to meet his needs, the connectors are where he wants him, or this connector is not rotated, so that he's going to have to hook the wires up from on the board rather than off the board.

All the little things that you must think about in that regard would be more milestones. Then, I'd route the thing. Once it's routed, it goes out again to the customer, look at it, make sure that I haven't routed his 50-watt lines into 10 mils, that sort of stuff. Then, off we go, off to the races.

Once it's routed, it goes out again to the customer, look at it, make sure that I haven't routed his 50-watt lines into 10 mils, that sort of stuff. Then, off we go, off to the races.

Holden: I'm using the term milestone, like what Bob talked about. If your design process is a flow chart of five boxes, then every box is a milestone, but if you take each box apart, let's say, placement, and instead of being one box, it's actually 10 boxes of a complex flow chart. If within that, before you decide on placement, you've asked the question, "Do we have to have design for test?" Do they have to have space to get in there and probe? Are they going to use boundary scan in which they need access to vias, but this is going to be an HDI board in which the vias are going to be under the BGAs and not accessible?

There are several questions and alternatives in terms of what is going to be your placement strategy. Does the component have access from all four sides? Does the component require certain reference planes that have to be matched? It must be paired up with other components that use the same reference planes. It can't be on opposite sides of the board. What I'm talking about is probably much more detailed—how we design a board in terms of 30 or 40 steps.

Dack: I've never seen a chart breaking the design process down into what's important for manufacturing, assembly, test, and fabrication, at least none that offer alternatives. What are the consequences of signal integrity, cost, or time? What options do we have?

It is as if PCB designers are steering through this obstacle course, and they're making the best decisions they can. It would be nice if you had two or three people on each side of you constantly giving you advice—one with expertise in fabrication and assembly, and another with expertise in test and field repair. We don't have that kind of a thing. Now, maybe with AI we will someday have those virtual people looking over our shoulders offering that advice. Happy, isn't that what our smart factories are about?

Matties: That's generations away.

Holden: Yes. But that's what they're promising. It might be 10 or 20 years, though.

Dack: I think the smart factory is going to explode once it catches on. It reminds me when flat screen TVs came out or as we presently transition to electric vehicles. I rolled my eyes about electric vehicles years ago, but now I drive an EV.

Shaughnessy: When I first started covering this industry, I asked for a list of milestones and everybody said, "Well, it depends on what you're designing, the technology, and the type of company." For instance, in a design bureau, you might have a different set of milestones for jobs based on the level of technology.

Matties: I think that there is agreement on the broad brush-stroke milestones, as Bob has depicted for us. First, you do this, then you do this. These are all milestones that every designer can hang their hat on. For me, what I really learned from this is, the way we design boards is changing, and it's going to be changing fast, and we had better pay attention to DWM, design with manufacturing. And greater collaboration. Are you seeing greater adoption of DFM and design with manufacturing, Bob?

Tise: Yes. You know, HP was already doing DFM when I got there. They had the full-meal deal, with engineers for everything. You had different manufacturing engineers, and if you had a question, you could just call them up and say, "I've got this thing. What do you think?" They'd give you the pros and cons, or they'd just say, "We don't do that here."

I worked in R&D at the printer shop, doing inkjet printers. You'd start out with just the chip. You'd say, "Can we route this thing out to the intended connectors?" You'd just route to the connector. Okay, we got it routed to the connectors, and they'd make the board, and it connected, and everything was good.

Then, while that was going on, somebody else is designing the next build. Then, while they're finishing that up, somebody else would then do the LVDS or the memory. You'd have a dozen different versions of the board, just leapfrogging on each other as you go. Each one of those things would be considered a milestone. How granular do want to make it?

Downstream, when I was working at a fab shop as a design engineer or layout engineer, I didn't have that. I had to basically leverage what I learned over the years and be my own DFM engineer.

Dack: It makes a lot of sense to follow the rules, stay between the lines. If you don't work for a large organization and you've got a product that you want to get on Shark Tank really fast and need to get it manufactured, you don't always have the luxury of standardization. But you still need to be aware of who's out there and set up these milestones to establish process and get things checked out. Position project stakeholders near every milestone.

Shaughnessy: This has been really great. Thanks for taking the time to speak with us.

Tise: Thank you, Andy. I enjoyed it. DESIGN007

Changes on the Horizon: Is Resistance Futile?

Feature Article by Stephen V. Chavez
PCEA

For printed circuit engineers, especially those of us who have been in the industry for some time now, change is inevitable. From customer requirements that lead to design changes and deadlines being pulled in, to decreasing budgets and resource reallocations, change is one area where we must be adaptable if we want to survive and be successful in today's industry.

Engineering change orders (ECOs), schematic/drawing redlines, component placement adjustments, and mechanical features modifications are among the changes that most of us usually deal with at one point or another during a project's design cycle.

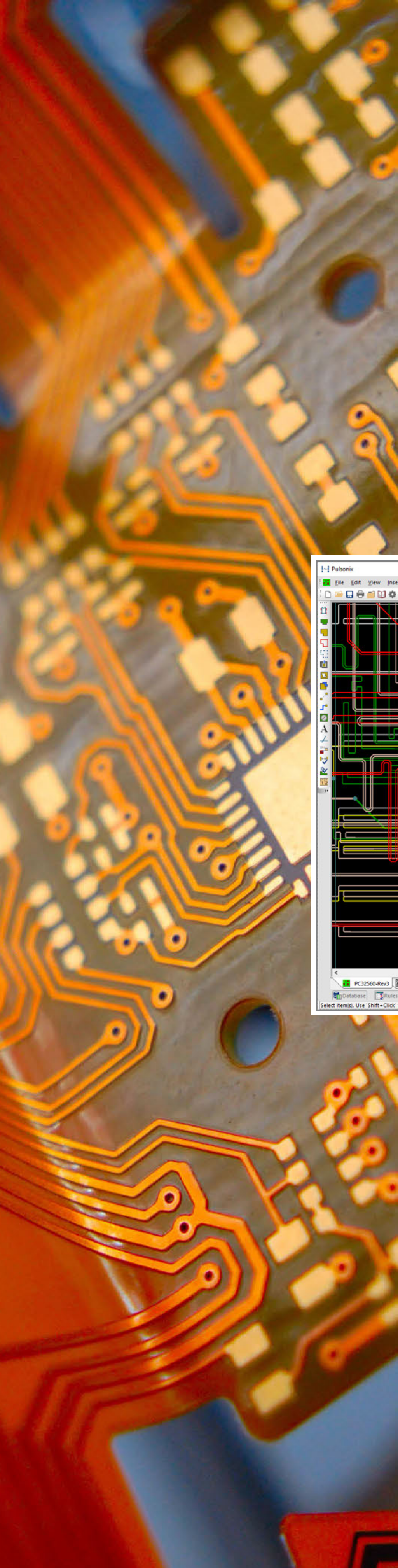
In my experience working for both small engineering firms and large OEMs, change typically translates into more time, which translates into more money. The big questions: Is the change billable or not? Who pays for this, or eats the cost of this change?

It all depends on how a contract was written, and how the purchase order (PO) was awarded. Potential changes are risk factors that are added into each quote, with a caveat that each change will be evaluated. This evaluation leads to a potential "out of scope" response to this change, adding an additional cost to the already agreed-upon PO, extension to the project schedule, or reset to the original project task duration. Every company handles purchase orders differently.

As I mentioned, change is inevitable in PCB design, and designers have become accustomed to it. But what about those other changes that may come at some point in our careers? Perhaps you've experienced a change in EDA tools, or a change in company culture, or both.

If so, you may have a special understanding of the famous motto of "The Collective" in Star Trek: Generations. "Resistance is futile."

I feel your pain. Usually, this sort of change happens if you switch jobs, your company is acquired by another company, or your department is reorganized. It happens, and if it hasn't happened to you yet, it probably will, if you remain in the industry long enough.



What if you must learn a new EDA tool after mastering another tool and utilizing it very successfully for many years? From your employer's perspective, what is your return on investment (ROI) to them?

Before the change to a new PCB design tool, your ROI was likely very high. Now, you're learning a new a tool, which usually leads to a drop in your ROI for a period. The duration of time depends on several factors.

How fast can you get up to speed on the new tool? What are the company's expectations for you to be assimilated into "The Collective"? And perhaps most importantly, what is your personal attitude? What is your willingness to adapt to this new software?

If you're early in your design career, you probably wouldn't hesitate to accept this type of change. But what if you have been with a company for 40 years, using the same tool for all that time, and now you are being directed to make this change to a new tool?

Every designer is different, so is every tool, and each tool has a learning curve. Do you assimilate into "The Collective" and learn the new tool?

Every designer is different, so is every tool, and each tool has a learning curve. Do you assimilate into "The Collective" and learn the new tool? Or do you simply find a new employer who uses the tool you have mastered and jump ship to that new company?

After spending years designing PCBs and integrating into many "collectives" through my career, I can honestly say it's not that simple.

Yes, learning a new tool has its advantages, especially in the long run, and especially if your company is willing to pay you to learn a new tool. Learning a new tool opens the door to more opportunities down the road. But again, what if you're 40 years deep with the company? Do you assimilate with the change, find another employer, or if your finances allow it, do you just ride off into retirement?

I have witnessed this scenario many times with friends and colleagues. What I have learned over the years is that there is no right or wrong answer, and each of us must make the decision that is best for us and our respective families.

Now, what if the change is not with a design tool, but rather a change in company culture? A company's culture can make or break a company. If a company's culture is negative, word will get out and people will not want to work there. But if a company's culture is sound, it will attract great talent.

What would you do if your division merges into a larger division and forces you into a new company culture that you find disagreeable? Again, if you have been with a company for a very long time, it's not that simple to just assimilate into "The Collective."

As PCB designers and design engineers, most of us are accustomed to dealing with near-constant changes in technology and the electronics industry overall. Many of us would just assimilate as needed and move on.

But don't be surprised if you find yourself sitting in your office, where you wielded your previous design tool as a true master, and you hear that familiar, low-pitched, digitized voice:

"Resistance Is futile." DESIGN007

Stephen Chavez is chairman of the Printed Circuit Engineers Association.

MilAero007 Highlights

Real Time with... IPC APEX EXPO 2022: Blackfox Mixes It Up with Training Opportunities ▶

Andy Shaughnessy speaks with Jamie Noland, master IPC trainer and marketing manager for Blackfox Institute, about current and upcoming training opportunities and what you can expect from them at the show.

Rising Material, Labor Costs Continue to Plague Global Electronics Manufacturing Supply Chain ▶

IPC's January 2022 global electronics manufacturing supply chain sentiment report found that materials and labor costs continue to be the largest issue facing the electronics supply chain, with nine in 10 electronics manufacturers reporting rising materials costs and more than three-fourths reporting rising labor costs.

Georgia Tech Leads Effort to Strengthen State's Defense Manufacturing Industry ▶

Department of Defense grant enables collaboration with Spelman College, Technical College System of Georgia, and the Georgia Department of Economic Development in pilot project.

BAE Systems, Embraer to Explore Potential Defense Variants for the Eve eVTOL Aircraft ▶

BAE Systems and Embraer Defense & Security have announced plans to embark on a joint study to explore the development of Eve's electric Vertical Take Off and Landing (eVTOL) vehicle for the defense and security market.

Lockheed Martin Achieves First Ultra-Secure, IIoT-based Smart Factory With IPC-CFX, Aegis' FactoryLogix ▶

Aegis Software announces that Lockheed Martin's Lufkin facility has connected key SMT machines of all key types using the IPC-CFX standard, providing IIoT data for Aegis' FactoryLogix MES platform in a single standard language, contextualizing accurate, timely, detailed data into smart manufacturing values, all in a highly secure environment.

Nano Dimension Sells DragonFly IV 3D-AME System to Leading Western Defense Force ▶

Nano Dimension Ltd., announced that it has sold another DragonFly IV 3D-AME Printer and FLIGHT Applications Software package, both first released in mid-November 2021, to a leading Western Defense Force.

Calumet Electronics Earns IPC-1791 QML Recertification ▶

Calumet Electronics further positions itself as an industry leader in printed circuit board and IC substrate engineering for the defense industry with its recent IPC-1791 Qualified Manufacturers Listing (QML) recertification.

Aegis' IIoT-based MES Platform to Improve Speed, Quality, and Traceability ▶

Aegis Software, a global provider of IIoT-based Manufacturing Execution Software (MES), announces that CARI Electronic, EMS partner in Valence near to Lyon, France, has chosen FactoryLogix as a key part of their MES Industry 4.0 journey.

Design for Test, Part 2

Designers Notebook

by Vern Solberg, CONSULTANT

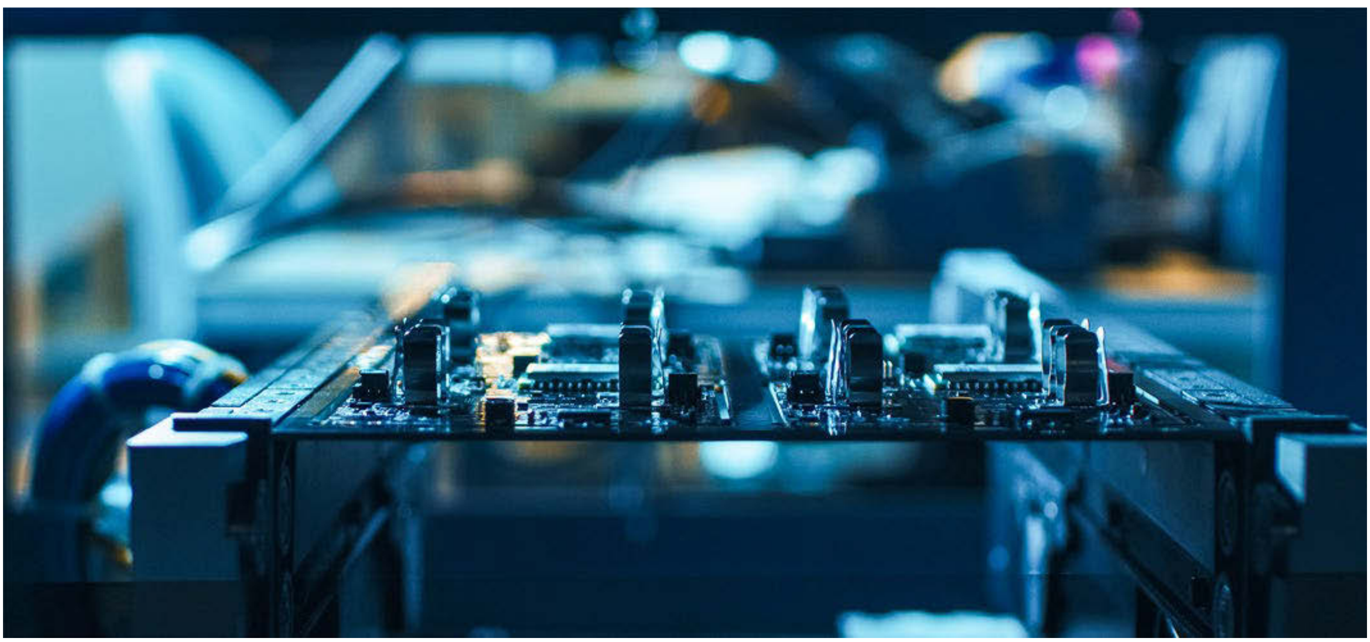
Current generations for PCB designs have increased in complexity. The product developer and assembly service provider, whether in-house or outsourced, must consider manufacturing efficiency, throughput, and process yield. While design for manufacturing is an absolute necessity for controlling manufacturing costs, design to accommodate product testing does need attention as well. The primary concern is to ensure that the end product will perform reliably without compromise.

