I-Connect007

APRIL 2023

DESIGN007 MAGAZINE

DIALING IN RF

DESIGN007 MAGAZINE

Dialing in RF

The last few decades have brought on a proliferation of wireless handheld devices, and almost all of them feature some type of RF circuitry. RF is increasingly cited in *Design007 Magazine* surveys as an area of concern. In this issue, our expert contributors will discuss the best practices for designing RF PCBs for wireless communications, and the many trade-offs involved, from material selection to board-level design techniques.

FEATURE INTERVIEWS

10 **RF and Wireless PCB Design**
with Kris Moyer

16 **RF PCB Design: The Devil's in the Details**
with John Watson

38 **RF and Wireless Design Means Fighting Interference, Loss**
with How-Siang Yap

FEATURE ARTICLE

22 **RF Antenna Design and Layout Tips for Your PCB**
by Cody Stetzel

FEATURE COLUMNS

8 **What's the Frequency, Kenneth?**
by Andy Shaughnessy

30 **Containing Electromagnetic Fields in Wireless PCB Design**
by Barry Olney

48 **Optimizing Thermal Management for Wireless Communication Systems**
by John Coonrod

DESIGN007 MAGAZINE

SHORTS

15 What Does a CAD Designer Do?

37 The AI-shortcut Requires Re-thinking of Examination

51 Flexibility, Communication Help Prevent 'Nightmare' Boards

57 Infographic: You know you're working for a great company when...

61 Robots Can Improve Mental Wellbeing at Work, If They Look Right

73 Magnetism Fosters Unusual Electronic Order in Quantum Material

COLUMNS

44 **Top 5 High-Profile Activities for Production Excellence**
by Kevin Beattie

70 **Catalyzing Change and Design Evolution**
by Joe Fjelstad

ARTICLES

52 **DFM 101: Data Package Guidelines**
by Anaya Vardya

58 **Design for Cost in Real-time**
by Paul Carpine

64 **DFM Analysis for Flex and Rigid-flex Design, Part I**
by Mark Gallant

HIGHLIGHTS

28 **MilAero007**

62 **Flex007**

74 **Top Ten Editor's Picks**

DEPARTMENTS

77 **Career Opportunities**

86 **Educational Resources**

87 **Advertiser Index & Masthead**

What's the Frequency, Kenneth?

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

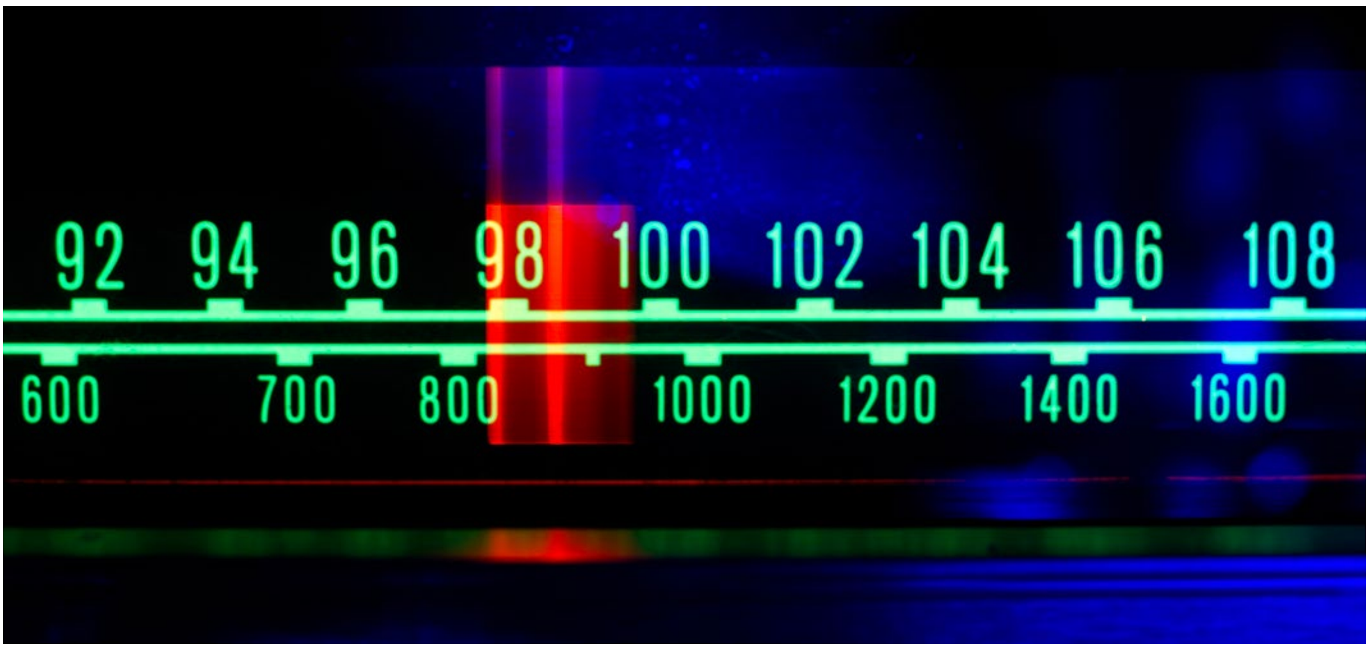
I've been fascinated with antennas since I was 5 years old. I remember my parents directing me to bend our rabbit ears hither and yon until we got a clearer picture of "Laugh-In." I remember thinking, "How does this thing push a signal into space?"

Now, many PCB designers are finding themselves pondering the same question while designing for radio frequency (RF) boards for wireless applications. Once a small but steady percentage of all PCB designs, RF is becoming more commonplace in this segment. The last few decades have brought on a proliferation of wireless handheld devices, and almost all feature some type of RF circuitry.

RF is increasingly cited in *Design007 Magazine* surveys as an area of concern. There's no room for error at these speeds, and RF designers must make constant tradeoffs throughout the design process.

Antennas are an entire discipline within RF. When should you design your own antenna, and when should you use a commercial off-the-shelf antenna instead? Just as importantly, how do you avoid creating accidental antennas?

DFM is more critical than ever. Certain components must be placed in a way that isolates their noise from the other potential "victim" components. Speaking of "victims," wireless devices often include multiple RF points,



which will interfere with each other if designers are not careful. Even a simple trace on an RF board isn't so simple.

Designers often find themselves adding extra vias to create "via shielding" to cut down on crosstalk on RF transmission lines. These vias resemble fences that surround the entire length of the RF trace. Another handy via trick is "via stitching," which involves linking together large areas of copper on multiple layers, which helps designers avoid long return loops and impedance issues.

Material selection is a huge part of RF PCB design. Thermal conductivity is a real concern, and designers must choose a high-frequency material that meets design requirements, and hopefully without overconstraining the board and driving costs sky-high.

Some of you are thinking, "I'm not an RF designer. Do I really need to care about this?" You bet your sweet bippy! Even if you're not working with RF yet, you're likely designing high-speed PCBs. Chances are that you're encountering the same challenges that your RF brethren have been dealing with for years. If you're lucky, you might learn a few things from their (expensive) mistakes.

In this issue, our expert contributors will discuss the best practices for designing RF PCBs for wireless communications, and the many trade-offs involved, from material selection to board-level design techniques, and much more.

We start off with a conversation with IPC design instructor Kris Moyer, who discusses some of the many critical decisions that RF designers must consider. Columnist John Watson focuses on some of the many unique details that a designer faces in the wireless realm. Cadence's Cody Stetzel provides a variety of tips for RF layout and antenna design. Columnist Barry Olney discusses best practices for controlling electromagnetic fields in wireless applications, as well as the theory behind it all. Keysight's How-Siang Yap explains a variety of methods to defeat two of the biggest villains in RF: interference and signal loss. And columnist John Coonrod explains how to take your thermal management skills to the next level when designing PCBs for wireless products.

We also have columns from our regular contributors Matt Stevenson and Joe Fjelstad, as well as another installment of Anaya Vardya's DFM101 series. And this month, we begin with Part I of a three-part series on DFM for flex and rigid-flex authored by Mark Gallant of DownStream Technologies.

We'll see you next month...and that's the truth. DESIGN007

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

RF and Wireless PCB Design

Feature Interview by the I-Connect007 Editorial Team

RF is becoming almost ubiquitous; how many devices in your home contain at least one antenna? Automotive, aerospace, defense, and IoT segments are all pushing the envelope for wireless communication. But designing an RF PCB is a lot different than designing typical boards.

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.

Andy Shaughnessy: Kris, tell us about the RF and wireless PCB market. How does it compare to the rest of the market for PCBs?

It's never huge, but with the increase of smart devices and IoT devices, it's definitely growing. It's not easy to design. Think about your modern cellphone, or your modern smart watch that has GPS, a Wi-Fi antenna, a cellular antenna, and a Bluetooth antenna.

Happy Holden: Now there's wireless charging.

Right. So, there are five different RF frequencies and modules running simultaneously on one board. There are probably more than just five that we're not aware of in terms of what the military, NASA, and other organizations like SpaceX are doing with hundreds of little satellites. The big question is: How do you get all of that to work together and not interfere with each other?

Shaughnessy: Our readers have a lot of questions about antennas and antenna design.


Kris Moyer

This means not only designing the antenna design itself, but also the Bluetooth and Wi-Fi protocols, FCC compliance, and the system-level design to integrate all these different communication methods. You have all kinds of different IEEE protocols involved with that, and how they may interact or interfere with each other. How do you deal with SMA connectors, controlled impedances, wave cavities, and so on?

Back in the radio days, it was literally just an amplifier, a transmitter, and a receiver. The design of the antenna was critical because it controlled your frequency and so on. That's why we used to have TV antennas back in the day, especially the old TV antennas that were triangle-shaped. They used the triangle shape because each of those lengths of antenna picked up a different frequency. Nowadays, it's not only the design of the antenna, but it's which chip you need to use. They actually sell not only 3D antennas, but pre-packaged antennas in chip packages. You can buy a Bluetooth antenna in the equivalent of, say, a 1206 or a chip package, and just solder it onto the board.

Shaughnessy: It seems like the easiest way forward would be to use an off-the-shelf antenna, since it's been validated, I assume, and that preliminary work has been done.

There's more math involved in designing your own antenna, which is technically a "copper geometry device." If you're trying to design the antenna out of the board trace itself, there is a lot of engineering mathematics and analysis that goes into that. Specifically, to get the antenna to work right, you have to convert to the SAP or mSAP process so you don't get non-ideal geometries that you get from the standard subtractive process. You must be much more diligent about the width and the clearance, especially if you're serpentining your antenna rather than making one long straight trace, to save board space.

With a serpentine pattern, now the gap width and the length before the turns will be much more critical than it is with just the simple serpentining I would do for length matching. There's a lot more mathematics involved if you're trying to design the actual antenna. If, on the other hand, all you're doing is buying parts and trying to connect them, you're still into RF board routing techniques, but without the complex mathematics of antenna design.

Holden: Years ago, we had a problem designing the phase array for anti-collision systems for cars because our EEs didn't have radar training. We eventually found a retired EE with the specific RF knowledge that we needed. Have you found that to be true today?

Yes. You need that specialized RF training, or at least access to an RF engineering specialist, to go through the three-dimensional geometry calculations for some of this design work.

Barry Matties: Are antennas needed in every electronic device we have now?

They are. But a lot of them are etched out of the board trace. They're soldered on as a component, so an RF engineer has already done the design work. You connect a proper coplanar waveguide trace to a pin of a three-pin surface mount device.

You need that specialized RF training, or at least access to an RF engineering specialist, to go through the three-dimensional geometry calculations for some of this design work.

Holden: The phase of a radar is multiple antennas, because they're on different layers of the board and they actually direct the electromagnetic field.

Exactly. They direct the field in different directions.

Matties: What does the typical PCB designer need to know about designing RF boards?

I'd start with comparing the additive process vs. the subtractive process, and what that means for the design. Part of that is the skin effect, and RF that needs to be taken into account and why additive rather than subtractive processes are necessary. For proper performance discussions, it's important to understand the complexities of dealing with multiple different RF frequencies running simultaneously, as well as interference and interaction.

At companies where I've worked, we've usually had a dedicated person to do this compliance verification and validation for FCC or EU, or whatever your region. How do you deal with FCC compliance when you've got four or five different frequencies running simultaneously, and that actually need to be EMI emissive? How do you deal with emissivity and susceptibility when you are the generating source and need to be the receiver of those energies but also not interfere with the surrounding devices? It's complex.

Shaughnessy: What's the criteria involved in deciding to design an antenna vs. using a COTS antenna?

Historically, you did it yourself with copper geometry because there simply were no COTS RF parts. Antennas were not commercially available a few years ago. The other reason is board area real estate. The COTS antenna may be too big and I have a very limited real estate that I need to fit this into, so I need to custom design my antenna to fit in this little corner of my board because that's the only area of my product that I can safely remove all shielding material so it can act as an antenna and not shield my RF signals from getting in or out.

Shaughnessy: Who are the new, young RF designers of today? Are they being mentored by the outgoing members of the "old guard?"

A lot of the old RF guys aren't in the field anymore. Honestly, you would find a lot of the new RF guys are making the smartphones. The military contractors in the U.S. would also have upcoming RF design engineers. A lot of the newest RF stuff is happening in IoT and smartphones.

Matties: If I'm looking to hire an RF designer, who am I looking for? An electrical engineer?

Yes, it's going to be someone with an EE degree with a specialty in RF, but very few engineers get straight RF degrees anymore. You need that specialized RF training, or at least access to an RF engineering specialist, to go through the three-dimensional geometry calculations needed for some of this design work.

Matties: What colleges graduate this type of engineer?

I would say MIT, UC Davis, Cal Poly, Penn State, the University of Washington, University of Wisconsin, and Purdue, for a start. There aren't many, though.

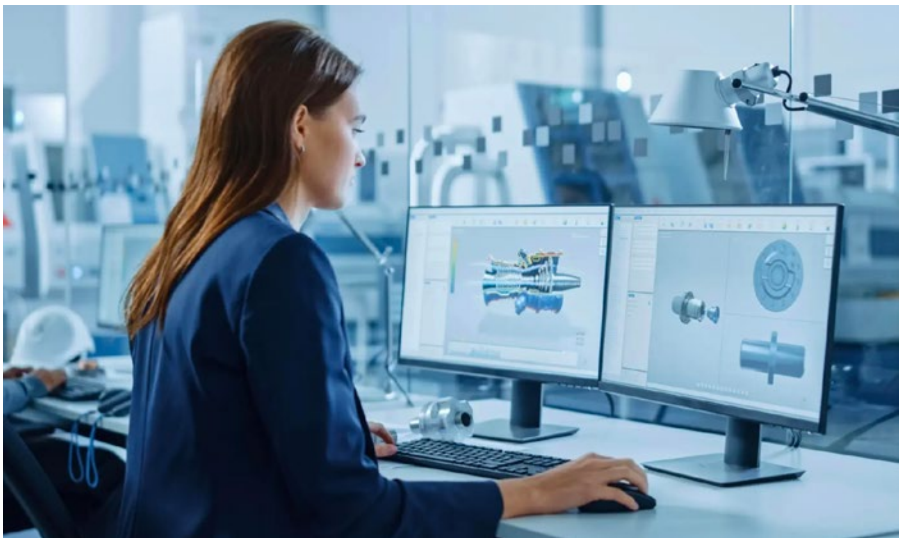
Shaughnessy: It sounds like RF is a discipline that would be worth studying if you're a young student working on an EE degree.

Absolutely. There are very few RF engineers, and we need more of them all the time.

Matties: Thanks so much for your time, Kris. It's always a pleasure.

Thank you all. **DESIGN007**

What Does a CAD Designer Do?



A computer-aided design (CAD) designer is responsible for creating plan outlines and project designs for a specific business need. A CAD designer utilizes various technologies and software applications to generate graphic illustrations. CAD designers must have strong knowledge of the technology designs, software management, engineering standards, and the construction industry fundamentals. CAD designers label their works with appropriate dimensions and specifications, performing necessary adjustments to meet the clients' needs. A CAD designer requires excellent critical-thinking and analytical skills to manage design structures and creative solutions.

Communication is also an essential part of CAD design. Using the computer-generated plans, CAD designers must work with other team members and departments to set timelines, budgets, and assist in making decisions regarding materials that will be used to complete the project.

CAD designers must know their industry. In order to be effective, they must have knowledge of machines, engineering standards, mathematics, and the materials used to make their plans a reality.

CAD designers use technology to help generate designs for complex projects. These designs may be used to create 2D images, a process known as surface modeling, or 3D images, or solid modeling.

