I-Connect007

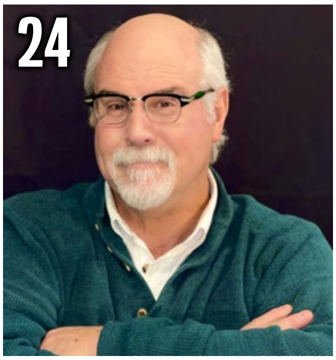
MAY 2023

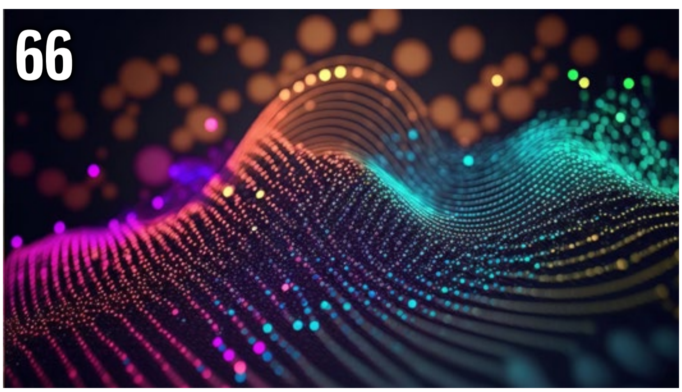
DESIGN007 MAGAZINE

DESIGN007 MAGAZINE

A New Materials Paradigm

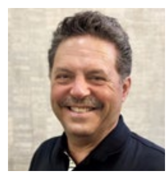
In this issue, our expert contributors examine the development of IPC slash sheets and the need for new, robust material guidelines created specifically for PCB designers. We will also discuss the dangers of over-specifying materials, particularly for offshore manufacturing, as well as the many risks and trade-offs associated with laminate selection.





FEATURE INTERVIEWS

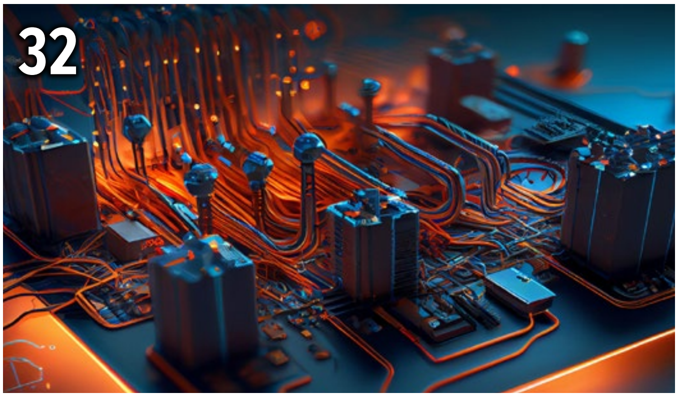
FEATURE ARTICLES



FEATURE COLUMNS

MAY 2023 • ADDITIONAL CONTENTS

DESIGN007 MAGAZINE



BENDING THE POSSIBILITIES

FLEXFINER

BY TAIYO

AVAILABLE IN AMBER

AMBER

- Excellent bendability
- High Resolution by photolithography
- High Reliability
- Simplified process
- UL94 VTM-0 certified
- Low spring back force
- Resolution Line/Space=50/50μm
- Solder mask defined opening (30μm diameter)

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

Visit www.taiyo-america.com for more information.

TAIYO AMERICA

Follow us on social media

A New Materials Paradigm

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

PCB designers are proud of their independent streak; this is one of the few careers in which being labeled "off-grid" is considered a resumé enhancement. Designers all have their own little tips and tricks for designing boards, and this trait carries all the way to the material selection process.

As we learned in a recent *Design007 Magazine* survey, when it comes to choosing the right material for their board, our readers are about evenly split. Almost 30% of respondents said they always consult IPC's slash sheets during the material selection process. One-third said they sometimes use slash sheets in their decision-making progress, but 39% said they never utilize slash sheets.

Once again, it all comes down to data. There just isn't one document that contains all the information that a designer needs to consider when selecting a PCB material. This is why so

many designers say, "I like vendor X for my aerospace boards and vendor Y for medical."

As you'll see in this month's issue of *Design007 Magazine,* slash sheets such as IPC-4101/126 were never meant to be used by designers when comparing PCB laminates. These documents were created to facilitate communication between purchasing and customer service departments. I guess you could say that slash sheets are made for administrative purposes, not engineering.

As the captain said in "Cool Hand Luke," "What we've got here is a failure to communicate." IPC does offer several design guidelines, created specifically for PCB designers, including one just for flexible circuits. But there isn't such a guideline for high-speed designs.

Perhaps it's time for a new set of design guidelines—a new materials paradigm. The ideal design guidelines would contain all the information designers would need to consider when choosing a laminate. These guidelines could also rank the materials by sector, such as aerospace, medical, or industrial. This month, we include a conversation with Doug Sober, IPC's director of materials and IEC engagement. Doug was instrumental in the development of the first slash sheets over 40 years ago, and he details IPC's original intent for these documents, which were not targeted at designers.

We have more features from Barry Olney, Kelly Dack, Geoffrey Hazelett, Tim Haag, and our newest contributor Michael Morando of PFC Flexible Circuits. We also have columns by Istvan Novak and Martyn Gaudion, as well as articles by Anaya Vardya and Mark Gallant.

What's *your* material selection process? **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 23 years. To read past columns, click here.

careers in **ELECTRONICS**

PRINCIPAL ENGINEER

Ever been to a mountain climbing adventure where you, along with the team, follow an experienced lead? You can think of the team lead as the principal engineer who leads a group of new and experienced engineers throughout the project. They are the science people who practice the art of managing their team and projects and complete them within provided resources. So many engineers pursue a master's in management sciences to polish their administrative skills, but that's not a must; instead, most principal engineers earn higher degrees in their relevant field.

What Else Does a Principal Engineer Do?

- Principal engineers are responsible for monitoring the overall design phase of the construction process
- Inspects materials and resources used to ensure projects stay within budget and on time
- Uses their excellent communication and organizational skills
- Works with engineering teams to discuss new ideas, issues, etc.
- Evaluates existing engineering processes and recommends improvements
- Recommends new technologies to improve system performance and reliability
- Provides technical guidance
- Participates in project analysis, design, implementations and commissioning phases
- Provides support in system acceptance testing and validation activities
- Prepares user requirements, technical specifications, design specifications and commissioning plans.

Learn More

Slash Sheets and Material Selection

Feature Interview by the I-Connect007 Editorial Team

In a recent I-Connect007 survey to PCB designers, about one-third of respondents said they consult IPC's so-called slash sheets, such as IPC-4101/126, that contain a variety of information about PCB materials. But many designers say these specs don't contain enough information. Some ask their fabricator to pick a material, and still others say, "It's an RF board, and I always use supplier X for RF." What's the best process for selecting your PCB materials?

Doug Sober helped pioneer the development of IPC's first slash sheets in 1996 for IPC-4101 Specification for Base Materials for Rigid and Multilayer Printed Boards and we asked him to discuss slash sheets—what they are, what they are not, and why PCB designers might benefit from an IPC materials guide developed specifically for designers.

Andy Shaughnessy: Doug, let's start with your work with IPC and slash sheet development.

I started in the laminate and prepreg business in 1978 for General Electric and was sent to my first IPC meeting in 1980. I got heavily involved right away with the task groups regarding specifications for laminates, prepregs, copper foils, glass fabric, resin coated foil, etc. The first standard we did was IPC-4101, but it did not just come out of thin air. We had a document from the military called MIL-S-13949H that had requirements for base materials. The requirements were physical such as flex strength and peel strength, electrical such as Dk and Df, thermal such as Tg and solder float, and environmental such as moisture absorption and fungus resistance. After IPC-4101 was all fin-

Doug Sober

ished, it had specification sheets for simple FR-4s and polyimide base materials of various types.

The military had these specification sheets which described specific base materials and, as laminate and prepreg producers, we had to qualify our products to the requirements of the sheets. With specification sheet 21, for example, a bevy of lab tests proved that we met the requirements. Then the Defense Electronics Supply Center (DESC) would list our product by commercial name on their qualified product list (QPL). For example, Westinghouse's FR-4 product 65M38 as listed on the QPL for specification sheet 21. Once a year, Lowell Sherman or Dave Corbett from DESC came to audit each base material supplier. When the 3-11 subcommittee created IPC-4101, we duplicated MIL-S-13949 because we wanted the military to sign off on it, discontinue their document, and use IPC-4101 in its place. In 1996, with the help of Dave Bergman negotiating with the military higher-ups , IPC-4101 was launched and Mil-S-13949 was discontinued.

In the IPC-4101, a simple FR-4 would also be qualified to specification sheet 21. It was called out the same as it was in MIL-S-13949 as IPC-4101/21. This is how the industry got to calling each specification sheet a "slash" sheet /24, /99, /126 and so forth. But a slash sheet, as we called it, was never intended to be a design guide. It is only there to make minimum requirements for base materials so each could be bought and sold. Each specification sheet had some generic chemistry commonality in the header section with other descriptive words. For example, IPC-4101/21 would say "majority epoxy resin" no filler, woven fiberglass reinforcement and a minimum Tg.

So, when you hear, "With slash sheets, it's hard for the designers to decide what to use," remember that's not what it was intended for. Once a designer has the product they want in terms of the characteristics, it goes through the purchasing department which uses that to make their purchases. We use the terms "slash sheet" and "specification sheet" interchangeably.

For a specification sheet like 4101/126, there might be 10 to 20 different laminate and prepreg suppliers that have materials to fit that. You should be able to buy those and they would reasonably interchange with one another.

Now, since we have so many low-Dk, low-Df, and ultra-high-Tg materials, I think it's time for us to create a design guide for the rigid materials that come from 4101. That is my thought process having looked at what the 3-11 Subcommittee has done since 1996. We have released a lot of base material specifications with specification sheets, but there should be a design guide to help the designers. We have design guides for flexible printed boards and high-speed/high-frequency printed boards. There should be one for standard rigid boards.

This concept would make a really good project: Put together a Tiger Team consisting of 6 to 8 designers along with some base materials experts and, using one of those other design guides as a pattern, we could devise a design guide for these materials fairly easily.

Shaughnessy: Are designers just not aware of the IPC design guidelines? Where's the disconnect?

The designers talk about high speed and high frequency, so they're looking for low Dk and low Df. We have minimums or, in the case of Dk and Df, maximums for those attributes. The maximum can be 0.008 for a Dk at 10 GHz, but the designers may want less than that. They want a better performing base material, but sometimes they cannot find a specification sheet that fits their needs.

The datasheets from the vendors are where they can find the actual Dk and Df they are looking for because the datasheets from the vendors are what's "supposed" to be the typical value you get when you buy the material.

That's where the designers, if they use the datasheets when they are planning, can find the specification sheets that are applicable in the bottom of those datasheets. Every base material supplier puts them on there. The certificate of conformance states that the specification sheet must be referenced.

That is where the designer can find those numbers if the 4101 standard is just too cumbersome to handle. But it would be a good idea for us, as industry experts, to create a design guide that lists the different types of materials and spec sheets that fit the applications, such as, "For high-speed routers, use slash sheet X or Y or Z." With a small group Tiger Team, they could come up with those pretty easily.

Shaughnessy: What's the process for updating a slash sheet?

It's simple and it's the same process we use to add anything to 4101. Somebody says, "I'd like to add a new slash sheet," and the chairman puts it on the agenda. The proposer of a new data sheet comes to the 3-11A meeting, it is put up on the wall, discussed, and a header is proposed for it. In a follow-up meeting, we vote whether it should be included in the balloting for the next revision of IPC-4101. It can also get some massaging along the way. If somebody wants to include that we need fracture toughness as a requirement or CAF resistance, for example, they can make a PowerPoint presentation. The proposer must promote the new slash and say why they think it's important to the industry.

Shaughnessy: Are slash sheets primarily meant for purchasing?

Yes, they were designed to be used by the purchasing department and customer service to make it easy for them to order back and forth.

Shaughnessy: It sounds like the way forward lies in creating designer-specific guidelines for materials, not amending the slash sheets.

I don't think that the group would allow putting on each spec sheet what the target application area might be. Although in reverse, if we had a design guideline, we would say, "For 5G routers, these are the slash sheets that are most likely utilized". If you are trying for high CAF resistance for the automotive application, you could list those spec sheets that would be optimum for those.

Barry Matties: Regarding the listings and spec sheets, that would include the material supplier as well?

Yes, you could have it that way. But IPC as a rule doesn't single out any suppliers' names. I also think that is going too far.

I might put out a survey from IPC Works that says, "We would like to have a design guide to help designers use 4101 materials. Do you think this is a good idea? Would you be willing to participate?" Based on that, IPC can create another task group for a design guide. I mean, I am not so old that I just want to see the status quo until I fall over. I'm still interested in new ideas and new places.

We need some designers to lend their knowledge to this, maybe from different areas like automotive designers, telecom designers, and aerospace designers. Denny Fritz writes a chapter about design in his book, and he asked me to review the chapters on base materials. Gary Ferrari's done the same, and some others as well. We just need to capture that knowledge and filter it into a typical IPC document.

Shaughnessy: Doug, some of our readers specify the material but others go to their fabricator and say, "What do you think we ought to use?" What's the best method?

> **If you're buying high-speed, high-frequency materials it is driven by the OEM.**

It's pretty straightforward. If you're buying high-speed, high-frequency materials, it is driven by the OEM. In the high-frequency end of it, there's no interchange of materials. Each OEM has to test the material to make sure that it works in their application, then they'll specifically tell the board shop they have to buy AGC A, B, C. It is never left up to the specification sheet. The companies that sell these high-speed and high-frequency materials go directly to the OEMs, skipping everyone in the middle, and then the fabricator is forced to buy the exact product that's on that list with no modifications.

At the lower end of the spectrum, the fabricator will tell the OEM, "This is my favorite. This is the design you have, and this is the product, the prepreg and laminate product that I like the best." So, they put that on the list. In the marketplace right now, there's more buying and selling based on exact specifications from the suppliers as opposed to a general specification sheet off the back of 4101 or 4103.

In mid-range and down, the OEM asks the board shop, "In this range, what's your favorite?" Sometimes they'll say, "You told us we had to build it with this one, but this one will work, and we know how to use it. Will you test it for us?" That's how it works with some of the more medium-range materials, but the high-end stuff is all specified by the OEM down this flow, right down to the board shop.

Shaughnessy: I'm looking at 4101/126, and there's a lot of information about the material. Do the designers need more than that?

Yes. The information starts in the header of the slash sheet with a description of the chemistry and the type of reinforcement, maybe glass, glass paper, or Kevlar. It describes whether it's brominated flame-retardant or halogen-free, which is important for the designer. If they have different rules, they can see that it says bromine, or whether it's done with bromine. Some are concerned about fillers, and the glass transition range, and those are in the header as well. Items are then listed in this order: peel strength, volume resistivity, surface resistivity, moisture absorption, dielectric breakdown, Dk, Df, flex strength, flex strength at elevated temperature, and the last one is arc resistance.

Now, maybe we could say that, for example, 134, 126, 150, and 130 are all great for making low-end servers and list those in our design guide. We could do the same for aerospace, automotive, or whatever the final application is.

Happy Holden: When you're focusing on what the designer needs, the history of IPC and the standards were set up for the OEM, fabricator, and the supplier. We've been weaker on the design area, so it would be good to set up a design guide to simplify the designing, especially with so many new people amidst a broader market. There's a reason we're talking about automotive: Because of electric vehicles, we're going to see 80 to 100 million additional circuit boards fabricated every year.

Our industry increases in complexity as it grows. It's difficult to do an excellent job of design because documents that apply to aerospace and high speed may not apply to automotive, which emphasizes more high reliability, high current, and high voltage.

Matties: Where should the designer turn their attention in terms of material selection? What are the most important metrics, Doug?

In today's world, everybody looks at Df at 10 GHz. If I'm making a decision tree and I'm designing something new, it usually has to do with low-frequency antennas and different things. That seems to be what I hear most. Dk goes along with that. It falls over to some of the reliability factors like the Z-axis expansion and Td, the decomposition temperature, and then spills over to glass transition temperature.

But you could almost put together a decision tree based on what we see today with low-Df materials. It starts with Dks and Dfs, then goes to the thermal reliability characteristics, which include Tg, followed by things like peel strength and CAF resistance. Those seem to be the key factors.

Matties: Doug, we thank you so much.