Designing the circuit board for testability is a key requirement in meeting the overall design for manufacturing (DFM) criteria. Throughout the circuit board's development process, the designer must contemplate the essential features required for electrical testing, testing to verify component functionality, and identifying PCB assembly defects. Imperfections discovered during testing may include defects that occur during the assembly process, opens and shorts attributed to solder operations, component placement errors, or the unknowing inclusion of counterfeit, defective or damaged components that can affect both performance and long-term product reliability.

Part two of this series focuses on three PCB assembly test methodologies:

- Functional testing
- Partitioned functional test
- In-circuit test

In determining what type of electrical testing to adopt, the product developer will need to consider production volume and specific end-user requirements. The requirements for con-



sumer electronics differ greatly from those implemented for medical equipment, automotive, or aerospace and defense applications. For example, low-volume assemblies may only require functional testing while higher-volume assemblies or those products requiring uncompromised operation, will most likely adapt a more sophisticated test strategy.

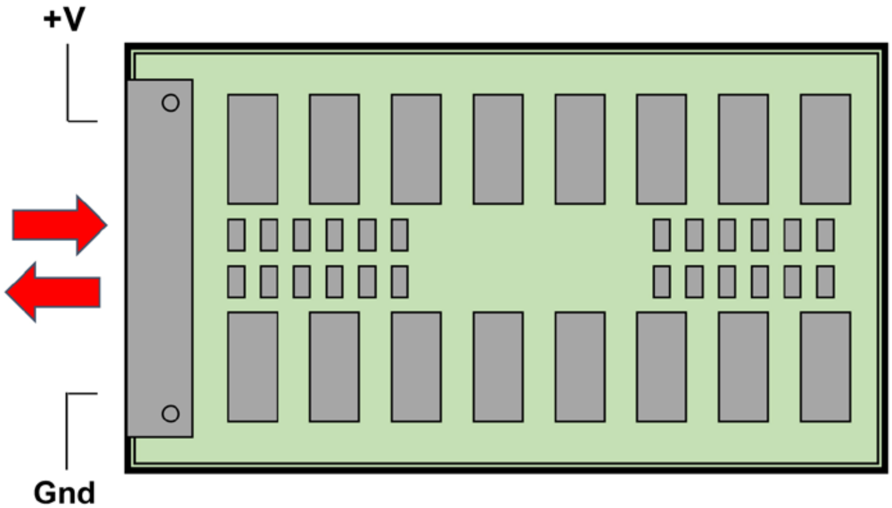


Figure 1: Functional circuit test access using I/O connector.

Electrical Test Strategy

When developing a test strategy, many factors must be considered to determine which test methods will be used. If possible, a test engineering specialist will review the schematic diagram before the circuit board design begins. For example, they may suggest that the designer divides complex logic into smaller, functional logic sections; provides additional capacitor elements that enable decoupling semiconductor devices; and furnish pull-up resistors on unused logic terminals. The goal is to maximize the fault coverage, but not always at any cost.

Low volume or less complex circuit board assemblies may rely on a basic functional, “go or no-go” testing while high-volume assembly or products having greater complexity will most likely adapt some form of automated or semi-automated functional partitioning or in-circuit testing (ICT). These methods, when used together, could achieve 100% fault coverage, ensuring that no circuit board assembly leaves the factory with a potentially compromising manufacturing defect or functional fault.

Most solder process-related defects (opens and shorts) can be detected visually using magnification. High-volume manufacturing may employ an automated optical inspection (AOI) system or even X-ray inspection systems, especially suited for detecting elusive solder defects, under BGA or QFN devices, for example.

Functional Test for PCB Assembly

To physically secure the assembly for functional and partitioned circuit test applications, a dedicated fixture is developed. Electrical test will be performed with power applied to evaluate the actual performance of the finished board assembly. Access for this type of testing is typically made through an interface connector or probe interface with input/output lands dedicated to the test procedure (Figure 1).

With fewer component parts to contend with, the detection of component failures or assembly related defects will be fairly routine. When the assembly fails due to component malfunction, identified components will be de-soldered and replaced before secondary inspection and re-testing.

Larger boards with high component density will require more complex test preparations. Partition testing is commonly semi-automated, requiring a dedicated test station and access to benchtop measurement gear. This type of testing is often utilized for measuring amplifier, RF and IF (intermediate frequency), and other sensitive circuit functions. The partitioning of functions allows the operator to isolate and make specific measurements on critical circuit

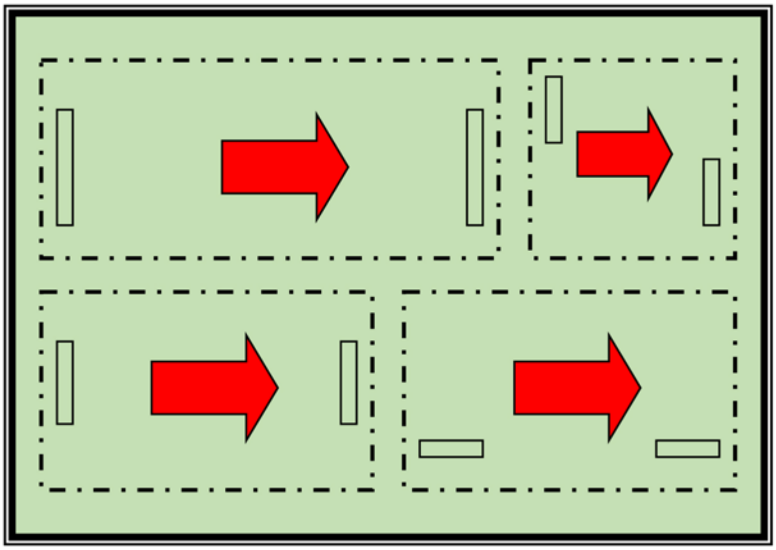


Figure 2: Circuit partitioned for focused functional access.

functions, enabling faster identification of the defects whether component level or defects related to the assembly process (Figure 2).

Additionally, the test engineer may request that the circuit board designer provide a means of disabling on-board clock functions. The purpose of furnishing this disabling capability is to expedite testing and improve test efficiency.

In-Circuit Testing (ICT)

To certify absolute assembly quality the industry commonly implements a more sophisticated electrical testing solution. The fixtures developed for in-circuit testing employ dedicated spring probe fixtures that interface with test systems pre-programmed to measure every net within the circuit board.

Preparing for assembly level testing on the board, the panel must first be furnished with tooling holes. A minimum of two non-plated tooling holes are needed to maintain a fixed position of the board under test.

Tooling hole requirements:

- At least two non-plated tooling holes must be furnished on the PCB
- Ideally, the tooling holes are located at diagonal corners of the board
- Position components and test probe lands away from tooling holes

Programming for ICT is designed to measure overall functionality, identify any component termination locations needing attention, confirm that the values specified for all passive devices are correct, and that semiconductors are functioning properly.

In-Circuit Net Access

To enable the in-circuit test system to function without compromise, 100% test probe access must be provided for every "net," the common interface between all component terminals. Anything less than the 100% net access will compromise test coverage. The example furnished in Figure 3 represents a high-density SMT assembly requiring two-side test probe access.

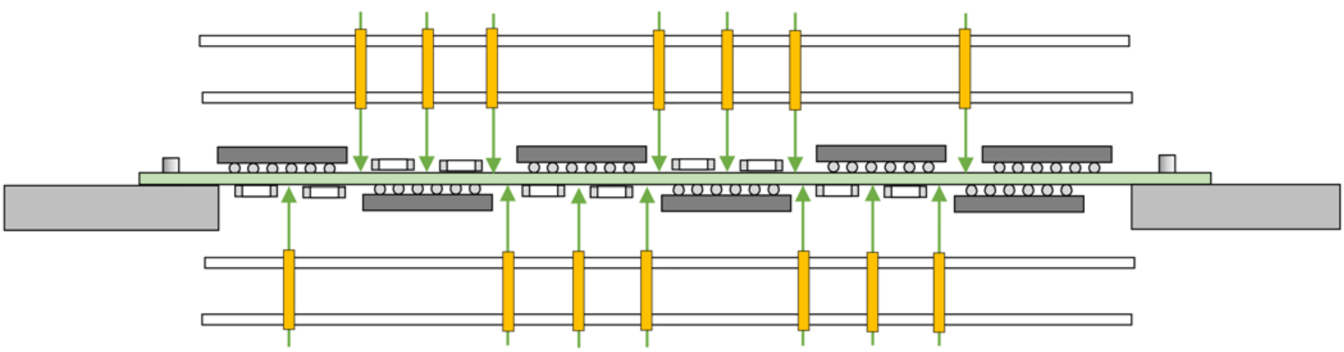


Figure 3: In-circuit test fixture with two-sided probe access.

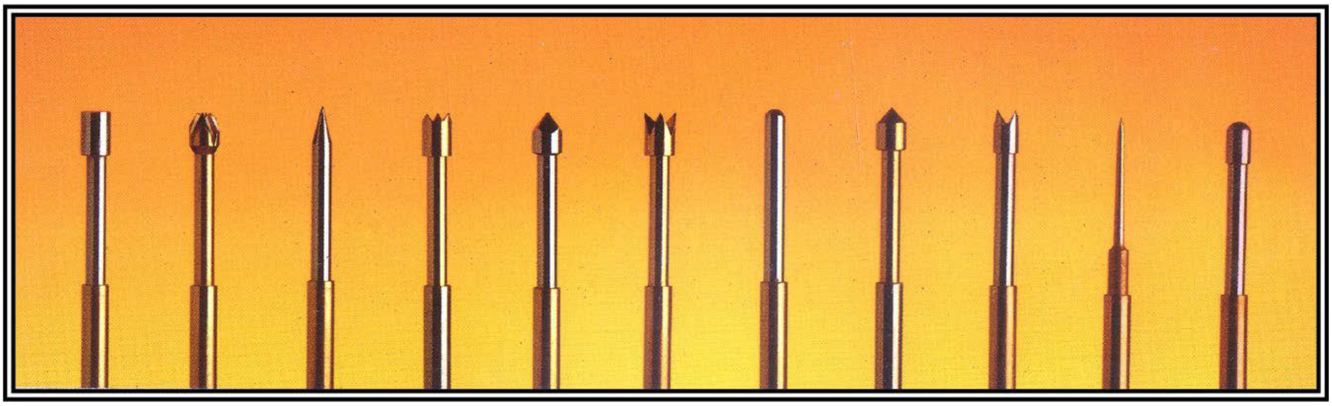


Figure 4: Test probe design variations.

Probe Designs and Test Pad Size

The minimum surface area provided for probe interface is influenced by the alignment accuracy of the test fixture to the circuit board's test sites and the probe contactor design. The test probes furnished for in-circuit test are designed to apply uniform contact pressure at all locations. These spring-loaded contactors have a broad range of tip designs to access even the most confined test pad locations (Figure 4).

Test fixture providers note these variables are not usually considered during the initial quoting process and can impact both fixture development time and cost.

Basic ICT Requirements

- One test node per net
- Probe spacing of 2.0 mm (0.080") preferred
- Probe spacing of 1.2 mm (0.050") acceptable[1]
- Probe land on PCB should be ≥ 0.3 mm (.012")
- All nodal access from one side preferred
- Double side test probe access may be required[2]
- Provide 1.2 mm (0.050") probe body to device body clearance

To enable the in-circuit text fixture developers to prepare the platform for accessing each net of the circuit they will need the schematic diagram for the product, a net list, and the bill of materials defining the components reference designator, as well as each component's physical attributes. The CAD files will determine the location and orientation of each component and the precise X-Y coordinates for test probe access.

In part three, the focus will be on boundary-scan testing, an integrated method for testing interconnects on populated printed circuit boards. An extremely rapid test method, it enables both in-circuit test coverage and system diagnostics. DESIGN007

References

1. The smaller needle probes are fragile and bend easily.
2. Double-sided text fixtures are typically three times in cost.

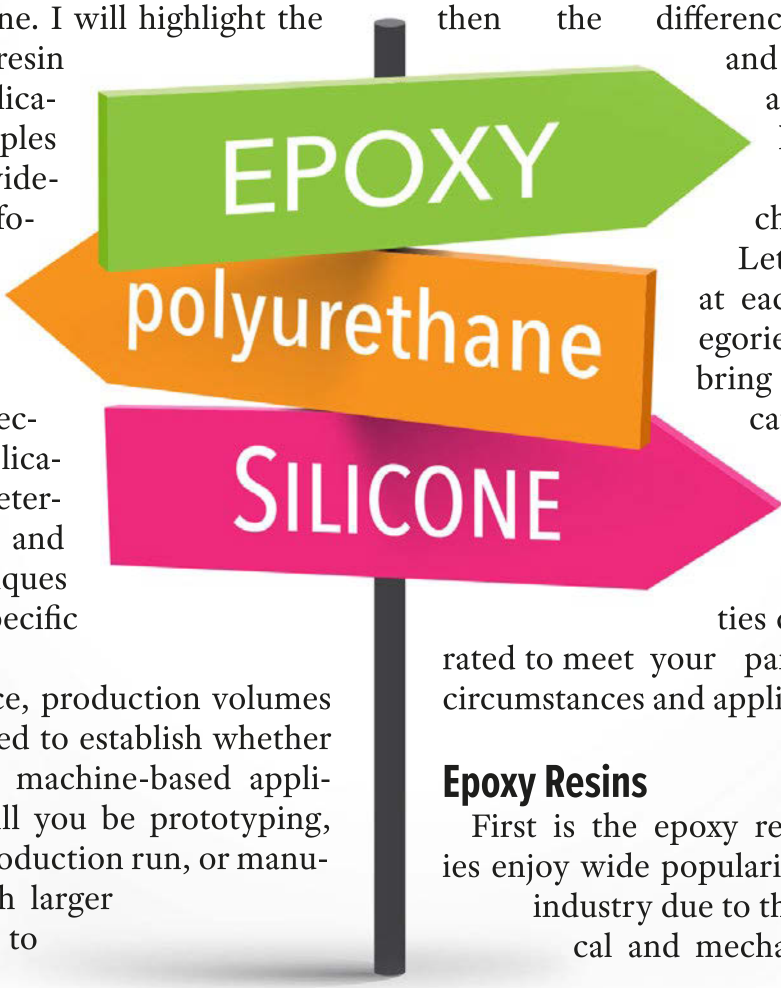
Vern Solberg is an independent technical consultant, specializing in SMT and microelectronics design and manufacturing technology. To read past columns or contact Solberg, click here.