A CAD designer utilizes computer-aided design software to create three-dimensional models of complicated structures for many fields including automotive, manufacturing, product development and many others.

CAD designers are a crucial part of any design team. They bring all the ideas of the team to life in a digital world. From there actual physical prototypes can be manufactured.

Learn more at: Careersinelectronics.com

careers in ELECTRONICS

RF PCB Design: The Devil's in the Details

Feature Interview by Andy Shaughnessy
I-CONNECT007

Altium's John Watson is a longtime designer and design instructor, not to mention a Design007 columnist. He's been dealing with RF issues for years; his previous employer, Legrand, is one of the pioneers in smart lighting and data center solutions.

John discusses the differences between designing RF and "typical" PCBs, how to avoid the missteps and miscues that can bedevil new RF designers, and why the tiniest details can make or break an RF design.

How does designing PCBs for wireless communications differ from designing a traditional PCB?

The basis of any basic PCB is a balance of physics in a controlled situation. When RF is introduced to the party, it's the equivalent to having a Bud Light on the hammock in the backyard vs. a full-blown kegger with all your neighbors and friends. When it comes to RF, everything becomes even more essential and critical to success—the physics change with high-frequency signals.

The term radio frequency (RF) refers to the oscillation rate of an alternating electric current or voltage or a magnetic, electric, electromagnetic field, or mechanical system in the frequency range from around 20 kHz to around 300 GHz. That envelops many of today's electronic systems. Also, by definition, RF exists to transmit radiated information from one point to many points. Therefore, the emitted energy must be controlled in a specific area on the PCB level so as not to cause problems.

Is RF the black magic of the PCB world? It is commonly believed to be, but I would disagree


Hmm, what is the recommended **minimum solder mask** width to be able to get a solder mask bridge **between two copper pads?**

with that. I believe it's a misunderstanding about the the physics and mathematics involved. Simulating how it all works together requires an in-depth study and understanding of various Maxwell equations. But many people don't want to take the time to learn Maxwell, because he's not an easy read.


John Watson

At RF speeds, there's no room for error. What sort of problems do designers face in the RF arena?

The problems designers will face specifically with RF are numerous, and they can't be ignored. With a basic PCB, we have a much broader "window of operation and flexibility." With RF, everything gets raised to a whole new level. I fully agree there is no room for errors, and understanding the details of the circuit is probably the first hurdle a designer must consider. Small items on a "basic" PCB become important issues quickly, and significantly impact the design.

But specifically to answer the question, one of the biggest problems is the massive amounts of energy flowing throughout the PCB that you must control. Having a lack of understanding or plan of how you intend to manage that energy is a failure at the offset.

Many of today's devices have multiple RF points—Wi-Fi, Bluetooth, GPS, and cellular. How can designers ensure that these signals "play nicely" together?

When I was raising my children, whenever they would become unruly, I used a common practice utilized by parents all over the world: divide and conquer. I would separate them into their specific areas—and it's much the same with RF.

First, always identify an RF design by its distinct design features. For example, with various RF points, the same method is used: divide and conquer. Identify the various circuits and the level that they are operating, and then isolate them. That is usually done by using a scaled-down version of a Faraday cage, which is an enclosure used to block or control electromagnetic fields in a PCB design. These are cages that are grounded to control the emitted radiation. Placing a grounded metal box around the circuitry will isolate those EMI signals inside that area.

When we bring up the discussion of Faraday cages, we must also discuss the physics involved with controlling such a massive amount of energy, and "skin effect." A Faraday cage works because of the skin effect and depth. The physics of current flow through a material is much higher on the surface and decays considerably more profoundly into the material. That can occur with a metal shield box and is very common in a PCB trace.

That is such a common practice that the edge of the PCB is plated. Without plating, the edge of the PCB can become a weak point in the design. So these signals are entirely isolated into their specific areas, and everybody co-exists.

One significant mistake often made in RF design is inadequate grounding. A shielding configuration such as PCB Faraday cage is just half the battle. Placing enough grounding and stitching them together to keep them at the same reference point, and not allowing stray currents in the grounding plane, helps to isolate the energy into a specific area. That is a common feature you will find in RF design: plentiful ground stitching to the ground plane, which is also connected to the Faraday cage.

Another point to consider is how the signals get routed into another area. The key is that the specific grounding on each side of the isolation is independent. For example, a standard method is to use an opto-isolator. Each side of that opto will have a different and isolated ground to allow a "clean" return path for each circuit.


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

Designers who are new to RF face a variety of decisions regarding antennas. Should they design the antennas or use a commercially available antenna? Should they consider dual-band antennas, which seem to save board real estate?

With the first part of this question, when you learn to design an RF PCB, it's only a matter of time until you must design an antenna. That is something you should take as a challenge in furthering your career.

Considering a dual-band antenna would be the starting point of any PCB that requires an antenna. With a dual-band antenna, you are designing for the worst-case scenario, with space being a significant consideration. I believe it is better to design for the worst-case scenario instead of having to shift gears halfway through and finding out that another solution will not fit.

With a dual-band antenna, you are designing for the worst-case scenario, with space being a significant consideration.

What are some of the considerations designers should weigh when selecting RF materials?

How much time do we have? Materials are an extensive part of RF, and this is where many designers make most of their mistakes, either by selecting the incorrect material for RF or designing a stackup that does not support such a design.

Consider each material's engineering characteristics, such as coefficient of thermal expansion (CTE), dielectric constant (Dk), along with the weave of the substrate and the roughness of the copper foil. Above all, be consistent. Whatever material you use, make it consistent throughout your entire PCB. Mixing various materials causes significant problems in the RF world.

Another point related to material is how the materials are used. When using the more densely stranded FR-4, even the direction of the weave in the stackup can cause an imbalance, resulting in losing control of the RF energy. The same holds true for copper foil. Again, keeping the smoothness consistent helps create a stable RF PCB.

I have found that one major mistake that RF designers make is not keeping the symmetry of the PCB in the Z direction. We must deal with thermal expansion in RF designs, and using different materials that expand at different rates will result in warping and most likely cracking vias and traces.

What advice would you give to designers in the RF world now?

First, be patient. If you expect to learn RF design in the next few months, I have some bad news for you: It's not going to happen. You're going to make mistakes. Some RF designers have studied this area for decades and still only scratched the surface.

Is that a subconscious skin effect joke?

It may be. With that said, study and don't stop studying. Understanding the basics of physics and Maxwell's equations in particular will help you grasp what is happening with RF. Pay attention to the details. Assumptions cause huge problems in an RF PCB design. Even a successful RF design built upon assumptions will be much more expensive than it should be.

Don't be afraid to ask the experts about what you've done. Various RF design experts throughout our industry are willing to evaluate what you've done. Get their input; this is how you learn.

Thanks for speaking with us, John.

Thank you, Andy. DESIGN007

RF Antenna Design and Layout Tips for Your PCB

Feature Article by Cody Stetzel
CADENCE DESIGN SYSTEMS

RF antenna design and layout is one area that requires careful attention to detail and some pointers from mixed-signal designers. If you're just getting started with high-frequency analog design, follow these tips to ensure isolation and signal integrity in your RF design.

These days, it's hard to think of a consumer product that doesn't contain an antenna. Even my garage door opener can connect to my phone via Bluetooth or Wi-Fi. 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?

Thankfully, there are some simple design choices you can make to help ensure your RF signals are not degraded by nearby digital components. These same design choices will help prevent multiple analog signals from interfering with each other. While there are plenty of topics in RF design to consider when designing mixed-signal or all-RF systems, antenna design and layout are probably two of the most important. Here's what you need to know about RF antenna design in your PCB and how to ensure analog signal integrity.

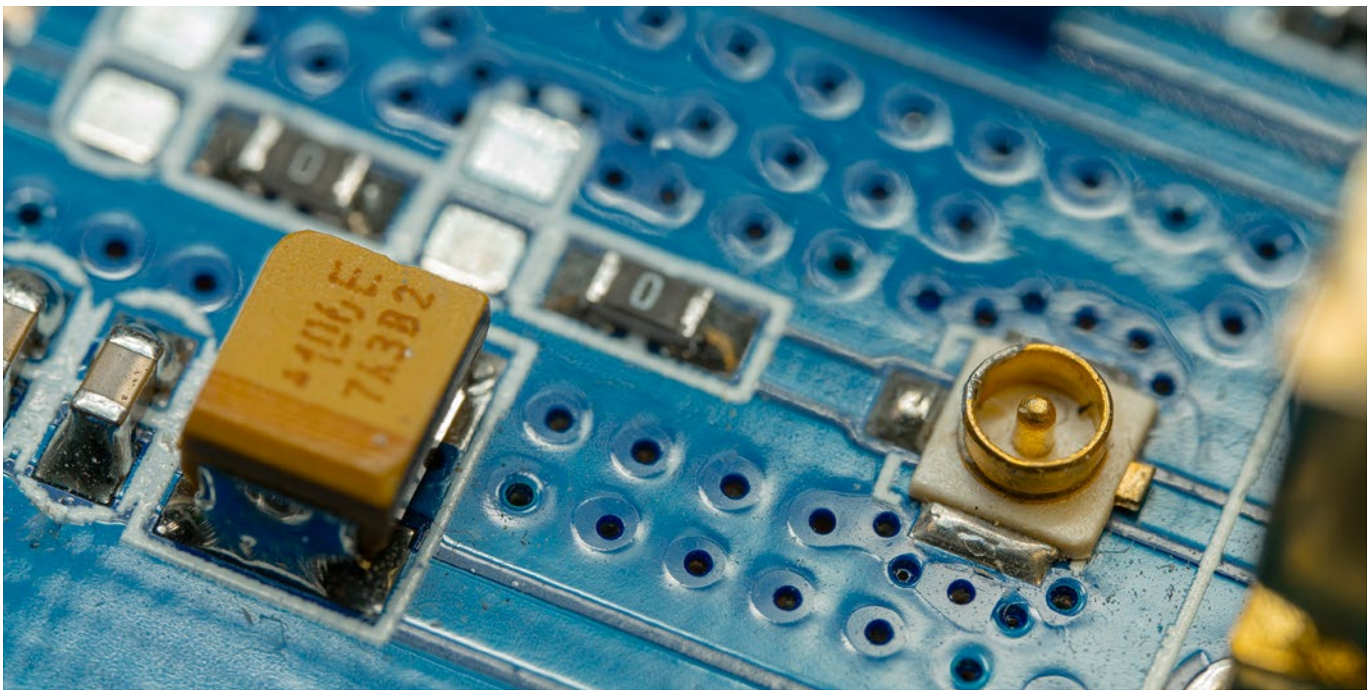


Figure 1: This SMA connector makes a coaxial connection with an RF antenna.


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
DEPARTMENT OF DEFENSE
UNITED STATES OF AMERICA
ITAR

RF Antenna Design Basics

There are a few basic points to follow when designing a custom antenna or choosing a commercial off-the-shelf (COTS) antenna for use in your RF PCB. All RF antennas share a few particular characteristics that should be considered during the design phase. Every antenna needs the following elements:

- **Floating conductive radiator:** This is the antenna element from which radiation will be emitted.
- **Reference:** The reference plane or element for an antenna helps determine the structure's directionality in each antenna mode.
- **Feedline:** The feedline routes the input signal from an RF component into the radiating antenna element.
- **Impedance matching network:** An antenna normally has ~10 ohm impedance, so it needs to be matched to the feedline impedance to prevent reflection and ensure maximum power transfer at the desired carrier frequency and bandwidth.

There are many standard antenna designs that have been thoroughly studied. You can find many reference designs online, which can then be copied into your PCB layout. You can also find many design formulas for standard antenna structures in microwave engineering textbooks.

Finally, if you want to use a COTS RF antenna, there are many inexpensive designs you can find on the market for low cost. No matter which RF antenna you choose to use, you'll need to carefully place it in your layout to prevent interference between board sections.

RF Antenna Layout Tips

Once you've designed your antenna, it's time to figure out where it should be placed on the PCB. RF designers should take some tips from mixed-signal designers (most RF boards are really mixed-signal boards) in order to prevent interference between multiple sections in the RF front end, back end, and digital sections.

- **Efficient radiation:** The goal here is to ensure radiation from antenna elements travels away from the board without being picked up by other structures in the PCB layout.
- **Isolation:** Similarly, we don't want multiple sections in the PCB layout interfering with each other.
- **Electromagnetic compatibility (EMC):** Finally, we need to ensure that the layout is resistant to reception of signals from other devices that may emit over a broad range of frequencies.

In a real PCB, most design goals are in competition, but there are two important points to follow that will help you balance these design goals.

1. Separate Circuit Blocks in Your PCB Layout

This is a fundamental mixed-signal PCB design topic, and it applies just as much to RF antenna layouts. You'll need to place the antenna section in a location on the board that is separate from other circuit blocks. Generally, it is best to place the antenna section near the edge of the board and away from other analog components. This confines strong emission to one location on the board and ensures interference between board sections is minimal.

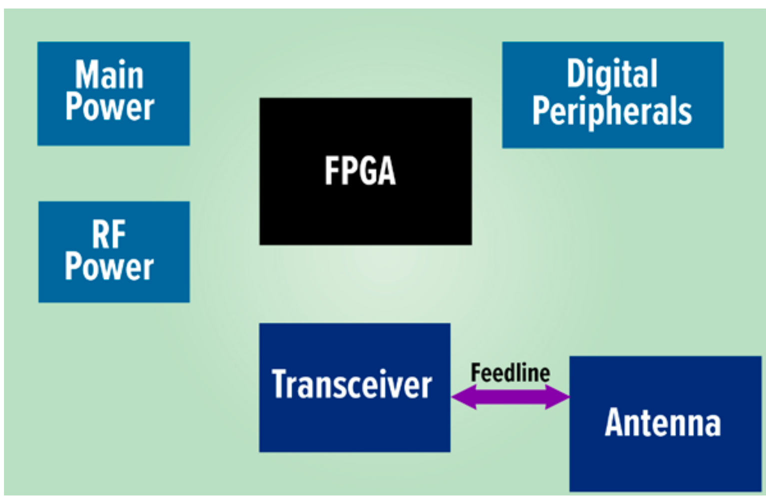


Figure 2: Gridded system layout on an RF PCB.

The challenge in gridding is to ensure your return paths in different sections do not interfere with each other, as this leads to noise coupling and crosstalk. Field solvers integrated into advanced PCB design tools can help you spot deviations from return paths as you create your layout. For high frequency designs, use a continuous ground plane structure to ensure consistent return paths.

Table 1: Pros and cons of various RF isolation structures

Isolation structure	Advantages	Disadvantages
Shielding can	High isolation value as long as gaps in the structure are small.	Can be bulky components, or they need to be custom-built.
Via fences	Similar effect as ground pour, but with less board space.	Lower isolation, narrowband only, low cutoff frequency.
Ground pour	For RF antenna feedlines, creates a coplanar waveguide with high isolation.	Takes up board space, not ideal in smaller boards with dense component arrangement.
Waveguide routing	Very high isolation, mode selection can be performed to enable bandwidth-specific routing.	Takes up board space, only appropriate for most critical lines.
Bandgap structures	Can be engineered to provide moderate to high isolation for particular bandwidths.	Ideal for high frequencies, which will take up less board space.

2. Isolate Antenna Sections

Modern cell phones and cellular equipment have become the gold standard for RF isolation techniques through the use of creative isolation structures. Very simply, isolation involves placing some shielding around an RF-sensitive element in the board to block propagation of waves between an emitter and receiver. Here are some options you can use in your RF antenna section to isolate components, feedlines, and the antenna from each other or external noise sources.

Isolation structures are generally placed between RF elements to block noise coupling and power exchange between them. Determining which isolation structure you should use to ensure RF antenna signal integrity is a complex design problem that has been thoroughly researched. If you're not an expert at elliptical integrals, you'll need to rely on an electromagnetic (EM) field solver to determine how these structures affect feedline/RF antenna impedance, as well as the level of isolation these structures provide.

If you have access to an EM field solver, you can use near-field and far-field simulations to identify areas where strong radiation occurs in your PCB layout. Once you identify these areas and which frequencies are being emitted, it's easier to see which type of isolation strategy you should use. It's best to work directly using a finite element method (FEM) solver rather than using Fourier transforms to convert from FDTD results.

While RF antenna design and layout requires careful attention to detail, this extra caution pays off, as you ensure isolation and signal integrity for your RF design.