Thank you. **DESIGN007**

The Journey to IPC-1791 Validation

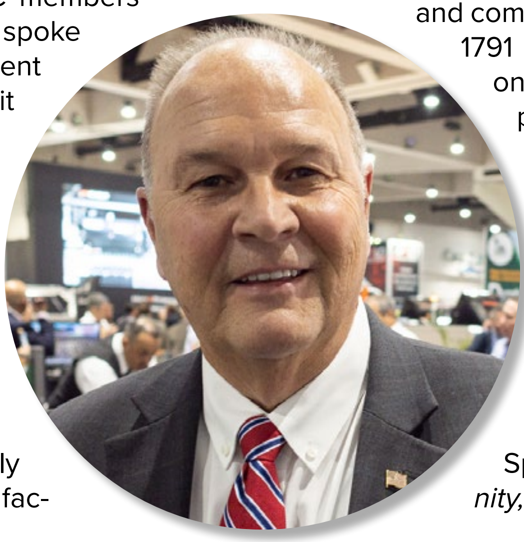
How does a company protect its most valuable electronics manufacturing information? How can designs and processes be kept safe? IPC-1791 is an industry-driven and industry-written standard that focuses on protecting two things: controlled unclassified information (CUI) and controlled technical information (CTI)—the information that would be devastating for a company to lose.

IPC Validation Services plays a critical role in ensuring that you can keep your information safe. This is the team that performs the Qualified Manufacturer List (QML) audits, validating the manufacturing process to the four pillars: quality, supply chain risk management, security, and chain of custody.

To learn more about how IPC members participate in the process, we spoke with John Vaughan, vice president of strategic markets at Summit Interconnect, who provides insight into his company's IPC Validation Services journey. If you're working with defense primes, he says, this certification is vital.

John, please tell us a little about Summit Interconnect.

Summit is the largest privately held printed circuit board manufacturer in North America, featuring eight highly integrated facilities, over one-half million square feet of advanced technology processing capability, and approximately 1,300 employees.

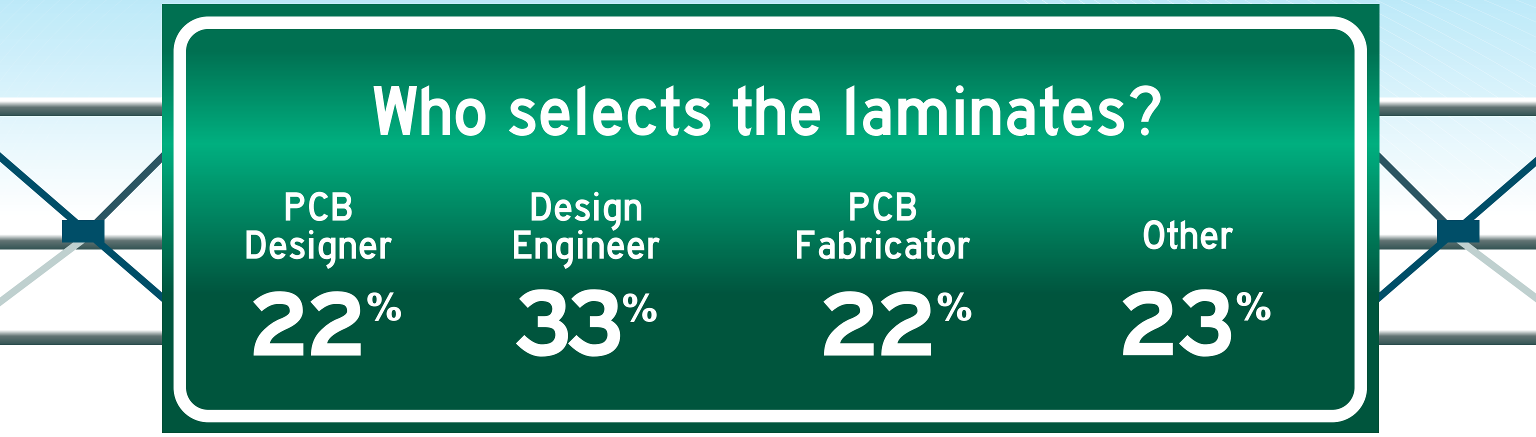


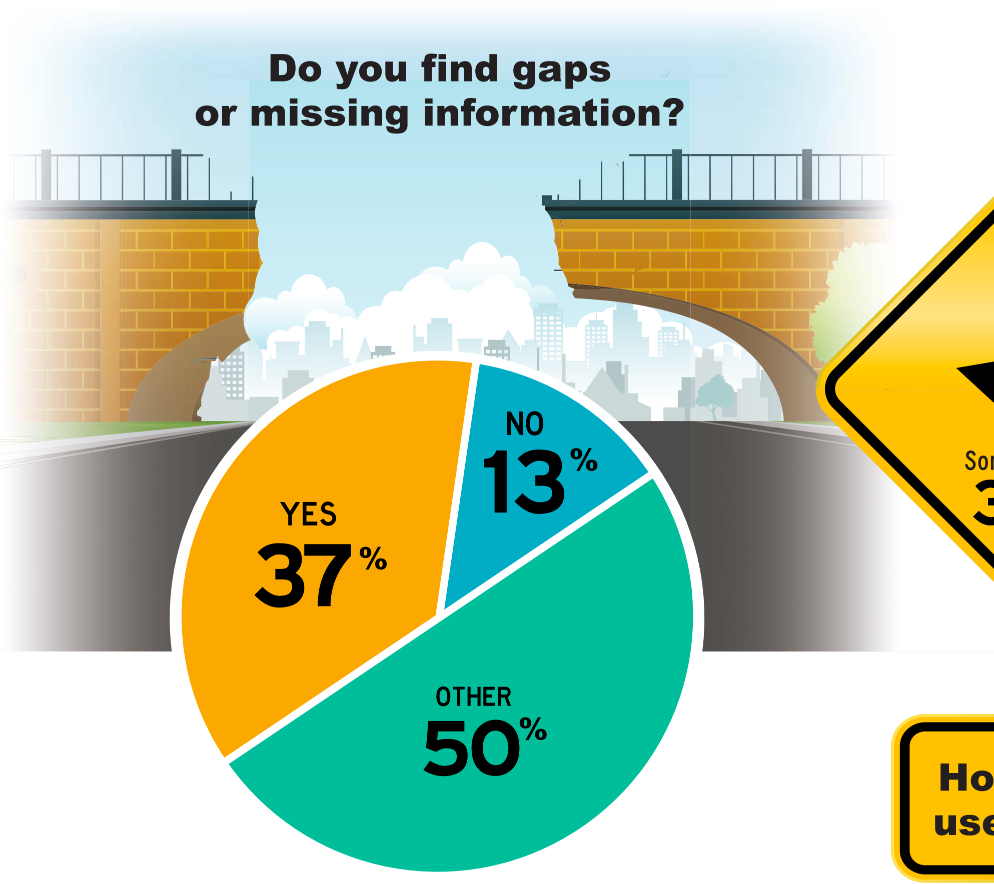
Why did Summit Interconnect decide to certify to IPC-1791?

We operate in very compliance- and certification-driven markets, and we support a heavily DoD and military prime customer set. Our customers have very high expectations in terms of Summit protecting controlled unclassified information (CUI), supply chain risk management (SCRM), chain of custody (CHoC), quality systems (AS 9100), ITAR/EAR, and compliance to NIST 800-171. The IPC-1791 audit and standard is focused on compliance to all these and our position on the IPC-1791 Qualified Manufacturers List (QML) as a Trusted Fabricator gives our customers third-party assurance through the IPC Validation Services that Summit meets specific criteria that are important to them.

To read the rest of this interview, which appeared in the Spring 2023 issue of *IPC Community*, click here.

The road to IPC slash sheets and material selection





Source: I-Connect007 survey

Select Dielectric Material With Precision

Beyond Design

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

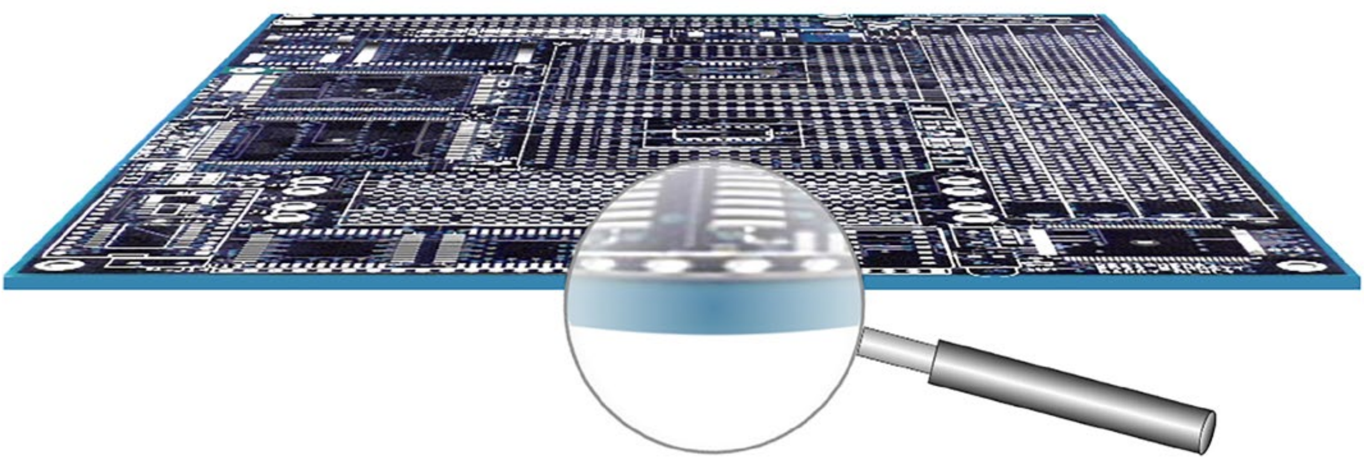
In the past, selecting a dielectric material for PCB fabrication was a no-brainer because we all just used FR-4. Clock frequencies were low and signal rise times were slow, so substrate performance was not an issue. However, in today's multi-gigabit designs, with their extremely fast rise times and tight timing margins, precise material selection is crucial to the performance of the product. This puts the materials selection process under tighter scrutiny. Materials used for the fabrication of multilayer PCBs absorb high frequencies and reduce edge rates, which is a major cause of signal integrity issues. But we're not all designing cutting-edge boards. Sometimes we tend to over-specify requirements, which can lead to inflated production costs.

Over the years, a huge range of materials has been developed for multilayer PCB fabrication. To give you an idea, iCD now has a choice in its dielectric materials library of more than 700 series of dielectric rigid/flex materials from more than 60 different manufacturers. When each material is used for the right target application, the resultant PCB will have the lowest possible cost, yet still satisfy the design and performance goals of the project.

Electromagnetic energy propagates in a vacuum or in air at the speed of light. But, as the field is enveloped in a dielectric material in the PCB media, it slows down. The signal trace in a PCB simply guides the wave as the electromagnetic energy propagates in the surrounding dielectric material. The signal rides on this carrier wave. It is the dielectric material that determines the velocity (v) of propagation of the electromagnetic energy:

$$v = \frac{c}{\sqrt{Dk}}$$

c is the speed of light (in free space) and Dk is the dielectric constant of the material (FR-4 is ~4.0). By contrast, the Dk of air is 1. Therefore, the velocity of propagation in FR-4 is about half



the speed of light, or 6 inches per nanosecond.

The electrical properties of a dielectric material can be described by two terms:

1. The dielectric constant (Dk) or relative permittivity (Er) is the ratio of the amount of electromagnetic energy stored in a material by an applied voltage. It describes how the material increases the capacitance and decreases the velocity of propagation in the material.
2. The dissipation factor (Df) or dielectric loss/loss tangent (tan δ) is a parameter of a dielectric material that quantifies its inherent dissipation of electromagnetic energy.

Dielectric constant and dielectric loss are not a function of the geometry of the transmission line. Rather, they are a function of the dielectric material in which the signal propagates their distribution in the PCB stackup and the applied frequency. These mechanisms contribute to the frequency-dependent loss and to degrade the speed of the signal. The signal quality transmitted through the media and picked up at the receiver will be affected by any impedance discontinuities and losses of dielectric materials. The glass epoxy material (FR-4) commonly used for PCBs has a negligible loss for digital applications below 1 GHz. But at higher frequencies, the loss is of greater concern.

If the signal has a fast rise/fall time, then the electromagnetic wave needs to propagate at a higher speed, and therefore the Dk needs to be low to enable this. If a material with a high dielectric constant is placed in an electric field, the magnitude of that field will be measurably reduced within the volume of the dielectric. Therefore, a lower Dk is desirable for high-speed design. Conversely, a high Dk material is very good at condensing electric fields, so having it between the planes increases planar capacitance.

An efficient dielectric material supports a varying charge with minimal dissipation of energy in the form of heat. There are two main forms of loss that may dissipate energy within a dielectric:

1. Conduction loss is the flow of charge through the material that causes energy dissipation.
2. Dielectric loss is the dissipation of energy through the movement of charges in an alternating electromagnetic field as polarization switches direction.

Dielectric loss is especially high around the resonant frequencies of the polarization mechanisms as the polarization lags behind the applied field, causing an interaction between the field and the dielectric's polarization that results in heating (Figure 1).

There are also thermal factors to consider. The most important is the glass transition temperature (Tg), which is the point at which a glassy solid changes to an amorphous resin/epoxy. If the reflow temperature exceeds the Tg for an extended period, the material rapidly expands in the Z-axis. Plus, mechanical material properties degrade rapidly—strength and

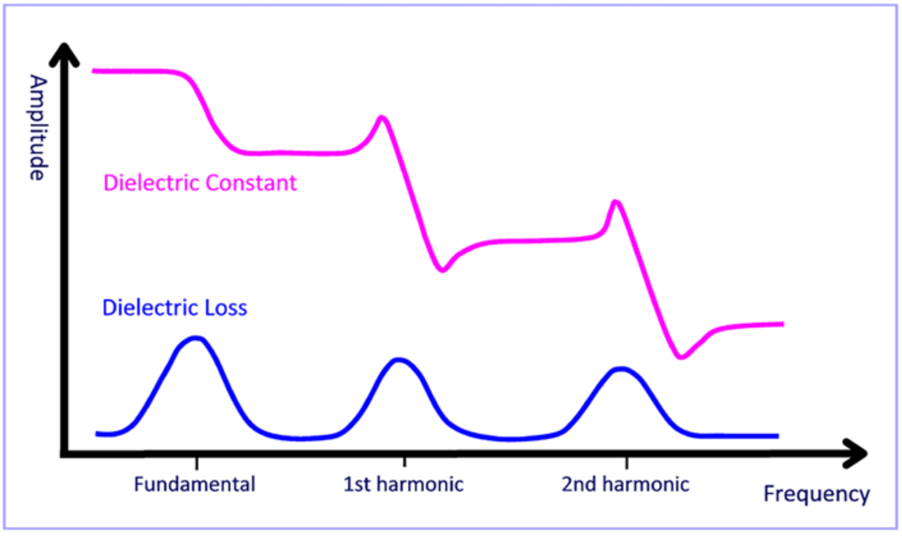


Figure 1: Dielectric constant and dielectric loss vs. resonant frequency.

Table 1: Loss profile ranges

Profile	Bit Rate (Gbps)	Frequency (GHz)	Dissipation Factor (Df)
Ultra Low Loss	≥ 50	≥ 25	≤ 0.005
Low Loss	25 - 50	12.5 - 25	0.005 - 0.010
Mid Loss	10 - 25	5 - 12.5	0.010 - 0.015
Standard Loss	2 - 10	1 - 5	0.015 - 0.02
Basic FR-4	≤ 2	≤ 1	≥ 0.02

bonds in the material. A high Tg guards against barrel cracking and pad fracture during reflow. Standard FR-4 has a Tg of 135-170˚C, whereas the high-speed materials are generally well over 200˚C.

Decomposition temperature (Td) is the temperature at which the material chemically decomposes. This is the maximum limit or the point of no return. Most materials have a Td of 320˚C, so it is not an issue. The coefficient of thermal expansion (CTE) in the Z direction is the rate of expansion as the material heats up. The CTE should be as low as possible (<70 ppm). There are many other properties of materials to consider, but the material manufacturers know the right combination for high-speed design. Fortunately, a material with a low Dk has a low Df, a high Tg, and a high Td, which is exactly what is needed for high-speed design. So, considering the Dk and Df alone can get you in the ballpark for the right material for your design.

With so many materials to choose from, which is the best for your specific product? Low cost generally means low quality. But the price of poor yields drives up the final material cost. Dielectric material selection is usually driven by the frequency and rise time of the digital signal, with lower values of loss most suitable for high-frequency applications. These materials generally exhibit lower values of dielectric constant, resulting in faster signal propagation. Table 1 shows the loss profile ranges of dielectric materials.