How to Select the Best Resin Category for Your Application

Sensible Design

by Beth Turner, ELECTROLUBE

Resin chemistries are extensively used for potting and encapsulation in the electronics and electrical industries, but not everyone gets it right the first time. With so many resin options available, deciding which resin is best suited for your application can present a challenge by itself. In this month's column, I will look at the features and benefits found in each of the three major categories: epoxy, polyurethane, and silicone. I will highlight the different types of resin properties and applications, using examples from Electrolube's wide-ranging resin portfolio, and provide an overview of the different chemistries available to improve product selection for various applications as well as determine which mixing and application techniques are appropriate to specific production needs.



In the first instance, production volumes need to be considered to establish whether you use manual or machine-based application methods. Will you be prototyping, delivering a short production run, or manufacturing on a much larger scale? Your answers to these questions will also define which resin types are compatible with these application techniques. For example, if you are potting prototypes or small volume runs you will most likely use manual methods, while much larger volume runs will be best served by automated, machine-based mixing and dispensing systems. Should you be looking to work up from the prototype stage to a full production run, then the differences between manual and machine mixing and dispensing will have to be taken into account when choosing your resin.

Let's start by looking at each of the three categories and what they can bring to the table. Each category has distinctive properties and your choice will be guided by how these properties can be best incorporated to meet your particular production circumstances and application needs.

Epoxy Resins

First is the epoxy resin category. Epoxies enjoy wide popularity in the electronics industry due to their excellent electrical and mechanical properties, as

well as their ability to offer protection against chemicals and high temperatures. Their use enables components to be protected against dust, moisture, and short circuits, and as standard epoxy resins have better thermal conductivity than air, they provide more efficient dissipation of heat from components, prolonging service life.

> **Today there are a wide range of different epoxy-based resin systems available, offering a comprehensive range of different properties.**

Today there are a wide range of different epoxy-based resin systems available, offering a comprehensive range of different properties. A range of different cure speeds and cured properties can be obtained using different hardeners; amines, amine modified polymers and polyamides are all available. Some reactions can be very fast, and also very exothermic, leading to the possibility of a runaway reaction, particularly with large potting volumes. This can be mitigated by modifying the hardener or by using a filler, which will help absorb the heat of reaction and can often help achieve flame retardant properties. It is important to note that the temperature at which a resin is cured will affect not only its cure speed, but also the quality of the end result. For example, post curing epoxy resins can often improve the glass transition temperative, so it is important to carry out some trials before committing to a specific cure profile.

One of our newest epoxy resins is a highly thermally conductive resin with low viscosity, ideal for potting cells within electric vehicle batteries. The UL94 V-0 approved epoxy potting compound can withstand long durations at high temperatures and can assist with securing cells in place whilst dissipating the heat away to the surroundings. This particular epoxy resin creates a protective shield around the battery and enables strong adhesion, high temperature resistance, and retention of characteristics throughout the thermal cycling process.

This epoxy resin portfolio includes a variety of clear, opaque, one- and two-part products with a host of useful properties to meet most requirements from potting and sealing to dipping, including exceptional electrical and thermal characteristics, flame retardancy, and resistance to chemicals and fuels. Optically clear resins may be desirable for applications such as LED lighting fixtures, as the cured resin will obviously need to retain its clarity for the life of the unit. Colourless resins are also useful for prototyping applications as the encapsulated components are easily viewed during and after environmental and mechanical testing. On the other hand, coloured/opaque potting and encapsulation resins conceal what lies beneath the encapsulant surface, providing an effective foil against counterfeiters or those wishing to copy a circuit layout, helping you to protect your intellectual property.

Polyurethane Resins

While epoxy resins are hard encapsulants when cured, polyurethane resins are typically elastomeric—more rubber-like—in their cured state, which is particularly useful if the circuit to be potted contains delicate components. Like epoxy resins, polyurethane resins provide chemical, dust, and moisture resistance, as well as excellent electrical insulation and good adhesion to most substrates, both metal and plastic. Unlike their epoxy counterparts, polyurethanes have a lower exotherm during cure, even for fast cure systems. Polyurethane resins typically have a lower continuous operating temperature range and should not be allowed to rise above 130°C for the majority of its lifetime.

One of our newest polyurethane resins has successfully been deployed in a car park sensor application, where the resin requirement needed to avoid RF interference and provide a dielectric constant of between 3-4 at 50Hz. It has a low viscosity ensuring sufficient flow in small gaps, as well as a wide temperature range of -60 to +125°C and a low exotherm (<35°C). With a Shore hardness of A80, the resin is suitable for protecting delicate components whilst providing good impact resistance to the external environment. It is a high-demand resin for IoT applications, due to its proven capability of protecting RF transmitters, sensors, and circuitry from harsh environments, vibration, temperature extremes, and water ingress.

It is much easier to tailor the cure speed with polyurethane systems, and the usable life and gel time of these can be adjusted to suit customer requirements, leading to faster process times and less work in progress. Polyurethanes show lower exotherm during cure than epoxies, and the heat generated is not usually a problem, even for fast cure systems.

Silicone Resin Chemistry

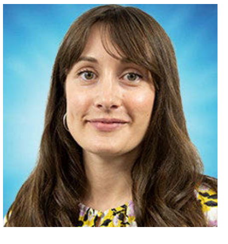
Though not as popular as epoxy and polyurethane resins, silicone resins do offer some distinct advantages when used as an encapsulating resin because the cured products have a high degree of flexibility across a wide temperature range, excellent chemical, dust and moisture resistance, and good electrical insulating properties. Silicone resins tend to be more expensive than epoxies or polyurethanes, but are ideally suited where high continuous operating temperatures (above 180°C) are required. Moreover, the exothermic temperature when working with silicone systems is very low indeed, ensuring compatibility with heat-sensitive components.

One product I work with is a two-part silicone potting and encapsulating resin which was primarily developed for the protection of LED drivers, however, its properties make it ideally suited for EV battery protection. The thermal conductivity is 1 W/mK, making it suitable for use in applications where the operating temperature will be up to 200°C. Its good flow characteristics make it an excellent choice for extremely tight spaces or difficult geometries.

What You Need to Know

As a rule of thumb, silicone resins have the broadest temperature range (-50 to +250°C), are generally soft resins, and are not as chemically resistant as some of the other chemistires. Some polyurethanes have a low glass transition temperature and are suited to lower temperatures than silicones (-60°C). Epoxies are more designed for higher temperature applications (-40 to +200°C), but have excellent adhesion to a wide range of substrates and excellent chemical resistance.

The majority of resins I use are two-part systems that, when mixed together in the correct ratio, react to form polymeric materials. By careful formulation, the properties of the cured resin can be tailored to meet individual customer requirements. If you foresee any challenges with matching resin types to your production procedures, be sure to contact your supplier's technical support team for further advice. In my next column, I shall be taking an in-depth look at some of the most frequently asked questions we get asked as resin experts and will be exploring various options in response to these enquiries. **DESIGN007**



Beth Turner is head of encapsulation resins at Electrolube. To read past columns from Electrolube, click here. Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments,* and watch the micro webinar series "Coatings Uncoated!"

SAP—Changing the Way You Look at PCB Design

PCB Talk

Feature Column by Tara Dunn, AVERATEK

Which side of these competing aphorisms do you most relate to: "The only thing constant is change" or "The more things change, the more they stay the same." I can build a case behind both when looking through the lens of someone working in the printed circuit board industry.

Electronics are rapidly evolving and challenging printed circuit board designers to add more functionality in smaller and smaller spaces. This pushes designers to search for and utilize new "tools" in their toolbox. Whether that is moving into the world of flexible circuits or rigid-flex, or jumping into technology created with semi-additive processes and suddenly being able to route with 50-micron line and space, 25-micron line and space or even finer, PCB designers are continually learning and adapting. Through that continuous change, designers are also expected to meet quality and reliability standards that have been a benchmark for decades.

One of the exciting changes and one of these new "tools" for printed circuit board designers and fabricators is the semi-additive PCB processes for printed circuit board fabrication. While these processes are not new to the electronics industry, they are new to the printed circuit board segment of the industry. These semi-additive processes gained recognition when the smartphone market started using these processing techniques to achieve 35-micron feature sizes in key PCB designs. There are a small handful of fabricators that cater to the high-volume consumer electronics market, yet until the past year or so, there have not been many options available for low-volume, high-mix work.

This is changing rapidly. Subtractive etch processes are typically constrained at about 75-micron line and space. There are fabricators that will take a purchase order for 50-micron feature sizes, formed by subtractive etch processes, but this comes with a significant cost

added, along with yield and reliability concerns. This has driven printed circuit board designers to increasingly rely on complex HDI structures to accomplish their routing. Stacked and staggered microvias and multiple lamination cycles are often required. These, too, come with yield and reliability concerns, not to mention increased cost.

The exciting news is that a handful of forward-thinking PCB fabricators have invested in semi-additive processing capabilities and now provide an option for those outside of the high-volume consumer electronics industry to take advantage of the benefits of routing with much smaller trace and space constraints. With any change, there is a learning curve to be expected and both fabricators and designers are navigating this now. Some embrace change and jump in with both feet. Others take a more cautious approach and wait to see how this develops.

From a fabrication standpoint, at a surface level, these semi-additive processes are a simple addition to the processes that they are currently using day in and day out. These processes move the constraint from the etching process to the photolithography process yet are still run with the existing laser direct-imaging equipment and all the same chemistries for electroless and electrolytic copper. Once the traces are formed, the circuit panel processes through fabrication in the same fashion as a subtractive etch layer would.

As with any change to process, there is a learning curve and things that can catch you off guard. For example, again from the fabrication perspective, there are certain resists that resolve these finer feature sizes better than others and some process tweaks that need to be done to the photolithography process for the best results. And, with these finer feature sizes, the handling and cleanliness become more important. Arguably, while this can seem to be a challenge, once resolved, yield and reliability increase for all product, not only the SAP layers.

Switching perspective to the PCB designers' adoption of features created with SAP processes, there is a similar learning curve. All the major design tools are able to accommodate these features sizes, with relatively little additional effort. At the same time, designers are working hard to understand the impacts. For some, simply reducing overall size, or reducing layer count is the critical element. For others, signal integrity is key. There is a continually growing body of knowledge around this design space. When you decrease line width, there is an impact on impedance and the best way to mitigate this or to use it to the best advantage is very design dependent.

There will be many tech sessions throughout the next several months that drill down into these considerations, looking both at the impacts at the board level and at the impact to the overall electronics design. One interesting case quickly showed the improvement in power consumption once the design was reviewed more holistically rather than just from the bare board perspective.

As much as things change, they stay the same. As this effort is going on, industry associations such as IPC are working to establish guidelines for acceptance and reliability. Designers are requesting design guidelines, and as mentioned in a previous column, I am not necessarily in favor of that until we understand more about how to be apply the new capability. This is our opportunity to think outside of the box and collaborate on how to best reset the traditional technology curve. This is an exciting time for the PCB industry. Change is inevitable. Lean in and take advantage of these new capabilities. **DESIGN007**

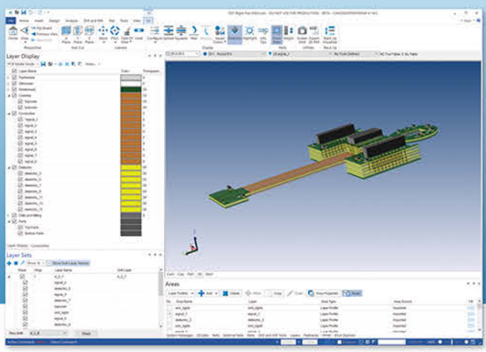
Tara Dunn is the vice president of marketing and business development for Averatek. To read past columns or contact Dunn, click here.

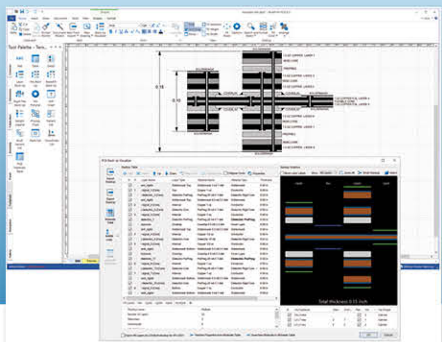
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

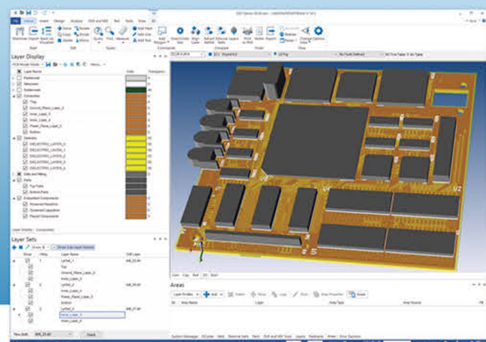
BluePrint-PCB®

CAM350®

- Import and Visualize Flex, Rigid-Flex and Embedded Component Designs
- 3D Visualization to Validate PCB Construction and Component Assembly
- Manage Variable Stackup Zones for Rigid-Flex Designs
- Easily Create Custom Flex or Rigid-Flex Fabrication and Assembly Documentation







DownStream Technologies

For more information visit downstreamtech.com or call (508) 970-0670

Managing Risk With Model-based Engineering

Digital Transformation

by David Wiens, SIEMENS EDA

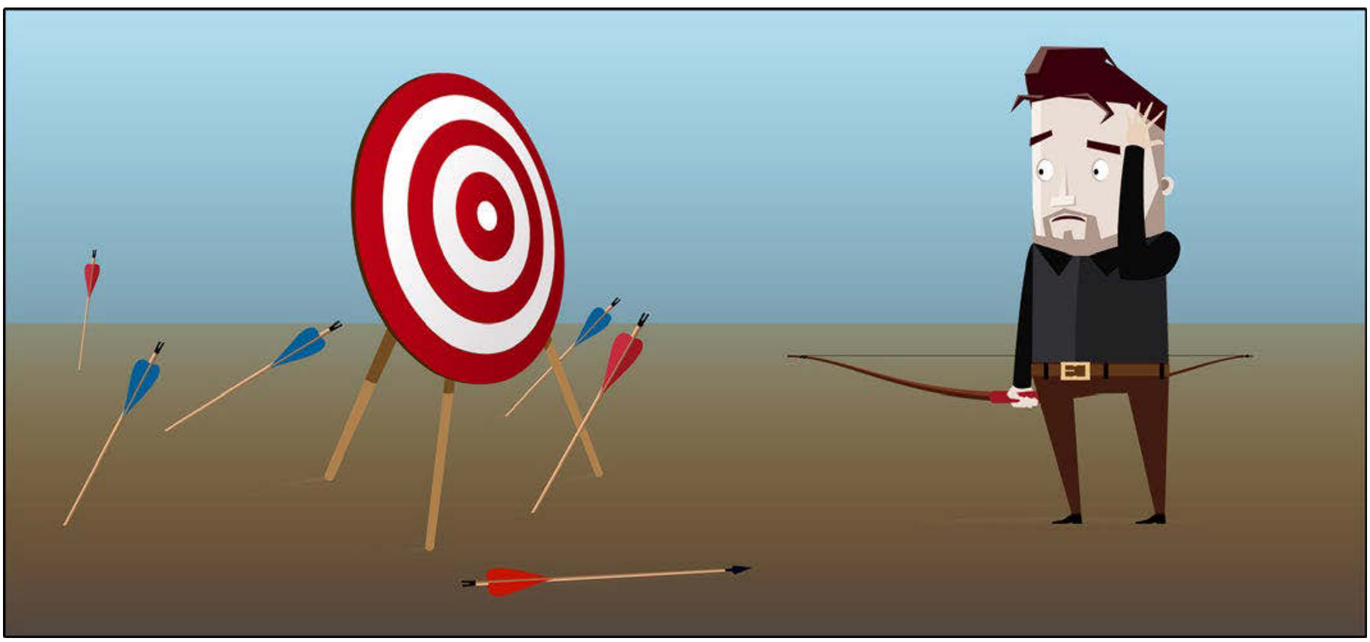
The more complex the system, the more likely it is to fail.