Key Takeaways

- RF antennas come in many form factors, ranging from flat chip antennas integrated into ICs to copper antennas printed directly on a PCB.
- Creating a layout with one or more antennas requires ensuring isolation between different circuit blocks in your PCB.
- When you need to design an RF antenna, you should use CAD tools that help you design isolation structures, transition structures, and even printed antennas for your PCB. **DESIGN007**

Cody Stetzel is lead technical marketing engineer for Cadence Design Systems. This article appeared as a Cadence Design Systems blog post.

MilAero007 Highlights

NextFlex Launches $4.4M Hybrid Electronics Funding Opportunity ▶

NextFlex, America's Flexible Hybrid Electronics (FHE) Manufacturing Institute, released Project Call 8.0 (PC 8.0), the latest call for proposals that seek to fund projects that further the development and adoption of FHE while addressing key challenges in advanced manufacturing that support Department of Defense priorities.

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

IPC, the global association representing the electronics manufacturing industry, welcomes the action of U.S. President Joe Biden 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).

American Made Advocacy: A Collective Stake in American Microelectronics ▶

Recently, hundreds of thousands gathered in Las Vegas for the annual Consumer Electronics Show, where amazing new tech was rolled out—everything from razor-thin TVs to smart robots that clean your home. As an executive in the materials science space, I was struck not just by the pace of innovation, but also by the incredibly complex supply chains that were necessary to bring those products from the drawing board to reality.

Hughes Circuits Earns IPC-1791, Trusted Electronic Designer, Fabricator and Assembler Requirements QML ▶

IPC's Validation Services Program has awarded an IPC-1791, Trusted Electronic Designer, Fabricator and Assembler, Requirements Qualified Manufacturers Listing (QML), to Hughes Circuits Inc., located in San Marcos, California.

Insulectro and LCOA Install R&D Lab With Partner Kyocera in Orange County ▶

Insulectro, the largest distributor of materials for use in the manufacture of printed circuit boards and printed electronics, has opened a testing and development laboratory for Kyocera tools in its Lake Forest, CA headquarters. The Lab was created in association with backup and entry materials manufacturer LCOA™ in that company's plant.

Fresh PCB Concepts: PCBs for Harsh and Extreme Environments, Part 1 ▶

At the end of 2022, NCAB's Ryan Miller completed IPC's six-week IPC training certification, PCB Design for Military, Aerospace and Other Extreme Environments. This in-depth course provided him with the knowledge and tools to provide support to customers who are designing within these harsh and/or extreme environments.

IPC Issues February Economic Outlook Report: Economic Data Provides a Mixed View of the Economy ▶

Economic data over the last month is providing a mixed view of the economy per IPC's February Economic Outlook report.

Containing Electromagnetic Fields in Wireless PCB Design

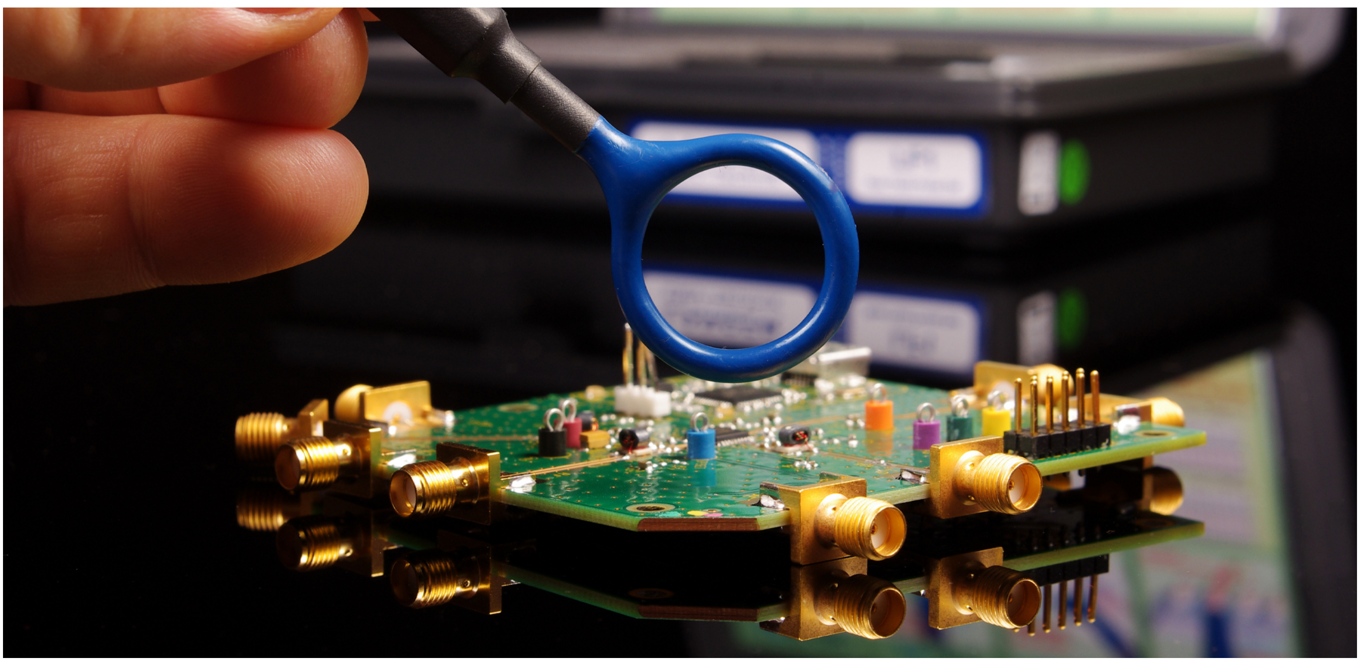
Beyond Design

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

The path of electromagnetic energy in multilayer PCBs is generally guided by a signal trace bounded by the plane(s). However, as the demand for high-density, high-performance microwave (µWave), and millimeter wave (mmWave) circuits increases in the latest wireless technologies, the electromagnetic fields require more stringent control as they tend to radiate more—particularly on microstrip (surface) layers. Thus, as we enter the realm of µWave (3-30 GHz) and mmWave (30-300 GHz), designers are compelled to implement waveguide techniques, used traditionally in RF design, to reduce radiation loss. At these ultra-high frequencies, the behavior of electromagnetic waves and their interaction with circuit components is significantly different from what is observed at lower frequencies. It requires specialized circuit design techniques and the use of components that are tailored to the specific requirements of RF circuits.

One of the most fundamental steps in the process of gaining proficiency in high-speed digital, and RF design (encompassing µWave/mmWave) is learning to think in the frequency domain. For most of us, the vast majority of our early experience with electrical circuits and signals remains within the context of voltages and currents that are either static or dynamic with respect to time.

An RF circuit is a special type of analog circuit operating at very high frequencies suitable for wireless transmission. One prominent feature of an RF circuit is the use of inductive



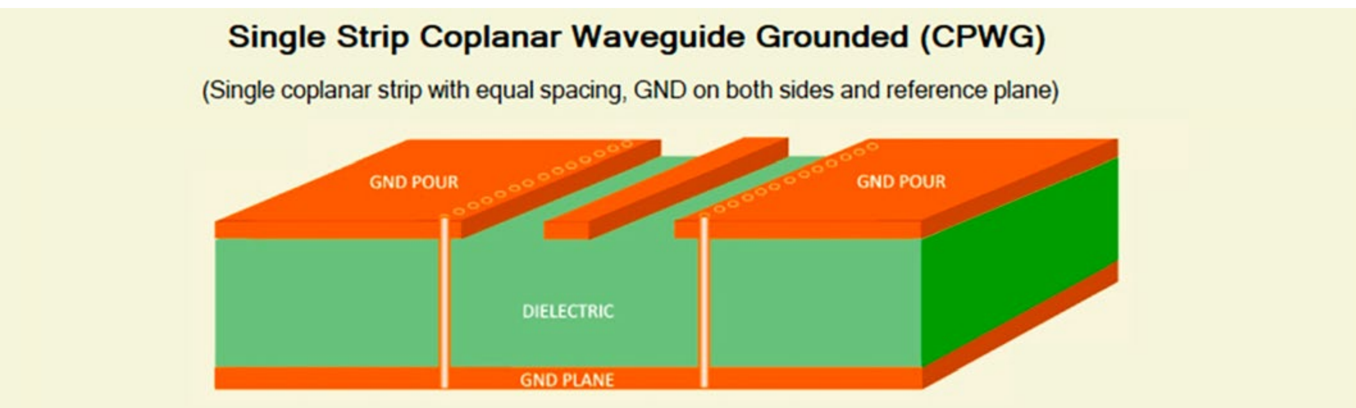

EMC
STRIVE FOR EXCELLENCE
www.emctw.com | EMC.INFS@mail.emctw.com
Enable 800GbE Ethernet Speed with EM-892K and EM-892K2 Materials
Industry's First Halogen-free Extreme Low Loss (ELL) Laminates
Key Material Features:
EM-892K @70%RC:
Dk: 2.84/ Df: 0.0017 @10GHz
EM-892K2 @70%RC:
Dk: 2.76/ Df: 0.0013 @10GHz
with Cavity Resonator Method
Lead-Free Soldering Test:
PASS LF-260°C/ 10X
with 32L 0.6mm hole pitch
Anti-CAF Test:
Pass 100V/ 85°C/ 85% RH
with 32L 0.6mm hole pitch
for 1K Hours
IST Test:
Pass Room temperature
to 150°C for 3K Cycles
with 32L 0.6mm hole pitch
Advanced Foil Availability:
Both HVLP3 and HVLP4 are
compatible with "Glicap"
non-etch oxide treatment
Hassle-free Hybrid Compatibility:
Fully compatible with EM-370(Z) (Mid. Loss) material for cost reduction and pass LF-260 10X with 32L 0.6mm hole pitch
ARLON ELECTRONIC MATERIALS
North American Master Distributor
(909) 987-9533 • arlonemd.com

elements to tune the resonant circuit operation around a specific carrier frequency. In the world of RF design, intentional coupling of electromagnetic fields is also used to create ports between individual circuits.

Typical communication designs incorporate analog, digital, and RF signals on the same substrate, so partitioning is particularly important. In applications where high power is not required, such as a smartphone, Wi-Fi, and Bluetooth, transceiver circuits often take the form of a silicon-based IC, reducing the footprint and simplifying the layout. However, routing all the essential power distribution networks and grounding is always a challenge.

The most effective board stackup for RF design is to have a ground reference plane immediately adjacent to the surface layers and to keep the RF traces on the surface as much as possible. Minimizing vias in the RF path reduces trace inductance, reduces the voids in the ground plane, and gives less opportunity for the electromagnetic energy to escape. Through-board vias should be avoided to prevent unwanted fields from transferring from one side of the board to the other or into the plane cavity. The field, emanating from the vicinity of the signal via, injects a propagating wave into the cavity which can excite the cavity resonances or any other parallel structure. Other signal vias passing through this cavity can pick up this transient energy as crosstalk. The usual technique to prevent this is to use blind microvias from both sides, effectively making two separate back-to-back boards. Digital signals and power can be routed in the internal layers. One can minimize the impact of the essential through-vias by placing them in an area that has no RF signals.

Microstrip transmission lines have been widely used in RF circuit design for decades. However, at high frequencies, microstrip lines can suffer from significant signal loss due to radiation and dielectric losses. Coplanar waveguides (CPWs), on the other hand, offer lower radiation loss and are becoming a popular alternative for high-frequency digital circuits. CPWs consist of a central conductor on the surface layer of the PCB, flanked by two ground planes on either side, and are usually ground referenced, which confines the electromagnetic field and reduces radiation losses (Figure 1).



Layer No.	Via	Description	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Characteristic Impedance (Zo)
		Soldermask		Dielectric	3.3	0.5				
1	8	Signal	Top	Conductive			1.7	25	20	48.98
		Prepreg		Dielectric	3.5	10				
2		Plane	GND	Conductive			1.4			

Figure 1: Single strip coplanar waveguide grounded. (Source: iCD CPW Planner)

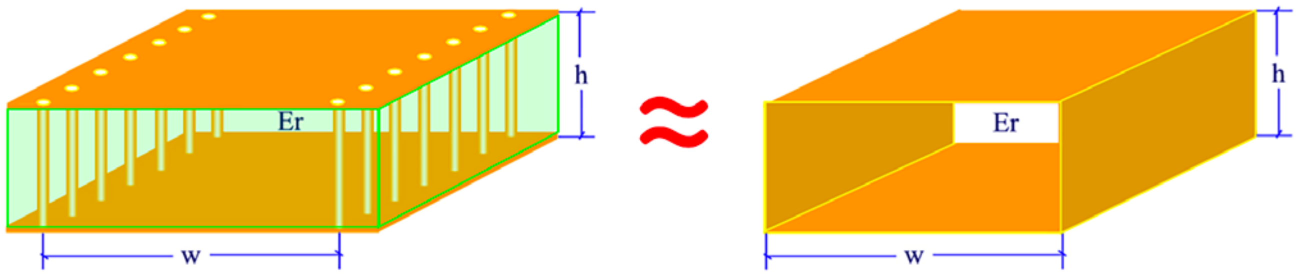


Figure 2: The SIW (left) features similar properties to the metallic waveguide (right).

The CPW offers several advantages over a conventional microstrip transmission line:

- Simplifies fabrication
- Facilitates easy shunt as well as series surface mounting of active and passive devices
- Eliminates the need for via holes and wraparound (ground plating on the edge of a substrate to provide a low inductance path)
- Reduces radiation loss at very high microwave frequencies

The impedance of a CPW is determined by the ratio of trace width to clearance, so size reduction is possible without limit, the only penalty being higher losses. In addition, a virtual ground plane exists between any two adjacent lines, as there is no field at that point. Hence crosstalk effects, between parallel trace segments, are extremely weak. Despite the efforts to evolve and improve the existing transmission line structures, it still remains a technological challenge, which necessitates the emergence of a revolutionary concept.

As digital transmission frequencies head toward 100 GHz and beyond, the current mainstream PCB technology—the copper interconnect—is reaching its performance threshold. Ultimately, dielectric loss, copper roughness, and data transfer capacity are the culprits. However, the biggest performance restriction for PCB interconnects is the size of the conductor. Metallic waveguides, on the other hand, are a better option compared to traditional transmission lines, but they are bulky, expensive, and non-planar in nature. Recently, substrate integrated waveguide (SIW) structures have emerged as a viable alternative and are ideally suited to the high-speed transmission of electromagnetic waves. SIWs are planar structures fabricated using two periodic rows of PTH vias or plated slots connecting adjacent copper ground planes of a dielectric substrate as shown in Figure 2 (left).

Several types of transition from SIWs to microstrip or CPW structures are possible but most are challenging to implement. They can be roughly divided into single-substrate or multilayer substrate applications. Dual-layered SIW transitions to microstrip or CPW structures have been successfully applied. But multilayer SIW circuits often suffer from alignment issues. Z-axis alignment of the multilayer laminate book has always been a major limitation of implementing any broadside-coupled application. However, due to the similarity between the traditional waveguide and microstrip modes, the dual-layer microstrip-to-SIW transition is undoubtedly the simplest to implement.