Figure 2 depicts the profile for dielectric materials with a Df < 0.005. The iCD materials

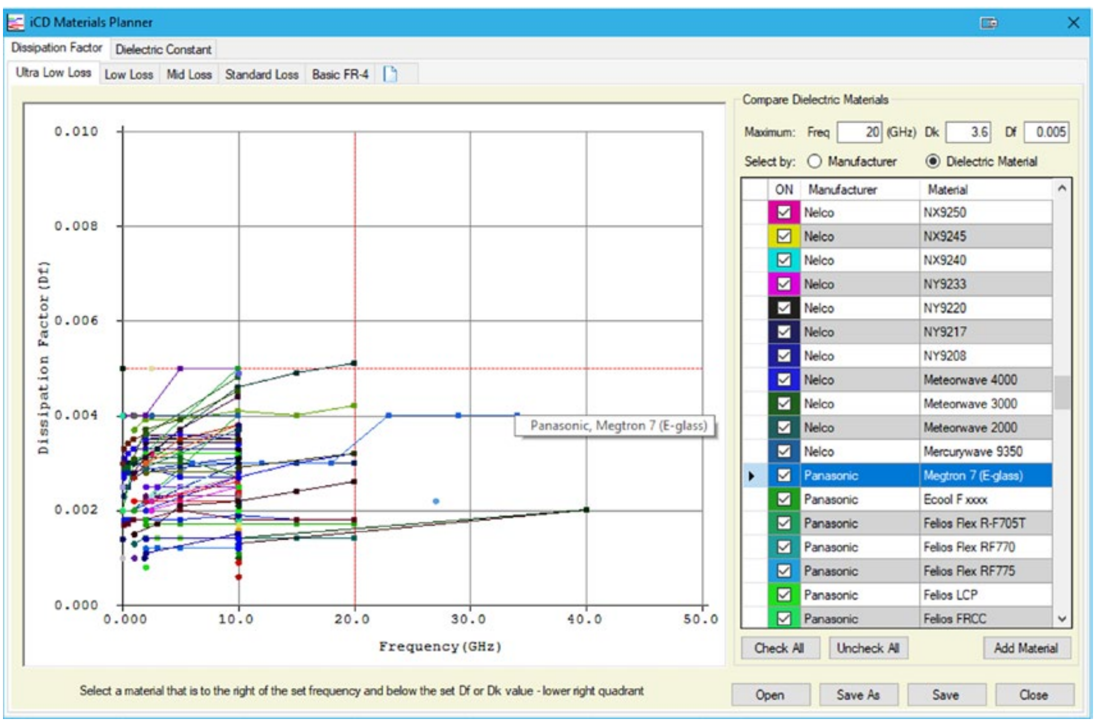


Figure 2: Loss profile for ultra-low loss dielectric materials. (Source: iCD Materials Planner)

planner has five default profiles ranging from basic FR-4 to ultra-low loss materials, as in Table 1. This enables the designer to compare dielectric materials based on manufacturer, fabricator, frequency, dissipation factor (loss), and dielectric constant.

Another issue is that materials available locally (vs. offshore) can vary from fabricator to fabricator. Typically, prototype boards are fabricated locally, whereas those made in Asia are a more economical option for mass production. Profiles of PCB fabricators can be set up to display the complete range of materials each fab shop stocks, enabling comparison among shops.

Figure 3 plots the loss properties of dielectric materials from an Asian fabricator. One can easily see which materials are best for high-speed applications and can choose among a few materials that are in stock. Cost-to-performance evaluations must still be done to ensure that the lowest-cost material is selected to do the job. Also, keep in mind that material costs vary with quantity.

Matching material performance numbers of the dielectric constant is also important. A small difference in the value between materials can significantly impact impedance, line widths/clearances, and thus losses. Also, the dielectric constant of a material determines the propagation velocity of the signal in the medium. So, if Dk values vary on different layers of the substrate, then bus signal timing may also become an issue. One should consider construction options that allow a drop-in material that matches the impedance, and Dk, for each layer of the stackup.

PCB designers need to quickly choose the best, most cost-effective material for their application from among the vast array of choices available. Sorting through numerous slash sheets and datasheets is very time-con-

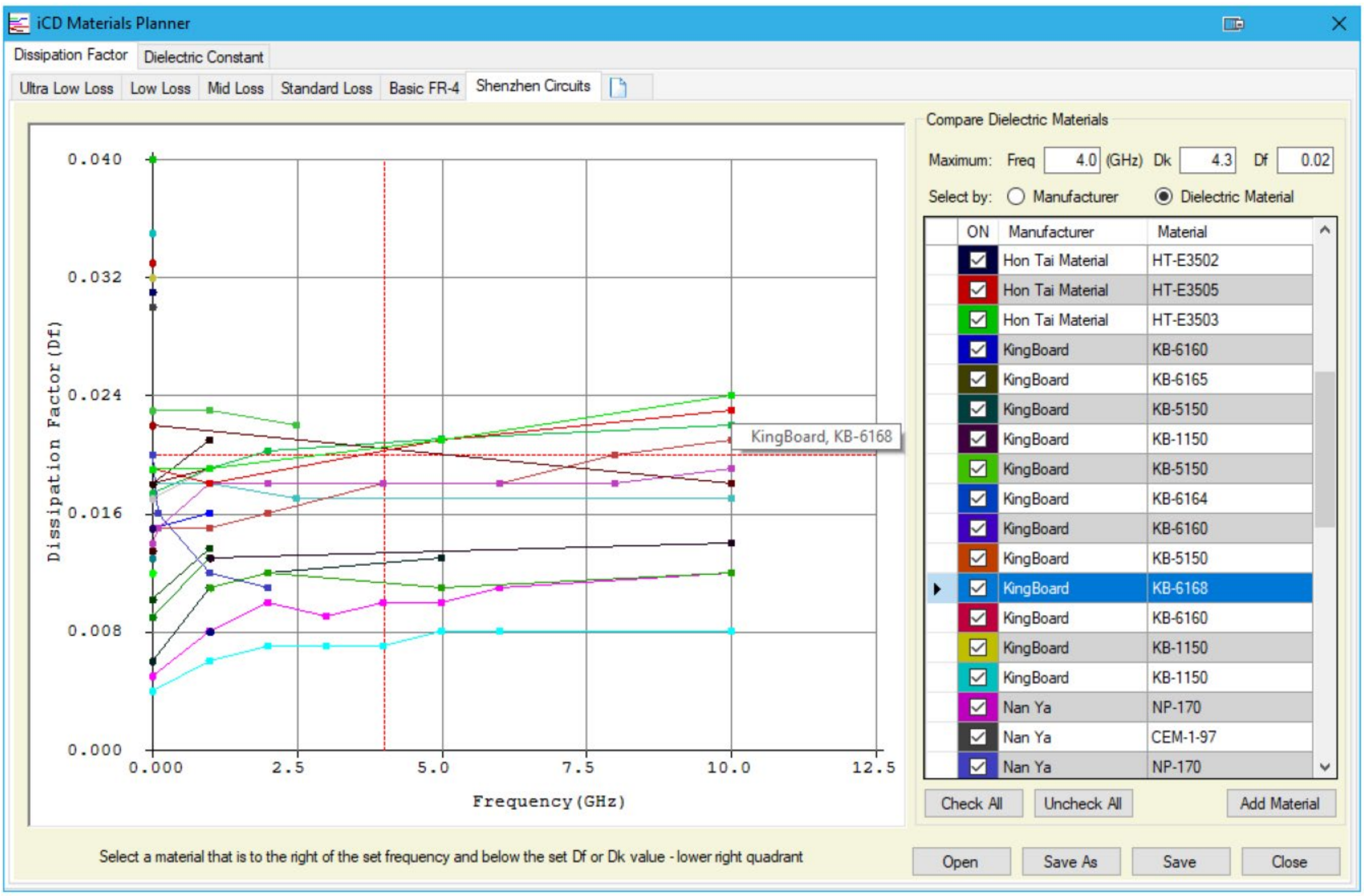


Figure 3: Example of a fabricator's dielectric materials loss profile. (Source: iCD Materials Planner)

suming. A direct visual comparison of dielectric materials—based not only on the manufacturer's product lines, but, more importantly, on one's preferred fabricator's stock—is undoubtedly the most efficient approach for material selection.

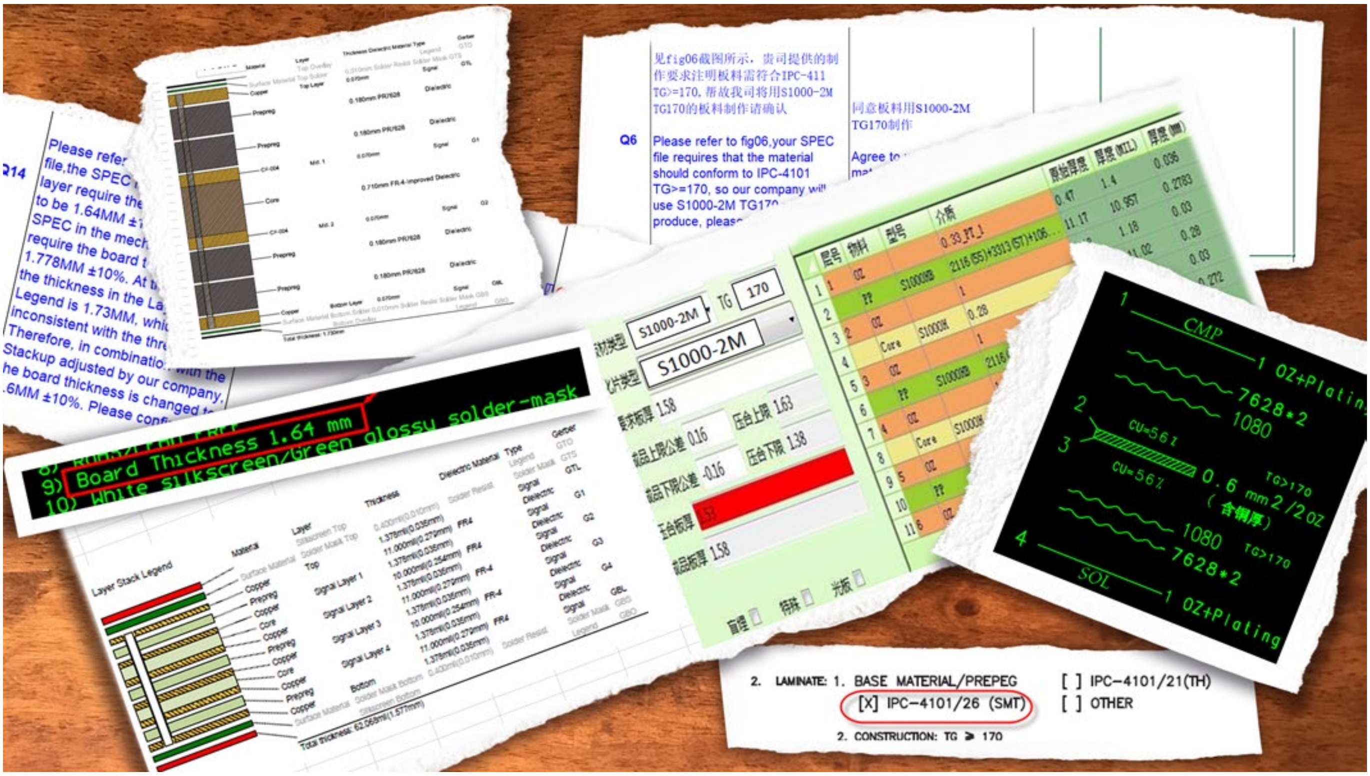
Key Points

- Materials used for the fabrication of multilayer PCBs absorb high frequencies and reduce edge rates, which is a major cause of signal integrity issues.
- When each material is used for the right target application, the resultant PCB will have the lowest possible cost, yet still satisfy the design and performance goals of the project.
- Electromagnetic energy propagates in a vacuum or in air at the speed of light. But, as the field is enveloped in a dielectric material in the PCB media, it slows down. FR-4 is about half the speed of light, or 6 inches per nanosecond.
- The glass epoxy material (FR-4) commonly used for PCBs has a negligible loss for digital applications below 1 GHz. But at higher frequencies, the loss is of greater concern.
- A lower Dk material is desirable for high-speed design. Conversely, a high Dk material is very good at condensing electric fields, so having it between the planes increases planar capacitance.
- The dielectric constant (Dk) is the amount of electromagnetic energy stored in a material by an applied voltage.
- The dissipation factor (Df) quantifies a material's inherent dissipation of electromagnetic energy.
- Conduction loss is the flow of charge through the material that causes energy dissipation.
- Dielectric loss is the dissipation of energy through the movement of charges in an alternating electromagnetic field as polarization switches direction.
- Dielectric loss is especially high around the resonant frequencies, causing an interaction between the field and the dielectric's polarization that results in heating.
- A material with a low Dk has a low Df, a high Tg, and a high Td, which is required for high-speed design.
- Matching material performance numbers of the dielectric constant is important. A small difference in the value between materials can impact impedance, line widths/clearances, and thus losses, significantly.
- The Dk determines the propagation velocity of the signal in the medium. If Dk values vary on different layers of the substrate, then bus signal timing may also be an issue. **DESIGN007**

Resources

Beyond Design columns by Barry Olney:
- Material Selection for SERDES Design
- Material Selection for Digital Design
- It's a Material World

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, click here.



Stop Over-specifying Your Materials

Feature Interview by the I-Connect007 Editorial Team

Columnist Kelly Dack has had a pretty wide range of experiences. As a PCB designer, he has sat behind the desk at an NPI company, an OEM, a fabricator, and now an EMS provider. We asked him to share a few thoughts on the materials selection process and how it could be improved.

Kelly also explains how overly zealous PCB designers make things too complicated by over-specifying their materials, which leads to confusion once the board goes to volume production overseas. Are you over-constraining your material choices?

Andy Shaughnessy: Kelly, what is your process for material selection? Walk us through it.

Sure. I actually wrote a guideline for our customers that explains the material selection process. As mentioned, over-specification in the EMS realm is rampant and problematic from the standpoint of scaling products to volume overseas. This guide has a section on laminated materials that includes a simple, tried-and-true material specification. It says, "Materials: laminated glass epoxy resin type FR-4 series or equivalent per IPC-4101 with a Tg of greater than or equal to xxx." This is a number that we can modify. We say, for instance, 170°C, and a Td (time to delamination) temperature of greater than 3XX°C. Those are all movable numbers that designers can edit.

That's how we specify our laminate materials for printed circuit boards, unless the performance criteria dictates that it needs to go further and get more specific—for example, high-performance signal integrity constraints, impedance control, or exotic materials. But

otherwise, FR-4 laminates cover 85–95% of our customers' design requirements.

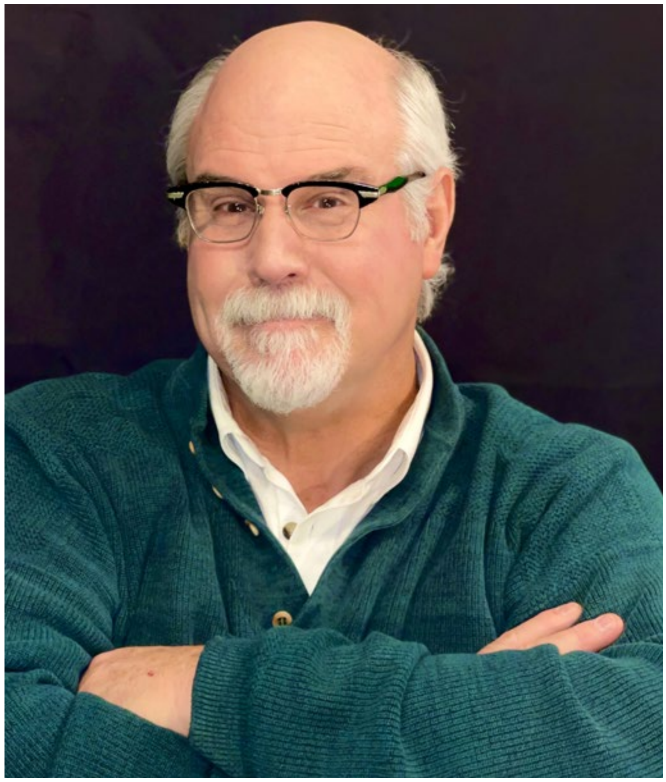
Shaughnessy: Where do designers typically go to find this information? What documents or guidelines should they use?

Many designers use their company's documentation template or go to their elders and learn through knowledge that has been passed down. I just went through a bunch of our customers' designs and fabrication drawings, and I found plenty of examples of customer material specification. Many of them call out a specific IPC-4101 slash number, and it's usually the same slash number each time. It's either 4101/26 or /21.

Shaughnessy: IPC has said that slash sheets were not ever meant for designers to use; they're mainly a way for PCB supplier purchasing and marketing departments to communicate with buyers.

I'm glad to hear that confirmation as it was my understanding as well. This all came from the MIL-S-13949 spec back in the mil-spec days, most of which has been replaced by IPC standards. But we are seeing some of our customers' designs specifying laminates by using slash numbers and I get the feeling it's just because of tribal knowledge. If you look at most of the designs, they appear to be basic, not really requiring a specific material. The board would work fine with a generic glass-epoxy laminate because it has no impedance control or performance criteria. We just print and etch some copper onto it, create a circuit, and it moves electrons.

It's usually only when we get into the super high-speed design in the gigahertz range where we have to start thinking about loss tangents and permittivities. Here's the challenge: How much do you constrain? If you're building your PCB design down at the local prototype shop, which will build a board any way you want, everybody seems fat and happy. But when you want to build 1 million boards, you must introduce a low-cost constraint because you can't pay $1,000 a board and make a profit. You scale to volume to get cost savings. To realize maximum cost savings, this has always been done offshore. But designers must realize that offshore suppliers don't have access to all the materials and capabilities that U.S. prototype companies have. Over-specification of laminate materials by composition, performance characteristics, or a trademarked source creates a terrible, but avoidable ordering situation, putting quotes on hold every single day. The simple fact is that it causes our offshore PCB suppliers to request and obtain approval for material substitutions before they can proceed.