Companies at the forefront of electronic systems engineering understand this basic tenet of risk analysis. They must face the many challenges of rapidly developing markets and futuristic products. Reaping the rewards of these new opportunities and innovations requires more complex products, processes, and, often larger more complex organizations.

Technology has not only increased the complexity of individual domains, but also the number of domains. And these complexities introduce a lot more risk of failures. These include failure to meet delivery targets and system requirements, failure to achieve manufacturing yield, and failure to function in the field and continue to do so over the expected life of a product.

Electronic systems themselves are becoming more complex, not just at the individual board level, but at the subsystem and system levels as well, with multi-board designs becoming increasingly common. Furthermore, this complexity spills over to mechanical engineering and manufacturing as all domains need to become more integrated and cohesive to move toward creating final products more efficiently and effectively. It's no longer possible to design the electronics without taking into consideration the other domains that come together to create that system—including the software, mechanical, and electrical domains.

All this brings new kinds of challenges. Complexity is no longer something we can look at as solving a single domain challenge. We must look at more sophisticated solutions that help us in a multi-domain context and help bring



In times of challenging allocations, you need a supplier that will LEAD you through thick and thin
We accept the challenge! Utilizing our global supply network, we DELIVER an All-in-One solution!

WE DO IT ALL ...

We offer an All-in-One solution through our experience, integrity, and clear communication. Get on board for our joint journey into the future.

www.PCB-technologies.com

ALL IN ONE

PCB TECHNOLOGIES™
leading the way

those multi-domain architectures together earlier in the product design cycle. That solution requires model-based engineering (MBE).

Model-based engineering has been around since Mylar and tape gave way to a revolutionary little thing called computer-assisted design. It's no surprise that what people mean by "MBE" has had a variety of shadings over time. But what do we mean by MBE today in this context?

Basically, it starts with a digital model (the digital twin) that's intended to represent something physical. The more we can use digital models throughout the design process, the more we can avoid using paper and handing off non-digital descriptions. In other words, we want to establish a digital thread, which is a communications framework that allows a connected data flow and integrated view of a product's data throughout its lifecycle—spanning ideation, realization, and utilization. The digital thread allows us to pass digital models from one step, one domain, and one discipline to the next, instead of having to regenerate new digital models with every incarnation of the design.

By leveraging the digital thread, the digital twin, and existing modeling capabilities, we can start addressing system design challenges. The digital thread enables continuity between domains and traceability from requirements all the way to manufacturing. As we increase the sophistication of the digital thread, more data and metadata can be shared along its ribbon, increasing the fidelity of the exchange.

The digital twin has evolved as well, allowing more complex interactions to be validated, analyzed, and simulated earlier in the design cycle. These digital models are not limited to design; they continue to be used in manufacturing processes and even in the matching models of a product in use. Through scalable, multi-domain flows that leverage the digital twin and digital thread framework, companies reap the benefits of reducing the need for expensive, time-consuming physical prototypes.

Thus, a model-based engineering approach consolidates advanced multi-domain system models, allowing for the definition of a multi-domain solution and early (shift-left) simulation at the system level. Importantly, all of this takes place before implementation and while the physical realization comes to fruition. As a result, model-based system engineering supports design, process, and system scaling, addressing exploration, design, verification, validation, and certification of complex electronics functions across all the development domains. Therefore, MBE is effective in mitigating the risks of electronic systems design and manufacture.

To enable multi-domain system design through MBE, we leverage the digital thread to integrate the flow from requirements through manufacturing and across multiple design

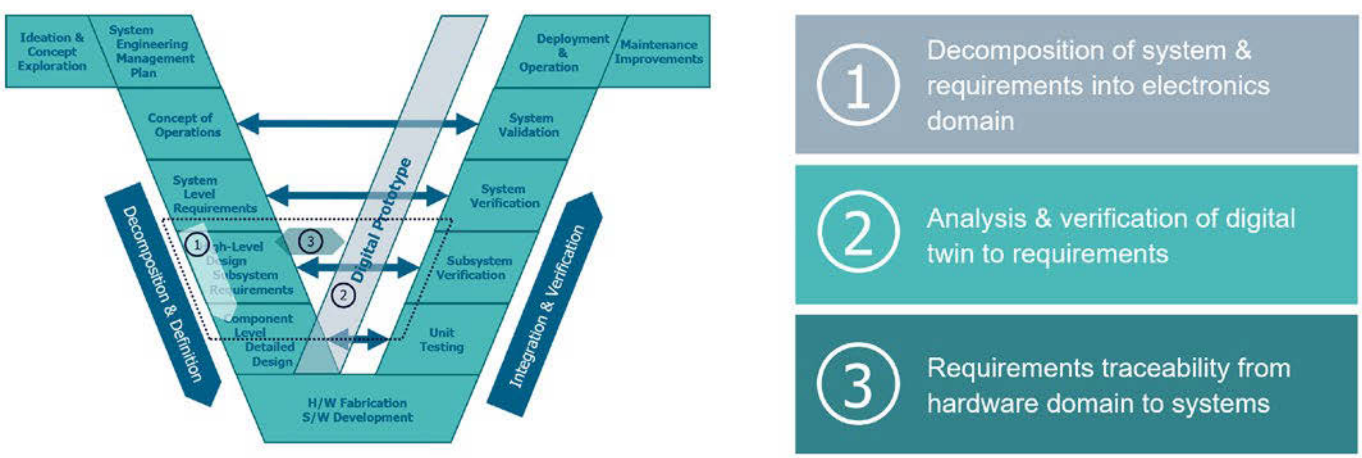


Figure 1: Orchestrating MBE for electronic systems require focus on three key areas.

domains. The digital twin is used to make system-level trade-offs, verify the design, and make optimizations through analysis.

This flow consists of three broad activities that are essential to closing on the architecture, development, and verification of complex electronic systems. It begins with decomposition of the product and requirements into the electronics domain. Decomposition involves taking the high-level architecture and design requirements and breaking them down into different domains, including mechanical, electronic/hardware, electrical, and software. There is often a gap between the requirements and domain architectures, requiring manual interpretation by the system engineer (think drawings and napkins) to decompose the requirements into multiple domains.

Once the electronics domain component is defined, it must be further decomposed into (potentially) multiple boards, with IC packages and ICs on them. Only then can traditional authoring tools like PCB or IC flows be leveraged to create the high-fidelity digital twin for the hardware domain.

In the next column, I'll address how MBE enables analysis and verification throughout the design process.

By embracing the digital thread and digital twin as they move through the MBE flow, multi-discipline, multi-domain engineering teams can build the most exciting and innovative products of tomorrow, today. DESIGN007

David Wiens is Xpedition product manager for Siemens Digital Industries Software. To read past columns or contact Wiens, click here.

The PCB Design Secret Sauce for RF Applications

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

Design and manufacture of PCBs for radio frequency (RF) technology is a unique animal. RF had been considered a niche, thought of only in terms of television broadcasts, commercial airline phones, and military radar systems. Now, light industrial and consumer applications ranging from remote meter reading to home security systems are just the tip of the RF iceberg.

More and more new applications are now relying on RF, meaning PCB designers increasingly need to become fluent in the use of high-speed low-loss PCB material used in the production of boards for RF applications. Sooner rather than later, you'll probably find yourself designing a board manufactured with these less familiar materials, so it's important to understand challenges and best practices associated with them.

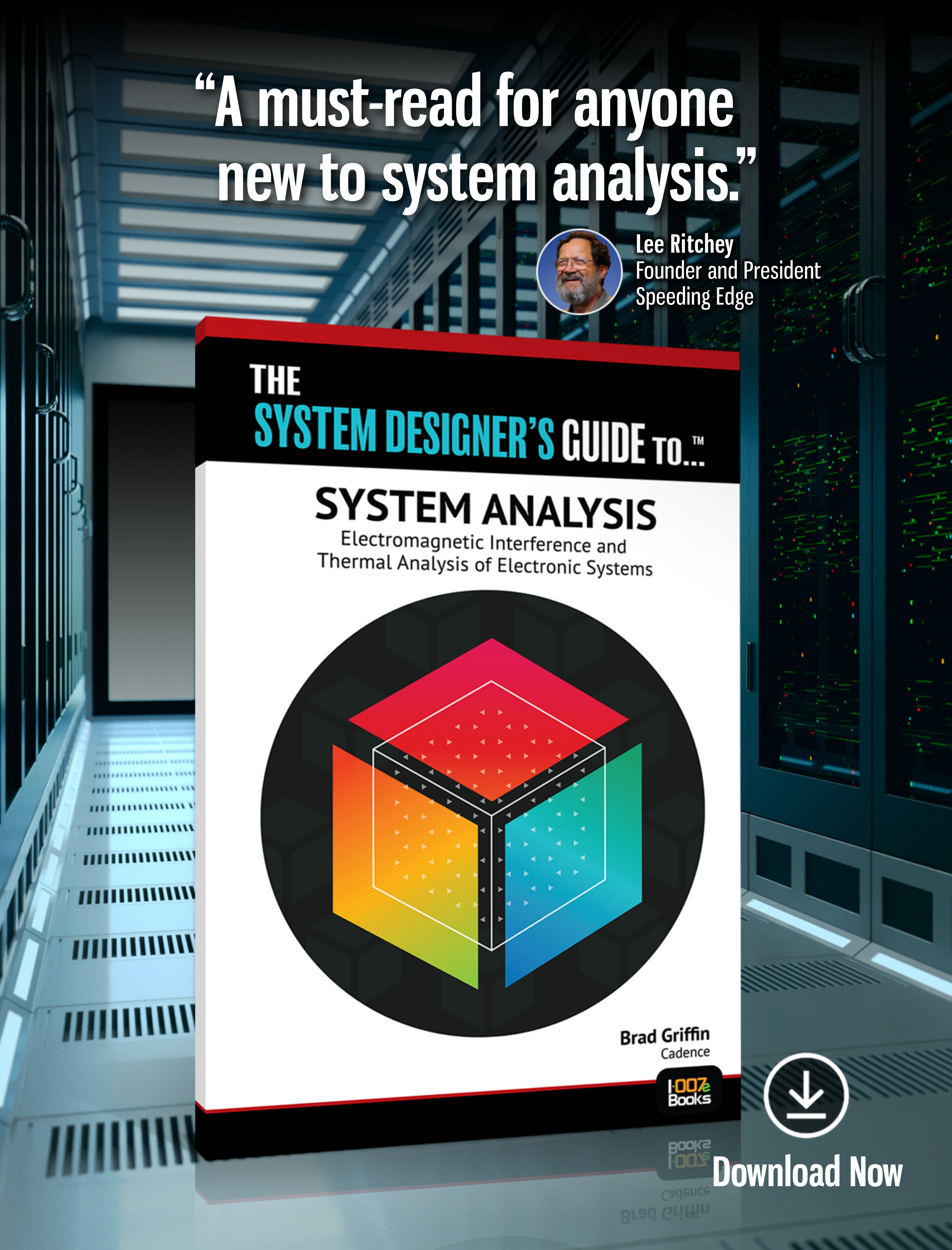
Are the benefits to high-speed, low-loss PCB materials worth it for my design? Yes, these materials are what enable innovation and have been vital to the creation of 5G wireless networks, autonomous vehicle development, and the internet of things (IoT). RF boards are more expensive to prototype, but if you adhere to best practices for working with the materials and specifications required for RF applications, you'll have more success and be more cost-effective in the long run.

High-Level RF PCB Design Practices

Best practices begin with focus on manufacturability. With RF boards, the materials chosen will impact every element of your design, from thickness to layout to surface mounts. This makes the decisions about materials critical for cost-effective prototype and manufacturing.

Designers accustomed to concentrating on circuit speed and density must adjust their thinking and focus on potential issues with frequency. RF boards should be made from materials that minimize energy loss and maintain dielectric (Dk) uniformity. To do both, designers have to choose among dozens of different controlled-Dk and low-loss materials. What you choose will depend on your outputs, budget, and design requirements.

And your design needs to be cost-effectively manufacturable by your chosen PCB fabrica-


"A must-read for anyone new to system analysis."
Lee Ritchey
Founder and President
Speeding Edge
THE SYSTEM DESIGNER'S GUIDE TO...™
SYSTEM ANALYSIS
Electromagnetic Interference and Thermal Analysis of Electronic Systems
Brad Griffin
Cadence
I-007e Books
Download Now

tor. Collaborating with them from the outset about materials will help avoid production issues related to copper surfaces, materials varying by layer, blind and buried vias, and surface prep for solder mask. All these elements are interrelated, and decisions made in one area can ripple through your design.

For example, designers must consider how the fabricator's copper treatments will impact RF performance. It's also important to factor in via structure as it relates to copper surface thickness. Multilayer RF PCBs can have layers whose properties differ a great deal, so it is vital to work closely with your manufacturer to balance performance with manufacturability.

The real key to optimizing your RF PCB designs is to choose tools, processes, and a manufacturing partner that meet your unique needs. Treat your PCB fabricator as a member of your design team—prioritizing open and persistent collaboration with each.

Best Practices at Work

Trust is everything in these relationships. We have adhered to the open channels mantra working with RF solutions-provider Krytar over the years. Offering the highest standard of RF test and measurement services available to the broadband industry, Krytar supports development of all types of systems, including radar and satellite.