Figure 3 illustrates the transition from a microstrip transmission line to a SIW. The propagating electromagnetic wave, guided by the microstrip trace, travels through the dielectric between layers 1 and 2 and radiates from the solder mask into the surrounding volume. However, as the wave enters the SIW, it begins to tunnel between the ground planes and as such, the dispersion losses are solely based on the losses of the substrate material. The simulation of the electric field shows how the losses

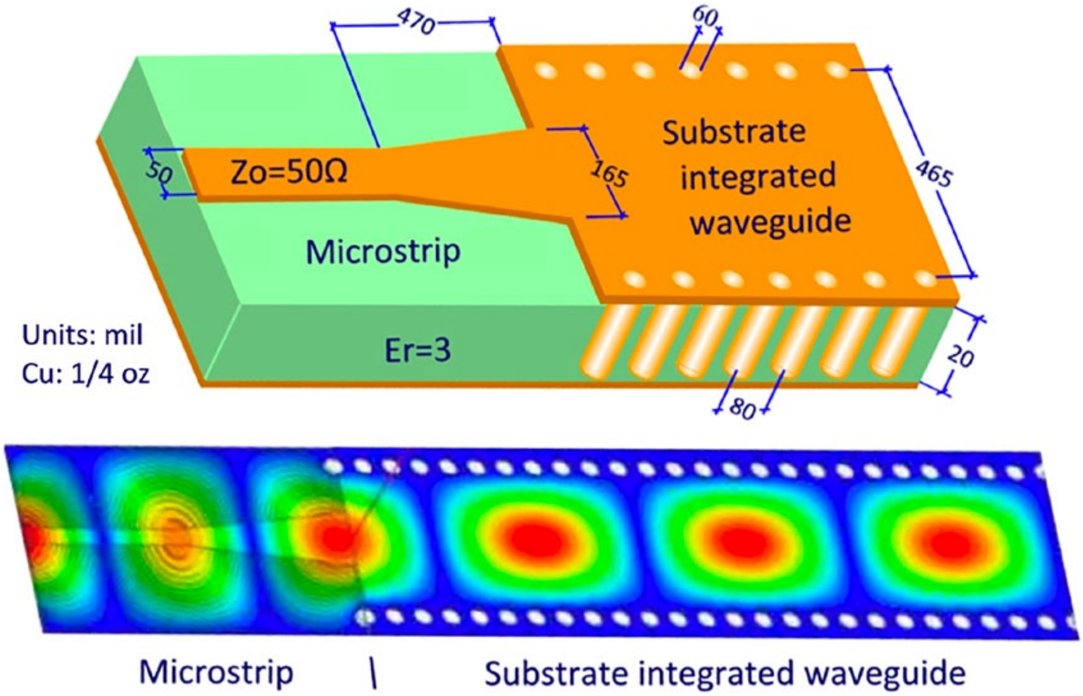


Figure 3: Microstrip-to-SIW transition and simulated electric field. (Source: Kumar[3])

reduce as the electromagnetic wave enters the SIW. Here the field becomes more intense and less distributed, providing clarity of signal and thus higher bandwidth. Obviously, another similar transition back to the microstrip, at the other end to receive the signal, is also required. The downside, of this configuration, is the distortion of the field—limiting bandwidth—at the transition from the microstrip to the SIW.

While it is relatively simple to interface SIW with microstrip lines, in order to exploit connectivity to surface-mount devices, CPW technology is more amenable to integration with leadless monolithic microwave integrated circuits (MMIC). An MMIC is a device that operates at μWave frequencies and typically performs functions such as μWave mixing, power amplification, low-noise amplification, and high-frequency switching. Figure 4 illustrates a back-to-back CPW grounded to SIW structure. Ultra-low loss RT/Duriod 5880 dielectric is used for the core material. The transi-

Layer No.	Via	Description	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Characteristic Impedance (Zo)
		Soldermask		Dielectric	3.3	0.02				
1	8	Signal	Top	Conductive			0.7	4	30	49.51
		Prepreg		RT/Duriod	2.2	20				
2		Plane	GND	Conductive			1.4			

Figure 4: Back-to-back CPWG to SIW. (Source: Taringou[4] and iCD CPW Planner)

tion performance can be controlled by three main parameters (Lt, Wt and W_{edge}) which are adjusted to optimize the return loss and insertion loss properties.

The drawback of the coplanar-to-SIW transition is the limitation of bandwidth due to higher-order mode propagation if the via holes are not placed in close vicinity to the CPW slots. Also, if the via pitch is increased while the via hole diameter is unchanged, the electromagnetic field starts to radiate outside the via hole arrays. This gives rise to leakage loss in addition to the dielectric and conductor loss of the waveguide. The energy escaping the SIW channel is shown graphically in Figure 5 where field propagation outside the functional region of the waveguide is observed. Thus, it is imperative to select SIW parameters such that the leakage loss is maintained within an acceptable range. The via pitch length, the hole diameter, and their ratio prove to play a key role in confining the fields in an optimal manner.

Substrate-integrated waveguides are similar to traditional waveguides in terms of their ability to support the propagation of electromagnetic waves with low loss and dispersion. However, SIWs achieve this performance in a planar structure by using a substrate material with a low Dk and a low Df. SIWs can be used in a wide range of µWave applications, such as filters, couplers, power dividers, and antennas. The combination of microstrip, CPW, and SIW technologies provides designers with a high degree of flexibility in the design of complex µWave/mmWave systems. Overall, the design of RF circuits requires specialized knowledge and techniques that are tailored to the unique challenges of high-frequency circuit design.

Key Points

- Wireless technologies require more stringent control of electromagnetic fields as they tend to radiate more—particularly on microstrip (surface) layers.
- An RF circuit is a special type of analog circuit operating at very high frequencies suitable for wireless transmission.
- The most effective board stackup for RF design is to have a ground reference plane immediately adjacent to the surface layers and to keep the RF traces on the surface as much as possible.
- To prevent unwanted coupling, use blind microvias from both sides, effectively making two separate back-to-back boards.
- At high frequencies, microstrip lines can suffer from significant signal loss due to radiation and dielectric losses.
- CPWs offer lower radiation loss and are becoming a popular alternative for high-frequency digital circuits.

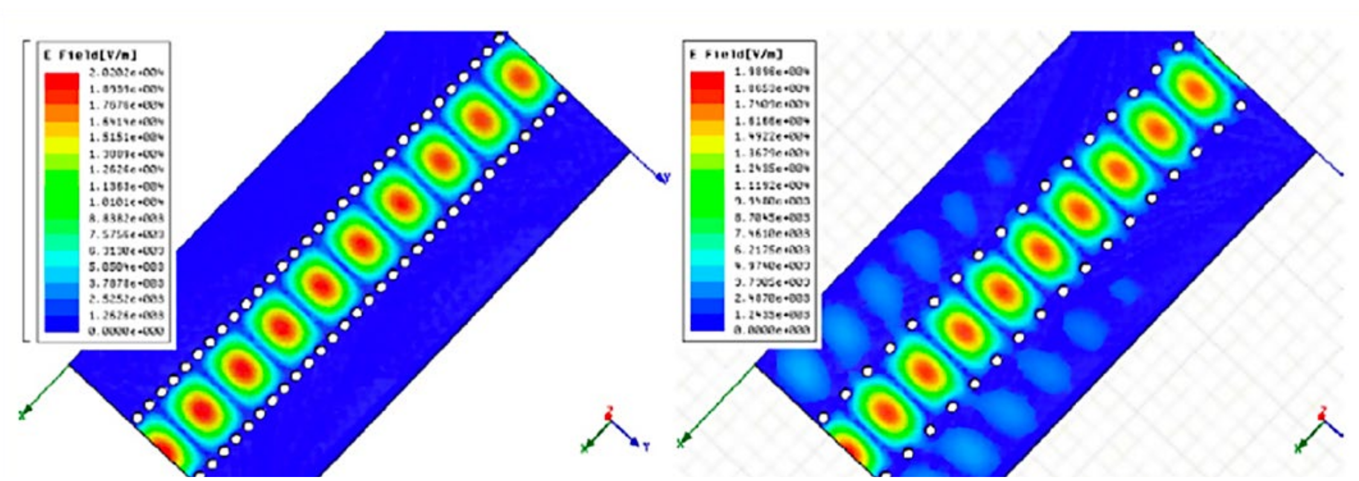


Figure 5: Electric field comparison for two SIWs with different via spacing. (Source: Taringou[4])

- The impedance of a CPW is determined by the ratio of trace width to clearance, so size reduction is possible without limit, the only penalty being higher losses.
- SIW are planar structures fabricated using two periodic rows of PTH vias or plated slots connecting adjacent copper ground planes of a dielectric substrate.
- SIW structures have emerged as a viable alternative to microstrip lines due to their low loss, and are ideally suited to the high-speed transmission of electro-magnetic waves.
- Several types of transition from SIWs to microstrip or CPW structures are possible but most are challenging to implement. The dual-layer microstrip-to-SIW transition is undoubtedly the simplest.
- CPW technology is more amenable to integration with leadless monolithic microwave integrated circuits. **DESIGN007**

References

1. Beyond Design columns by Barry Olney: "The Frequency Domain," "Next Gen PCBs: Substrate Integrated Waveguides," "Microstrip Coplanar Wave Guides," "It's a Material World, Plane Cavity Resonance."
2. "Partitioning for RF Design," by Andy Kowalewski (RIP, old friend).
3. "A Review on Substrate Integrated Waveguide and its Microstrip Interconnect," by Kumar, Jadhav, Ranade.
4. "Transitions from Substrate Integrated Waveguide to Planar Transmission Lines and their Applications to Amplifier Integration," by Farzaneh Taringou.
5. "What is RF Circuit Design? How to Design RF Circuits," by Synopsys.

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 icd.com. au. To read past columns, click here.

The AI-shortcut Requires Re-thinking of Examination

Changes are ahead when it comes to testing students' knowledge. The arrival of ChatGPT challenges both the university's teachers and its disciplinary board.

Artificial intelligence helping students write essays and assignments is already a reality. One problem is that university plagiarism checks cannot identify texts not written by the students themselves.

"AI is a technology that brings fantastic opportunities. We shouldn't focus only on the risk of cheating brought on by AI, but also on how to benefit from this technology in our teaching and examinations," says Karin Axelsson, deputy vice-chancellor at Linköping University and chair of its Disciplinary Board tasked with handling attempts at cheating, etc.

So far, the Disciplinary Board has not handled any cases relating to the use of Chat GPT. According to Axelsson, this is because this technology is currently difficult, if not impossible, to detect.

"Like all other education providers, LiU realises that Chat GPT presents new opportunities for students wishing to take a shortcut in certain types of examinations. Course coordinators should try to find solutions to how examination of texts not written in an examination hall can be complemented with other ways of testing students' knowledge," says Axelsson.

(Source: Linköping University)

RF and Wireless Design Means Fighting Interference, Loss

Feature Interview by Andy Shaughnessy
I-CONNECT007

Keysight EDA has been in the RF and microwave design industry issues 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.

How does designing PCBs for wireless communications differ from designing a traditional PCB?

The major differences in designing PCBs for wireless communications are:

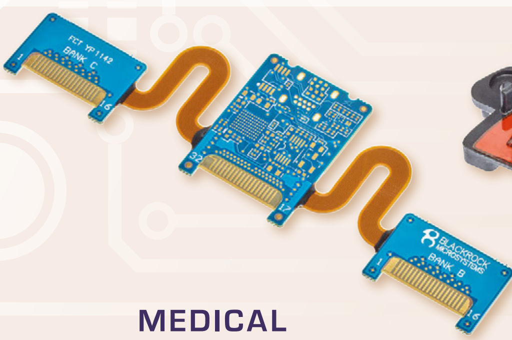
- Layout of traces, vias, interconnects, and packaging must consider the electromagnetic behavior of these physical structures on wireless communication performance.
- Layout must be optimized for RF (radio frequency) sensitivity to prevent interference and noise coupling from non-wireless and digital sections by shielding, absorption, or isolation.
- Layout must consider impedance matching between components and to antennas for optimal power transfer and low noise performance. This means that trace ground reference, shape, widths, and interconnecting transitions are not arbitrary, and they must obey RF signal propagation and radiation geometries.

PREMIER GLOBAL SUPPLIER of **FLEXIBLE CIRCUITS**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

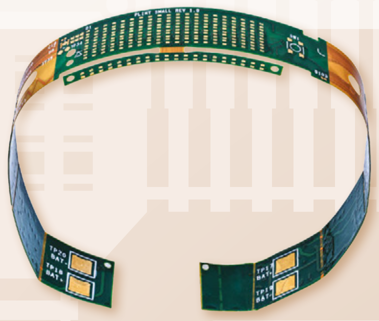
- **Superior design consultation & support**
- **Flexible Circuits**
- **Rigid Flex**
- **Flexible Heaters**
- **Flex/Rigid Flex Assemblies**
- **Membrane Switches**
- **Plastic Moldings**
- **Specialized EMS/Assembly**
- **Product Module to Complete Box Builds**

EXPLORE the POSSIBILITIES!

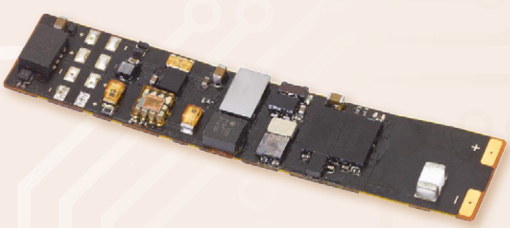


MEDICAL

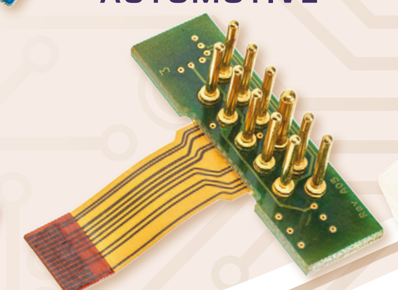
AUTOMOTIVE



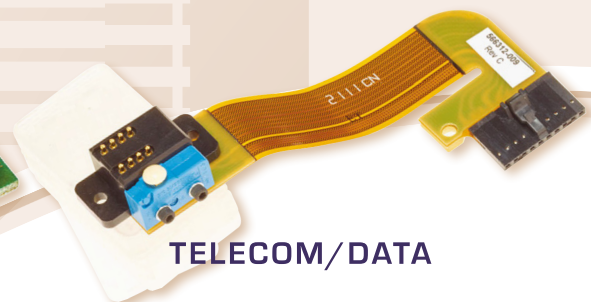
CONSUMER PRODUCTS



WEARABLES



INDUSTRIAL



TELECOM/DATA

FLEXIBLECIRCUIT® Technologies

Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

- Select material properties of PCB board and layer stackup arrangements to ensure minimal loss of precious RF wireless signals to optimize performance.

Shaughnessy: What sort of problems do designers face in the RF arena that they don't see with typical PCB designs?

Based on these differences, we can see that RF PCB design is not just designing interconnecting traces to fit onto a board. We must also consider how RF signals are protected from interference from non-RF sections on the same board and propagated along RF transmission lines such as co-planar waveguides (CPW) or microstrips with careful shaping of transitions and bends to prevent signal reflection and insertion loss.

RF PCB design minimally requires simulation of critical RF signal paths for loss and unwanted coupling from neighboring non-RF components and traces. It is important for the RF PCB designer to understand how to simulate and optimize critical RF paths on a mixed-signal board.

Many of today's devices have multiple RF points—Wi-Fi, Bluetooth, GPS, and cellular. How can designers ensure that these signals "play nicely" together?

Designers need to understand the antenna characteristics and where to locate the antennas for these signals on the PCB board to satisfy these prioritized requirements:

- Antenna efficiency and impedance matching

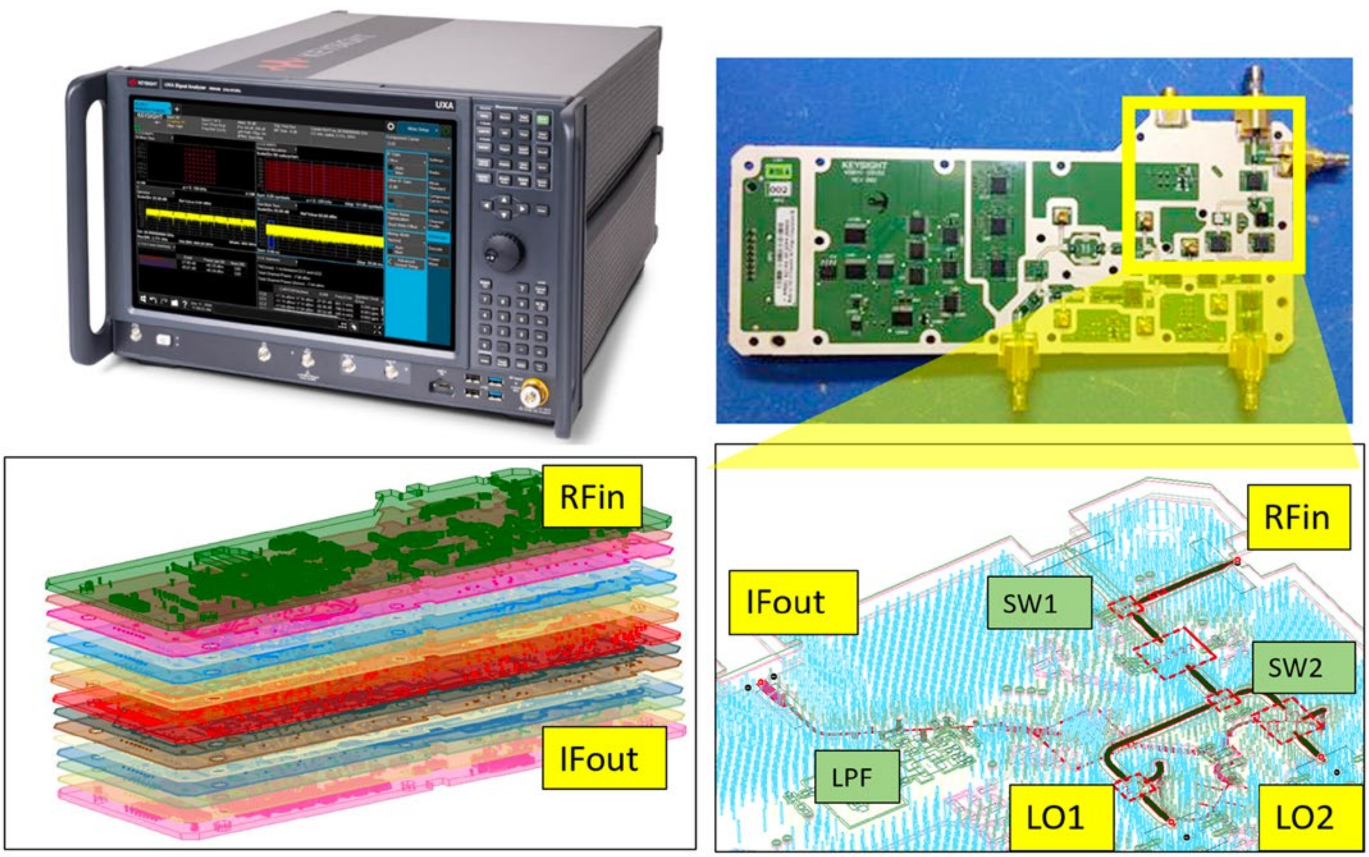


Figure 1: Examples of multilayer mixed-signal RF PCB simulations generated by a Keysight UXA signal analyzer utilizing Keysight RFPro software. Note that the critical RF paths traverse 14 layers of the PCB. The design data was input in the OBD++ format.