Kelly Dack

Shaughnessy: How can our PCB designers re-think material specification for low-cost volume production?

I like to use a hamburger analogy. If I go to In-N-Out Burger, I know within a small margin of error how my hamburger will taste, and how many calories it has. I don't know what's in the bun, the burger, or the magic sauce, but I know it's really good.

I don't care, because at the end of the day I put the burger in my food hole, I feel good, and everything is fine. In a simple sense, it's the same with circuit board material specification. I'm not talking about high-performance, signal integrity type of boards, but general circuit boards. As I said, about 85% of what we do is an assembly-line hamburger, and it just needs to taste like a hamburger at the end of the day. Yes, the designer should specify the material, but they shouldn't over-specify it. Over-specification—over-constraint—is the problem. Most boards we see could surely work without specifying a slash sheet. When we go offshore, it's the slash numbers and brand name or trademark indicators that tend to stop the procurement process.

Happy Holden: Ideally, the material selection process should be a conversation with the design engineer, designer, assembler, and the fabricator. That's ideal. I've always cautioned customers not to put this decision in the lap of the EMS provider. The EMS company is only concerned with two surfaces, and they don't know a lot about the interior or the stackup.

That communication would be ideal, but the reality is, many PCB designers are not in touch with the fabricator—prototype or offshore. They may not even know who will fabricate the prototypes. PCB designers are so far removed from the fabrication and assembly stakeholders who build their products that there's almost no way to execute solid DFM processes. Glass transition temperature rating (Tg) specification is a good example of this. PCB designers have no business specifying Tg if they're trying to save money.

Many of our customers have standard notes, and I used to do this. I wanted the best quality board, and I always specified Tg 170°C or Tg 180°C, which is a higher temp performance range for laminate material. I did that every day, all week long, not realizing that it was extremely costly in volume. We had the other stakeholders on our projects and in our corporation who wanted to make money, not spend it.

After working for a bare board supplier and now an EMS supplier, I've found that there are other stakeholders involved who plan the assembly of the board. They know how to effectively modify the material's Tg and aim it toward the number of thermal excursions that the board will experience—and which the designer has no idea about.

For instance, a multilayer board design has surface mount on one side, no surface mount on the other side, and a couple of through-hole parts. How many thermal excursions will this board go through? It may go one pass on the top side through the oven, and then one pass down the wave solder line for the through-hole ports. That's two thermal excursions. If the volume assembly supplier were given an opportunity to change the spec to Tg 150°C, there would be an opportunity to realize a cost savings here. But if I've specified 180°C Tg as the standard, the EMS provider must get it quoted that way.

Shaughnessy: Is there one document or guideline that contains these metrics? You might go to laminate suppliers' datasheets or somewhere else to get the information, but can you trust it?

You must keep it simple from a designer standpoint, because the biggest culprit working against driving your board to success in volume is over-specification. If we over-specify, we might be spending dollars unnecessarily. If we partner with suppliers, we build a relationship of quality, delivery, and low cost. If we build trust with our suppliers, they become part of our stakeholder community and we can rely on them to do the right thing—to build the hamburger the right way and use the right materials to build the hamburger so it gives everybody what they want.

There are a lot of variables in circuit board manufacturing. We have this long-time philosophy about dimensioning, tolerancing, and specification of any part. You specify what you want to end up with, or for the end-type performance, but you don't tell anybody how to do it, or too much about how to make it right. If we can be as general as possible and allow our stakeholders to have a say and dial in the materials to achieve the performance specification that we've outlined, we'll be set up for flexible cost savings.

Shaughnessy: I like that; let the stakeholders have a say.

Holden: Unfortunately, when you select components, there's no information about the manufacturing repercussions of selecting this component. The design engineer selects components. If he starts mixing technologies—through-hole, surface mount, and bottom-leaded components, for instance—the mixture makes it really difficult to say, "Do this and don't do that," because of the permutations and combinations. That's why it's best to communicate early on with the manufacturer.

Exactly. If we define our stackups with what I call the recipe approach, the fabricator must put together stackup ingredients like a recipe. They will need to put together core layers and prepreg the way the PCB designer shows it in the stackup recipe when it might not be appropriate at all. They should untie the supplier's hands to be able to use prepregs or whatever they need to meet the performance and thickness requirements of the board.

There are a lot of variables in circuit board manufacturing. We have this long-time philosophy about dimensioning, tolerancing, and specification of any part.

Holden: Many don't appreciate that quick-turn prototype companies make their money off their knowledge; they can build what you specify in record time. They're the Burger Kings of the industry: Have it your way. They'll build it any way you want.

That does not necessarily mean that the board is suitable for high-volume production. Designers who go offshore spend much more time talking to suppliers and assemblers. If there's some secret that makes the prototypes work, that had better be incorporated somehow into the final specifications.

Knowing the terms and the anatomy of a slash sheet is a good thing, but specifying it on a drawing is another. We must know how to specify, to know the language of the industry for appropriate, effective specification.

Shaughnessy: Some designers have said that they'd love some sort of material guidelines that are final application-sortable, where you'd say, "Okay, I'm working in automotive," then it would give you a list of each type of material that fits under that. Would a one-stop shop approach be helpful?

Yes, in one sense, but dangerous from the standpoint of taking that product out of prototype and launching it into production. "Here's the slash sheet, it has specs here, and that's all there is to it." What has been suggested is that these slash sheets were never intended for designers. That's a good thing to hang our hat on.

We do a lot of daisy-chaining. We say, "Inspect our acceptability requirements for IPC-A-600," and that says the supplier has to build and inspect it to IPC-A-600. That one sentence gets us the specification that covers everything.

It's the same with assembly. We've got the yellow book—IPC-A-610—which covers the acceptability requirements for all the assemblies. It's full of color pages of the target conditions (my column name), acceptable criteria, and nonconforming criteria for each class. The project that you're suggesting might be a daunting project to specify, or be an index for materials used in design because now you're getting down into the granular composition of the materials. Every supplier will be fighting against the competition and trademarking, so they will tweak their composition just slightly so they can get around the rules.

Shaughnessy: It sounds like you're saying that the current methods for specifying materials work pretty well, and if designers get too granular with their metrics, they can wind up over-constraining the board and raising the cost when they don't need to.

Yes. I can show you. The two big over-constraining areas on bare board PCB fabrication specification are materials, and dimensioning and tolerancing. Design vs. manufacturing is so dynamic and can be subjective; sometimes you have to rob Peter to pay Paul.

The two big over-constraining areas on bare board PCB fabrication specification are materials, and dimensioning and tolerancing.

Holden: The problem in materials is that the suppliers selling materials have a reason to make specifications and information available. When we talk about design, we open a can of worms. What market is the material designed for? What is the performance, reliability, lifecycle, and cost? Those trade-offs would be very difficult to document.

The designers who know the most don't have time to sit down and write it out, whereas the suppliers have that incentive because it helps them sell. A designer is not selling anything. If the designer doesn't care about the board, he can use any 130°C Tg material. They need to know where to care.

Happy, I like that. PCB designers need to "know where to care." Here's a quick story: I got a circuit board from a customer and it had gone through years of evolution. It was a very simple circuit board, and it had a carbon-wound through-hole resistor on one side. It had a surface mount part on the bottom side, and another surface mount part on the top side. From a thermal excursion standpoint, they had to run it through one pass from the top side, one pass from the bottom side, and then it had to get hit with the leads of the resistor. For three thermal excursion passes, we typically recommend that we go to the higher-temp materials. We also do cost-down efforts; we'll make recommendations through the DFM studies that we do.

I said, "How much are you paying for that axial lead resistor?" It turned out to be $3 because they're just not as available as the surface mount parts. We recommended switching it to a surface mount part, putting it on top, taking the other surface mount part on the bottom, and putting it on top. We got all three of those part types on the single board surface, one pass through the oven, lowered the material requirement to 130°C Tg, and saved a ton of money on a board that's being made in the millions.

Shaughnessy: We're seeing more young designers now. What advice would you give someone who's looking at these different sources of information about specifying and selecting material?

I used to tell them to visit their board fabrication shop, but I've readjusted. You need to visit a volume EMS provider who is processing boards down conveyor belts, because you can learn so much, not just about slash sheets and materials, but you'll know what the requirements are. You'll find out what the requirements are for panelization, flexi boards, and un-panelizable boards.

Every designer should visit the bare board manufacturing stakeholder—EMS, the assembler, and the bare board supplier—and then become involved in IPC education. This helps new designers tie it all together.

Shaughnessy: Kelly, any closing comments?

With the hundreds of designs I see, I suspect that at least 85% of the boards are normal multilayers that can use just about any glass epoxy FR-4 material. The rest of the designs will be high-performance boards. In other words, there are not many designs that require super specification and exotic materials.

We're talking about creating a new guideline or index for specifying materials that will really only help 10–15% of PCB designers. There are bigger fish to fry just in helping the designers who are designing the 85% of boards that are FR-4 boards, because we can't get these FR-4 boards to run down our line due to bad DFM.

Shaughnessy: Good stuff, as usual. Thanks for speaking with us, Kelly.

Any time. Designers: "Know where to care."

DESIGN007

Is It Time to Shake Up Materials Standards?

In a past issue of *Design007 Magazine,* the I-Connect007 team spoke with Ventec's COO Mark Goodwin and Technology Ambassador Alun Morgan about material standards. In this interview, they describe how they feel current standards do not sufficiently recognize the needs of end customers today with new processes and materials being shoehorned into old standards based on dated ideas of classifications, and how this makes choosing the right material challenging for designers. They suggest implementing a simpler system that is based on performance.

Barry Matties: Mark, please start with what you see as issues around the standards.

Mark Goodwin: There are two particular areas: one is very product-specific, and the other is standard-specific. I'll start with the product-specific one. We have an increasing global market for thermal management products, insulated metal substrates, thermally conductive laminates, and prepregs, and we have no industry standards for comparing test methods to allow an end user to adequately compare the thermal performance of those materials. It's the Wild West out there on those products. The other area for me is IPC-4101—the slash sheets—where there is a hang-up on resin chemistry rather than functionality; there's a whole history to that. And the world has moved so far forward, the specification has not kept up, and it needs an industry effort to reconfigure and realign that specification. Those are the starting points for me.

Alun Morgan: We consider standards from two perspectives. One is a mandatory side, so that means both fire and electrical safety, which are pretty clear and there's very little compromise. The other is performance standards or classification standards, which Mark alluded to, such as IPC-4101 or IEC series, where the intention is to define a standard or specification that gives designers a choice within a range of performance for materials; that's somewhat broken now. To read this entire conversation, which appeared in the September 2019 issue of *PCB007 Magazine,* click here.

3D Effects in Power Distribution Networks

Quiet Power

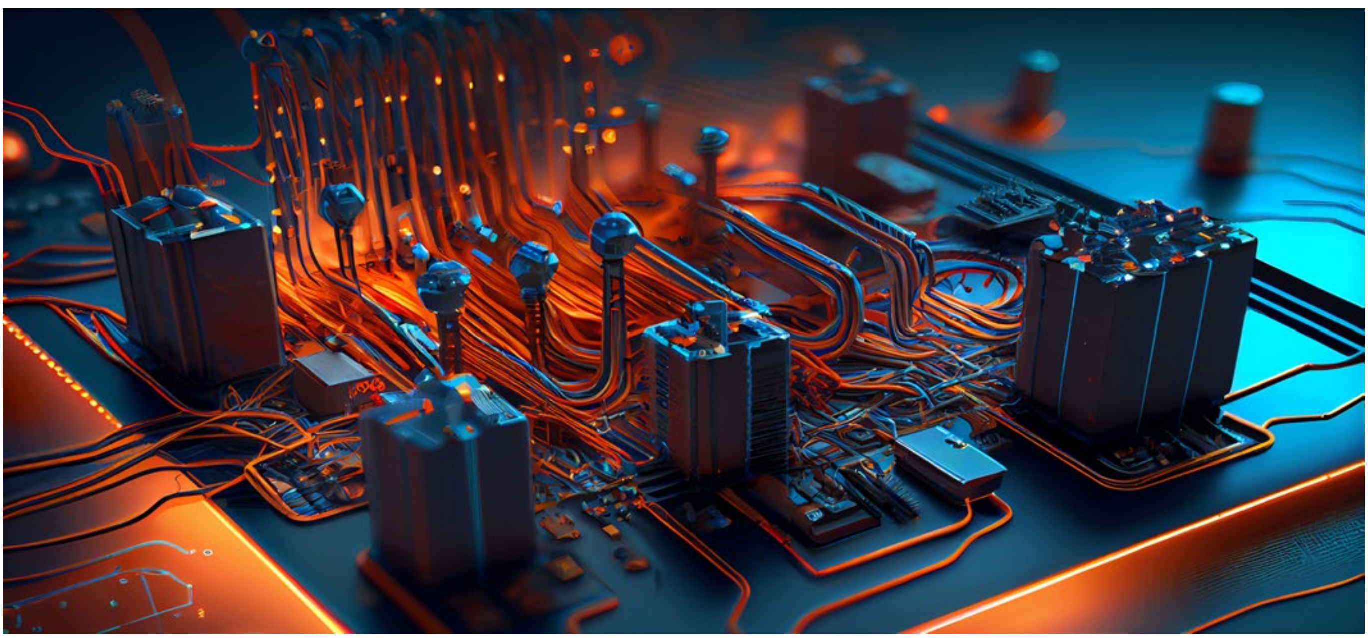
by Istvan Novak, SAMTEC

Signal integrity (SI) grew out of the electromagnetic compatibility discipline in the early 1990s and gradually became a hot topic in its own right. Although the term SI never appeared in the title, one of the first books on the topic was *Introduction to Electromagnetic Compatibility* by Clayton Paul. About a decade later, power integrity (PI) became the new hot topic, creating a separate discipline on which a multitude of books and conference sessions were based.

SI was based on high-frequency RF and microwave knowledge, whereas PI tasks were handled primarily by AC/DC and DC/DC power supply designers, who preferred the low-frequency behavior of circuits. Although the basic rules and principles of physics apply to both SI and PI, the separate high-frequency and low-frequency considerations led to seemingly disconnected design rules.

Take, for instance, three-dimensional (3D) effects, which we usually associate with high-frequency SI problems, where the full-wave solution of Maxwell equations becomes necessary. They can happen when we analyze a right-angle turn in a PCB trace or look at the high-speed behavior of a plated through-hole. In SI, the full-wave effects become more noticeable at higher frequencies, where the physical dimensions are not negligibly smaller than the wavelength.

Still, it may be surprising to learn that in PI, 3D effects can equally show up at very low frequencies, sometimes in the kHz region and even at DC. These 3D effects are created by specific patterns and changes in the current



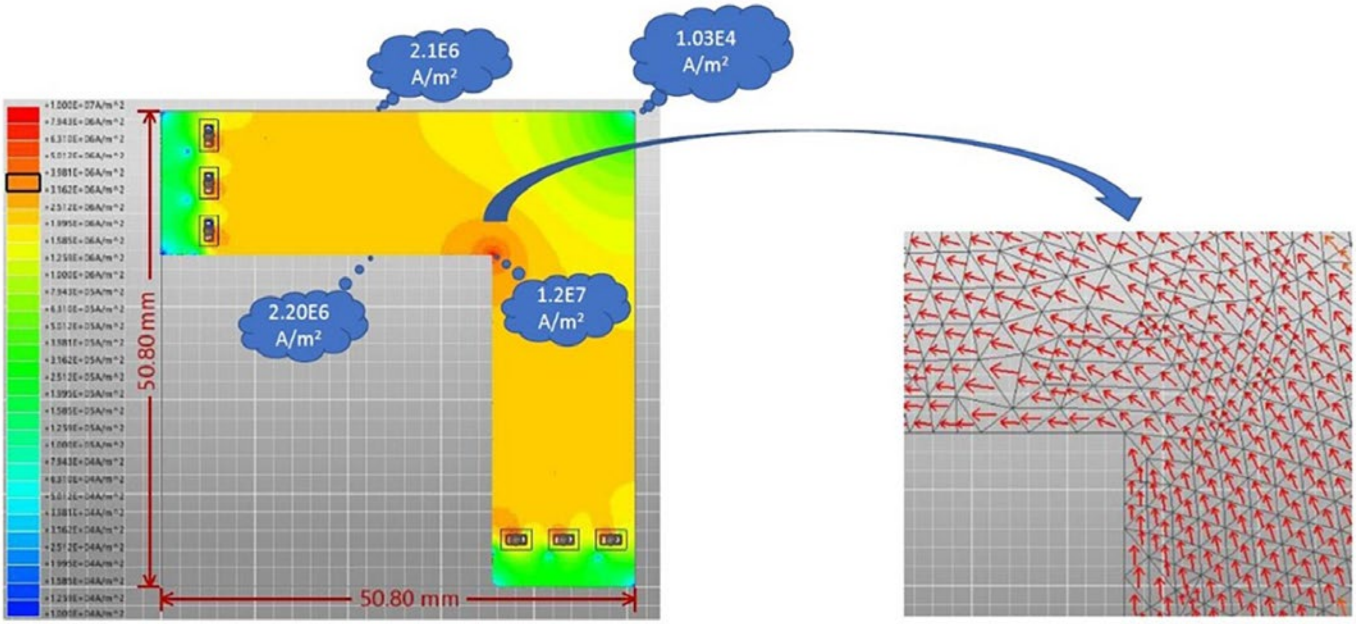
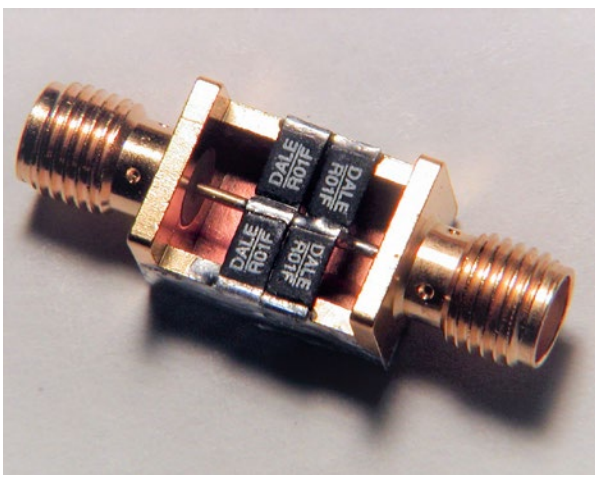


Figure 1: Simulated DC current density of a power strip making a right-angle turn[1]. There is a total of 1A through a 1-ounce, half-inch-wide copper strip.

density in conductors. As seen in Figure 1, the density of DC current at a sharp right-angle turn of a power strip will be very small at the outer point of the corner and could be very high at the inner corner.