For Krytar's microwave components and test systems to work as designed, the RF circuit function must be optimal. If the board specs deviate from the PCB design by even a tiny fraction of millimeter, it can compromise functionality. The higher the frequency, the smaller the wavelength—making it critical to maintain tolerances and stick to design specification. That's no small feat, even in the best of circumstances.

As Krytar's PCB manufacturing partner, we evaluate the customer's design to ensure the manufactured RF circuit will function as intended and are not afraid to provide input on the design if we think we have a better idea about how to build a board that will do what it's supposed to do. This commitment to each other is what ensures positive outcomes.

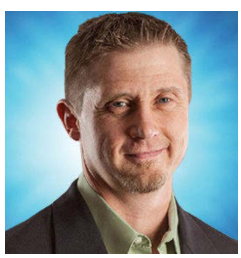
Focusing on Continuous Improvement

RF applications will keep growing in number and increasing in complexity. For designers and manufacturers alike, this requires continuous evaluation of capabilities and improvement of processes. As RF applications became a bigger part of our business, our focus on continuous improvement grew in importance.

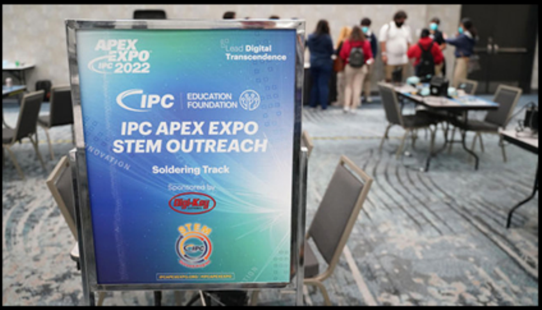
Commitment to quality is just as important as the commitment to the customer relationship. How we deliver quality matters, and that is never truer than during collaboration with designers. Those interactions are where we constantly find new ways to improve both process and product in the RF arena.

Designers should expect their manufacturing partners to exceed their expectations and create value in every aspect of the production process by fostering communication and constantly evaluating customer needs. The production process is not unlike the symbiotic relationship between components on a board. Each element of production should be viewed by all parties as part of a larger ecosystem where individual processes are managed as part of the interrelated whole.

The margin for error in the RF arena is tiny. Even a pixel-width flat spot on an arc can become a false antenna and create bad behaviors. The designer and manufacturer working together—able to focus on granular detail without losing sight of the bigger picture—have the best outlook for producing a quality RF PCB cost-effectively. **DESIGN007**



Matt Stevenson is the VP of sales and marketing at Sunstone Circuits. To read past columns or contact Stevenson, click here.


APEX EXPO IPC 2022
Lead Digital Transcendence
IPC EDUCATION FOUNDATION
IPC APEX EXPO STEM OUTREACH
Soldering Track


APEX EXPO IPC
WELCOME


STEM
IPC
PCB DESIGN


The Commitment To Reinvest Now
To Ensure Success In The Future!
This Year, APCT Will Invest $10M In New Equipment & Integrated Systems
• Enhanced Productivity
• Reduced Cycle Times
• Support of New & Emerging Technologies
The Solution Is APCT
APCT
Passion I Commitment I Trust
512.548.7435 | APCT.com
Printed Circuit Board Solutions
APCT.com
APCT Santa Clara HQ 408.727.6442
APCT Anaheim 714.921.0860
APCT Orange County 714.993.0270
APCT Wallingford 203.269.3311
APCT Global 203.284.1215


girls who
CODE
KYZEN


CLASS OF
22


I-Connect007
REALTIME with...
EXCLUSIVE EVENT COVERAGE
APEX EXPO
IPC
2022
Thank you to our sponsors:
BLACKFOX
1991 2021 BÜRKLE NORTH AMERICA, INC.
CalcuQuote
elisa company
ICAPE GROUP
The power of precision
SUPER DRY®
Totech EU
TAIYO AMERICA
realtimewith.com

BOOK EXCERPT

Chapter 7

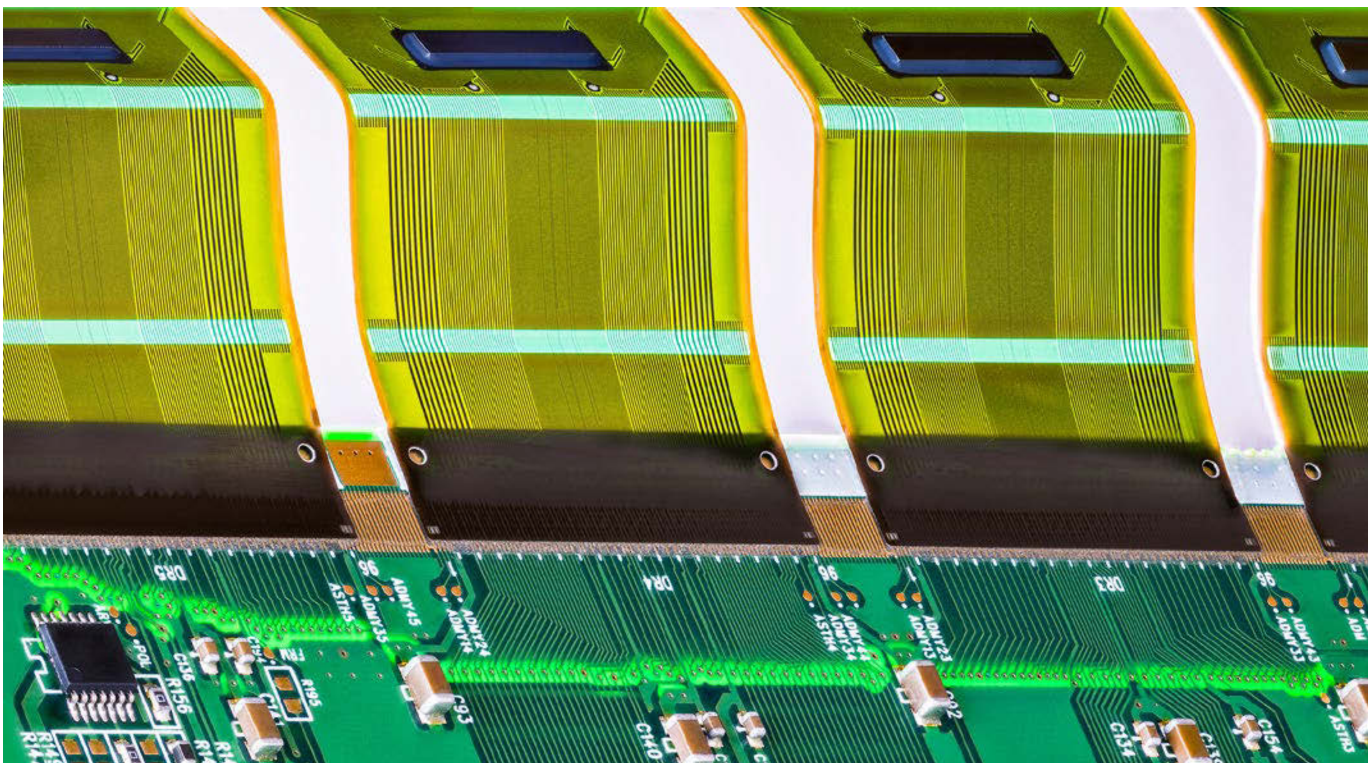
Rigid-Flex Materials

Article by Bill Hargin
Z-ZERO

The following is an excerpt from the recently-released, The Printed Circuit Designer's Guide to... Stackups: The Design within the Design *by Bill Hargin. Click here to download this entire book.*

For most of my career, I worked exclusively with rigid PCBs. Rigid-flex boards were those things in my cellphone. Then, about six years ago, while working in the laminate space, I started getting requests for low-flow prepregs and getting pulled into rheology testing. Since then, I have learned a lot more about rigid-flex stackups and included rigid-flex support in our stackup-software solution. This chapter shares some of the lessons I learned along the way.

Flex and rigid-flex designs refer to a PCB that is partially or entirely constructed using dynamic, flexible substrates instead of being exclusively made of rigid substrates. Flexible



materials allow for denser designs because components and traces can be placed and routed in three dimensions, eliminating physical connectors. This reduces cost and impedance discontinuities while increasing reliability. Rigid-flex designs also have improved electromechanical functionality, including dynamic bending, vibration and shock tolerance, heat resistance, and weight reduction.

Their increased shock and vibration tolerance makes flexible designs popular within medical, automotive, military, and aerospace applications that necessitate dependable field operation. The ability to bend portions of the circuitry facilitates dense constructions, leading to widespread use in flat panel displays and cellular devices, while also making wearable technology possible.

Rigid-Flex Materials

The materials used in rigid-flex are a bit different than the cores and prepregs we introduced in Chapter 1. Figure 7.1 shows the different types of flexible/rigid-flex PCBs according to the IPC-6013 specification.

- Type 1 are single-sided flexible printed boards containing one conductive layer, with or without stiffeners.
- Type 2 are double-sided flexible printed boards containing two conductive layers with PTHs, with or without stiffeners.
- Type 3 are multilayer flexible printed boards containing three or more conductive layers with PTHs, with or without stiffeners.
- Type 4 are multilayer rigid and flexible material combinations containing three or more conductive layers with PTHs.
- Type 5 are flexible or rigid-flex printed boards containing two or more conductive layers without PTHs.

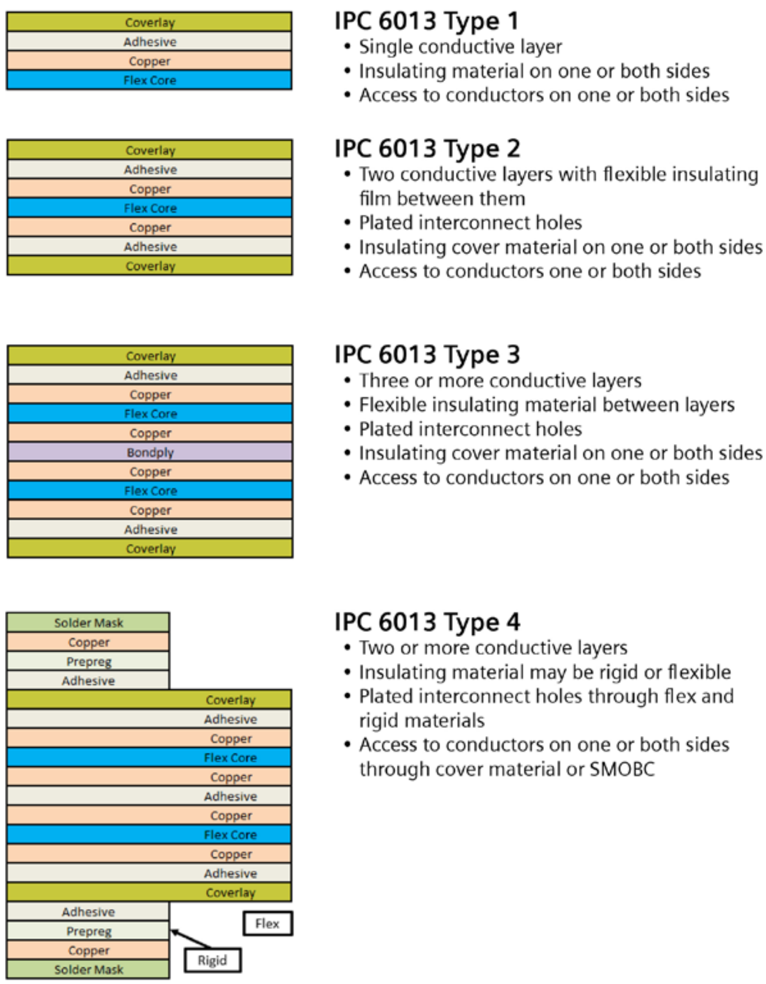


Figure 7.1: Flex/rigid-flex PCB types.

Copper in rigid-flex designs is adhered to flex dielectrics in various ways. Adhesive may be added where copper is bonded directly onto the base material. Stiffeners are sometimes added to reinforce a flex area for component placement or routing holes. Copper can either be electrodeposited (ED, less flexible, lower cost) or rolled annealed (RA, more flexible, higher cost). Flex planes often have a hatched pattern etched into them because the reduction in copper makes them more flexible.

Coverlays are flexible materials, typically on the outside of a flex substack. They protect and insulate the flex circuitry on the surfaces, preventing it from lifting. Coverlays are typically made from acrylic, polyimide, or polyester. A typical coverlay construction, as shown in Figure 7.1, has a polyimide dielectric on top and an adhesive on the side facing the copper.

Bondplies are similar to coverlays, but they are used on inner stripline layers, with adhesive on both sides and the polyimide dielectric in the middle. If you were gluing two flex cores together, this is where a bondply would come in.

Flex cores typically span all substacks, carrying the copper from one end to the other. Common brands for flex cores are DuPont Pyralux or Panasonic Felios, with lots of sub-flavors within these two product families. Flex cores without adhesive are used for high-performance rigid-flex applications. Adhesive-based flex cores are less expensive and generally used for single- or double-sided or low-layer-count flex applications.

Low-flow and no-flow prepregs are used on the rigid substacks in lieu of traditional prepregs to keep the resin from flowing onto the flex substack, which would cause a portion of the flex region to become brittle and a create a potential failure mode. No/low-flow prepregs are typically available in 106 and 1080 glass styles. As the names suggest, low-flow prepregs flow a bit more than no-flow prepregs. Based on my research, low/no-flow prepregs are available from the following manufacturers: AGC-Nelco, Arlon, Isola, TUC, Showa Denko (formerly Hitachi), and Ventec.

Low-flow prepregs are typically stacked one above the other, with the outer ply going to the rigid edge, while the inner layer is recessed by 50 mils so that the coverlay from the flex substack can nest into the rigid board and be held onto by the outer low-flow prepreg (Figure 7.2).

Flex cores and bond plies, polyimide films, and coverlays don't have glass weaves, but as noted, no-flow prepregs are glass-reinforced.

Design Considerations

You can choose from different flex circuit strategies depending upon your application environment. Operating temperature range and duration along with thermal cycling requirements will be crucial in your decision whether to use rigid-flex or not.

For your design, the number of cycles and the bend method, whether flex-to-install or dynamic flexing, is also important. Flex-to-install indicates that the board needs to be bent to be installed in an end-product or system. Dynamic flex is when it gets bent repeatedly

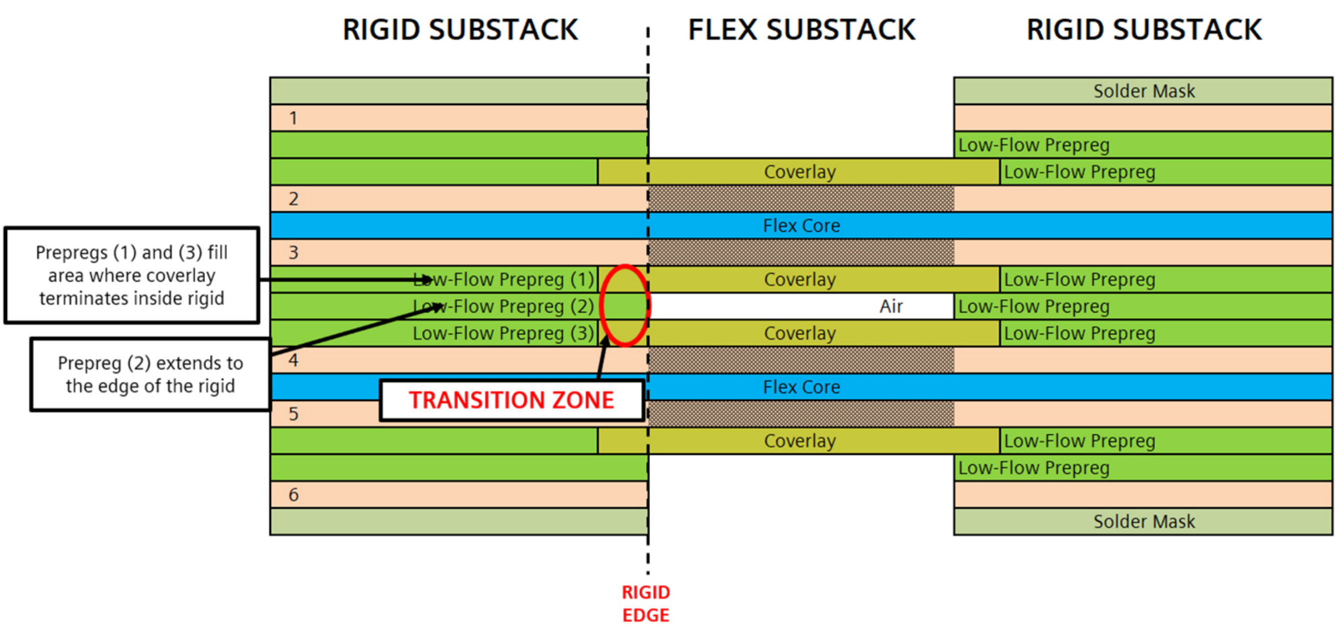


Figure 7.2: Coverlays from flex substacks are nested within a transition zone in the rigid substack.

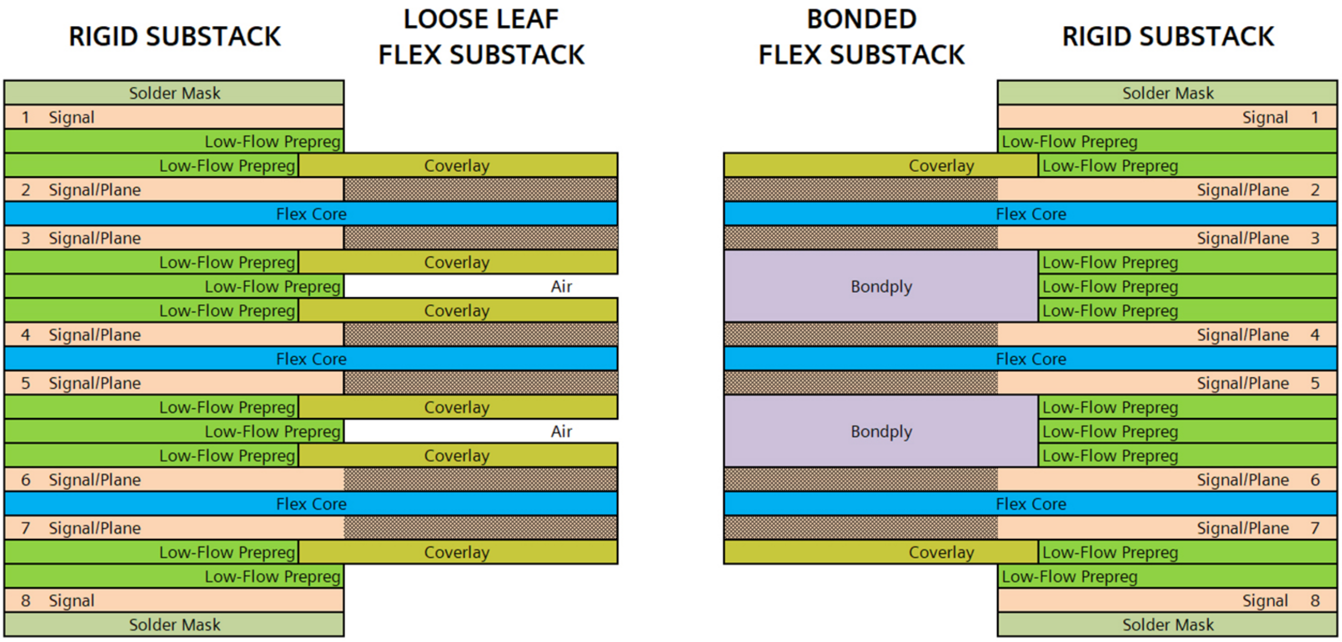


Figure 7.3: Loose-leaf designs are most flexible for 90- and 180-degree bends. They are the simplest to manufacture but not recommended for impedance control. Bonded is less flexible, thus requiring a more generous bend radius.

out in the field during use. The minimum bend radius is something to consider, as well, where tighter bend radiuses require more careful attention.

There are some important differences relative to rigid PCBs that you should consider when designing a rigid-flex project. You can connect flex cores using a loose-leaf or bonded approach (Figure 7.3).

With a loose-leaf design, you have air gaps between individual flex cores, and you get a cheaper and more flexible design. Bonded designs require an additional sublamination with bondply and prepreg, resulting in a stiffer design. Although at least 20 times thicker than the loose-leaf design, it improves stripline impedance control because you have tightly controlled current return paths.

Figure 7.4 is example rigid-flex stackup with a four-layer rigid substack on the left and the right connected by a two-layer flex substack. The polyimide flex core runs end-to-end through the middle. The outside of the flex board has adhesive and coverlay, and the rigid substack has low-flow prepreg layers adjacent to the coverlay and coverlay adhesive.

Dynamic Flexing and Flex-to-Install Recommendations

With flex substacks, thinner overall thickness will outlast thicker constructions. The construction should be balanced in the Z-direction to avoid concentration of stresses. Unbalancing of materials, including copper, dielectric, and adhesive thickness, can decrease flexure life because it causes stress to be distributed nonuniformly.

Copper thickness can affect flex life too, depending on the type of flexure. For maximum flexibility, use rolled annealed copper because of its greater ductility and elongation. The copper grain direction should be parallel to the conductors. Manufacturers that do rigid-flex know how to handle that for you.

The bending and flexing action should be perpendicular to the conductors wherever possible; use radiused corners where perpendicularity is not possible. If you have to do some type of angled routing or changing direction in the flex portion, radius the corners. And no plated through-holes should be used in the flexing area. For dynamic flex and flex-to-

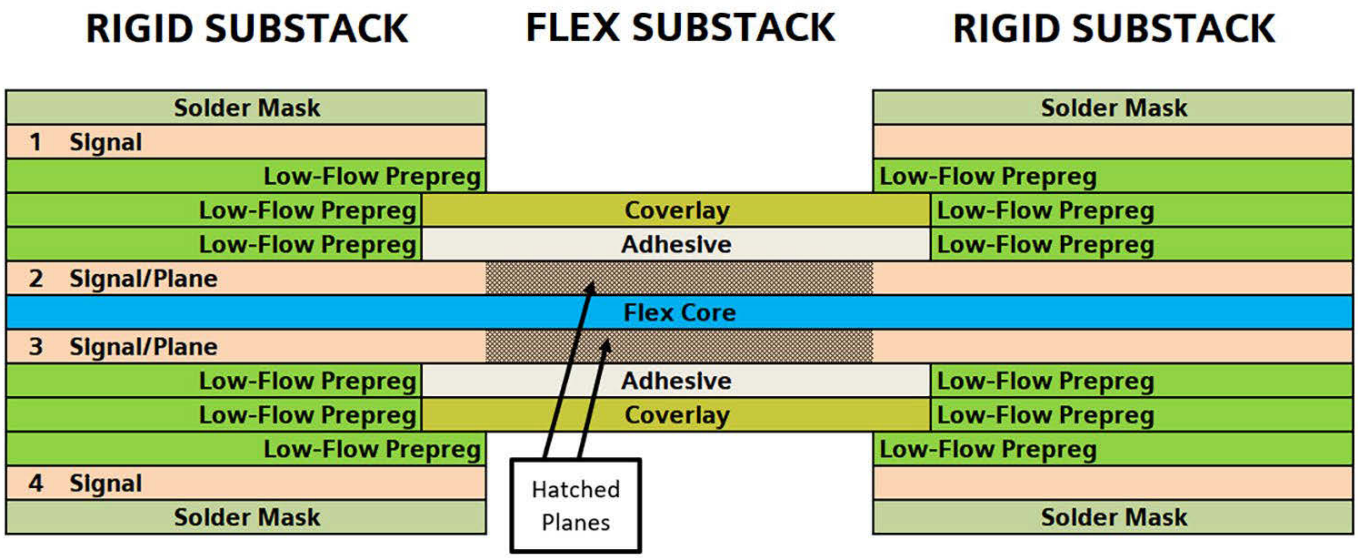


Figure 7.4: A 4-2-4 stackup design. Although this example has an even number of layers, rigid-flex stackups can also have an odd number of layers, somewhat unique to rigid-flex designs.

install applications, it's advisable to use materials without adhesive where possible because materials with adhesive will fatigue sooner.

Flex Reference Planes

Often, a cross-hatched pattern is etched into flex reference planes in order to increase

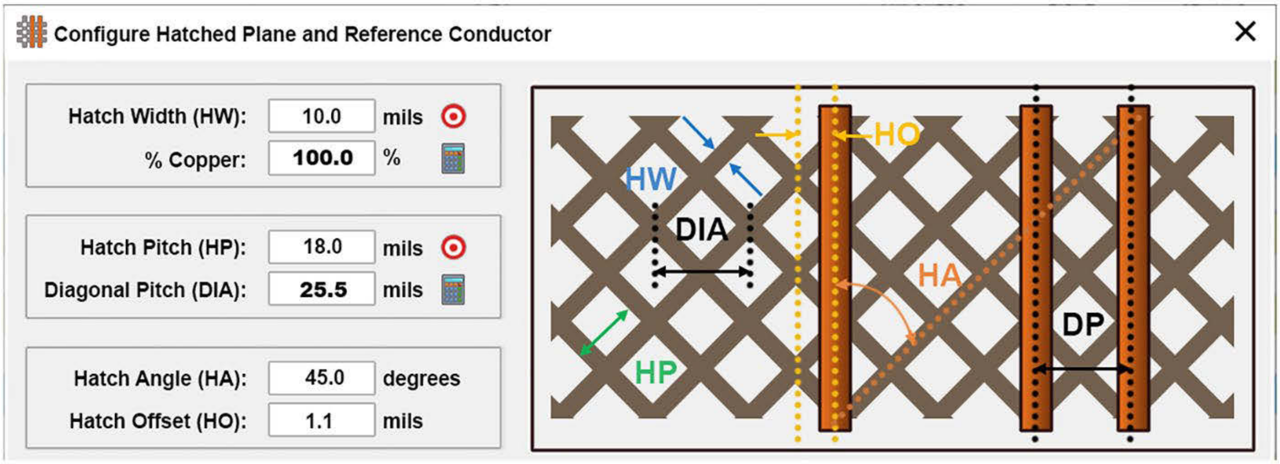


Figure 7.5: In this example, the hatch pitch is 18. The diagonal pitch is 25.5. Hatch angle is 45 degrees, and the hatch offset is 1.1 mils, but ideally would be zero. (Image from Z-planner Enterprise.)

flexibility. When designing a flex hatched plane, your traces should go over the knuckles in the crosshatching. If you have a differential pair and you are concerned about the impedance, the differential pair pitch and the diagonal pitch should be equal. Put your traces right over the knuckles. Figure 7.5 illustrates the hatched-plane parameters to consider.

Minimum Bend, Radius, and Failure Modes

For flex types 1 and 2 as shown in Figure 7.1, the minimum bend radius should be at least six times the overall thickness of the flex substack. For flex types 3 and 4, loose leaf, the minimum bend radius should be at least 12 times the overall thickness, and for flex Types 3 and 4, with bonded flex layers, the minimum bend radius should be at least 20 times the overall thickness.

If these minimum radii are not considered, the result can be an open circuit from conductor fracture, distortions in the adhesive and polyimide, or you can have actual delamination just like you could with a rigid design.

The "bookbinding" technique in flexible-circuit design incorporates greater length of the flex layers on the outside of the bend, similar to the spine of a book. This is the most expensive approach, so it is something to consider when all other methods to achieve 90- or 180-degree flexure have been exhausted. Artwork lengthening in the flex area depends on whether the bend is 90 or 180 degrees. A 180-degree bend means twice the length, which adds a significant cost to the manufacturing process.

IPC-2223 highlights preferred and undesirable bend-area routing practices. Some high-level examples follow:

- Avoid changing the trace width in the bend region; instead, keep it the same width all the way through the bend area.
- Distribute the traces evenly. Lack of symmetry increases the likelihood of stress buildup.
- Route traces perpendicular to the bend region.
- Do not put vias in the bend area. FLEX007

Bill Hargin is the chief everything officer at Z-zero, developer of the PCB stackup design and material selection software, Z-planner Enterprise and author of *The Printed Circuit Designer's Guide to... Stackups: The Design within the Design.*


all flex
Flexible Circuits • Heaters • Assemblies
• Design application assistance
• Component assembly
• Fast response
• Quick delivery
All Flex Flexible Circuits & Heaters
1705 Cannon Lane
Northfield, MN 55057
(800) 959-0865
(507) 663-7162
allflexinc.com

Flex007 Highlights

Testing Flexible Circuit Assemblies ▶

Bert Horner of the Test Connection Inc., talks about the current state of flex and rigid-flex assembly testing and explains some of the differences between testing flex and rigid PCBs, the requirements for testing specialties such as high-voltage flex, and the added demands of handling flexible circuits.

Nano Dimension Establishes First AME NanoLab Facility to Advance Multilayer Printed Electronics ▶

Nano Dimension Ltd. announced a collaboration with TTM Technologies, Inc. to open its first AME NanoLab at TTM's Advanced Manufacturing Center in Stafford Springs, Conn.

DuPont Announces Completion of New Production Line in Circleville, Ohio ▶

DuPont Interconnect Solutions has announced it has completed the expansion project at its Circleville, Ohio, manufacturing site.

Cirexx Adds to Excellon Cobra Laser Fleet ▶

Cirexx International announced that they have acquired and installed a fifth Excellon Cobra laser machine. The hybrid Laser is equipped with both CO2 and UV laser technology.