- Radiation patterns don't overlap in the same frequency bands
- Isolation and separation amongst antennas
- Alignment with polarization of these signals (e.g., right-hand circular, vertical, or horizontal)

Add lumped or printed filters along signal paths for these different signals to reject interference outside of their carrier frequencies and bandwidths.

Route and shield RF transmission lines to minimize interference with these signals. Simulate the above structures at various operating signal frequency bands to verify and tweak until they "play nice together" with minimal interference prior to prototyping.

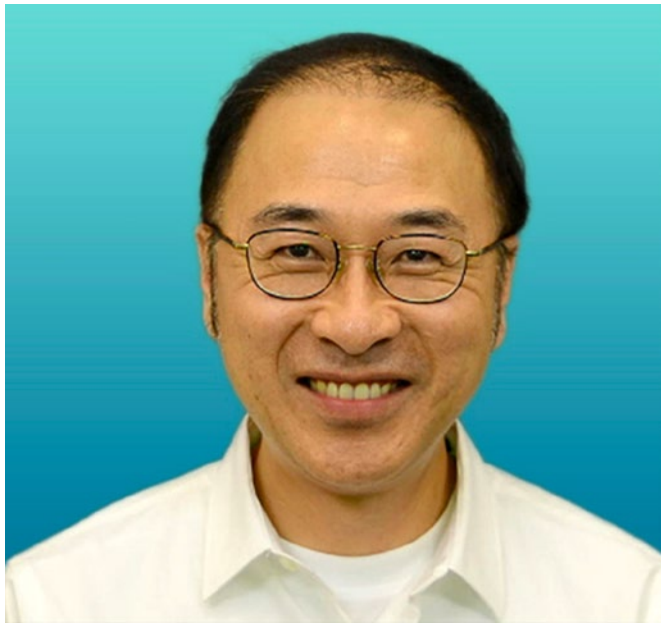
What do "traditional" PCB designers and EEs need to know about measurement and analysis of RF circuits? How is this different from analysis at lower speeds?

Measurements and simulation analysis always go together in the design of RF PCBs, because designers need to understand how physical layout and 3D packaging affects RF performance and then account for these effects. Designers can use simulation analysis to reproduce the non-ideal effects seen in measurements in order to prescribe corrective design tweaks for the next prototype. At lower speeds, the usual measurements will be based on connectivity checks.

Shaughnessy: What are some of the considerations that PCB designers should take into account when selecting RF materials?

Some of the most important considerations when choosing RF materials are:

- Cost
- Signal loss at the intended frequency of operation
- Tolerance of electrical and mechanical properties



How-Siang Yap

- Stability of electrical and mechanical properties over a range of temperature and humidity
- Manufacturer's availability schedule and risk of delays or discontinuation
- Risk mitigation from second source availability or replacement

What advice would you give to designers who find themselves in the RF world now?

Learn how to use the latest RF circuit and EM simulators. This is critical. Learn how to use EM and circuit simulators together in co-simulation and co-optimization of electrical and physical layout parameters to preserve given RF specs. This will help reduce the number of design iterations.

Thanks for your time.

Thank you. I appreciate it. DESIGN007

Top 5 High-profile Activities for Production Excellence

Connect the Dots

by Kevin Beattie, SUNSTONE CIRCUITS

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.

In this effort, manufacturers should challenge themselves by constantly seeking to make operations run better, increase profitability, and improve the customer experience. By focusing on activities that move the performance needle, organizations can attain a higher state of production excellence.

Identifying High-payoff Activities

What exactly is a high-payoff activity (HPA)? HPAs are activities that directly improve processes, enhance best practices, and evolve the production facility to the next level of production excellence.

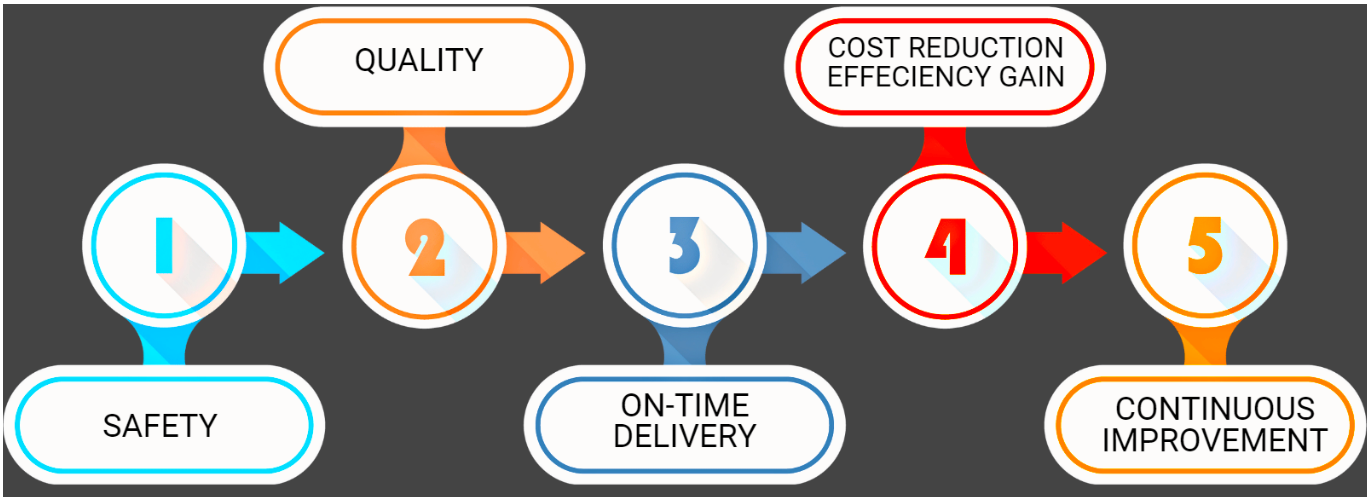
HPAs fall into five categories. They are:

- Safety
- Quality
- On-time delivery
- Cost reduction/efficiency gain
- Continuous improvement

When deployed, these five types of HPA can improve production performance.

Safety

Safety should be the top priority for every PCB manufacturer. Protecting employees on the production floor is not only the right thing to do, but it is also a business best practice. Employees are happier and more productive working in a safe workspace, using safe processes, and operating throughout the day under an umbrella of safety.



Engineering controls to error-proof (Poka-Yoke) safety standards can be utilized to attain a higher level of safety and decrease the probabilities of injuries. Common examples of error-proofing in manufacturing include:

- Interlock door switches on machine doors, so the machine will not run with the door open
- Using a self-retractable box knife which automatically retracts the knife when not in use
- Fixing uneven production floors to avoid slips, trips, and falls

Chemical processes play a big role in PCB manufacturing, and establishing exceptional standards for proper use of personal protective equipment (PPE) can improve the safety of a working environment. Goggles, face shields, aprons, gloves, and chemical boots are common forms of PPE used on the production floor. When choosing PPE equipment, manufacturers should make sure it is impermeable to the chemicals being used.

Ongoing training of both new and current employees is perhaps the most important aspect of ensuring safety on the production floor. Training should be diverse and comprehensive, covering everything from proper lifting techniques to electrical safety.

Quality

Quality is about more than quality products. It should be top of mind with respect to every aspect of production. The quality cycle really begins with process development and evaluation. Manufacturers need to scrutinize production output data in search of process variances. This will help to determine where in the manufacturing cycle process that quality output meets or exceeds expectations.

There are several tools available to control production quality. They include:

- DMAIC (Define, Measure, Analyze, Improve, and Control) refers to a five-step, data-driven improvement cycle aimed at improving, optimizing, and stabilizing operational processes
- SPC (Statistical Process Control) is a quality control method used by manufacturers to maximize efficiency and minimize waste
- FMEA (Failure Mode and Effects Analysis) can be employed to create a known production quality output

Quality does not begin and end with the production process. It should be embedded in the organization's culture and a priority for everything from the overall customer experience to individual elements of employee relations such as training or incentive programs.

On-Time Delivery

Quality products do not arrive late. The first thing customers ask during the quote process is often, "When can I expect delivery?" PCB manufacturers are often among the most mission-critical supply chain components for producers of electronic products. This means manufacturers must understand with great accuracy every component of their production process before committing to a shipping due date.

Every manufacturer strives for perfection, 100% on-time, all the time. If an organization is not at 100%, improvement is possible. To ensure on-time delivery, manufacturers should focus on the following:

- Process the order correctly the first time. Customers won't wait for a supplier to build an order twice.
- Know the capacity limitations of the facility. Correlate capacity level to the volume and timing of customer orders, paying particular attention to where the largest constraints are located in the production process.
- Continuously learn to overcome processing challenges using root cause analysis. This will make order processing less variable, more predictive, and less constrained.

Cost Reduction/Efficiency Gain

Efficiency gains and cost reductions sound like the same thing, but they are more like two sides of the same coin.

- Cost reduction is measured by use of fewer consumables
- Efficiency gains are the result of utilizing less labor

To improve at both, manufacturers need meaningful, granular metrics in place to measure how much of each is being used.

The metrics for raw materials consumed during production should be in the purview of the accounting and finance team. They can set metrics for the volume of consumables used during production. How those materials are used on the production floor requires analysis of manufacturing processes. Once an organization has a clear vision of how and how much of a raw material is being used, then strategies and best practices for cost reduction can be put into place.

The process for making efficiency gains is similar to material cost reduction initiatives. It is essential to understand labor costs at a granular level, including how process and technology impact operational efficiency down to the individual level. When a facility learns how to increase output at the per person level, the opportunities for improvement increase substantially.

Continuous Improvement

Continuous improvement is arguably the most important of the five HPAs.

Repeatedly challenging the status quo with an eye toward improvement will keep production excellence at its highest level. Manufacturers who adopt continuous improvement programs and implement quality management systems (QMS) will realize improvement efforts across the production floor.

Common elements of successful continuous improvement programs include:

- Utilize Lean Six Sigma, 5S, and 5-Why root cause analysis.
- Perform daily process walks, known as Gemba in the Lean manufacturing philosophy. These provide daily observation of processes occurring in real time and can uncover resource gaps.
- Plan-Do-Check-Act (PCDA) is a method to both continually improve and measure results—setting the stage for solid countermeasures that will prevent reoccurrence of an observed weakness or gap in the process.

Putting the five HPAs into practice on a daily basis into any production process will help keep any production team laser-focused on achieving production excellence through the continual improvement cycle. **DESIGN007**

Guest columnist **Kevin Beattie** is a quality assurance manager at Sunstone Circuits.

To read past columns from Matt Stevenson, click here.

Download *The Printed Circuit Designer's Guide to... Designing for Reality* by Matt Stevenson.

Optimizing Thermal Management for Wireless Communication Systems

Lightning Speed Laminates

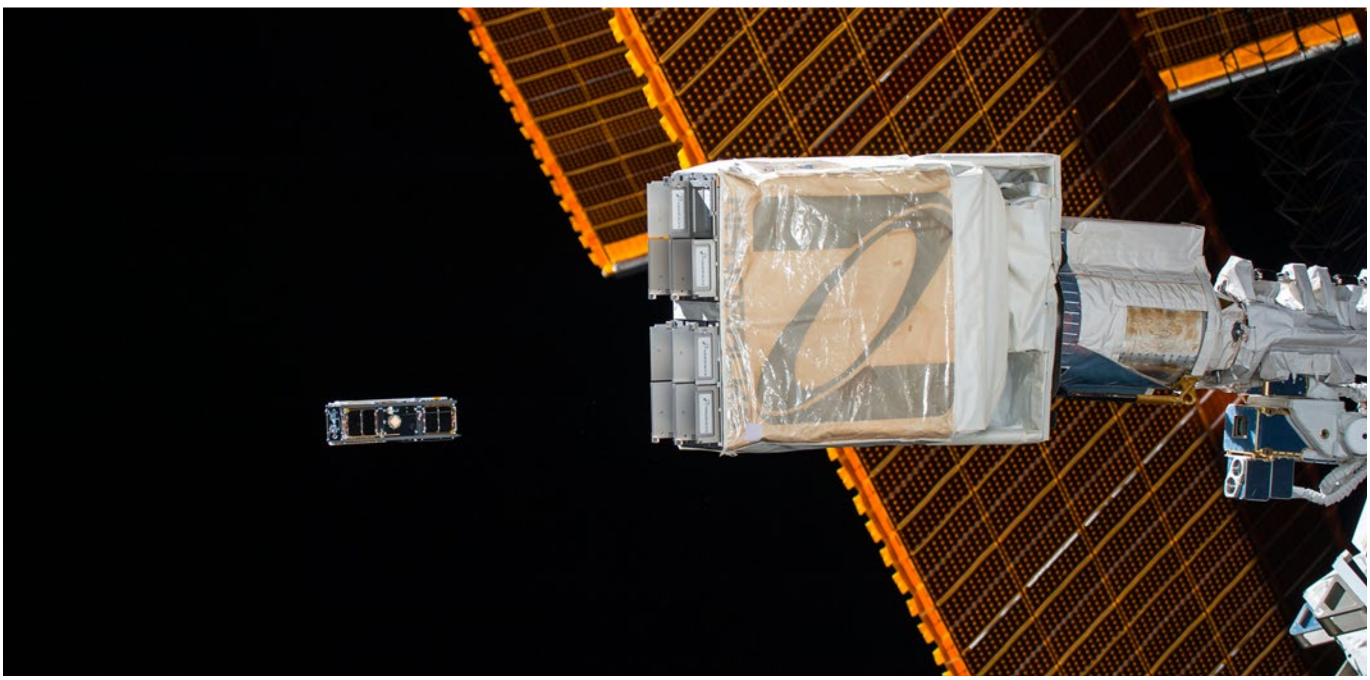
Feature Column by John Coonrod, ROGERS CORPORATION

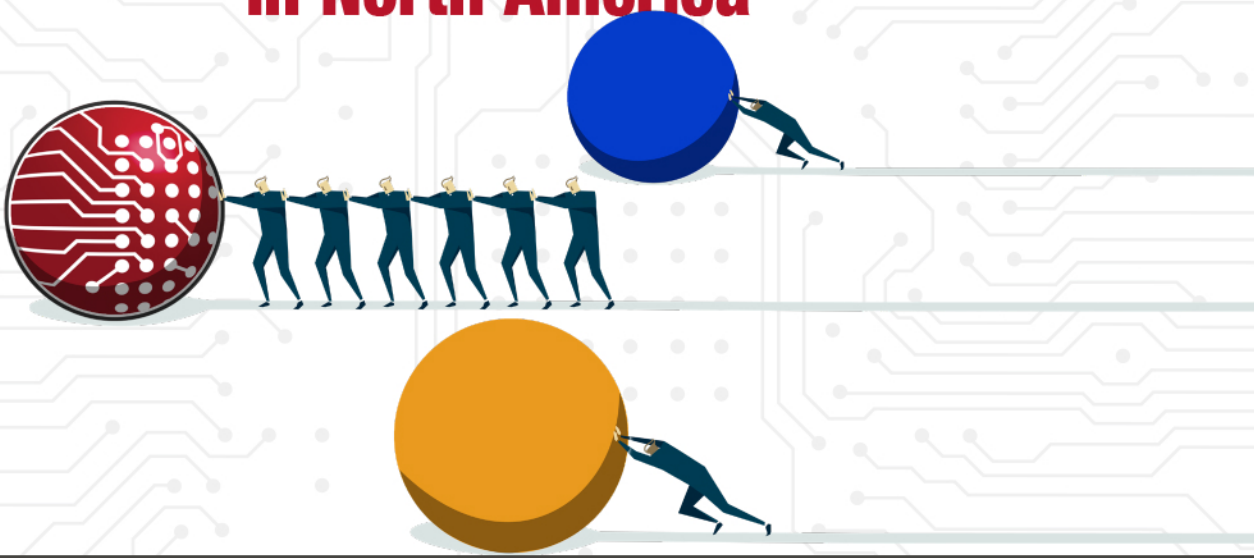
The term wireless communication has been around for many years, and it can mean many different things. The wireless communication between your mouse and your computer is very different than the wireless communication between a satellite and its ground station. The PCBs which are used for wireless communications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

Depending on the wireless system, the requirements for the PCB can be diverse. Even within a system, the different modules or components can have very different requirements. A good example of a complex wireless communication system would be an application for a LEO (Low Earth Orbit) satellite system. The ground station will have different requirements than the space-based station. Two major differences between these two components are power level and power management.