We know that high-speed traces with a sharp bend will suffer from the capacitive loading at the outer "unused" portion of the turning trace. We see in Figure 1 that, at DC, the outer corner is equally "unused" and does not cause problems; it is just not necessary to have copper there. Instead, the problem at DC shows up on the opposite side of the power strip, at the inner corner, where the current density may exceed the safe limit.

Another example of complex 3D behavior at low frequencies was documented by measured and simulated data[2], where the conductor geometry and the component placement interacted in a way that created a negative phase and negative slope of the magnitude of impedance. In Figure 2, the circuit in question



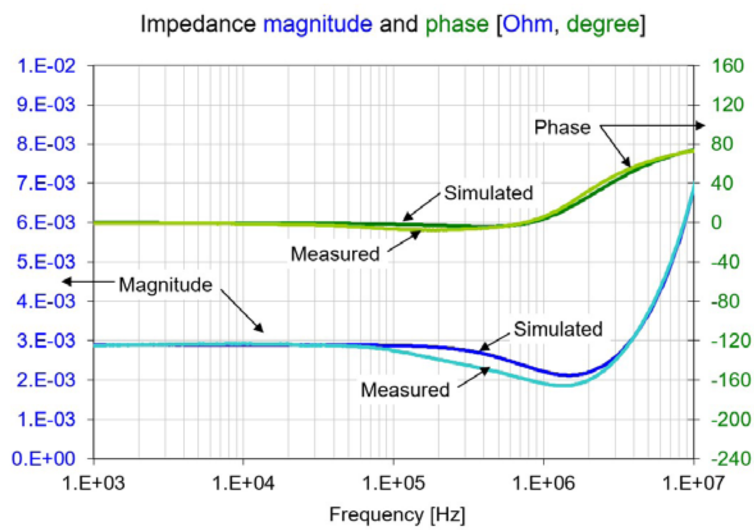


Figure 2: Small fixture with two coaxial connectors and resistors is seen on the left, while measured and simulated impedances are seen on the right[2].


Hmm, what is recommended
minimum distance for
copper to board edge?

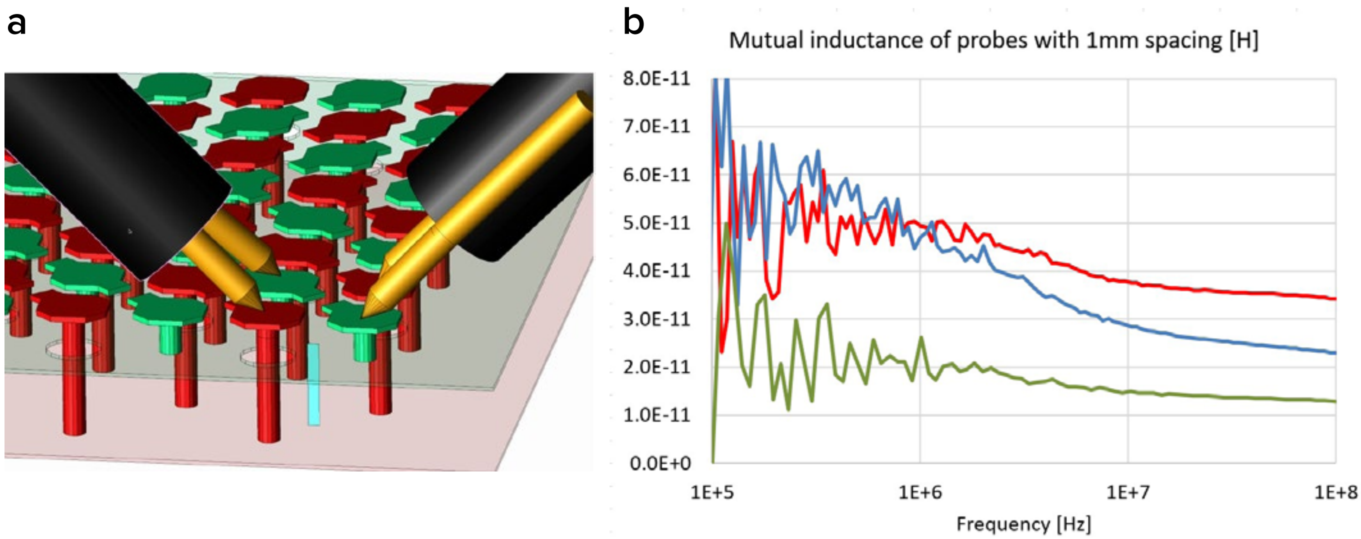


Figure 3: a) Wafer probe landing on a via-grid array; b) the mutual inductance between the probe-tip loops as a function of frequency.

is a small fixture consisting of two back-to-back SMA edge-mount coaxial connectors and four 10 milliohm surface-mount resistors.

With just metal pieces and resistors, we would expect an R-L-like impedance profile. But the frequency dependency in this case looks as if we had the series resonance of a low-Q bulk capacitor at around 1.5 MHz. However, there is no capacitor in this circuit—not one that could create that low of a series resonance. Instead, the reason for the impedance dip is the distributed nature of the resistance and inductance of the ground pegs of the SMA connectors, which interact with the multiple pieces of R-L parallel shunt elements created by the four resistors[2].

3D effects at relatively low frequencies also show up in wafer-probe PDN measurements[3]. Figure 3 shows a computer rendering of two wafer probes measuring a chip's core power supply across two adjacent power-ground via pairs in a 1-mm array. The inductive coupling between the two loops formed by the probe tips has to be removed from the measured data, either by de-embedding or by calibration. The coupling can be characterized on appropriate calibration substrates. The mutual inductance between the probe tip loops for three different probe tip geometries is shown in Figure 3b. Note that the mutual inductance has different frequency dependency at low frequencies for the three probe geometries.

Conclusion

Regardless of the frequency, 3D interactions among electrically small features noticeably impact PI simulations and measurements.

DESIGN007

References

1. "The Perils of Right-Angle Turns at DC," by Istvan Novak, gEEk spEEk, May 14, 2020.
2. "Accuracy Improvements of PDN Impedance Measurements in the Low to Middle Frequency Range," by Istvan Novak et al., DesignCon 2010, Feb. 1–4, 2010, Santa Clara, California.
3. "3D Connection Artifacts in PDN Measurements," by Istvan Novak et al., DesignCon 2023, Jan. 30–Feb. 2, 2023, Santa Clara, California.

Istvan Novak is the principal signal and power integrity engineer at Samtec with over 30 years of experience in high-speed digital, RF, and analog circuit and system design. He is a Life Fellow of the IEEE, author of two books on power integrity, and an instructor of signal and power integrity courses. He also provides a website that focuses on SI and PI techniques. To read past columns, click here.

Slash Sheets: Don't Fall Into the Trap

Feature Article by Geoffrey Hazelett
TECHNICAL SALES

Slash sheets can be confusing, and this is a big topic, so let's start big and drill down from there.

Here's the big picture regarding slash sheet references: They were designed to provide handy groupings of PCB materials (laminates, polyimides, etc.) that go into a stackup. These groupings are designed around mechanical characteristics to provide insight for PCB fabricators to identify similar laminates with similar properties.

These documents were not designed to be used by PCB designers to help select the correct material for their job. For high-speed designers in particular, these group references can be a trap. While two material families may have similar mechanical processing characteristics for fabricators, the materials can have wildly dissimilar electrical characteristics. Dielectric constant (Dk), dissipation factor (Df), copper roughness values, and glass weave styles (106/1080/3313, etc.) are all significant for signal integrity and are not appropriately distinguished within slash sheets.

For example, due to this ambiguity of electrically significant values, two electrically dissimilar performing materials could be substituted for one another. The two materials may have similar processing temperatures and roughly similar characteristics; to savvy PCB designers with an eye for signal integrity, slash sheets fall short.

For example, IPC-4101/126 has a maximum Dk value of 5.4, and a Df of 0.035. To use Isola

materials as an example, this lumps 185HR, 370HR, IS550H, IS415, FR408HR, and I-Speed all together. So, if a designer needed high-frequency performance with something like I-Speed, but allowed a substitution with /126, their fabricator would likely swap the material to something else. That other material could likely be a 185HR or 370HR, which have significantly differing Dk/Df values as well as differing glass weave types.

A designer who wants to control the materials in their stackup is often pressed to communicate specific acceptable alternative supplier laminates, such as "Panasonic Megtron 6 (or Isola I-Speed)," to their fabricator.

To use shoes as an analogy: Shoes generally all have a sole, laces, breathable top, etc. A slash sheet reference would help identify "shoes," but doesn't help identify "running shoes" vs. "hiking boots." Nike and Adidas both make running shoes, so a designer could use "/shoes" (slash sheet reference), but they may be in trouble if they are running a marathon and want "Nike or Adidas running shoes." A limitation of the slash sheets is that there are significant differences between the brands and the types of shoes, and unfortunately they could all be lumped together in the same slash sheet for "shoes."

A designer should clearly communicate with the fabricator regarding the desired glass weave, resin type, etc., going into their board if they wish to ensure their signal integrity characteristics will be met. Don't take unnecessary shortcuts when a few more words can save a world of headache. **DESIGN007**

Geoffrey Hazelett has a bachelor of science degree in electrical engineering, and was vice president of sales at Polar Instruments for several years. He is currently home with his new baby daughter, Sage.

BOOK EXCERPT

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

Chapter 5: Control of Electrical Performance

With many things influencing electrical performance, like resin system, glass weave, copper treatment roughness and impact of alternative oxide, what options are available for the user to make sure that the PCB works as intended?

Electromagnetic field simulations have improved significantly in recent years, but they all depend on defining the structures correctly. Geometries can be found quite easily by cross-sectioning. For Dk/Df values, it's important to understand the methods that have been used at the CCL supplier to determine them. There really is no magic Dk number to plug into a simulation tool to get everything right. Even so-called 'design-Dk' values are just measurement by a different test method that may or may not reflect field orientation in a given design.

As covered above, we know that the issue of copper roughness is making things very complicated. The roughness created by the alternative oxide will be different from PCB supplier to PCB supplier, so there are no good rules of thumb that one can apply in a model with 100% confidence.

The only way to be sure is to measure characteristics on the final PCB.

Download this book to continue reading.