Nan Ya PCB Reports Strong Finish in 2021 ▶

Taiwan-based Nan Ya Printed Circuit Board Corp. has posted unaudited sales of NT$4.9 billion ($177.2 million at $1=NT$27.70) in December 2021, up by 31.5% year-on-year and by 3% from the previous month.

Flexium Reports 19% YoY Growth in 2021 ▶

Taiwan-based flexible printed circuit manufacturer Flexium Interconnect Inc. has posted sales of NT$4.5 billion ($162.88 million at $1:NT$27.65) in December, up by 23.15% year-on-year (YoY) and an increase of 5.3% from November.

Fabrication of Flexible Electronics Improved Using Gold, Water-Vapor Plasma ▶

Researchers at the RIKEN Center for Emergent Matter Science and the RIKEN Cluster for Pioneering Research in Japan have developed a technique to improve the flexibility of ultra-thin electronics, such as those used in bendable devices or clothing.

SEMI FlexTech Funds 5 New R&D Projects for Flexible Hybrid Electronics ▶

SEMI FlexTech announced more than $5 million in funding for five new research and development (R&D) projects aimed at flexible hybrid electronics innovations for sensors, medical devices, automotive electronics, and other consumer and industrial microelectronics products.

Tikehau Ace Capital Enters Exclusive Negotiations for the Acquisition of the Group ELVIA PCB ▶

Tikehau Ace Capital has entered into exclusive negotiations to acquire 100% capital of ELVIA PCB, a specialized manufacturer of electronic printed circuits boards for demanding markets, such as aerospace and defense, automotive, industrial, medical, railway, and telecom.

Kris Moyer Discusses New IPC Design Role

Kris Moyer

Interview by the I-Connect007 Editorial Team

The I-Connect007 Editorial Team spoke with Kris Moyer, a longtime PCB designer who has just joined IPC's Education Foundation. Kris was one of the judges and creators of the IPC Design competition that culminated at IPC APEX EXPO, and he was eager to discuss his new job and the cutting-edge technology he's seen lately, including additive, flex, and rigid-flex circuits.

Barry Matties: Kris, what's your role at the IPC?

Kris Moyer: I'm on the Education Foundation. I'm actually the instructor who teaches all the printed circuit board design courses at IPC.

Matties: Fantastic. And when you're teaching the design courses, who are your students.

Moyer: Typically, it's all levels of professionals already in industry. Everywhere I've had students who are fresh out of college who got a job, and they said, "Hey, you need to learn this, go take it," to folks who have been in the industry 30 or 40 years and want to expand their knowledge. Basically, it's for anybody in the industry who's working on printed circuit board design and wants to expand their knowledge based on the IPC standards, or just other aspects of board design they might not have been exposed to.

Matties: Doing your program, then, one is learning what the latest trends and demands are for designers.

Moyer: Latest trends, demands, and the latest and greatest IPC standards. It's the latest revision of all the IPC standards, so you're up to date with the latest requirements, standards, methodologies, and so on.

Matties: Why did you join the IPC? What was the appeal here? Why is the IPC so important to the design community?

Moyer: I'll start with the first question. I've been an evening instructor at Sacramento State University, and I recognized the need several years ago to train the next generation of board designers. We weren't seeing a lot of that. We saw folks who were experienced and starting to retire out, but there weren't a lot of people wanting to go into that career. So, I created a course for my university to teach some board design. I was speaking with IPC about that, and they were very interested. They worked with me, we came to an agreement, and I developed the courses for IPC to teach at the professional level. I've been doing that for several years now, and they decided to bring me on full time to develop additional courses.

Matties: One of the things that we're hearing out there in the world is that additive manufacturing is creating a challenge for designers. How are we going to overcome that?

Moyer: Part of it will be with courses like mine that teach how to design with the additive manufacturing processes, while others are just added as that technology gains acceptance and use out in the industry. It will just become heritage knowledge through fabricators, through assembly houses, and so on. It will then be able to convey that information back and forth more efficiently to fabricators just like we do now with traditional boards.

Matties: Are the designers coming to you and saying they want to learn this, or are you going to the designer and saying that you offer this kind of training?

Moyer: Both. The courses I've developed were partly based on feedback. I started with a two-part fundamentals course on the basics of design. The feedback from that, along with my own experience in the industry, said, "These are the areas where we need to add focus on advanced topics."

Matties: What is the hottest topic that the designers are asking for more education on right now?

Moyer: Right now, military aerospace is extremely high in demand, and flexible and rigid-flex printed circuit design is highly in demand. And then there's one that I call advanced packaging, which is not quite the semi-additive yet, but it's the one dealing with microvias and sequential lamination. We're getting to the finer pitch without quite getting to the additive process yet.

Nolan Johnson: So, that's the technology. Now my question is a slightly different spin on that. Everybody has a compelling event that causes them to make the decision to change, to learn, to grow in some way. Are you seeing some trends in those moments for your students as to why they're coming in?

Moyer: The commercialization of space is getting a lot more demand for the aerospace market. The reduction in package sizing, the higher density packaging, is one of the main drivers, both for the rigid-flex and for the advanced packaging course, as parts are getting smaller and smaller into smart watches and the like. How do you get all the parts into something that small? You can't do that with traditional packaging. You have to go to advanced packaging methods, also getting rid of interconnects—getting rid of your wiring interconnects and going with your rigid-flex packaging are really the things I'm seeing drive those.

Johnson: What about going to a more personal level? Sometimes we have career issues or challenges at that level, not just the technology, but our own personal growth. Are you hearing trends from your students in that regard?

Moyer: Not really. I will honestly say that has not really come up as an issue. I guess I'm getting students from all walks of life, but I really haven't heard a lot of them saying, "Oh yes, I had to make a change of this, that, or the other thing." Most of them are usually pretty happy with where they're at. They're just wanting to advance their careers.

Matties: In terms of additive, is that going to be applied in the flex market as well?

Moyer: It can be applied in the flex market as well, primarily in the rigid-flexes, just by the nature of how it's done. Most of the flexes are already very thin coppers to begin with, so the need to do the additive may or may not be there. I haven't seen a whole lot of trend that way yet.

Matties: So when you talked about flex, what is the demand for knowledge in the flex arena?

Moyer: Primarily what I'm getting is in the rigid-flex demand area, for example, where you're basically replacing all your traditional cables and wire harnesses with a piece of flex circuit connecting your two boards. It becomes one integrated assembly, which is then what we call flex to install where it's not a machine that's constantly moving and continuously flexing. You're folding up all your individual board pieces into one shape, putting in your box and packaging it again to reduce your packaging, increase your density, and so on.

Matties: What advice would you give somebody who is entering a design career today?

Moyer: Number one, absolutely get familiar with the IPC standards. Those standards and methodologies will absolutely help you to be successful in creating boards that will not only meet your design needs, but will also be manufacturable, reliable, and high quality. The type of courses we offer is definitely going to be a big help that way.

Matties: Kris, what is your background?

Moyer: I have over 30 years of experience in board design across multiple market segments. I spent about 10 years of my career doing high-precision sensors and very low voltage along with their associated signal conditioning. I had another 10 years in factory automation and industrial controls, such as elevator control systems, and another 10 years in military aerospace, where I got to work on some really cool space and military systems. Part of that time, I worked at Blue Origin for a year, working on the crew module. They just took William Shatner, Captain Kirk, up into space, so I got to work on that module.

Matties: That's exciting.

Moyer: Yes. There's also stuff that's gone to Mars, stuff that's in orbit right now and stuff that's deployed that I can't talk about.

Matties: Great. Well, Kris, we certainly appreciate your time today and your insights.

Moyer: Thank you so much. **FLEX007**

Real Time with... IPC APEX EXPO 2022:

Flexible Circuit Technologies: Rigid-Flex Specialists

As flexible circuit specialists growing their world-wide business, Flexible Circuit Technologies is planning to expand its workforce. Chris Clark, senior applications engineer, discusses the company's 25% growth in the last year, as well as its new plant in Zhuhai, China, which will focus on rigid-flex circuitry. Chris just joined the company in the past year, but he has spent 31 years in the industry wearing whatever hat was required, and he lays out the company's plans for the next year.

TOP TEN

EDITOR'S PICKS

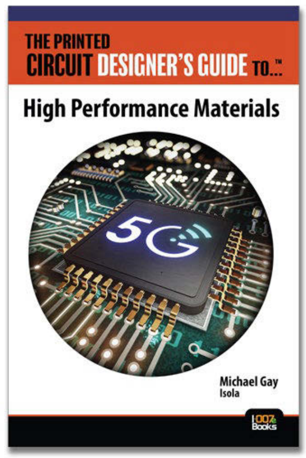
Real Time with... IPC APEX EXPO: Siemens' Supply Chain Solutions

Nolan Johnson speaks with Oren Manor of Siemens Digital Industries Software about the company's booth at IPC APEX EXPO. Highlights include discussion of a DSI platform meant to help designers find and use components in their designs during these tough supply chain challenges.

Tamara Jovanovic Discusses Lessons Learned at AltiumLive

Shortly after graduating three years ago, design engineer Tamara Jovanovic attended her first AltiumLive in San Diego and discovered the global community of PCB designers. With a few years of experience designing PCBs for smart baby beds with Happiest Baby, Tamara is now working on her master's degree.

New Book From Isola Highlights Importance of Material Selection



In *"The Printed Circuit Designer's Guide to... High Performance Materials,"* the latest release from I-007eBooks, readers will learn how to overcome challenges associated with choosing the right material for their specific application. Author Michael Gay of Isola provides a clearer picture of what to know when determining which material is the most desirable for which products.

The Digital Layout: Live or Via Zoom, Knowledge is All Around Us

For the past year-plus, this column has been written by Kelly Dack, our erstwhile communications director, and Stephen Chavez, our chairman. In the next couple of months, the PCEA is transitioning from an all-volunteer organization to one with a fulltime staff, which will allow the board of directors to focus on higher-level strategy.

If You're Not Measuring Inductance, Ask Yourself: Do You Feel Lucky?

Dr. Bruce Archambeault is retired from IBM now, but as he explains, he's still teaching. He taught a class at the show that focused on layout considerations and inductance, and why inductance needs to be measured. There weren't many empty seats in his class.

Sensible Design: The Rise of Resins in IoT Applications

The Internet of Things (IoT) is a platform enabling embedded devices connected to the internet, to collect and exchange data with each other. Devices can begin to interact and work with each other, even learning from each other's experience as humans do. What role do resins play?

Elementary, Mr. Watson: The Five Pillars of Your Library, Part 1

I have recently had some great conversations with many of you, and the same question keeps coming up: What does it takes to have an excellent component library? So, I have decided to kick off the new year by taking a deep dive into your PCB component libraries and looking in detail at the five pillars of your library.

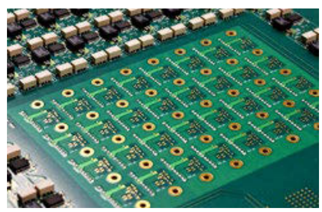
Living in a Material World: High-Speed Design Strategies

We recently spoke with high-speed design expert Lee Ritchey of Speeding Edge, and electronic materials veteran Tarun Amla of Avishtech and Thintronics, about the relationship between advanced PCB materials and high-speed design techniques. They discuss the challenges facing designers and engineers working with materials at speeds that were considered unreachable not long ago.

Kelly Dack Teases AltiumLive 2022

Editor Nolan Johnson chats with guest editor and columnist Kelly Dack, one of the event hosts for AltiumLive Connect 2022. Kelly updates Nolan on the latest about the upcoming AltiumLive virtual conference.

The Art of Using Symmetry—and Asymmetry—in PCB Design



An empty board outline is a PCB designer's empty canvas. Components are the designer's paint palette, and the traces are the brush strokes used to blend and mesh the components together on the canvas. The subject matter is defined by the schematic entry and the tone is often set according to the purpose of the design.

For the latest news and information, visit **PCBDesign007.com**

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

pluritec®
MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

Pluritec North America, ltd., an innovative leader in drilling, routing, and automated inspection in the printed circuit board industry, is seeking a full-time field service engineer.

This individual will support service for North America in printed circuit board drill/routing and x-ray inspection equipment.

Duties included: Installation, training, maintenance, and repair. Must be able to troubleshoot electrical and mechanical issues in the field as well as calibrate products, perform modifications and retrofits. Diagnose effectively with customer via telephone support. Assist in optimization of machine operations.

A technical degree is preferred, along with strong verbal and written communication skills. Read and interpret schematics, collect data, write technical reports.

Valid driver's license is required, as well as a passport, and major credit card for travel.

Must be able to travel extensively.

apply now

Ucamco

Sales Engineer

Germany, Austria, Switzerland, Southeastern Europe e.g. Italy

Ucamco is looking for a sales engineer for our front-end software in the German-speaking area (Germany, Austria, German Switzerland) as well as adjacent markets in the South and East.

Ucamco is a market leader in PCB CAM, pre-CAM software and laser photoplotters with more than 35 years' experience developing and supporting leading-edge, front-end tooling solutions for the global PCB industry.

Responsibilities:

- Selling software solutions
- Selling support contracts and upgrades
- Developing and implementing customer acquisition plan
- Organizing and taking part in roadshows, seminars, exhibitions
- Follow up of current customers and sales
- Contributing insights into the marketing plan
- Reporting to Ucamco's sales director

Requirements:

- Fluent in German, good knowledge of English; other languages a plus
- Frequent traveling to prospects and customers–live contact is important
- Feeling for technical software
- Motivated to succeed as a solution seller
- Strong empathy for the customer
- Self-starter, able to work independently, organized
- Honest, trustworthy, dependable, credible
- Sales and technical expertise in PCB industry a big plus
- Knowledge of market and customer base in German speaking area a big plus
- Used to working from home office
- Traveling to headquarters in Gent (Belgium) for sales and customer meetings
- Good feeling for software is more important than strong sales experience

This is a salary-based position with a commission plan, company car, expense reimbursement, and benefits like health insurance.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

SMT Operator
Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for a **surface-mount technology (SMT) operator** to join their growing team in Hatboro, PA!

The **SMT operator** will be part of a collaborative team and operate the latest Manncorp equipment in our brand-new demonstration center.

Duties and Responsibilities:

- Set up and operate automated SMT assembly equipment
- Prepare component kits for manufacturing
- Perform visual inspection of SMT assembly
- Participate in directing the expansion and further development of our SMT capabilities
- Some mechanical assembly of lighting fixtures
- Assist Manncorp sales with customer demos

Requirements and Qualifications:

- Prior experience with SMT equipment or equivalent technical degree preferred; will consider recent graduates or those new to the industry
- Windows computer knowledge required
- Strong mechanical and electrical troubleshooting skills
- Experience programming machinery or demonstrated willingness to learn
- Positive self-starter attitude with a good work ethic
- Ability to work with minimal supervision
- Ability to lift up to 50 lbs. repetitively

We Offer:

- Competitive pay
- Medical and dental insurance
- Retirement fund matching
- Continued training as the industry develops

apply now

SMT Field Technician
Hatboro, PA

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

Duties and Responsibilities:

- Manage on-site equipment installation and customer training
- Provide post-installation service and support, including troubleshooting and diagnosing technical problems by phone, email, or on-site visit
- Assist with demonstrations of equipment to potential customers
- Build and maintain positive relationships with customers
- Participate in the ongoing development and improvement of both our machines and the customer experience we offer

Requirements and Qualifications:

- Prior experience with SMT equipment, or equivalent technical degree
- Proven strong mechanical and electrical troubleshooting skills
- Proficiency in reading and verifying electrical, pneumatic, and mechanical schematics/drawings
- Travel and overnight stays
- Ability to arrange and schedule service trips

We Offer:

- Health and dental insurance
- Retirement fund matching
- Continuing training as the industry develops

apply now

Career Opportunities

Product Manager

MivaTek Global is preparing for a major market and product offering expansion. Miva's new NG3 and DART technologies have been released to expand the capabilities of Miva's industry-leading LED DMD direct write systems in PCB and Microelectronics. MivaTek Global is looking for a technology leader that can be involved guiding this major development.

The product manager role will serve as liaison between the external market and the internal design team. Leadership level involvement in the direction of new and existing products will require a diverse skill set. Key role functions include:

- **Sales Support:** Recommend customer solutions through adaptions to Miva products
- **Design:** Be the voice of the customer for new product development
- **Quality:** Verify and standardize product performance testing and implementation
- **Training:** Conduct virtual and on-site training
- **Travel:** Product testing at customer and factory locations

Use your 8 plus years of experience in either the PCB or Microelectronic industry to make a difference with the leader in LED DMD direct imaging technology. Direct imaging, CAM, AOI, or drilling experience is a plus but not required.

For consideration, send your resume to N.Hogan@MivaTek.Global. For more information on the company see www.MivaTek.Global or www.Mivatec.com.

apply now

Field Service Technician

MivaTek Global is focused on providing a quality customer service experience to our current and future customers in the printed circuit board and microelectronic industries. We are looking for bright and talented people who share that mindset and are energized by hard work who are looking to be part of our continued growth.

Do you enjoy diagnosing machines and processes to determine how to solve our customers' challenges? Your 5 years working with direct imaging machinery, capital equipment, or PCBs will be leveraged as you support our customers in the field and from your home office. Each day is different, you may be:

- Installing a direct imaging machine
- Diagnosing customer issues from both your home office and customer site
- Upgrading a used machine
- Performing preventive maintenance
- Providing virtual and on-site training
- Updating documentation

Do you have 3 years' experience working with direct imaging or capital equipment? Enjoy travel? Want to make a difference to our customers? Send your resume to N.Hogan@MivaTek.Global for consideration.

More About Us

MivaTek Global is a distributor of Miva Technologies' imaging systems. We currently have 55 installations in the Americas and have machine installations in China, Singapore, Korea, and India.

apply now

Career Opportunities

Career Opportunities

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB shop, is looking for sales representatives for all territories.

Reasons you should work with Prototron:

- Serving the PCB industry for over 30 years
- Solid reputation for on-time delivery (99% on-time)
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- AS9100
- MIL-PRF- 31032
- ITAR
- Global sourcing
- Engineering consultation
- Completely customer focused team

Interested? Let's have a talk.
Call Dan Beaulieu at
207-649-0879
or email to
danbbeaulieu@aol.com

apply now

SIEMENS

Siemens EDA
Sr. Applications Engineer

Support consultative sales efforts at world's leading semiconductor and electronic equipment manufacturers. You will be responsible for securing EM Analysis & Simulation technical wins with the industry-leading HyperLynx Analysis product family as part of the Xpedition Enterprise design flow.

Will deliver technical presentations, conduct product demonstrations and benchmarks, and participate in the development of account sales strategies leading to market share gains.

- PCB design competency required
- BEE, MSEE preferred
- Prior experience with Signal Integrity, Power Integrity, EM & SPICE circuit analysis tools
- Experience with HyperLynx, Ansys, Keysight and/or Sigrity
- A minimum of 5 years' hands-on experience with EM Analysis & Simulation, printed circuit board design, engineering technology or similar field
- Moderate domestic travel required
- Possess passion to learn and perform at the cutting edge of technology
- Desire to broaden exposure to the business aspects of the technical design world
- Possess a demonstrated ability to build strong rapport and credibility with customer organizations while maintaining an internal network of contacts
- Enjoy contributing to the success of a phenomenal team

***Qualified applicants will not require employer-sponsored work authorization now or in the future for employment in the United States. Qualified Applicants must be legally authorized for employment in the United States.*

apply now

Career Opportunities

ARLON ELECTRONIC MATERIALS

Arlon EMD, located in Rancho Cucamonga, California, is currently interviewing candidates for open positions in:

- Engineering
- Quality
- Various Manufacturing

All interested candidates should contact Arlon's HR department at 909-987-9533 or email resumes to careers.ranch@arlonemd.com.

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

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2015 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customers' requirements.

For additional information please visit our website at www.arlonemd.com

apply now

ICAPE GROUP

The power of precision

PCB Field Engineer–North America Operations

ICAPE Group is a European leader for printed circuits boards and custom-made electro-mechanical parts. Headquartered in Paris, France, we have over 500 employees located in more than 70 countries serving our +2500 customers.

To support our growth in the American market, we are looking for a PCB Field Engineer.

You will work in our North America technical center, including our U.S. technical laboratory, and will be responsible for providing technical and quality support to our American sales team.

You will have direct customer contact during all phases of the sales process and provide follow-on support as required.

RESPONSIBILITIES INCLUDE

- Feasibility recommendations
- Fabricator questions and liaison
- Quality resolutions
- Technical explanation (for the customer) of proposals, laboratory analysis or technology challenges

REQUIREMENTS

- Engineering degree or equivalent industry experience
- 5 years' experience with PCB manufacturing (including CAM)
- Excellent technical understanding of PCBs
- Experience with quality tools (FAI, PPAP and 8-D)
- Good communication skills (written and oral)

Communication skills are essential to assist the customer with navigation of the complex process of matching the PCB to the application.

SALARY

Competitive, based on profile and experience. Position is full time in Indianapolis, Ind.

apply now

Career Opportunities

American Standard Circuits
Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

Summary of Functions

The CAD/CAM engineer is responsible for reviewing customer supplied data and drawings, performing design rule checks and creating manufacturing data, programs, and tools required for the manufacture of PCB.

Essential Duties and Responsibilities

- Import customer data into various CAM systems.
- Perform design rule checks and edit data to comply with manufacturing guidelines.
- Create array configurations, route, and test programs, panalization and output data for production use.
- Work with process engineers to evaluate and provide strategy for advanced processing as needed.
- Itemize and correspond to design issues with customers.
- Other duties as assigned.

Organizational Relationship

Reports to the engineering manager. Coordinates activities with all departments, especially manufacturing.

Qualifications

- A college degree or 5 years' experience is required. Good communication skills and the ability to work well with people is essential.
- Printed circuit board manufacturing knowledge.
- Experience using CAM tooling software, Orbotech GenFlex®.

Physical Demands

Ability to communicate verbally with management and coworkers is crucial. Regular use of the telephone and e-mail for communication is essential. Sitting for extended periods is common. Hearing and vision within normal ranges is helpful for normal conversations, to receive ordinary information and to prepare documents.

apply now

BLACKFOX®
Premier Training & Certification

IPC Instructor

Longmont, CO; Phoenix, AZ; U.S.-based remote

Independent contractor, possible full-time employment

Job Description

This position is responsible for delivering effective electronics manufacturing training, including IPC Certification, to students from the electronics manufacturing industry. IPC instructors primarily train and certify operators, inspectors, engineers, and other trainers to one of six IPC Certification Programs: IPC-A-600, IPC-A-610, IPC/WHMA-A-620, IPC J-STD-001, IPC 7711/7721, and IPC-6012.

IPC instructors will conduct training at one of our public training centers or will travel directly to the customer's facility. A candidate's close proximity to Longmont, CO, or Phoenix, AZ, is a plus. Several IPC Certification Courses can be taught remotely and require no travel.

Qualifications

Candidates must have a minimum of five years of electronics manufacturing experience. This experience can include printed circuit board fabrication, circuit board assembly, and/or wire and cable harness assembly. Soldering experience of through-hole and/or surface-mount components is highly preferred.

Candidate must have IPC training experience, either currently or in the past. A current and valid certified IPC trainer certificate holder is highly preferred.

Applicants must have the ability to work with little to no supervision and make appropriate and professional decisions.

Send resumes to Sharon Montana-Beard at sharonm@blackfox.com.

apply now

Career Opportunities

U.S. CIRCUIT

Plating Supervisor

Escondido, California-based PCB fabricator U.S. Circuit is now hiring for the position of plating supervisor. Candidate must have a minimum of five years' experience working in a wet process environment. Must have good communication skills, bilingual is a plus. Must have working knowledge of a plating lab and hands-on experience running an electrolytic plating line. Responsibilities include, but are not limited to, scheduling work, enforcing safety rules, scheduling/maintaining equipment and maintenance of records.

Competitive benefits package.
Pay will be commensurate
with experience.

Mail to:
mfariba@uscircuit.com

apply now

eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

Opportunities are available in Canada, New England, California, and Chicago. If you love teaching people, choosing the classes and times you want to work, and basically being your own boss, this may be the career for you. EPTAC Corporation is the leading provider of electronics training and IPC certification and we are looking for instructors that have a passion for working with people to develop their skills and knowledge. If you have a background in electronics manufacturing and enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Ability to travel required. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

Qualifications and skills

- A love of teaching and enthusiasm to help others learn
- Background in electronics manufacturing
- Soldering and/or electronics/cable assembly experience
- IPC certification a plus, but will certify the right candidate

Benefits

- Ability to operate from home. No required in-office schedule
- Flexible schedule. Control your own schedule
- IRA retirement matching contributions after one year of service
- Training and certifications provided and maintained by EPTAC

apply now

Career Opportunities

APCT

Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

APCT, a leading manufacturer of printed circuit boards, has experienced rapid growth over the past year and has multiple opportunities for highly skilled individuals looking to join a progressive and growing company. APCT is always eager to speak with professionals who understand the value of hard work, quality craftsmanship, and being part of a culture that not only serves the customer but one another.

APCT currently has opportunities in Santa Clara, CA; Orange County, CA; Anaheim, CA; Wallingford, CT; and Austin, TX. Positions available range from manufacturing to quality control, sales, and finance.

We invite you to read about APCT at APCT.com and encourage you to understand our core values of passion, commitment, and trust. If you can embrace these principles and what they entail, then you may be a great match to join our team! Peruse the opportunities by clicking the link below.

Thank you, and we look forward to hearing from you soon.

apply now

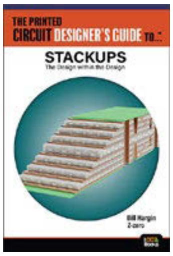
YOUR JOB AD HERE

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

I-Connect007
GOOD FOR THE INDUSTRY

I-007e EDUCATIONAL RESOURCE CENTER

Latest I-007eBooks



The Printed Circuit Designer's Guide to... Stackups: The Design within the Design

by Bill Hargin, Z-zero

Finally, a book about stackups! From material selection and understanding laminate datasheets, to impedance planning, glass weave skew and rigid-flex materials, topic expert Bill Hargin has written a unique book on PCB stackups. Get yours now!

The Printed Circuit Designer's Guide to... High Performance Materials

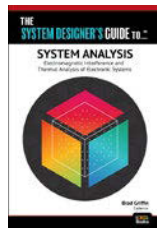
by Michael Gay, Isola

This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. Get your copy now!

The Systems Designer's Guide to ... System Analysis

by Brad Griffin, Cadence

In this book, the author, Brad Griffin of Cadence, focuses on EM and thermal analysis in the context of data center electronics systems. Be sure to also download the companion guide for end-to-end solutions to today's design challenges.

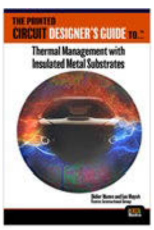


I-007eBooks The Printed Circuit Designer's Guide to...

Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

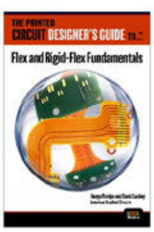
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



Thermal Management with Insulated Metal Substrates

by Didier Mauve and Ian Mayoh, Ventec International Group

Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.



Flex and Rigid-Flex Fundamentals

by Anaya Vardya and David Lackey, American Standard Circuits

Flexible circuits are rapidly becoming a preferred interconnection technology for electronic products. By their intrinsic nature, FPCBs require a good deal more understanding and planning than their rigid PCB counterparts to be assured of first-pass success.

Our library is open 24/7/365. Visit us at: I-007eBooks.com

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

MANAGING EDITOR: **ANDY SHAUGHNESSY**
(404) 806-0508; andy@iconnect007.com

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

EDITOR | COLUMNIST COORDINATOR: **MICHELLE TE**
michelle@iconnect007.com

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

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

SALES MANAGER: **BARB HOCKADAY**
(916) 365-1727; barb@iconnect007.com

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © CRIMSON**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

February 2022, Volume 11, Number 2
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering.................. 37
All Flex.................. 83
Altium.................. 57
American Standard Circuits.................. 61
APCT.................. 73
Burkle North America.................. 91
Cadence.................. 23
Candor Industries.................. 33
Downstream Technologies.................. 13, 63
Eagle Electronics.................. 15
EMA Design Automation.................. 19
Flexible Circuit Technologies.................. 87
I-Connect007.................. 75
I-007 eBooks.................. 2, 3, 25, 69
In-Circuit Design Pty Ltd.................. 49
IPC.................. 71
MacDermid Alpha Assembly Solutions.................. 77
NCAB.................. 5
PCB Technologies.................. 65
Polar Instruments.................. 29
Prototron Circuits.................. 51
Pulsonix.................. 47
Rogers Corporation.................. 27
Siemens Digital Industries Software.................. 41
Summit Interconnect.................. 9
Taiwan Union Technology Co.................. 7
Taiyo.................. 85
US Circuit.................. 43
Ventec International Group.................. 31

Problems solved!
PCB007 MAGAZINE
NOVEMBER 2021
PCB007 MAGAZINE
DECEMBER 2021
PCB007 MAGAZINE
JANUARY 2022
The European Influence
DECEMBER 2021
DESIGN007 MAGAZINE
JANUARY 2022
DESIGN!
HIGH VOLTAGE
DESIGN007 MAGAZINE
FEBRUARY 2022
CHANGE:
RESISTANCE IS FUTILE
DECEMBER 2021
SMT007 MAGAZINE
JANUARY 2022
SMT007 MAGAZINE
FEBRUARY 2022
RISING COSTS
What's Your X?
SUBSCRIBE NOW