The satellite will be designed to be very efficient for the use of electric power, due to the normal limits of operating in space. The ground station is typically less concerned with power management and can offer much higher power levels than the satellite, although with higher power levels will usually come tradeoffs between thermal management and system performance.

The satellite systems will have concerns with thermal management, but often different than the concerns for thermal management of the ground station electronics. The thermal management issues for the ground station PCBs are





usually focused around using high-frequency circuit materials with low loss, high thermal conductivity, low CTE (coefficient of thermal expansion), and low TCDk (thermal coefficient of Dk). The PCB design and fabrication are also considered for thermal management concerns and, when practical, design features like via farms or via fences are included.

Most dielectrics used for high-frequency circuit materials have poor thermal conductivity when compared to the excellent thermal conductivity of metals. As a simple comparison, the thermal conductivity of copper is about 400 W/m·K and most PCB dielectric materials have a thermal conductivity around 0.3 W/m·K. Because of the big difference in thermal conductivity, some PCBs with thermal management challenges will have cavities built into them, which will minimize the amount of dielectric material in the heat flow path. The heat flow path originates from the heat source that resides on the PCB or a RF trace on the circuit, and the migration of that heat to a heat sink that will absorb the heat.

Most dielectrics used for high-frequency circuit materials have poor thermal conductivity when compared to the excellent thermal conductivity of metals.

Over many years of dealing with thermal management issues, the PCB industry has informally adopted a rule of thumb that a dielectric with thermal conductivity of 0.5 W/m·K or higher is considered good for thermal management concerns. Many of the ceramic-filled high-frequency laminates have this thermal conductivity value or higher but there are a few special materials with significantly higher thermal conductivity.

RF designers must consider the different properties of the high-frequency circuit materials used in the board. For thermal management issues, a thicker substrate will increase the heat flow path, and that is not desirable. However, a thicker laminate is often desired when operating at lower microwave frequencies because there will be less insertion loss. This assumes the use of a low-loss material, with a low dissipation factor. Insertion loss is directly related to heat generation due to RF power heating the circuit, and higher insertion loss will cause more heat to be generated. A thin circuit will have a shorter heat flow path and that is desired for good thermal management. However, a circuit using a thinner substrate will have higher insertion loss and more heated generated from the applied RF power.

The tradeoffs for thermal conductivity are described here, but an application using a thick substrate should consider a material with high thermal conductivity and low dissipation factor. Of course, an application using a thin substrate will also benefit from these properties, but typically more attention is put on minimizing insertion loss so there is less heat generated. In minimizing insertion loss for a circuit based on a thin substrate, copper surface roughness is often a consideration.

Copper surface roughness at the substrate-copper interface can have significant impact on insertion loss and that is especially true for circuits based on thinner substrates. A rougher copper surface will increase conductor loss and conductor loss is usually a large portion of the overall insertion loss for a circuit based on thin material. Basically, when the copper planes are close together, which is the case for a circuit using a thin substrate, the effects of the copper surfaces will be more impactful on RF performance. The effects on RF performance are related to phase angle, wave velocity, effective Dk and, as already stated, insertion loss.

An example of material formulated specifically for thermal management concerns is a laminate with a thermal conductivity value of 1.24 W/m·K, considered very good for this property, designed for high-frequency circuit materials. Additionally, the laminate has a low dissipation factor of 0.0017 and is available with a very low-profile copper offering a smooth copper surface. Another property not previously mentioned but that can be significant for thermal management, is moisture absorption. The moisture absorption for this laminate is extremely low at 0.05% and that will greatly minimize the undesired effects that moisture can have on RF performance.

There are several different test methods which can be used to determine the thermal conductivity of a substrate. We always use a test method that does not include the effects of copper, so the stated thermal conductivity of the laminate is that of the substrate only. There are other suppliers of high-frequency laminates, which will include the effects of copper in their testing for thermal conductivity and that will cause the property to appear much better than is actually true of the substrate. For this reason and many others mentioned, it is always good to contact your material supplier when working on a new design that is sensitive to thermal issues. DESIGN007

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

Flexibility, Communication Help Prevent 'Nightmare' Boards

by Barry Matties

I recently spoke with American Standard Circuits CEO Anaya Vardya about ways to bridge the gap between designers and fabricators, and the need to stay flexible and not be locked into designing boards in one certain way.

What issue should designers be most concerned about in circuit board fabrication today?

The thing we preach to our customers is this: When you start to design difficult or complex boards, form a partnership with your board shop. Really partner with them, and work with them through the design phase. You can do a lot of things on a computer screen, but can you take that and translate it into something that's manufacturable? Left to their own devices, many times the design ends up being unmanufacturable, or it could be very expensive to manufacture.

We've built designs that are just perfect nightmares to build, and customers will pay a lot of money to get those boards because very few fabricators will even tackle them. In the end, they could have probably simplified their design along the way, but they are somehow locked into a design and now they can't change it.

Do you see more designers engaging in that?

We do have a lot of designers who come to us ahead of time with very complex boards that we spend quite a bit of time on. We also have cases where we've had people fly down just to do in-person meetings on particularly difficult designs.

Thanks for your time, Anaya.

Thank you, Barry.

DFM 101
Data Package Guidelines

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Introduction

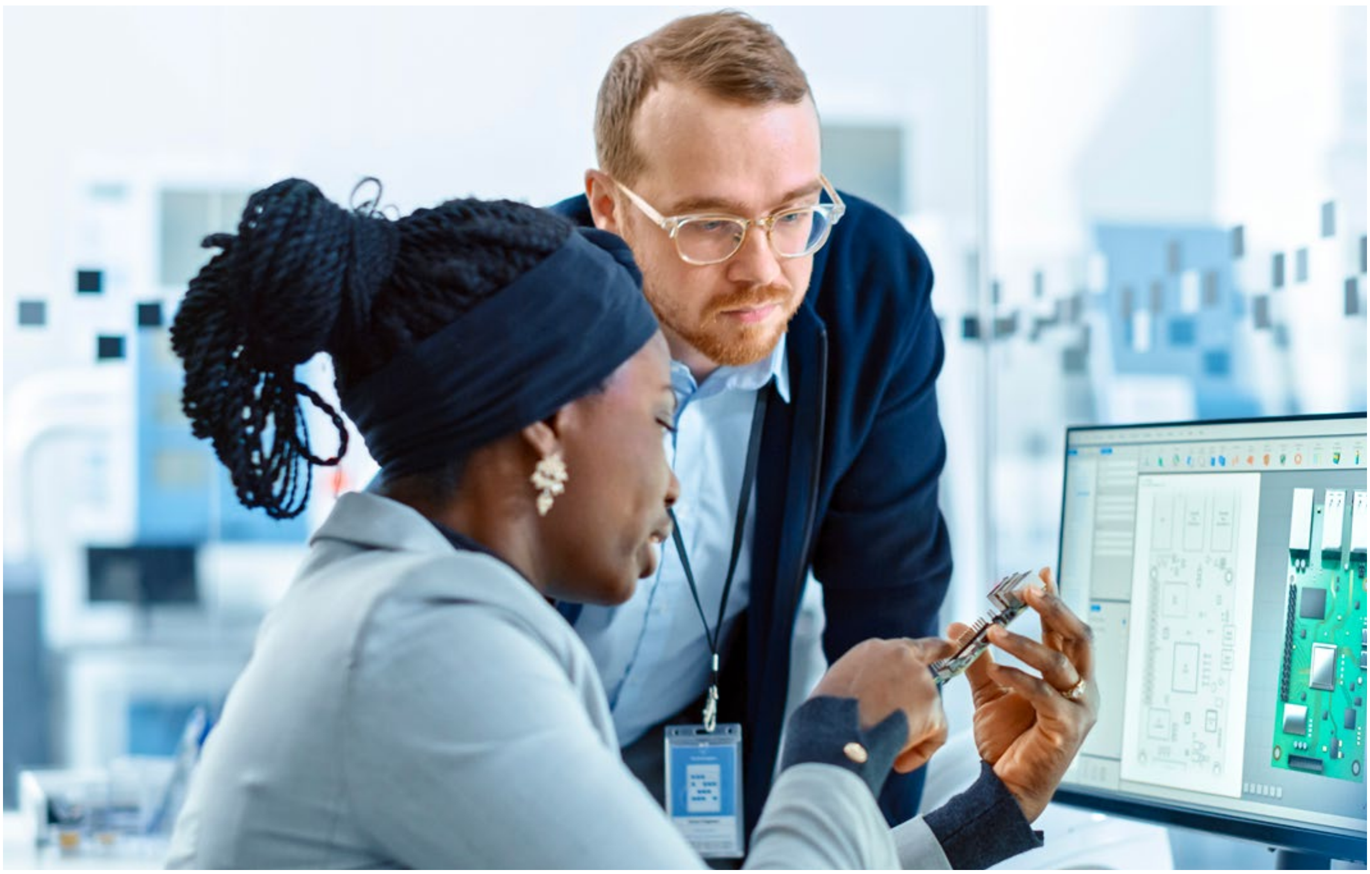
One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. A critical cost driver that often goes unrecognized is the data package that is sent to the PCB fabricator. Costs associated with data packages include engineering delays, data translation issues, and reliability concerns.

Data Formats

There are many common data formats in use, depending on the design provider's software and preferences. PCB fabricators need to be able to work with all of the formats; however, the one that is overwhelmingly preferred is ODB++ because it is the purest format between the designer and the fabricator.

Common data formats:

- ODB++
- 247D Gerber
- 274 Gerber
- IPCD356 Net List
- Mentor Neutered Netlist
- HPGL
- DXF
- DWG
- NC Drill
- Excellon



Part Drawing/Data Package Requirements

Drawing media includes paper, Gerber, HPGL, PDF, Film Plot, and AutoCAD.

Note: The "drawing" can consist of a "drilling drawing" and detailed "readme" file.

The drawing/data package should include:

- Profile dimensions and tolerance for the board
- A hole-to-board edge dimension in two axes
- List of finished hole sizes and hole size tolerance
- Material type, thickness, and finished copper weight
- Applicable acceptability specifications as required
- Solder mask color, type, and number of sides
- Surface finish: SMOBC, white tin, electroless nickel/gold, etc.
- Silk screen/legend color and number of sides

Array Drawings

Drawing media includes paper, Gerber, HPGL, PDF, Film Plot, AutoCAD.

If the board is to be supplied in array form, the array drawing shall have the following:

- Profile dimensions and tolerance for array
- Array edge to board edge/or hole in 2 axes and orientation of pieces in the array
- Dimensional location, size and tolerance of array tooling holes
- Dimensional location and size of array fiducials on copper and mask layer
- Breakaway slot detail and holding tab locations or scoring cross-sectional detail
- Number of "X outs" allowed per array

Artwork

Common media include Gerber RS274X, Gerber, film.

Gerber RS274X is the preferred format because it has embedded apertures and formatting within its file, which reduces translation/interpretation errors. Standard Gerber files require a separate aperture list. Film plots shall be used as received.

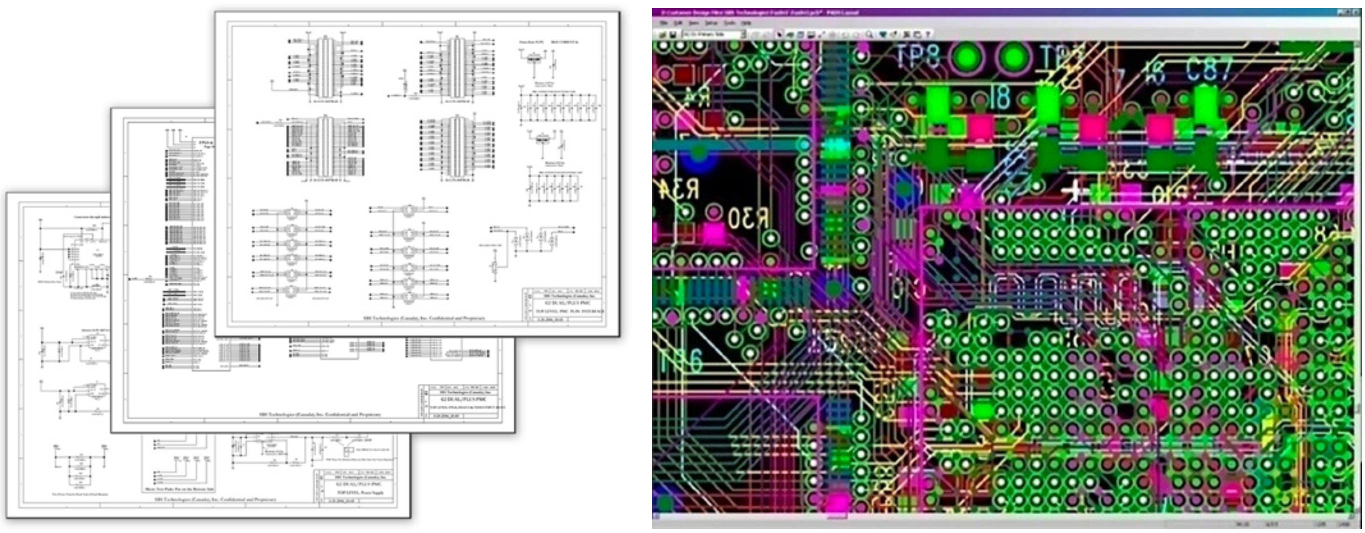
Drill Data

Common media is Excellon format ASCII data.

Paper coordinate lists and "bomb plotting" are to be avoided.

Artwork Approval

Customer artwork approval is strongly encouraged to ensure proper interpretation of



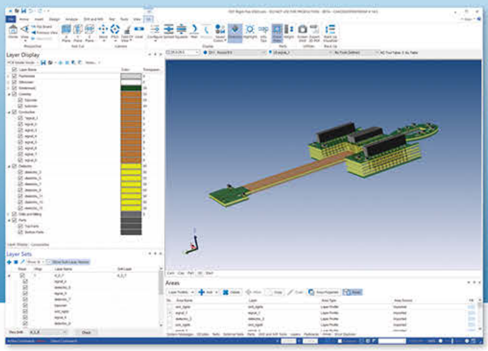
Part drawings and data example.

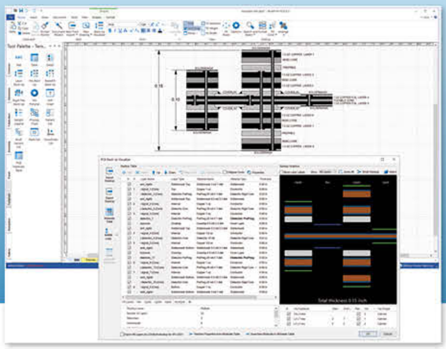
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

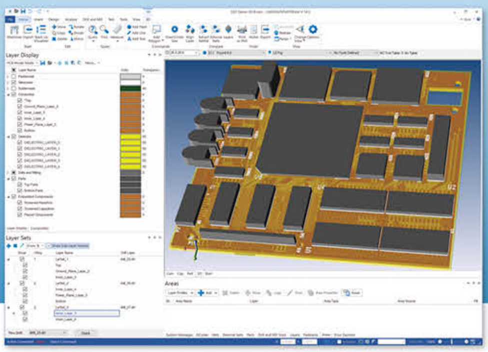
BluePrint-PCB®

CAM350®

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







DownStream Technologies

For more information visit downstreamtech.com or call (508) 970-0670 data received. It is not intended as an approval of line widths, pad sizes, etc. The intent is to ensure that all layers are present, requested merges are properly interpreted, acceptable logo and date code location, etc. This approval does not exempt the fabricator from responsibility for properly processing the data but provides a check and balance of the working data.

Common Data Issues

Two types of data issues are frequently encountered. Minimization of these issues will decrease tool generation time. The following lists present the most common errors:

CRITICAL: Can be substantial enough to stop the tooling process.

- Bad compression (cannot unzip)
- Missing FAB prints of files
- Formats not recognized
- FAB prints are not legible
- No README.TXT file
- Information does not match
- Missing Gerber files
- Outside ASC manufacturing capabilities
- Missing drill files
- Missing aperture list
- Missing "D" codes
- Unclear aperture list
- Revisions do not match

NON-CRITICAL: Simply result in additional edits that need to be performed.

- Netlist not supplied
- No README.TXT
- Design, as supplied, violates the OEM provided specifications/documentation
- Design, as supplied, violates the contract manufacturer provided specifications
- Violations on rules that improve yield

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.*

You know you're working for a great company when...

Source: I-Connect007

Design for Cost in Real-time

Article by Paul Carpine
SIEMENS EDA

When do you first start thinking about the selling price of your new product? It's impossible to avoid the question of what the price will be. The worst answer is: "I don't know, let's see after the first samples."

Let's say the product's selling price is $49; then the manufacturing cost will be $20. During each design change, the manufacturing cost should be recalculated to make sure the product is still competitive. This means you should involve people from both the development and cost calculation teams. This circle will be repeated until the final design is released, and the final calculation is done. Of course, if a small design change with "no impact for production" is not sent for evaluation, this is already a risk.

What if we had a software that was able to recalculate the price of the product in real-time during the design? What if we had something implemented into the design tool that was able to understand the change and recalculate the manufacturing price?

It is hard to find a product without a PCBA inside, and calculating the manufacturing cost for this component is not easy. It can be made in the beginning of the project based on several inputs, but what factors should you consider when you start the design? These same factors should be implemented inside this magic cost calculation software.

The cost calculation activity is irritating when your focus is hardware development and/or

layout design. How do you evaluate whether a small change will increase a PCBA's final cost? Here, I list the most important factors in the design that can impact the manufacturing cost.

The first way to keep the cost low is to use a surface mount device (SMD). Electronics manufacturing production is expensive, and each surface and machine should be utilized efficiently. Having only SMDs in your design means it is not necessary to use another line just for the through-hole devices (THDs). This will save time in production and will reduce the risk of quality issues. The SMDs are placed by the pick-and-place machine and soldered in the reflow oven, but for THDs, the process is very time consuming.

The SMDs are placed by the pick-and-place machine and soldered in the reflow oven, but for THDs, the process is very time consuming.

If THDs cannot be avoided, make sure the components are designed for pin-in-paste and can be assembled on the SMT line. This will keep the manufacturing price low because it is not necessary to use special machines and all the components will be assembled on the same production line.

After you have managed to have just SMDs on your board, let's try to keep all the components on the same side. This will make the guys in production very happy, and you will get a discount. Having one single-sided population will reduce the production time and some tooling costs can be eliminated. If a one-side population is not possible due to product-specific reasons, make sure the number of components is balanced between top and bottom sides. Having the same cycle time for both sides will reduce the manufacturing cost.

Another idea in "design for cost" is to reduce the PCB surface because the board is expensive. You should increase the component density and avoid making the PCB with the same dimension as the product. Don't fill the empty space inside the product with the PCB. If the product is a sphere with a circular base, add a square PCB inside. A complex PCB outline will increase the cost: rectangular PCBs are preferred by manufacturers because they can better optimize the panel usage. PCBs are not assembled as singles, but as part of a panel with multiple PCBs. Those panels can be better optimized if the PCB outline is a rectangle, and less material will be wasted. Better to keep the nice design for the outside part of the product and place rectangular shapes inside.

The bill of materials (BOM) cost is influenced by the component types and quantity. The next cost reduction factor is reducing the number of unique component types. Instead of using 50 different resistor types, it's better to add 60 resistors from just 10 unique part numbers. This action will reduce both the inventory cost and quality issues during assembly.

Exotic components are expensive and hard to find. During the design phase, it is recommended to check component availability and to adapt the schematic if one component is not available. There is no reason to push your supplier to find a non-standard component. It's not just that delivery time will be high, but the BOM price will increase dramatically.

Having a cost calculation feature as part of the design tool will give you the flexibility to make quick decisions. A small design change will be evaluated in real-time, and the designers can decide what to do next.

This tool should cover component price, PCB fabrication cost, and assembly costs. Of course, some settings are necessary and variable inputs will be collected from the design tool. To evaluate component cost, the system will be connected to a real-time component availability platform. After displaying the available components, the cost is added to the total product cost. This is updated after each component change or when the quantity is modified.

The PCB dimension and outline will influence the fabrication cost and panel design. Each PCB fabricator works with big FR-4 production panels and the goal is to utilize the entire panel, otherwise the waste is reflected in the PCB cost. If we can set the production panel inside the calculation tool, it is possible to simulate utilization and therefore, estimate the PCB cost. It is enough to see how the PCB price will change by modifying the dimensions. Sometimes, if the PCB surface increases by 5%, the price stays the same, but 5.5% could have a big impact because the fabricator cannot optimize the panel usage as well.

In the assembly phase, some costs can be easily estimated. Moving one component from one side to another will change the manufacturing cost and this should be visible in our new calculation tool, letting the designer decide what's best for the project. A decision to use an SMD connector compared with a pin-in-paste, press fit, or THD will be evaluated in real-time considering all the aspects of the manufacturing.

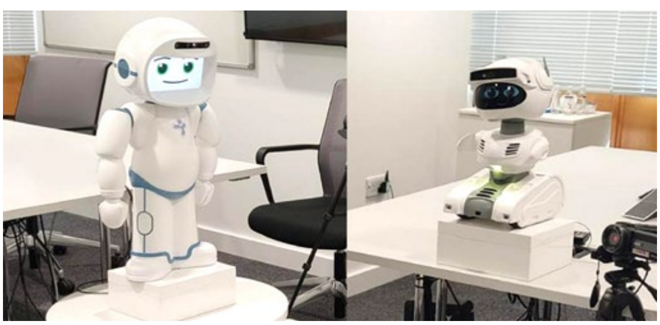
This calculation tool is partially available, but is not integrated in the design software and divided in BOM or manufacturing costs. Since the information is available, it will just be a matter of time until the function will be available and the manufacturing cost calculation becomes a transparent process. DESIGN007

Paul Carpine is a technical marketing engineer at Siemens EDA. Supporting the PCBFlow from Siemens, Paul works with PCB designers to improve their projects for more efficient manufacturing. He started his career as an SMT process engineer at Flextronics more than 15 years ago, and for the past 12 years has worked for important players in the automotive industry.

Robots Can Improve Mental Wellbeing at Work, If They Look Right

Robots can be useful as mental wellbeing coaches in the workplace, but perception of their effectiveness depends in large part on what the robot looks like.

Researchers from the University of Cambridge conducted a study in a tech consultancy firm using two robot wellbeing coaches, where 26 employees participated in weekly robot-led wellbeing sessions for four weeks. Although the robots had identical voices, facial expressions, and scripts for the sessions, the physical appearance of the robot affected how participants interacted with it.



Participants who did their wellbeing exercises with a toy-like robot said that they felt more of a connection with their "coach" than participants who worked with a humanoid-like robot. Participants who worked with the toy-like Misty robot reported that they had a better working connection with the robot than participants who worked with the child-like QT robot. Participants also had a more positive perception of Misty overall.

"It could be that since the Misty robot is more toy-like, it matched their expectations," said Spitale. "But since QT is more humanoid, they expected it to behave like a human, which may be why participants who worked with QT were slightly underwhelmed."

(Source: University of Cambridge)

Flex007 Highlights

Trackwise's Flex Technologies Inspire Diverse New Applications ▸

Trackwise Designs plc has revealed that its length-unlimited flexible printed circuits (FPCs) are driving product innovation in many sectors, as design engineers apply the Trackwise-patented Improved Harness Technology (IHT) FPC technology to solve their design challenges and enable smarter and more advanced products.

FLEX Conference 2023 to Spotlight Flexible Hybrid Electronics, PE, Advanced Packaging ▸

The FLEX Conference will gather industry experts July 11-13, 2023, at the Moscone Center in San Francisco for keynotes, panel discussions, technical sessions and product-based demonstrations highlighting the latest innovations in flexible hybrid electronics (FHE), printed electronics and advanced packaging including heterogeneous integration.

Asia/Pacific Wearable Device Market Slowed to 1.8% Growth in 2022 ▸

According to the International Data Corporation (IDC) Quarterly Wearable Device Tracker, wearable device shipments in the Asia/Pacific (excluding Japan) region slightly increased by 1.8% in 2022 to 259.1 million units, recording its lowest annual growth ever.

Nick Koop: A Commitment to Lifelong Learning ▸

Nick Koop, director of flex technology at TTM Technologies, reflects on the powerful impact the electronics manufacturing industry has had on the world, as well as the many learning opportunities offered by actively participating in IPC. Nick urges aspiring young professionals to reach out and create the lasting connections that will guide them throughout their careers.

NextFlex Launches $4.4M Hybrid Electronics Funding Opportunity ▸

NextFlex, America's Flexible Hybrid Electronics (FHE) Manufacturing Institute, released Project Call 8.0 (PC 8.0), the latest call for proposals that seek to fund projects that further the development and adoption of FHE while addressing key challenges in advanced manufacturing that support Department of Defense priorities.

Insulectro and LCOA Install R&D Lab With Partner Kyocera in Orange County ▸

Insulectro, the largest distributor of materials for use in the manufacture of printed circuit boards and printed electronics, has opened a testing and development laboratory for Kyocera tools in its Lake Forest, California headquarters. The lab was created in association with backup and entry materials manufacturer LCOA in that company's plant.

A Quantic Leap into Foils and Embeddeds With John Andresakis ▸

Andy Shaughnessy talks with John Andresakis about how the merger of resistive foil technologies from Ohmega and Ticer has evolved under the new ownership of Quantic. Andresakis also shares how these materials are finding new applications, especially in the embedded component application space, as the company reaches out to the new generation of PCB designers and design engineers.

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

Article by Mark Gallant
DOWNSTREAM TECHNOLOGIES

Introduction

Flex and rigid-flex PCB constructions are not new concepts. It has become commonplace as engineers look for alternative circuit packaging for ever-shrinking electronic products. A flat, one-sheet schematic for a straight ribbon cable is analogous to its physical flat substrate. A flat, multi-sheet schematic that details circuitry for a rigid-flex design bears little visual resemblance to its three-dimensional, variable material rigid-flex assembly. However, in both schematic examples, schematic-based analysis tools are applied equally. This same truth also applies to common FR-4-based two-layer or multilayer PCBs.

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.

Signal integrity analysis such as impedance, coupling, crosstalk and noise is complicated by variable stackups across flexible designs. A single transmission line can be stripline in a rigid-flex area and microstrip in a flex area. Material types and dielectric constants above or below a trace as it traverses a design also vary. While the challenges for signal integrity analysis for flexible designs is worthy of conversation, this article will focus on the current challenges to design-for-manufacture analysis of flex and rigid-flex designs.

Companionship at its Best
THE SOLUTIONS GUIDE TO...
Thermal Management
ventec INTERNATIONAL GROUP
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

Contrasting Rigid PCBs With Flex and Rigid-flex

Some designers design flexible PCBs as simple bendable circuit boards, but there are vast differences between rigid and flexible. Both technologies produce an electrical interconnect function, but are manufactured using different types of materials and processes. They also have varying applications. No need to design a rigid-flexible PCB for the motherboard of a desktop PC, but rigid-flex is required for most medically implanted devices.

A typical rigid PCB is comprised of electrodeposited copper-clad fiberglass substrates bonded together. While there are variations on materials used to bond substrates, it is commonly sheets of cloth pre-impregnated with uncured epoxy. This bonding material composition is not engineered to be flexible. The copper is chemically etched to create a circuit pattern. The hardness of the bonded substrates requires mechanical routing to trim the raw PCBs. All layers of the PCB are commonly identical in size and shape unless cavities, embedded components, or other such exotic construction is present. The rigid PCB layer stackup is identical across the entire PCB area. Solder mask and legend are almost always applied.

Flexible PCBs are comprised of rolled annealed copper over flexible polyimide substrates. Flexible layers or cores are produced with or without adhesives. Adhesiveless flex is prevalent in applications requiring higher performance, while those with adhesives are often found in low layer-count applications. The most common usage is copper foil laminated to a substrate with epoxy or acrylic adhesive. Both substrate material and adhesive are engineered for bending to minimize trace fracture. Like rigid PCBs, a chemical etching process is used to create a circuit pattern. The flexible nature of the materials requires die cutting or "blanking" rather than mechanical routing. Each layer of a flexible double-sided core has an identical shape. However, multiple-layer flex is likely to have variations in shape for each layer or core. Flexible PCBs require a thin film insulator over the conductors know as a coverlay. Unlike rigid PCB solder masks, coverlays are die cut much like the flexible layers they insulate. The stackup of a multi-layer flexible PCB can vary across the PCB area. This is especially true with multilayer flex where layer shape varies among the collection of layers or cores. A flexible ground or power plane area is typically crosshatched versus solid for rigid PCBs. The crosshatch reduces potential for fracture of conductors. Alternatively, flexible layers can be shielded with a layer of copper or silver foil. Masking and screening over flexible layers is not rare, but uncommon.

Mark Gallant

Rigid-flex PCBs are obviously a combination of rigid and flexible materials. Rigid-flex is, in essence, a hybrid PCB combination of materials and processes from both rigid and flexible PCBs. The two material types are generally processed separately and bonded together later in the fabrication process. The layer

stackup commonly varies greatly across the entire PCB. There may be areas of rigid-flex, flex only, various combinations of rigid and flex layer count, and so on.

There is also rigidized flex where blank FR-4 or other rigid materials are selectively bonded to flexible substrates to provide stiffness. The rigid stiffener material rarely has conductors present.

Rigid vs. Flex and Rigid-flex

Rigid PCBs are a foundational technology in today's electronic products. Rigid PCBs offer mechanical integrity, electrical conductivity and reliability, but are limited by their two-dimensional profile. Their flatness limits designers to two dimensions which severely limits design flexibility, especially as electronic devices decrease in size. Flexible PCBs are bent to take advantage of a three-dimensional space, while also accommodating components. Flexible PCBs enable maximal utilization of space to package electronics but at a premium cost compared to conventional PCBs.

Rigid PCBs offer mechanical integrity, electrical conductivity and reliability, but are limited by their two-dimensional profile.

Rigid and flexible PCBs are present in many electronic products. However, some applications benefit more from one type of circuit board. Rigid PCBs make sense for products such as televisions, desktop PCs, Blu-ray players, and other larger electronic products. Flexible PCBs are present in smartphones, smart watches, tablets, cameras, printers, and laptops. They are a fundamental requirement for implanted miniature medical devices such as pacemakers, cochlear implants, and implanted defibrillators. Complex multi-PCB assemblies interconnected with wires or cabling are often redesigned with rigid-flex PCBs to improve reliability and reduce weight and space. This is the catalyst for many military and aerospace products being designed with rigid-flex. One example is a single-use smart bullet that can change its trajectory if its intended target moves.

The introduction of small-outline and surface mounted semiconductors ushered in a revolution of miniature repackaging. Think Sony Walkman vs. a typical boom box. For years, flexible PCBs were relegated exclusively to replacement of multi-wire cables. Who would not recall the presence of a flexible flat cable connected to the head of a dot matrix or impact printer. The head would bob back and forth across the paper while the cable dynamically flexed and provided a more reliable interconnect between printer head and motherboard.

The introduction of rigid-flex was not quite the same level of game-changer as surface mounted packages due to its somewhat limited application and cost differential. We shouldn't expect a new collection of desktop PCs to be designed with rigid-flex motherboards as a means to reduce cost. However, miniaturized and reliable technology such as pill cameras, foldable cellphones or implanted medical devices would not exist without rigid-flex technology.

We'll pick this up this discussion with Part 2 in the next issue of *Design007 Magazine*. **DESIGN007**

Mark Gallant is a senior product marketing manager for Downstream Technologies.