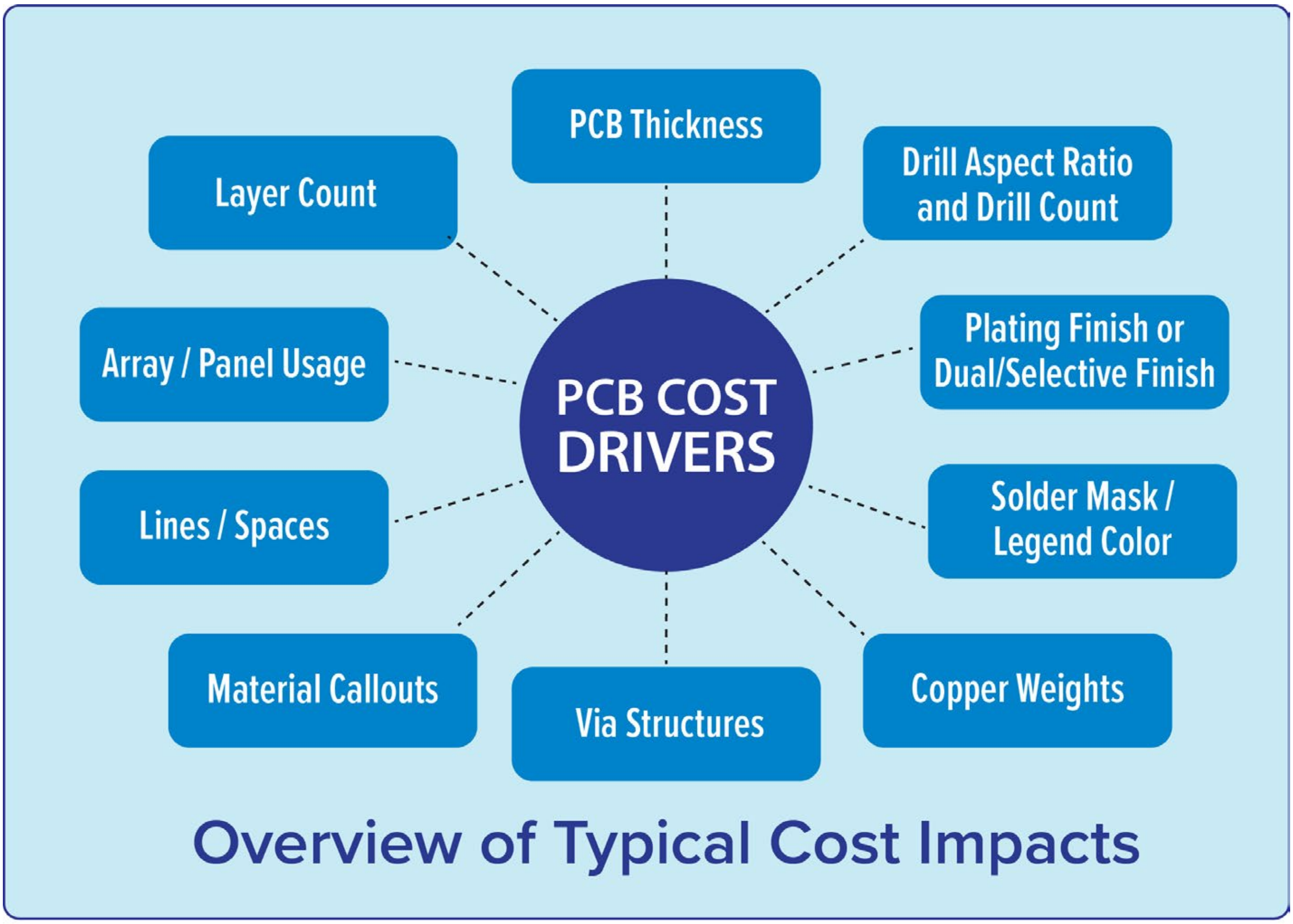
DFM 101
Cost Driver Summary

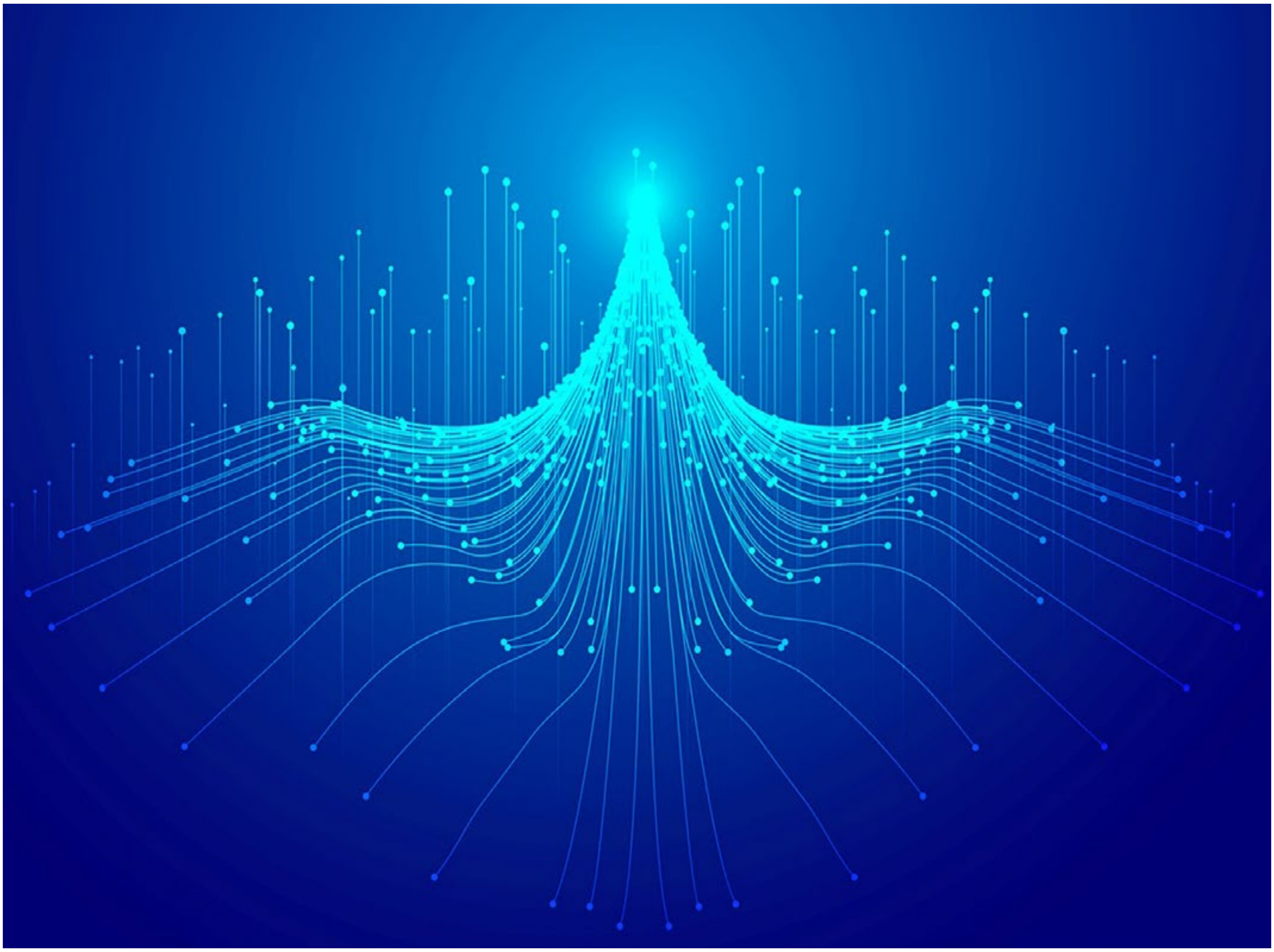
Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

One of the biggest challenges facing PCB designers is in not understanding the cost drivers in the PCB manufacturing process. We will wrap up this DFM series with a summary of cost drivers that impact delivery, quality, and reliability. It is categorized by low-, medium-, and high-cost adders.

Low-cost Adders (< 10% board cost)

Complex routing and scoring mean a small increase in process time, but a process is still required which will be driven by NC programming; it may limit tool life as a function of diameter. Thicker or thinner PWBs (>0.093", <0.030") means a material cost variation but it is minimal. Via plug or button print requires a small process change to the screen-print mask dot.





Medium-cost Adders (10–25%)

Regarding drilled hole quantity, there is a cost adder for high-density design-driven hole count and process time. With smaller drilled hole size, the small drill diameter (<0.010”) limits throughput and stack height.

Embedded resistors with Ohmega/Gould technology will need additional core testing and finished board verification. Non-FR-4 materials, like PTFEs, can be 10 to 20 times the FR-4 cost, and material cost is generally 25–50% of the board cost. Edge plating will have additional processes required prior to plating.

High-cost Adders (>25% board cost)

Advanced technologies may become “science projects” with industry non-standard processing or materials, or “bleeding edge” technology (<0.003” L/S, 1:1 aspect ratio microvias, 0.4 mm BGA technology, etc.). Sequential lamination and complex via structures, as well as metal core or external heat sink requirements also add cost.

If layer count will be >30 layers, the yield impact can be significant. Combination/hybrid material sets can be high-cost adders. Regarding material/panel utilization, test coupons and board size can greatly reduce panel utilization; array configurations reduce panel utilization with unusable real estate; and there may be limited availability of panel sizes.

Selective plating means multiple surface finishes or multiple thicknesses, complex processing requirements (e.g., masking), and the yield risk of combining non-standard processes. There are also cost adders for line width

and spacing below industry standards for any given copper weight:

- ½ ounce copper: 0.003”/0.003”
- 1 ounce copper: 0.004”/0.004”
- Ultra high-density interconnects
- Sub 1 mil line/space (25 and 15 micron)

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 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.

Industry Organizations Keep Knowledge Alive

The Pulse

by Martyn Gaudion, POLAR INSTRUMENTS

When looking at the PCB industry, an outsider may have the illusion that the typically green-colored substrates populated with components use only a small amount of technology. The term "printed circuit," which in principle is accurate, does not even begin to do justice to the sheer magnitude of chemical, mechanical, and metallurgical—not to mention CAD and CAM engineering—that goes into today's highly reliable and complex interconnection substrate. The term "printed," along with the notion that you can simply lay out a board and press "print," as you would a piece of paper, couldn't be further from the realities of a high-tech PCB fabrication process and, importantly, the complex supply chain of chemistry and laminates that feed the factory with raw materials.

The Benefits of Industry Organizations

Design and fabrication have become so specialized that engineers of any discipline can easily become absorbed in their own niche. When working in those environments, it is easy to slip into thinking that your own specialist discipline takes priority over all others. In some cases, that may be true, but in practice all disciplines are important in delivering the best specified product at the best price to the end-user.

Membership in one or two industry bodies provides a broader worldview and a chance to network with people outside your usual circle. IPC, EIPC, ICT, FED, and ZveI are but a few bodies that help improve communication, understanding, and awareness within PCB manufacturing. They often balance a range of


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

conflicting requirements to get compromise on and achieve optimum design. Stretched supply chains have perhaps worsened the "silo mentality" among designers, so it's important to keep other disciplines involved through, for example, their increasing number of webinars. Recently, with supply-chain length and security coming into more focus, there is more thought being given to the importance and contribution of all parts of the supply chain.

The capabilities of PCB base materials continue to increase, even in a changing regulatory environment.

Beyond the Organizations

Technical support specialists for PCB industry material suppliers are omnipresent at these groups' industry events. These specialists are a superb resource for designers who either need to build boards for unfamiliar applications or find a more cost-effective or reliable way to produce existing products on newer generations of materials. The capabilities of PCB base materials continue to increase, even in a changing regulatory environment. You should always keep in mind that when you're looking to update a product or are moving into a higher speed or more environmentally demanding environment, that materials have changed significantly in recent years.

Play a Role

As a designer, fabricator, or supplier, you may feel that you have something important to contribute to ensure that designs are optimally produced. These organizations provide a friendly environment in which to share knowledge, advice, and technical papers that benefit the whole industry, raise your own profile and that of your company. These groups are always looking for new faces and fresh perspectives on the design and fabrication of electronic interconnects. Who knows, you could be one of the industry's next go-to specialists in your discipline.

Cross-discipline Influences

A PCB may be standard or high reliability. It may be functioning in a stable temperature environment or subject to the huge temperature swings of a space application. It may be in the hot, cold, or humid environment of a vehicle engine bay, or immersed in oil in a gearbox application. A designer needs to consider reliability, the desired lifetime, and the operating speed (i.e., high-speed digital, radio frequency, radar frequency, etc). A base material supplier can provide invaluable assistance in choosing material. Industry organizations provide a forum for meeting a mix of specialists from diverse suppliers. However, if you are a designer for low volumes, it may be beneficial to work with a specialized PCB broker. Many of them have extensive material experience and can help match you with fabricators and material suppliers that best suit your application.

Conclusion

As PCB applications become more diverse and PCB designs become increasingly specialized, it is well worth PCB designers' time to meet fabricators and material suppliers through these industry organizations. They provide a good place to network and source the most appropriate materials. Always remember, you need more than just a materials datasheet to specify a PCB construction; you need extensive knowledge and the benefit of suppliers' application specialist support. **DESIGN007**

Martyn Gaudion is managing director of Polar Instruments Ltd., and author of *The Printed Circuit Designer's Guide to... Secrets of High-speed PCBs, Parts 1 and 2*.

MilAero007 Highlights

The Government Circuit: PCBs, Advanced Packaging Key to CHIPS Act Success ▶

In this column, we'll discuss the recent efforts made by the United States and Europe to invest in the entire semiconductor supply chain and strengthen the electronics industry. Also, read on for fresh environmental policy updates, as well as ways to get involved in IPC's advocacy efforts.

Watch Now: Episode 4 of the Micro Webinar Series: 'Smarter Manufacturing Enabled with Inspection Data' ▶

Episode 4, "Envisioning the Factory of the Future" from Koh Young's new micro webinar series 'Smarter Manufacturing Enabled with Inspection Data' is available now.

Raytheon, Northrop Grumman Team Down-selected for U.S. Army's Next-generation Precision Strike Missile ▶

The U.S. Army selected Raytheon Technologies to advance its design for the Long Range Maneuverable Fires program, intended to become Precision Strike Missile (PrSM) Increment 4.

Aegis Combat System Successfully Intercepts Target During Flight Test ▶

USS Daniel Inouye (DDG 118) successfully intercepted a Medium Range Ballistic Missile (MRBM) target using upgraded Aegis software during a recent flight test.

Bigelow: Bullish on Fab's Future ▶

Twenty-plus years is a long time to lead a business during a long decline in the industry, but IMI President and CEO Peter Bigelow remains quite confident about the future. What's his biggest challenge? He may not have all the answers, he's clearly got insight to share.

IPC Welcomes U.S. Presidential Determination Prioritizing Domestic Development of Printed Circuit Boards and IC Substrates ▶

IPC welcomes the action of U.S. President Joe Biden today in issuing a "presidential determination" that prioritizes the domestic development of printed circuit boards (PCBs) and advanced packaging, including IC substrates, under Title III of the Defense Production Act (DPA).

Panel Discussion: CMMC and Cybersecurity ▶

With the introduction of the Cybersecurity Maturity Model Certification (CMMC) framework, businesses will soon be required to meet specific, more stringent cybersecurity standards to bid on Department of Defense contracts. This has made cybersecurity hygiene and CMMC compliance more important than ever for businesses in the sector.

Invested in Growth ▶

The I-Connect007 team paid a visit to American Standard Circuits in West Chicago, Illinois. While there, we talked at length with CEO Anaya Vardya about the issues on his mind as he pushes technology, expands his floor space, and considers the implications of the CHIPS Act, staffing issues, and what's happening in China.


The RF Specialists
• ISO 9001:2015 Certified
• AS 9100 Rev D Certified
• MIL-PRF-31032-C Qualified
• MIL-PRF-55110 Qualified
• ITAR Registered

Batter Up: Stepping Up to the Substrate

Tim's Takeaways

Feature Column by Tim Haag, FIRST PAGE SAGE

It is officially spring, which is typically celebrated with flowers, Easter baskets, and warmer and sunnier weather (except some quite chilly spring days here in Oregon). However, there is one thing that signals spring in my house more than anything else—baseball season. Regular readers of my column know that major league baseball is extremely important around my house. My wife certainly knows it. (Even now, she is in the other room rooting for her Seattle Mariners.) From the perspective of TV viewers, watching baseball is pretty much the same from game to game: Cheer for the good, and boo at the bad.

But especially for those who are new to the game, have you ever considered what it is like from the players' perspective? In a video I watched recently of Adley Rutschman[1] walking onto the field for the first time as a major league catcher in May 2022, he slowly turns around toward home plate, soaking in what must have been an amazing moment for him.

What about a rookie stepping up to the plate for the first time? Consider the incredible focus it takes to evaluate the situation and correctly anticipate the trajectory of a ball traveling toward you at 90 mph. It takes an even greater depth of focus to block out the thousands of

park spectators and millions of TV viewers who are scrutinizing every move you make. The pressure must be unimaginable.

We step up to all kinds of "plates" every day to perform our jobs, don't we? We may not be staring down a baseball fastpitch, but we are sizing up our own skills, experiences, and untold variables to achieve maximum results. It's not the pressure of a national "hit it out of the park" performance like a major league baseball player, but what we do still feels just as important. Now, let's apply that to PCB design.

Personally, I did most of my design work when the work environment was very controlled. I was told what the board was used for, what its mechanical outline and features would be, and how many layers it would contain. Fabrication and assembly decisions were made or provided by someone else, as were the materials, design requirements, parts, and schematics. All I did was lay out the circuit board.

Layout designers are also putting more effort into ensuring that parts specified for the board are available for manufacturing at an approved price and in a timely manner.

The design paradigm has changed quite a bit since then. Circuit board layout designers are now asked to step up to the plate more than ever. Their process includes the mechanical development of the board in order to better synchronize with other portions of the system. PCB CAD tools allow us to co-design with other boards, and it's our job to ensure that the overall system design works together to properly mate cables and other interfacing components.

Layout designers are also putting more effort into ensuring that parts specified for the board are available for manufacturing at an approved price and in a timely manner. Thankfully, many CAD systems today are able to help both component vendors and PCB manufacturers connect online. Designers even have a hand in choosing the appropriate layer stackup for the board and selecting the best materials for optimum performance, mechanical durability, and efficient manufacturability.

PCB layout has come a long way since I first started in the industry. Designers today are being asked to level up their game in many ways. They may not be under as much pressure as a rookie ball player stepping into the batter's box for his first major league at bat, but it's still a lot for a designer who is new to this kind of responsibility. The key is to be prepared and know where to turn to for help when it is your turn to step up to the substrate. In that spirit, here are some tips:

1. Ask for help

Larger companies usually have a community of designers with various levels of experience that can field industry-related questions. However, in a smaller company, you may not have as many people to turn to. In this case, visit online designer forums and other social media platforms for more information. Many designers routinely use these for layout, tool, manufacturing, and other industry-related issues, including looking for the latest information on application-specific materials.

2. Attend design reviews

Most designers are already familiar with attending reviews—including schematic, critical placement, routing, and final design reviews—during key points of the PCB development cycle. There are also pre-design and system reviews that may yield information pertinent to your work. Attending these will give you a better idea of requirements for your current design and will help prepare you for next-generation design.

3. Keep your finger on the pulse

Design techniques, standards and requirements, new technologies, and tool strategies are evolving quickly in our business. To succeed, it is essential to stay current and be prepared for what's ahead. Trade publications like this one are excellent for keeping up with what's new. For example, this issue of *Design007 Magazine* provides information that designers need to make good materials selection choices. The ideas offered here by the experts in our industry will help you with these decisions.

4. Pursue continuing education

Thankfully, we live in an era where almost anything we want to know is widely available. If you haven't done so in a while, get online and explore the webinars, seminars, conferences, and classes that are available. Industry experts routinely offer group presentations on different design topics at PCB events. There are demonstrations by component or material vendors and PCB manufacturers that highlight how their products or services are used in various applications.

5. Work with your manufacturers

Finally, be sure to work with your manufacturers to understand their capabilities and the kinds of materials they recommend working with. Building circuit boards is what they do, so it is in their best interest to ensure that your project will work as you designed it. To that end, they can provide you with a wealth of information on what materials will be best for your design, both electrically and mechanically, and how cost-effective they are. Armed with that information, you will be better prepared to make the best material choices for your board and system requirements.

What do you think? Does stepping up to the plate to face increased design responsibilities still seem intimidating? With so many resources available, I sure hope not. So, grab your bat, or a convenient layout tool, step into the box, keep your eye on the ball, and swing for the fences. You've got this. Keep on designing everyone, and I'll see you next time. **DESIGN007**

References

1. "Adley Rutschman taking it all in during his Major League debut gives you all the feels," youtube.com.

Tim Haag writes technical, thought-leadership content for First Page Sage on his longtime career as a PCB designer and EDA technologist. To read past columns, click here.

Flex007 Highlights

Trackwise Awarded Prestigious King's Award for Enterprise for Innovation ▶

Trackwise Designs plc has been recognised with a prestigious King's Award for Enterprise. Trackwise has been recognised for its excellence in Innovation for its length-unlimited, multilayer flexible printed circuits. Trackwise was previously recognised for export success with a Queen's Award in 2005.

DownStream Discusses Rigid-flex and DFM Trends ▶

Kelly Dack recently sat down with Joe Clark, founder, and Ray Fugitt, technical marketing manager, of DownStream Technologies to discuss trends in tool design and manufacturing, including rigid-flex design and refreshed graphical user interfaces (GUI).

DFM Analysis for Flex and Rigid-flex Design, Part 1 ▶

Today's PCB analysis tools are applicable across all combinations of rigid PCBs regardless of layer count or size. However, due to the unique properties of flexible substrates and combined flexible and rigid substrates, flexible designs require a specific collection of analysis both functional and manufacture oriented.