Catalyzing Change and Design Evolution

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

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.

Today, the so-called metaverse seeks to blur the lines between reality and fantasy. Things that were pure speculation and pipe dreams in the late 1960s, such as the *Star Trek* communicators, are now physically manifest in our daily lives. The source of these developments has been the unfettered imagination of countless visionaries within the electronics industry.

The Desire for Change

Change, or the search for and openness to it, will arguably be a cornerstone of the electronics industry. It is the fuel that runs the global economic engine; as economists have observed, this growth is predicated on change and is based on the axiom that success of an economy is founded on the notion that an individual's wants must exceed their needs. If we collectively lose our appetites for change, everything grinds to a halt.

The electronics industry is continuously being pressured to develop newer and better products with more functions and at lower

cost. One might think that this is the result of consumer demands; however, Steve Jobs once made a great observation:

"Some people say, 'Give the customers what they want.' But that's not my approach. Our job is to figure out what they're going to want before they do. I think Henry Ford once said, 'If I'd asked customers what they wanted, they would have told me, a faster horse.' People don't know what they want until you show it to them. That's why I never rely on market research. Our task is to read things that are not yet on the page."

It is difficult to argue with Jobs' observation because it has turned out to be largely accurate. Customers are not the product visionaries; that is not their role, but they do know what they like and want when they see it. The desire for change is normally sparked by something outside of us rather than something from within—though it is likely that each of has said to ourselves at some point: "I wish somebody would invent a..."

Customers are not the product visionaries; that is not their role, but they do know what they like and want when they see it.

In recent years, it is our current dissatisfactions that most often initiate a move to change. It is sparked more with the product designer or design team than by the consumer. There is an almost biological aspect to the growth and evolution of electronic products. There is also, of course, a rational force driving the development and introduction of every new electronic product but at a certain point, electronic products seem to take on a life of their own.

Moreover, in their development and growth (i.e., change), there is a Darwinian-like quality to the process. Electronic products that adapt quickest and most readily to the winds of change are able to thrive; those that don't are pushed back to wither and die. Adaptation is key to survival and the synergistic (or symbiotic, if you wish) linking of adaptive technologies that offer obvious and beneficial potential and a prospective path to securing such benefit. It is not difficult to assume that evolution is often preceded and influenced by some moment of inspiration.

The Most Adaptive of All

Unquestionably, flexible circuits are among the most adaptive and adaptable of all electronic interconnection technologies and perhaps the most inspirational as well. Their adaptability has not been overlooked by keen product developers, from assemblers to packagers. Over the last couple of decades, the range of applications for flexible circuits has grown at an impressive rate as the technology has been adapted to a host of new interconnection opportunities.

The historical roles of flex circuits, such as wire harness replacement, 3D interconnection enabler, and dynamic interconnection scheme to connect parts of an electronic assembly designed to move relative to one another. They continue to be exploited, but branching into new areas has accelerated with the creation of consortiums such as NextFlex and its rebranding of flexible circuits as "flexible electronics," "flex hybrid electronics," "stretchable circuits," and more importantly, creating an environment where suppliers of materials, processes, and equipment can come to explore, prototype, and demonstrate their dreams for the future.

Changes continue to roll forth with enabling technologies that support those dreams. This includes e-textiles to the materials mix, recently supported through IPC standards IPC-8921, IPC-8951, and IPC-8972. These cover the material requirements, design, and testing requirements needed to support the emerging technologies which are certain to spawn yet a new range of products that the consumer does not yet know they need to make their lives better, more interesting, and enjoyable as we begin to explore in greater depth the relatively new category of electronics called "wearables."

Inspiration's Role in Change

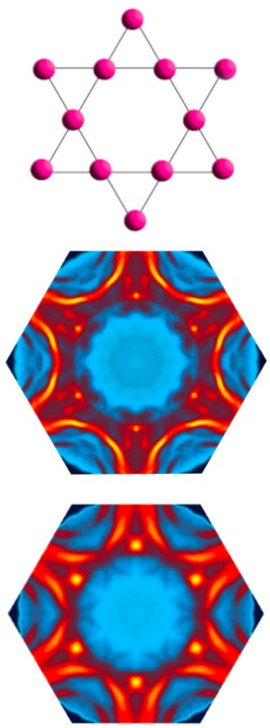
Again, this commentary has been about change—inspiring, enabling, adapting to, and mastering change. Inspiration, insight, or whatever you might want to call it, is vital in the change process. We can now see with ever greater clarity how the formerly sharp lines between fundamental elements of electronics assembly are blurring. The components and substrates, especially flexible substrates and even assembly technologies, are being used in a more coherent and cooperative way than ever before, as evidenced by the new era of heterogeneous integration which is upon us. The relative strengths of often very different technologies, especially flexible circuits, all adaptive and adaptable, are enabling us to get ever more value from our electronic products. Ironically and perhaps fittingly, the only thing that will never change is change. **DESIGN007**

Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, click here. Download your free copy of Fjelstad's book *Flexible Circuit Technology, 4th Edition,* and watch his in-depth workshop series "Flexible Circuit Technology."

Magnetism Fosters Unusual Electronic Order in Quantum Material

Physicists were surprised by the 2022 discovery that electrons in magnetic iron-germanium crystals could spontaneously and collectively organize their charges into a pattern featuring a standing wave. Magnetism also arises from the collective self-organization of electron spins into ordered patterns, and those patterns rarely coexist with the patterns that produce the standing wave of electrons physicists call a charge density wave.

In a study published in Nature Physics, Rice University physicists Ming Yi and Pengcheng Dai, and many of their collaborators from the 2022 study, present an array of experimental evidence that shows their charge density wave discovery was rarer still, a case where the magnetic and electronic orders don't simply coexist but are directly linked.



The iron-germanium materials are kagome lattice crystals, a much-studied family of materials featuring 2D arrangements of atoms reminiscent of the weave pattern in traditional Japanese kagome baskets, which features equilateral triangles that touch at the corners.

"Kagome materials have taken the quantum materials world by storm recently," Yi said.

Dai added, "When put onto kagome lattices, electrons can also appear in a state where they are stuck and cannot go anywhere due to quantum interference effects."

Dai said the findings confirmed the team's hypothesis that charge order and magnetic order are linked in iron-germanium. "This is one of the very few, if not of the only, known example of a kagome material where magnetism forms first, preparing the way for charges to line up," he said.

(Source: Rice University)

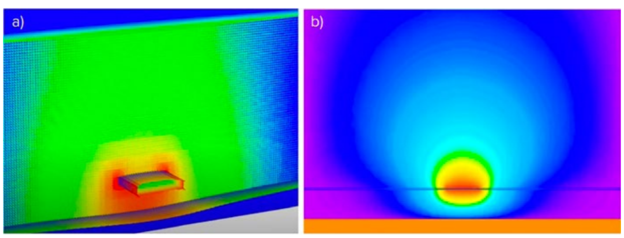
TOP TEN EDITOR'S PICKS

Design007 Survey: Slash Sheets and Material Selection

In this survey, we're asking our PCB designer readers to discuss their material selection processes and the use of references such as slash sheets during this process. The results will be published in upcoming issues of *Design007 Magazine*. From the survey responses we've seen so far, designers are split when it comes to slash sheets. What is your material selection process?

Beyond Design: The Interaction of Electromagnetic Fields

When two (or more) electromagnetic fields overlap or meet, they add vectorially at each point in space. Fields have direction and polarity. At any point in space, there can be only one field, so at some spatial points, they will cancel each other, and at others, they will re-enforce each other. James Clerk Maxwell described electromagnetic fields as being linear.

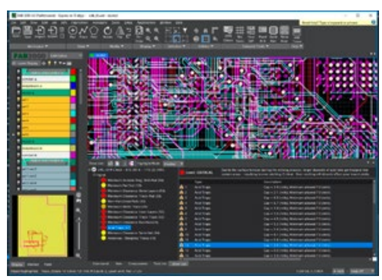


Tempo Automation to Acquire Optimum Design Associates

Tempo Automation Holdings, a leading software-accelerated electronics manufacturer, announced that it has entered into a definitive agreement to acquire Optimum Design Associates, Inc. and Optimum Design Associates Pty. Ltd., a fast-growing electronic design services company with offices in the United States and Australia that has delivered over 10,000 PCB designs to blue-chip customers.

Numerical Innovations Releases ACE 2D/3D Translator V8

Numerical Innovations (NI) announced its official release of ACE 2D/3D Translator Version 8.3.0 (64-bit). ACE 2D/3D Translator is a "Full-Featured" CAD Software solution that will easily convert between ALL common EDA and CAD formats: DXF, Gerber, GDS-II, DWG, Postscript, PDF, HPGL, NC Drill/Rout, OASIS, Image files, ODB++, IPC-2581, SVG, and 3D formats STEP, STL.



Runner-up Discusses IPC Design Competition

PCB designer Adam Thorvaldson of Innovex was a finalist in this year's IPC Design Competition at IPC APEX EXPO. He came in second place in this final heat. We asked Adam to share his thoughts on the competition, what it means to be one of the winners, and any ideas about improving the contest for 2024 in Anaheim.

Elementary, Mr. Watson: If Not You, Then Who?

If you haven't noticed recently, the PCB design industry is struggling. It is an understatement to say that we are facing a talent shortage. I constantly get phone calls regarding open positions and the need for more designers. At one time, there were designated PCB designers. But with the first economic downturn in 2008, talent shortages hit many companies. Next in line to fill the designer vacancies were electrical engineers (EE). But that came with inherent problems.

Optimizing Communication Between Fabricators and Designers

James Hofer

During DesignCon, I spoke with James Hofer from Accurate Circuit Engineering about some of his customers' biggest challenges. We discussed various ways to increase the level—and quality—of communication between designers and fabricators. James also offered some interesting observations about bridging the gap between designer and fabricator.

DFM 101: Final Finishes: OSP

One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. The next final finishes to discuss in this series is OSP. As with all surface finishes, there are pros and cons with the decision of which to use. It is a combination of application, cost, and the properties of the finish. OSP is RoHS-compliant as there is zero lead content in the finish.

Tim's Takeaways: Tribal Knowledge—Not the Villain You Thought

To put it simply, tribal knowledge is information or skills known by an individual or group that is not known outside that group. One of the trademarks of this definition is that it's commonly used to describe functional—but undocumented—knowledge essential to the operation of an organization.

True Experts Can Cite Their Sources

We've heard a lot lately about the need to identify tribal knowledge within our organizations. How do you know whether an "expert" is sharing documented knowledge? We asked IPC design instructor Kris Moyer to explain his process for separating the wheat from the chaff, so to speak, in design knowledge. As he points out, a true expert will not be afraid to cite the sources and data sets behind their arguments. Ask questions; maybe there is a reason why they "always did it this way."

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

Career Opportunities

Production Manager

Sunstone Circuits, a leading provider and manufacturer of custom printed circuit boards (PCBs) located in Mulino, Oregon (20 miles south of Portland), is searching for an experienced and talented manager to lead our Production team.

Responsibilities

Directing and coordinating activities of the manufacturing department in processing materials and products by performing the duties personally or through subordinate supervisors.

Sunstone is a fast-paced, custom electronics manufacturing environment that requires a "hands on" approach, extreme customer focus, and sense of urgency to ensure all products manufactured are built to quality standards and on-time product delivery. Successful candidate will perform these duties in conjunction with department leads, partners and suppliers.

Qualifications

Bachelor's degree (B.A.) from four-year college or university; or five to six years related manufacturing management experience and/or training; or equivalent combination of education and experience.

Competitive salary, bonus program, benefits, and robust wellness package. Offer of employment will be contingent upon passing drug screen, background check, and reference check.

Email resume to jobs@sunstone.com

apply now

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

Career Opportunities

The Test Connection, Inc.

Sr. Test Engineer (STE-MD)

The Test Connection, Inc. is a test engineering firm. We are family owned and operated with solid growth goals and strategies. We have an established workforce with seasoned professionals who are committed to meeting the demands of high-quality, low-cost and fast delivery.

TTCI is an Equal Opportunity Employer. We offer careers that include skills-based compensation. We are always looking for talented, experienced test engineers, test technicians, quote technicians, electronics interns, and front office staff to further our customer-oriented mission.

- Candidate would specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly Agilent & HP), Teradyne/GenRad, and Flying Probe test systems.
- Strong candidates will have more than five years of experience with in-circuit test equipment. Some experience with flying probe test equipment is preferred. A candidate would develop, and debug on our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Proficient working knowledge of Flash/ISP programming, MAC Address and Boundary Scan required. The candidate would also help support production testing implementing Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks. An understanding of stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

Contact us today to learn about the rewarding careers we are offering. Please email resumes with a short message describing your relevant experience and any questions to careers@ttci.com. Please, no phone calls.

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

Career Opportunities

Career Opportunities

pluritec®

MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

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

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

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

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

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

Must be able to travel extensively.

apply now

ventec

INTERNATIONAL GROUP

騰輝電子

European Product Manager

Taiyo Inks, Germany

We are looking for a European product manager to serve as the primary point of contact for product technical sales activities specifically for Taiyo Inks in Europe.

Duties include:
- Business development & sales growth in Europe
- Subject matter expert for Taiyo ink solutions
- Frequent travel to targeted strategic customers/ OEMs in Europe
- Technical support to customers to solve application issues
- Liaising with operational and supply chain teams to support customer service

Skills and abilities required:
- Extensive sales, product management, product application experience
- European citizenship (or authorization to work in Europe/Germany)
- Fluency in English language (spoken & written)
- Good written & verbal communications skills
- Printed circuit board industry experience an advantage
- Ability to work well both independently and as part of a team
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on offer:
- Salary & sales commission--competitive and commensurate with experience
- Pension and health insurance following satisfactory probation
- Company car or car allowance

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits. Please forward your resume to **jobs@ventec-europe.com.**

apply now

Career Opportunities

KOH YOUNG AMERICA

Technical Service & Applications Engineer

Full-Time — Midwest (WI, IL, MI)

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement-based inspection technology for electronics manufacturing. Located in Duluth, GA, Koh Young America has been serving its partners since 2010 and is expanding the team with an Applications Engineer to provide helpdesk support by delivering guidance on operation, maintenance, and programming remotely or on-site.

Responsibilities

- Provide support, preventive and corrective maintenance, process audits, and related services
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

- Bachelor's in a technical discipline, relevant Associate's, or equivalent vocational or military training
- Knowledge of electronics manufacturing, robotics, PCB assembly, and/or AI; 2-4 years of experience
- SPI/AOI programming, operation, and maintenance experience preferred
- 75% domestic and international travel (valid U.S. or Canadian passport, required)
- Able to work effectively and independently with minimal supervision
- Able to readily understand and interpret detailed documents, drawings, and specifications

Benefits

- Health/Dental/Vision/Life Insurance with no employee premium (including dependent coverage)
- 401K retirement plan
- Generous PTO and paid holidays

apply now

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

Career Opportunities

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

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

Career Opportunities

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

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

Summary of Functions

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

Essential Duties and Responsibilities

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

Organizational Relationship

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

Qualifications

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

Physical Demands

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

apply now

Career Opportunities

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

Educational Resources

I-007e

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

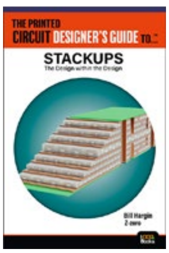
by Didier Mauve and Robert Art, Ventec International Group

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!

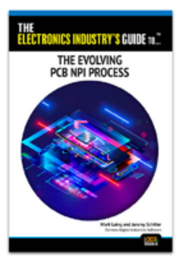


Stackups: The Design within the Design

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

ADVERTISING SALES: **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 © DAN TALSON**

DESIGN007 MAGAZINE

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

April 2023, Volume 12, Number 4
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX

Accurate Circuit Engineering.......... 35
All Flex Solutions.......... 59
Altium.......... 45
American Standard Circuits.......... 11
APCT.......... 49
Burkle North America.......... 25
Cadence Design Systems.......... 7
Candor Industries.......... 5
Downstream Technologies.......... 53, 55
Elite Materials.......... 31
EMA Design Automation.......... 13
Flexible Circuit Technologies.......... 39
I-Connect007 eBooks.......... 2, 3, 65
In-Circuit Design Pty Ltd.......... 27
IPC Community.......... 21
IPC.......... 67
NCAB Group.......... 17, 19
Polar Instruments.......... 71
Prototron Circuits.......... 43
Show & Tell Magazine.......... 69
Summit Interconnect.......... 29
Taiyo America.......... 63
U.S. Circuit.......... 23
Ventec International Group.......... 41