Rogers Corporation Reports First Quarter 2023 Results ▶

Net sales of $243.8 million increased 9.0% versus the prior quarter resulting from higher ADAS, general industrial and renewable energy market revenues, and favorable currency exchange rate fluctuations. AES net sales increased by 8.4% primarily related to higher ADAS and renewable energy revenues and favorable currency exchange rates, partially offset by lower EV/HEV revenues following strong fourth quarter sales.

DuPont Publishes 2023 Sustainability Report ▶

"We're pleased to issue this year's report that demonstrates how our teams are making meaningful progress to achieve our sustainability commitments and make a positive impact for our customers, employees, shareholders and communities," said Ed Breen, DuPont Executive Chairman and Chief Executive Officer.

Flexible Thinking: Catalyzing Change and Design Evolution ▶

Electronics have wormed their way into our daily lives in ways few of us could ever have imagined. In their early days, especially post-World War II through the 1950s, electronics were largely used to entertain us—the sound of radio and the pictures of television. Those two drivers have clearly not gone away; if anything, they are woven into our lives by internet and smartphone technologies where sound and images demand our attention nearly every waking moment.

Challenges of DFM Analysis for Flex and Rigid-flex Design, Part 2

Article by Mark Gallant
DOWNSTREAM TECHNOLOGIES

(Editor's note: This is the second of a three-part series. To read Part 1, click here.)

Flex and Rigid-flex: Different Layer Types

Everyday rigid FR-4 PCBs have a well-known layer stackup recipe: dielectrics, PCB conductor layers, plane layers, mask, and silkscreen (nomenclature or legend). More advanced layer types may include embedded or screened components or cavities with bonded bare die.

Flex and rigid-flex stackups include those similar to rigid PCBs, such as dielectrics, conductor, mask, and silkscreen layers, but that is where the similarity ends. There are many additional layer types present for this genre of PCB. They include types like coverlay, adhesive, conductive film, conductive foil, conductive adhesive, bondply, and stiffener.

- **Coverlay:** An external polyimide film that encapsulates and protects flexible conductor layers.
- **Adhesive:** Bonding agent used to bond copper foil to a polyimide substrate or coverlay.
- **Conductive film:** Transparent but electrically conductive film found in LCDs, touchscreens, and other optical applications.
- **Conductive foil:** Thin sheet of copper or silver commonly applied to a flexible PCB as shielding.
- **Conductive adhesive:** Adhesive applied for electrical interconnect, thermal or structural bonding applications.

- **(Flexible) bondply:** A polyimide film, coated both sides with B-staged acrylic adhesive. This is essentially a coverlay with adhesive on both sides.
- **Stiffener:** Blank FR-4 or other material bonded to flexible materials to add stiffness in a specific area.

These layer types are found in typical flexible PCB stackups. The presence of these layer types is not accounted for in most current DFM analysis. This is one area where specific DFM analysis is required. Rigid DFM analysis can detect issues with a solder mask but cannot be used to detect trace corners in a bend area.

The fabrication process used to bond rigid and flexible layers also has its inter-layer dependencies to be managed.

Inter-layer Dependencies

While less common in rigid PCBs, flexible PCBs have many inter-layer dependencies that, if not managed well, may lead to manufacturing issues or field failures. The fabrication process used to bond rigid and flexible layers also has its inter-layer dependencies to be managed. Here are a few examples of inter-layer potential issues.

Squeeze-out

Even with best design considerations and best manufacturing practices, adhesive can "squeeze out" and bleed into unwanted areas on adjacent layers. Coverlay adhesive is a common source of squeeze-out that flows during lamination onto conductors, pads, cavities, fingers, or other features. This requires analysis that compares the size of coverlay exposure against the annular ring of adhesive below it.

Missing Coverlay Exposures

Like solder masks, coverlay is intended to cover exposed conductors on flexible layers. When button plating is used, vias on flexible layers must be exposed (non-tented) on the coverlay. Analysis must detect only tented vias in a bend area.

I-Beaming

When two opposing conductor layers of a flexible core have parallel traces, coincident trace segments on the layers may fracture when the PCB is bent. The analysis must analyze only double-sided flexible cores having parallel traces on opposite sides.

Book Binding

In multi-layer flex, each flexible core bent on the same axis in the same location will result in what's known as book binding. The inner layer bends are at the smallest radius, while each successive outer layer bend requires a larger radius to prevent trace fracture. This requires the outer layers to be length-compensated before imaging.

Partial Exposure

Exposures created to facilitate interconnect between flex layers and adjacent shielding layers is not correctly sized, resulting in poor or no connection. Analysis must compare exposure shape and location against its mated shielding layer conductive surface.

Solder Mask Encroachment

In some cases, both coverlay and solder mask are present on a flexible design. Because mask materials can be more brittle than coverlay, there should be a maximum overlap of the two in or adjacent to bend areas. Analysis must detect maximum layer overlap between solder mask and adjacent coverlay layers.

Orphaned Layer Content

Traces, copper, board cutouts, vias, or other layer content are physically occupying space outside, or off of, the actual layer profile. Because of some PCB CAD tool limitations, layer shapes cannot be analyzed for this condition. Analysis must compare a layer's conductors against its layer profile, not the board outline.

There are many other such inter-layer dependency analyses that are not possible with rigid DFM analysis tools that don't recognize coverlays, bend areas, multiple stackups, and other flexible specific data.

Limited Support for Intelligent Formats

Gerber remains the de facto standard for transitioning a design to fabrication. For single- or double-sided substrate flex designs, Gerber is a sufficient format to define fairly simple interconnect. However, Gerber only describes part of the process. It must be augmented with a combination of other files and drawings with extensive details and notes. A typical rigid-flex stackup detail presents all stackup variants across a design, not a single stackup found in rigid designs.

In reality, the majority of flex fabrication packages received by manufacturers often require clarification. Rather than providing manufacturers with multiple files of different formats along with many ancillary files, a flex or rigid-flex design should be transferred with an intelligent data format such as ODB++ or IPC-2581. Intelligent formats have evolved over time to support newer technologies such as screened resistors and capacitors, flex and rigid-flex, and other such data. Almost all PCB CAD tool providers have developed new features for designing flex and rigid-flex. They updated their databases to support multiple stackups, new layer types, new material types, definition of bend areas, and more. However, many are laggards when it comes to exporting flex-specific data to intelligent formats.

For a rigid-flex design, the design processing phase of the fabrication would benefit greatly from a transfer via intelligent data. Most layers of a rigid-flex design involve variable stackups and uncommon layer shapes—details that cannot be effectively communicated with Gerber. PCB CAD tools carefully designed new content in their databases to accurately define both rigid and flexible design data. Yet for many, their ODB++ and IPC-2581 exports omit critical data that would clearly aid a fabricator in expediting design processing.

Without layer profile data, CAD tooling requires a user to interpret documentation to derive actual layer size or shape. Without stackup zone data, boundaries between different stackup areas have to be derived from the

Table 1: The type of flex and rigid-flex data that's not present in Gerber, but supported in intelligent formats

	ODB++	IPC-2581	Gerber	
Layer Profile	Y	Y	N	Defined shape for each layer (or substrate of a core)
Bend Area	Y	Y	N	Defined area where bending is applied to a flex layer
Flex Area	Y	Y	N	Defined area where only flexible layers are present
Rigid Area	Y	Y	N	Area where rigid and flexible layers are present
Stackup Zone	Y	Y	N	Area where a specific stackup is present in a design
Stiffener Area	Y	Y	N	Area where stiffener material is present in a design

same documentation. Without bend areas, it is not possible to determine fracture potential.

In addition, the actual content of some special layer types is often absent. On a solder mask Gerber output, where there is to be mask, there is a void (or shape) on a layer dependent on polarity of the file. A coverlay (layer) is like a solder mask and should visually be the same. In our experience, the coverlay is defined as an empty layer in intelligent files exported from PCB CAD tools.

Stackup zones are critical to getting an accurate DFM analysis in rigid-flex designs. Stackup zones delineate the boundary of each different stackup in a rigid-flex design. An example is an area where only flex layers are present between sections of rigid-flex. Currently, stackup zone definition and export is exclusive to a short list of PCB design tools.

Most DFM analysis tools available today start with translated or converted design data. The native PCB CAD design is converted to Gerber, ODB++, IPC-2581, or other data formats and imported into OEM or third-party analysis tools. More effort should be taken to enhance the conversion (or export) tools to enable a fully intelligent DFM analysis for flex and rigid-flex designs. The limited support for the intelligent formats is currently an impediment to fully assessing DFM for flex and rigid-flex. **DESIGN007**

Mark Gallant is a senior product marketing manager for DownStream Technologies, and author of *The Printed Circuit Designer's Guide to... Documentation.*

Cooking Up Plasmas With Microwaves

Lead author Yurii Victorovich Kovtun, despite being forced to evacuate the Kharkiv Institute of Physics and Technology amid the current Russia-Ukraine war, has continued to work with Kyoto University to create stable plasmas using microwaves.

Plasmas—soups of ions and electrons—must be held at the right density, temperature, and duration for atomic nuclei to fuse together to achieve the desired release of energy.

One recipe involves the use of large, donut-shaped devices with powerful magnets that contain a plasma while carefully aligned microwave generators heat the atomic mixture.

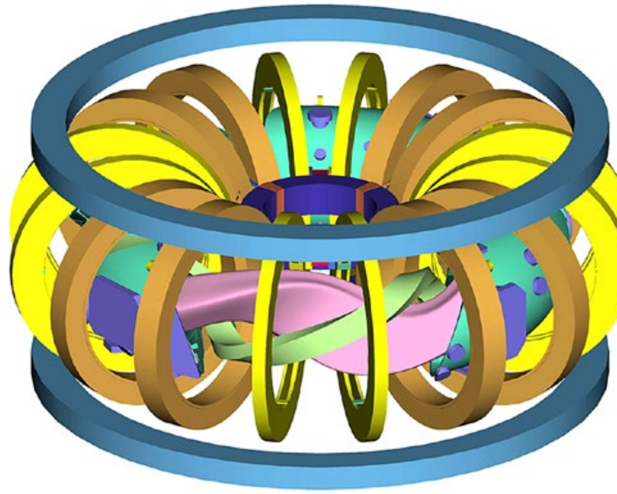
Now, the Institute of Advanced Energy at Kyoto University, together with the Kharkiv Institute and the Max Planck Institute for Plasma Physics have collaborated to create plasmas with fusion-suitable densities, using microwave power with low frequency.

The research team has identified three important steps in the plasma production: lightning-like gas breakdown, preliminary plasma production, and steady-state plasma. The study is being conducted using Heliotron J, the latest iteration of experimental fusion plasma devices at the Institute of Advanced Energy, located on Kyoto U's Uji campus in south Kyoto.

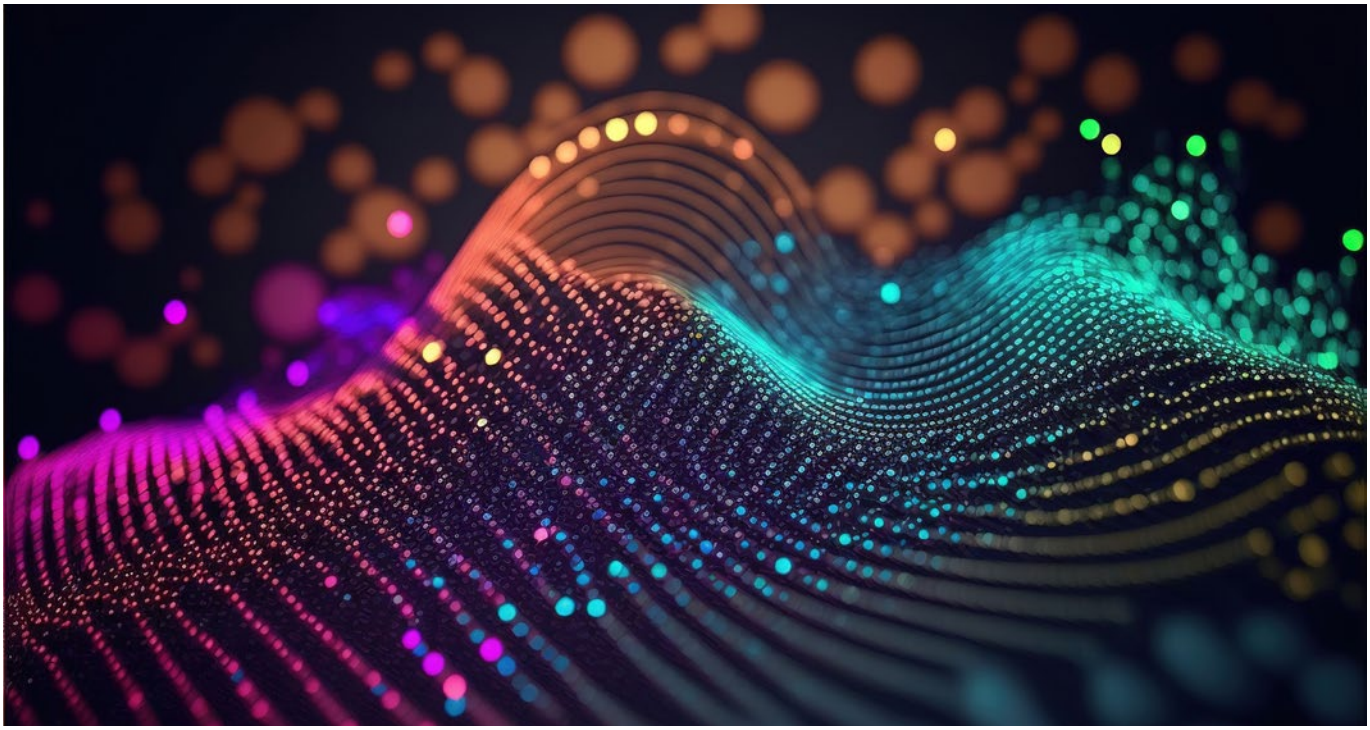
"Initially, we did not expect these phenomena in Heliotron J but were surprised to find that plasmas were forming without cyclotron resonance," group leader Kazunobu Nagasaki explains.

"Unexpectedly, we found that blasting the microwaves without alignment of Heliotron J's magnetic field created a discharge that ripped electrons from their atoms and produced an especially dense plasma," marvels Nagasaki.

"Our findings about this method to generate plasmas using microwave discharge may simplify fusion research in the future." (Source: Kyoto University)




Companionship at its Best
THE SOLUTIONS GUIDE TO...
Thermal Management
ventec
I-Connect007
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...
Thermal Management with Insulated Metal Substrates
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...
Thermal Management with Insulated Metal Substrates
Volume 2
Didier Mauve and Robert Art
Ventec International Group
This must-read sequel to Ventec's book series on Thermal Management describes the applications, IMS products and support services to help you understand and overcome thermal management challenges.
Download Now
I-007eBooks



Selecting Flex Materials: Do Your Homework

Feature Article by Mike Morando
PFC

Flex circuits have become more complex in the last decade, driven in part by shrinking designs and components' higher speeds and signal integrity requirements. Choosing the correct materials for your flex and rigid-flex circuits is a critical part of the design process.

While the layout of the circuit gives us much of the electrical characteristics of the design, your choice of materials can affect the mechanical and electrical characteristics of the circuit. Material choices affect not only the design of the circuit for its environment, but also the manufacturing and assembly processes.

Flex Material Choices: Comprehensive

Your manufacturer's expertise in flex materials is a true asset, especially with today's complex multilayer flex and rigid-flex circuits, as well as high speed and signal integrity requirements. In my opinion, it is a requirement to work with your flex manufacturer to communicate all aspects of the circuit, as well as the environment the circuit may encounter.

Here is a list of flex material criteria that you should discuss with your manufacturer.

- **Approved process:** What materials has your vendor run through their process and approved? All flex vendors etch copper on polyimide, but material availability, equipment, processes, and even humidity can affect processing flex and rigid-flex. Make sure your vendor is familiar with the materials they are proposing and have approved their process. You don't want your project to become a science experiment.

- **UL requirements:** Does the flex need to be UL-approved? What material stacks does your vendor have in their UL library? UL approval is affected by the flammability of the adhesives used in their proposed stack.
- **Flex stack:** Polyimide and copper adhesive-based or adhesive-less? Adhesive-based materials use acrylic adhesive to adhere the copper to the polyimide material. The acrylic adhesive has disadvantages such as via cracking and squeeze-out, and it is more apt to absorb moisture.
- **Electrical Characteristics:**

  › **Power:** Copper thickness, temperature, and adhesive requirements to meet your current requirements.

  › **Impedance/signal integrity:** Your manufacturer should recommend trace, space, and polyimide thickness, as well as appropriate adhesive requirements.

  › **Signal speed/low loss:** In most cases, standard flex materials can be used up to about 10 Gbps. For speeds above 10 Gbps, your manufacturer should recommend alternative materials. PTFE and LCP are the current popular choices. In addition to the base materials, different adhesives and epoxy films can be used in the lamination process that provide low Dk and thinner material stacks. In addition, there are high-speed coverlays to be considered and tested, depending on the Dk values required.

  › **Shielding:** Material stacks may require shielding either for EMI or for support of high-speed low-loss solutions. You can also add more copper layers, cross hatching, metallized epoxies, and some of the new ferrous cover materials. Adding materials to the stack will always affect the bending of the circuit, so keep that in mind.

  › **Bend radius:** IPC standards recommend the bend radius of a circuit to be 10X the thickness of the material. This likely affects material choices and design. The bend of

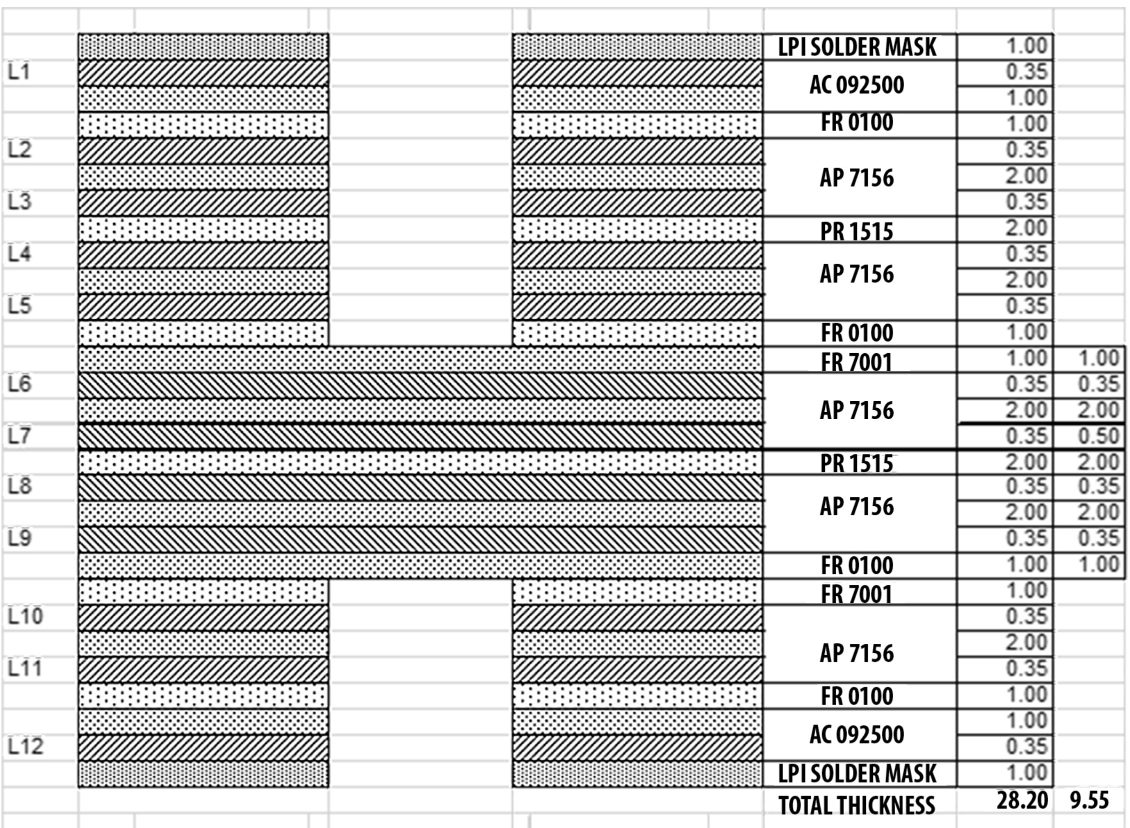


Figure 1.

the circuit may be in a particular section of the circuit only. Share the location of the bend and the number of times the circuit will bend. Your vendor should propose to reduce thicknesses in the required areas by adjusting the material thicknesses.

› **Temperature:** What temperature is the flex going to encounter? High temperatures affect the adhesives used. Make sure to discuss the environment with your vendor. We have used aluminum for heat dissipation.

› **Stiffeners:** FR-4 is the most popular stiffener material. It is used for stiffening SMT areas, through-hole connectors, and for ZIF connector interfaces, but we have used additional polyimide as a stiffener, and ceramic and stainless steel, depending on the requirements.

› **Coverlay and solder mask:** Coverlay (polyimide layer and adhesive over traces) is used in most flex areas, but now with the shrinking of components, the use of solder mask in dense areas has become more common. Remember: Solder mask can crack when bent.

› **Plating:** We are seeing more requirements for wire bonding on flex. Electroplated soft gold or ENEPIG are the best choices. ENIG is the most common for standard assembly.

Ultimately, after working with your manufacturer and going through this list, you should end up with a material stack that meets all your requirements.

Flex circuits and rigid-flex are a more complex solution than a standard printed circuit board. There are far more processes and components to consider vs. a rigid PCB. You must partner with a vendor that has industry knowledge and relationships, experience with the materials, and a full understanding of your environment and requirements. DESIGN007

Michael Morando is director of sales and marketing for PFC Flexible Circuits.

A Sensor That Might Someday Enable 'Mind-Controlled' Robots

It sounds like something from science fiction: Don a specialized, electronic headband and control a robot using your mind. But now, recent research published in ACS Applied Nano Materials has taken a step toward making this a reality. By designing a special, 3D-patterned structure that doesn't rely on sticky conductive gels, the team has created "dry" sensors that can measure the brain's electrical activity, even amidst hair and the bumps and curves of the head.

Most non-invasive versions involve the use of "wet" sensors, which are stuck onto the head with a gloopy gel that can irritate the scalp and sometimes trigger allergic reactions. As an alternative, researchers have been developing "dry" sensors that don't require gels, but thus far none have worked as well as the gold-standard wet variety.

When combined with an augmented reality headset displaying visual cues, the electrodes could detect which cue was being viewed, then work with a computer to interpret the signals into commands that controlled the motion of a four-legged robot—completely hands-free. Though the new electrodes didn't yet work quite as well as the wet sensors, the researchers say that this work represents a first step toward developing robust, easily implemented dry sensors to help expand the applications of brain-machine interfaces.

(Source: ACS)

TOP TEN EDITOR'S PICKS

Simplifying Simulation and Analysis

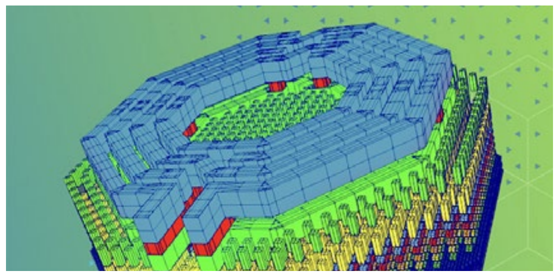

Todd Westerhoff

Gone are the days when simulation meant a simple logic simulation of your circuit, or a basic SPICE run to check signal waveforms. The simulation complexities that are required often fall "above the paygrade" of all but the most experienced members of a hardware design team. How do you keep simulation from becoming a design bottleneck? Todd Westerhoff, product marketing manager at Siemens, dives into the discussion with Andy Shaughnessy.

Lightning Speed Laminates: Optimizing Thermal Management for Wireless Communication Systems

The wireless communication between your mouse and your computer is very different than the wireless communication between a satellite and its ground station. The PCBs which are used for wireless communications are as diverse as the term. As a general statement, a more complex wireless communication system will require a more complex PCB.

Cadence Introduces EMX Designer



The EMX Designer solution lets customers rapidly synthesize DRC-clean passive devices at a touch of a button, based on electrical and geometrical requirements. With unparalleled flexibility, EMX Designer PCells can easily be modified to meet the designers' exact layout requirements, utilizing a long list of options from a user-friendly interface inside the Virtuoso platform.

Elementary, Mr. Watson: Responsible PCB Design Must Go Beyond RoHS

In the field of PCB design, a massive paradox exists. No doubt, the PCB industry is a fascinating field. The ever-changing design environment faces constant and more complex challenges to make products smaller, faster, and cheaper. There are endless ways that electronic innovation changes lives for the better, as seen when advanced medical systems provide patients a new lease on life. But there also is a downside, an unfortunate side, to PCB design, and it has bothered me for some time.

Connect the Dots: Top 5 High-profile Activities for Production Excellence

For electronics manufacturers, consistently producing quality products is the baseline for success. Even as pressures created by supply chain disruptions and labor scarcity persist, organizations need to focus on continuous improvement to remain competitive.

RF and Wireless Design Means Fighting Interference, Loss

Keysight EDA has been in the RF and microwave design industry for decades. We asked How-Siang Yap, product manager for Keysight EDA's RF and microwave software tools, to share his thoughts on RF design tips and techniques, and how designing boards at RF speeds compares with designing PCBs at slower speeds. As he explains, much of RF design involves battling interference and loss.

RF and Wireless PCB Design

We recently met with IPC instructor Kris Moyer for a discussion about designing RF PCBs and wireless applications in general. Kris teaches RF design, among other things, so we asked him to discuss RF design techniques, how designing for wireless applications differs from laying out traditional PCBs, and when to design your own antenna vs. using commercial off-the-shelf (COTS) antennas.

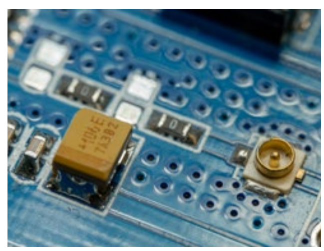
LogicSwap Solutions' Software Converts Third-party PCB Design Data to eCADSTAR

LogicSwap Solutions, a U.S.-based company specializing in data migration software for the EDA industry, is now offering a free data conversion package to transfer third-party PCB design data to Zuken's eCADSTAR format. The software enables eCADSTAR users to utilize Altium Designer library, schematic, and PCB design data for design, analysis, and manufacturing within the eCADSTAR toolset.

Electronic System Design Industry Logs $3.9B in Revenue in Q4 2022

Electronic System Design (ESD) industry revenue increased 11.3% from $3,468.2 million in the fourth quarter of 2021 to $3,858.7 million in the fourth quarter of 2022, the ESD Alliance, a SEMI Technology Community, announced in its latest Electronic Design Market Data (EDMD) report. The four-quarter moving average, which compares the most recent four quarters to the prior four, rose 12.6%.

RF Antenna Design and Layout Tips for Your PCB



Each time a new RF antenna gets added to a PCB layout, it can create a new headache for RF designers, especially as analog design skills start to become critical again. With so many RF capabilities being added to new PCBs, how can designers ensure the signals in their system are not corrupted and signal integrity is preserved?

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

Career Opportunities

THERMAL SYSTEMS

Sales Engineer SMT North Mexico

Rehm Thermal Systems, a leading German manufacturer of reflow soldering systems with convection or condensation and drying and coating systems, has produced energy-efficient manufacturing equipment for the electronics and photovoltaics industry since 1990. We also offer tailor-made applications related to the soldering, coating and hardening of modules.

Responsibilities:

- This position is responsible for expanding our customer network and maintaining existing customer relationships in the Northeast Mexico region. The Sales Engineer would work closely with the German headquarters and the General Manager Rehm Mexico to implement the sales strategy.
- A candidate's proximity to Monterrey, Mexico, is a plus.

Qualifications:

- An engineering degree or comparable qualification with a strong technical background is required.
- Sales-oriented attitude, good communication skills and willingness to travel frequently within Mexico is essential.

We offer innovative products, a great dynamic work environment and exciting training opportunities in our German headquarters.

To learn more about Rehm Group, please visit our website at www.rehm-group.com.

Please send resumes to: Mr. Luis Garcia at luis.garcia@rehm-group.com.

apply now

TAIYO AMERICA

Europe Technical Sales Engineer

Taiyo is the world leader in solder mask products and inkjet technology, offering specialty dielectric inks and via filling inks for use with microvia and build-up technologies, as well as thermal-cure and UV-cure solder masks and inkjet and packaging inks.

PRIMARY FUNCTION:

1. To promote, demonstrate, sell, and service Taiyo's products
2. Assist colleagues with quotes for new customers from a technical perspective
3. Serve as primary technical point of contact to customers providing both pre- and post-sales advice
4. Interact regularly with other Taiyo team members, such as: Product design, development, production, purchasing, quality, and senior company managers from Taiyo's group of companies

ESSENTIAL DUTIES:

1. Maintain existing business and pursue new business to meet the sales goals
2. Build strong relationships with existing and new customers
3. Troubleshoot customer problems
4. Provide consultative sales solutions to customers technical issues
5. Write monthly reports
6. Conduct technical audits
7. Conduct product evaluations

QUALIFICATIONS / SKILLS:

1. College degree preferred, with solid knowledge of chemistry
2. Five years' technical sales experience, preferably in the PCB industry
3. Computer knowledge
4. Sales skills
5. Good interpersonal relationship skills
6. Bilingual (German/English) preferred

To apply, email: BobW@Taiyo-america.com with a subject line of "Application for Technical Sales Engineer."

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Instructor
Longmont, CO

This position is responsible for delivering effective electronics manufacturing training, including IPC certification, to adult 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 primarily conduct training at our public training center in Longmont, Colo., or will travel directly to the customer's facility. It is highly preferred that the candidate be willing to travel 25–50% of the time. Several IPC certification courses can be taught remotely and require no travel or in-person training.

Required: A minimum of 5 years' experience in electronics manufacturing and familiarity with IPC standards. Candidates with current IPC CIS or CIT Trainer Specialist certifications are highly preferred.

Salary: Starting at $30 per hour depending on experience

Benefits:
- 401k and 401k matching
- Dental and Vision Insurance
- Employee Assistance Program
- Flexible Spending Account
- Health Insurance
- Health Savings Account
- Life Insurance
- Paid Time Off

Schedule: Monday thru Friday, 8–5

Experience: Electronics Manufacturing: 5+ years (Required)

License/Certification: IPC Certification–Preferred, Not Required

Willingness to travel: 25% (Required)

apply now

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the Southeastern U.S. territory. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

Reasons you should work with Prototron:
- Solid reputation for on-time delivery (98+% on-time)
- Capacity for growth
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- 5-day standard lead time
- RF/microwave and special materials
- AS9100D
- MIL-PRF- 31032
- ITAR
- Global sourcing option (Taiwan)
- Engineering consultation, impedance modeling
- Completely customer focused team

Interested? Please contact Russ Adams at (206) 351-0281 or russa@prototron.com.

apply now

Career Opportunities

Career Opportunities

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

INSULECTRO

Are You Our Next Superstar?!

Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers to build better boards faster. Our nationwide network provides many opportunities for a rewarding career within our company.

We are looking for talent with solid background in the PCB or PE industry and proven sales experience with a drive and attitude that match our company culture. This is a great opportunity to join an industry leader in the PCB and PE world and work with a terrific team driven to be vital in the design and manufacture of future circuits.

apply now

Career Opportunities

MivaTek Global

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

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

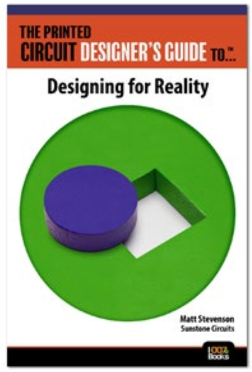
apply now

Career Opportunities

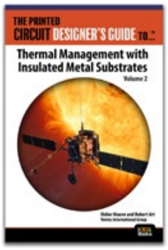
Educational Resources

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

Designing for Reality by Matt Stevenson, Sunstone Circuits



Based on the wisdom of 50 years of PCB manufacturing at Sunstone Circuits, this book is a must-have reference for designers seeking to understand the PCB manufacturing process as it relates to their design. Designing for manufacturability requires understanding the production process fundamentals and factors within the process that often lead to variations in manufacturability, reliability, and cost of the board. Speaking of making better decisions, **read it now!**



Thermal Management with Insulated Metal Substrates, Vol. 2

by Didier Mauve and Robert Art, Ventec International Group

This book covers the latest developments in the field of thermal management, particularly in insulated metal substrates, using state-of-the-art products as examples and focusing on specific solutions and enhanced properties of IMS. **Add this essential book to your library.**

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!**

Stackups: The Design within the Design

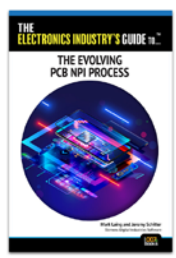
by Bill Hargin, Z-zero

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 ELECTRONICS INDUSTRY'S GUIDE TO... The Evolving PCB NPI Process

by Mark Laing and Jeremy Schitter, Siemens Digital Industries Software

The authors of this book take a look at how market changes in the past 15 years, coupled with the current slowdown of production and delivery of materials and components, has affected the process for new product introduction (NPI) in the global marketplace. As a result, companies may need to adapt and take a new direction to navigate and thrive in an uncertain and rapidly evolving future. Learn how to streamline the NPI process and better manage the supply chain. ***Get it Now!***



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

BUSINESS DEVELOPMENT: **TAMARA SITES**
tamara@iconnect007.com

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

ART DIRECTOR/PRODUCTION MGR. **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER DESIGN: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK** © **RICK**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by IPC Publishing Group, Inc.
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

May 2023, Volume 12, Number 5
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX