THE pcb design MAGAZINE
AN I-CONNECT007 PUBLICATION
June 2014

Designing Flex Circuits

Why Procuring Flexible Printed Circuitry is Different

by Thomas Stearns, page 12

THE pcb design MAGAZINE™
June 2014

This Issue: DESIGNING FLEX CIRCUITS

FEATURED CONTENT

Flexible circuits are becoming more popular in everyday electronics. But when it comes to design, rigid PCBs and flexible circuits are completely different animals. This month, our expert contributors Tom Stearns, Dave Becker, and Tom "Flexdude" Woznicki bring you the inside information behind designing flex circuits.

JUNE 2014
VOLUME 3
NUMBER 6
thepcbdesignmagazine.com

THE pcb design™ MAGAZINE

AN I-CONNECT007 PUBLICATION

THE OPTIMUM
MAGAZINE
DEDICATED TO
PCB DESIGN

CONTENTS

The EDA Tool Conundrum

by Andy Shaughnessy
I-CONNECT007

You've likely been using EDA tools for 30 years or so. And you probably use your PCB design software just about every day. Does your tool do what you want it to do?

I don't know if EDA tools are getting better or if designers are just mellowing with age, but I don't hear as many of you griping about your tools. You all used to complain constantly about your software. If four or five designers got together at a conference, eventually the conversation would devolve into a bitch session.

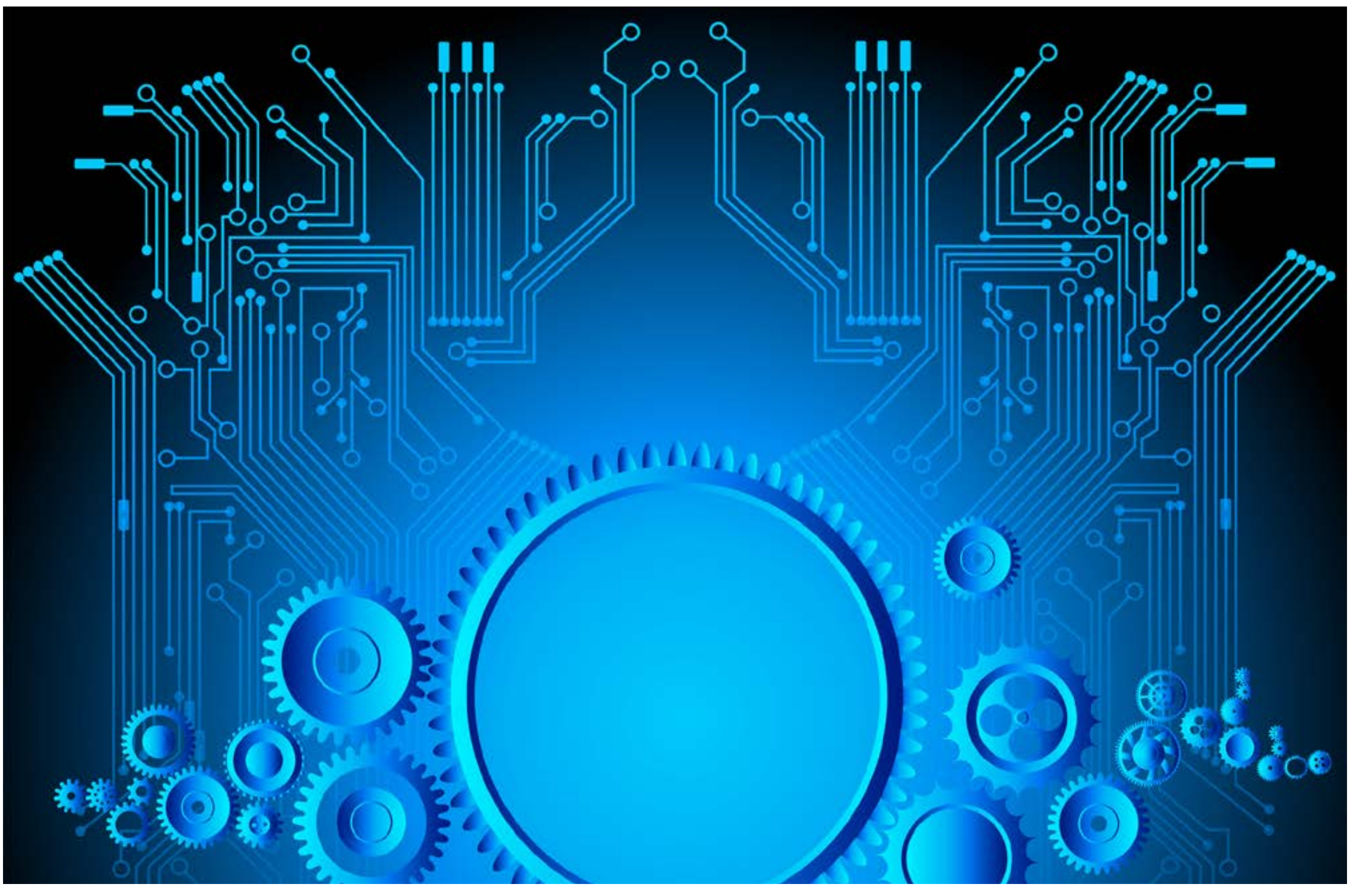
To hear you all tell it, your tools were crap, barely able to get the job done, and only after you set up patches and workarounds. Also, your managers were idiots for forcing you to use Mentor Graphics or Cadence or Zuken or Altium or Intercept, because it's just not designed to do what you're doing. How did they get to be managers, anyway?

And boy, wasn't P-CAD just the best tool ever? I still hear that a few times each year, actually.

It was almost a point of pride that you knew how to use this buggy software. You were part of a special club. And you UNIX users had an almost cultlike affinity for your tools. How many of you swore you'd never move to Windows tools? With UNIX, you could revel in the knowledge that no one else in your office could operate your software.

What job security!

Over time, the EDA companies started to take note. Now, every new rev of a design tool




CHECK OUT THE
DRAGON DRONES
INSTAGRAM FEED!

boasts that it's more "user-friendly" and has an "improved GUI." And they seem to be trying to make PCB design tools so simple that even electrical engineers can use them. (OK, I stole that line from one of the designers at IPC APEX EXPO. But it is funny.)

You may recall last month's column in which I shared feedback from the attendees at the SMTA Atlanta Designers Roundtable. I asked the group, "What do you wish your tools would do that they don't do now?"

One attendee wished his tool would slap the engineer, and another wished her Altium tool was more stable. But a third wished EDA companies would stop changing the very things he likes about his tool, and heads nodded in agreement.

One gripe I hear every so often: "Why did they add this new, useless feature? Were actual PCB designers involved in developing this tool, or is this just an exercise in self-indulgence for the software engineer?"

Designers feel as if the EDA companies aren't marketing the tools to them anyway. They say that they have little influence in which design tools their company uses, and that these decisions are made by their executives and the top guys at the EDA firm over three-cocktail lunches and golf outings.

Of course, if your company is buying multiple seats of Mentor, Cadence or Zuken, naturally that decision is going to have to be signed off by people above your pay grade. But too many designers tell me that they're never even consulted about EDA software decisions. Not asking for input from the person who will be using the tool seems like a bad idea.

If you were buying new accounting software, wouldn't you ask your accountants for their opinions?

So, it's a double-edged sword for the EDA companies: If they keep evolving their tools, making them "better" by whatever criteria you apply, they risk angering their users who've gotten used to the old versions of the tools. But they have to keep improving the tools, if only to get your boss to buy the next version.

If only that old P-CAD would work for the design you're working on now…

DESIGN 50 AUTOMATION CONFERENCE

Happy 50th Anniversary to DAC

It's hard to believe that it's been 50 years since the first Design Automation Conference took place in Atlantic City. Have any other conferences, of any type, been around that long?

Founded by Pat Pistilli, the conference was originally known as SHARE (Society to Help Avoid Redundant Effort), and the early DAC conferences featured sessions on design automation of all stripes, including architecture. The conference eventually focused solely on EDA.

Later, an exposition was added, and the event grew from 137 attendees in 1964 to 10,000 or more around 2000. Attendance is down since those heady days, but DAC is still a pretty big show by North American standards.

Unfortunately for PCB folks, DAC now focuses mainly on chip design, and there's very little PCB content. But I attended DAC in Los Angeles in 2000, Las Vegas in 2001, and New Orleans in 2002, back when there were a lot more PCB exhibitors and sessions.

Those shows were gigantic. Every booth had a fake Jerry Springer or a fake Phil Donahue hosting a fake talk show. "Beer on the floor" night was a zoo. A Synopsys rep told me they spent $1 million on the LA show.

And in Los Angeles, I played with the Porch Dawgs for the first time on the Soul Train stage at Paramount Studios. Huey Lewis was playing Denali's party next door…it was a different time before 9/11.

So, congratulations to Pat Pistilli and his family for keeping his show going, through good times and bad, for 50 years. **PCBDESIGN**

Andy Shaughnessy is managing editor of *The PCB Design Magazine.* He has been covering PCB design for 13 years. He can be reached by clicking here.

INTRODUCING COOLSPAN® TECA

thermally & electrically conductive adhesive

Rogers can help by being your reliable conductive adhesive film source

Get the heat out of those high-power PCBs. COOLSPAN® Thermally & Electrically Conductive Adhesive (TECA) Films are ideal for dissipating heat in high-frequency circuits. COOLSPAN adhesives feature outstanding thermal conductivity (6 W/m/K) and reliable thermal stability. Keep things cool, with Rogers and COOLSPAN TECA film.

CONTACT YOUR SUPPORT TEAM TODAY

ROGERS CORPORATION

www.rogerscorp.com

MEET YOUR COOLSPAN® TECA FILM SUPPORT TEAM

Leading the way in...

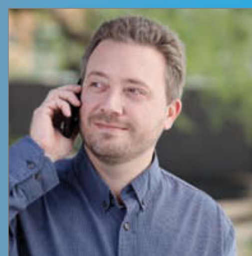
• Support • Service • Knowledge • Reputation

SCAN THE CODE TO GET OUR CONTACT INFO.

Greg Bull
Applications Development Manager
Northern Territory (U.S.) & Canada

Dale Doyle
Applications Development Manager
Western Territory (U.S.)

John Dobrick
Applications Development Manager
Southern Territory (U.S.) & South America



Scott Kennedy
Applications Development Manager
Eastern Territory (U.S.)

John Hendricks
Regional Sales Manager
Europe

Kent Yeung
Regional Sales Director
Asia

If you are unable to scan a VR code please visit our Support Team website at www.rogerscorp.com/coolspan

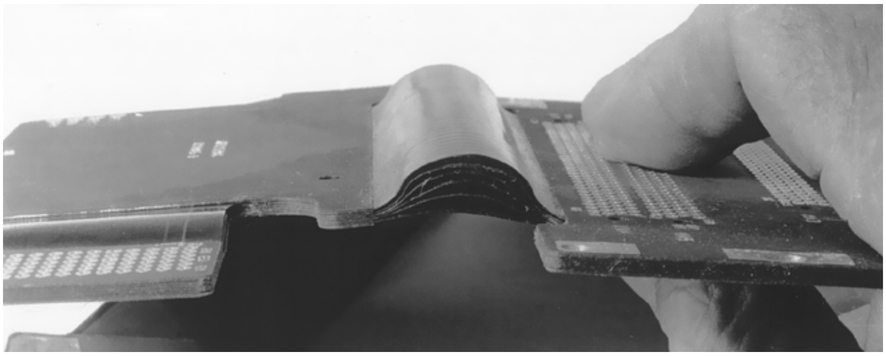
Why Procuring Flexible Printed Circuitry is Different

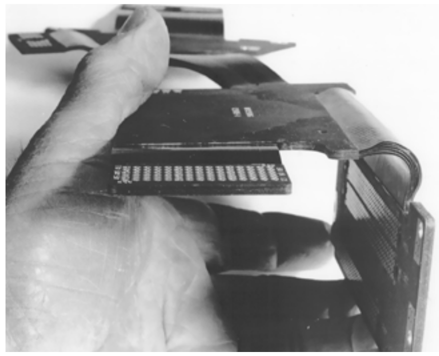
by Thomas Stearns
BRANDER INT'L CONSULTANTS

An article in the September issue of *The PCB Magazine,* How to Select a Flex Circuit Supplier, featured an excellent article on how to select a flexible printed circuit (FPC) vendor. Herewith two thoughts on why procuring FPC is different from other products.

Firstly, FPC specifications and quality controls tend to be too tight; FPC is, by nature, a flexible, easily formed product of thin sheets of plastic film and metal foils bonded together with flexibilized adhesives. It is not carved out of billets of metal in a machine shop and it is not reasonable to assign thousandths-of-an-inch tolerances to every dimension.

FPC appeared as a commercial product in the late 1950s. Back then, the manufacturing process consisted of fusion-bonding high-performance polymer films onto treated copper foil to form an initial laminate. This was imaged and etched to form the conductor pattern, then subjected to a second fusion bond process in which the covering film had pre-punched apertures to expose terminal pads, sometimes with an added piece of film attached to the backside of the pad to force it upward into the punched openings to stiffen and reinforce the area against solder heat and stress. Typical bonding temperatures reached 530°F or so, and distortion and residual stress were constant problems. But this was a product with outstanding chemical and environmental durability and superb electrical performance. There were no flexibilized adhesives here!





Figures 1 and 2: Bookbinder, shown flat (l) and bent (r). In rigid-flex circuits with bookbinder construction, which is very difficult to produce, FPC layers that will be on the outside of bent areas are designed slightly longer to provide ease in the formed position.

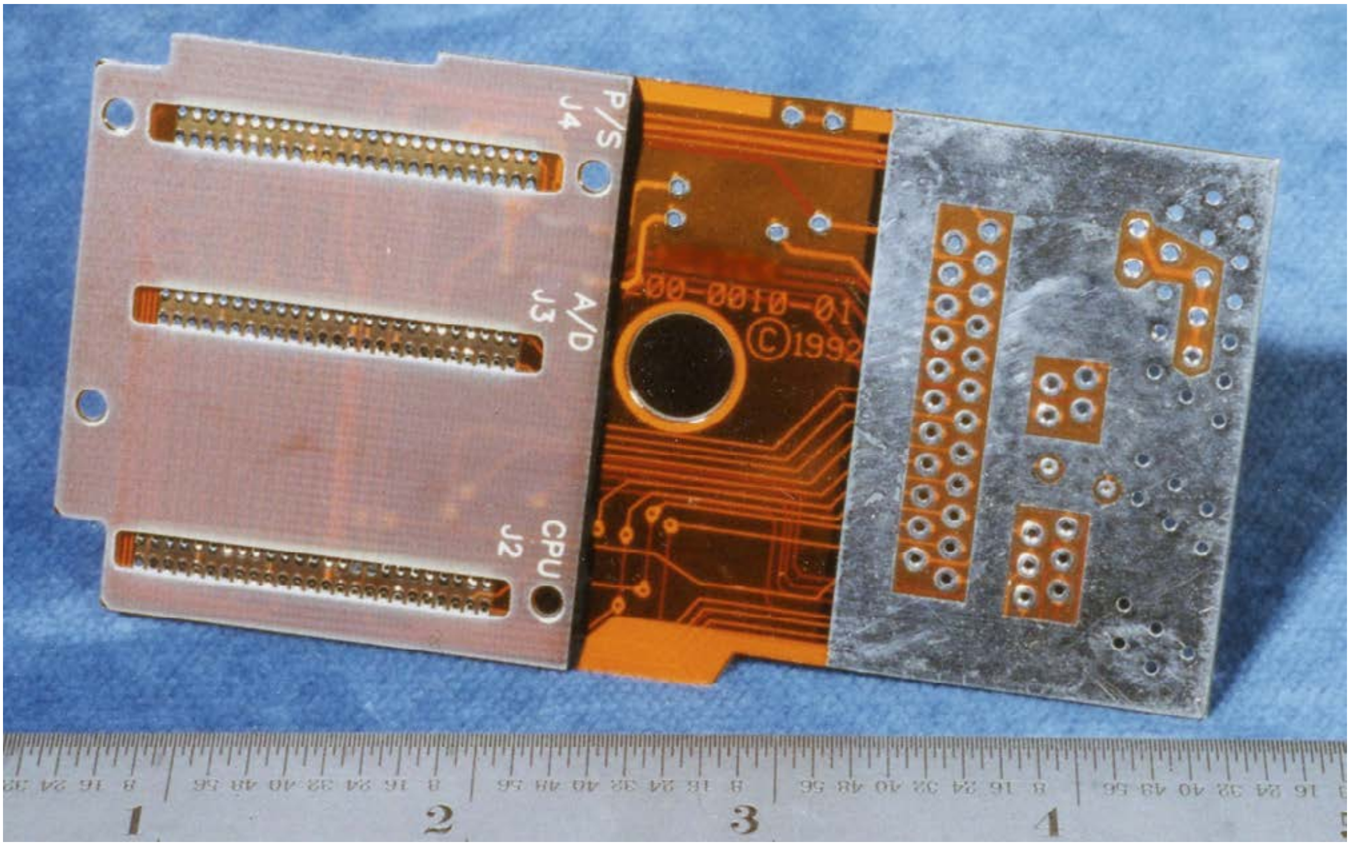


Figure 3: Stiffeners: FPC mimics PWBs when reinforced by an added structural layer; in this case (left end), epoxy-glass. The right end shows a ground plane or shield, which is part of the circuitry.

At that time, almost all of the applications were in military or space equipment and nobody had a lot of experience with FPC. These quality-oriented customers were understandably suspicious—how much heat could it withstand? Could you flex it for long periods of time? Did salt spray affect it? And so forth. So naturally, there was a need for complete characterization and quality control which led to overspecification. This was the era of "If you can build one

part in 100 to plus/minus 0.002", then let's use that as a spec."

Neither price nor yield was important to those uses—the only thing that mattered was performance, and FPC provided that wherever it was used. As the years passed, industry organizations came on the scene, and they wrote whole catalogs of specifications and test procedures, all with the intent of completely buttoning down everything from raw material to production code marking. This documentary effort lowered the variation from vendor to vendor and improved product quality, but at a considerable effect on cost, hampering the use of FPC in commercial goods. Latterly, with increasing familiarity with the benefits of FPC in volume production—near elimination of interconnect errors, steep reduction in labor content and so forth—these excessively tight specs have been perceived as unnecessary, with the result that today we're moving towards the "form, fit and function" end of the spectrum. But we're not there yet: FPC is still subjected to excessive physical and visual inspection to tolerances which are not necessary to assure reliable function, but the situation is improving. A procuring engineer will always review the product specifications and quality control requirements to be sure they cover everything that matters and have limits to produce good product, and work with the prospective vendor to agree on specifications just tight enough to yield good product at the best price.

"FPC is still subjected to excessive physical and visual inspection to tolerances which are not necessary to assure reliable function, but the situation is improving."

Secondly—and this may seem obvious and unimportant, but is a big issue—FPC is not a catalog item. It is a custom manufactured item and that makes a world of difference.

A short story to illustrate: Once upon a time a consortium of PWBpeople bought control of an FPC manufacturing company. The PWB guys with their "commodity production" mindset thought the FPC managers were unimaginative and behind the times and that PWB management ideas and efficiencies would quickly turn a poorly managed business into a winner. This rosy thinking lasted through the first year, at which point the PWB guys realized that things were very different from what they first thought. In another year the company was dead and the assets sold at auction.

Unfortunately for the PWB guys (and their bankers), this lesson was learned too late for financial recovery. What looked like an inefficient operation ripe for profit tweaking was actually a pretty savvy and experienced one struggling with an overload of variables and tight specs in a manufacturing environment significantly (perhaps 3X) more complicated than the one they knew. The biggest mistake in this tragic tale lay in the overpowering assumption that producing and selling FPC is the same as making PWBs, or razor blades, or any commodity.

Buying PWBs or FPC—anything that is built to a custom design—is very different from other procurement. To begin, you are trying to buy something that does not even exist yet! Both of these products, but particularly FPC, have proven their worth as cost reducers in electronic assemblies. PWBs have reached the maturity of a commodity; FPC isn't there yet, but it's gaining. Use of FPC is rising rapidly: In this modern world we would not have digital cameras or smartphones or any of a hundred other high functioning, complex, yet reasonably priced and rugged products we use every day without a piece of FPC, deep inside, providing error-free, semi automated, rugged interconnections. If you expect your new electronic gadget to make a profit, FPC is almost mandatory, and if you understand why the buying process is different, you can approach FPC procurement with confidence.

Sales or Contract Negotiation

Every manufacturer has a sales department, but in the FPC business it functions quite differently. In the FPC world, bringing customers in the door isn't selling; it's contract negotiation.

The result of an FPC sales meeting is a document that reads something like, "based on your drawings and specifications, and assuming we get X and Y by Z date, and that the dam don't burst and the creek don't rise, we will (may be able to) deliver by the third week of…."

"Sales" is what you think is going on but the much more truthful "contract negotiation" is really how an FPC project begins. FPC customers come to the procurement meeting with an idea of what they want—those drawings and specs—not a catalog number. Selling other products involves a catalog and a price list; FPC sales involves discussion and estimation of new requirements based on similar products and past production experience and ends with a quotation that is embroidered with many clauses and contains much wordsmithing.

Suppose that we look at selling as practiced at Paradise Bolt & Screw and compare it with what goes on at InterGalactic Flex. PB&S has a warehouse full of bolts and screws of known quality and production cost, ready to ship. The selling price and profit are already determined because any wrinkles in the process have been ironed out and the cost to produce is already known. Long ago, and well before the sales meeting, somebody did a market survey, decided which bolts and screws to make, fired up the machinery and tuned up the process so that high-quality product fell out the end at high yield and lowest cost. If you're looking for nuts and bolts, you simply get with a Paradise salesman who shows you samples, quotes prices, and guarantees delivery. It's cut and dried.

> ***"Remember that existing orders, particularly those from established customers, are already on the factory floor; if they get into trouble, they will take priority."***

At IGF, the quoting-negotiating-order closing process, although carried out in similar language and with the same goal, (i.e., "closing the sale") is completely different. At IGF, each order taken is a ticket to a custom manufacturing adventure. There's nothing in stock, no samples of product to fondle and discuss, and the cost to produce is unknown because each design, regardless of how well vetted to industry guidelines or scrupulously productionized, is a mystery until a pre-production pilot run has been made. Quality requirements can be scoped out by the use of standard classifications and requirements, but unexpected interactions between design, materials and production equipment can lead to ugly surprises. Much of this uncertainty disappears with a repeat order, but the basic problem remains: The partners to the deal are buying and selling something that does not exist yet!

Delivery

A hugely important difference between procuring custom manufactured products and commodity products is delivery. In our example let's assume that IGF has a well-equipped factory, one that is just as good as PB&S's. Nevertheless, unless this is a repeat order, IGF's factory has no experience at all in the production of the new design. And if InterGalactic is any good at its business, the factory is already loaded with orders and delivery commitments to other customers. Delivery estimates assume that not only will the new design run smoothly, but that none of the existing orders will run into a disaster. Remember that existing orders, particularly those from established customers, are already on the factory floor; if they get into trouble, they will take priority. At PB&S, the factory is irrelevant—it could have even burned down—since your parts are already in stock and need only be dusted off, packaged and shipped; at IGF you and your project are just entering a complicated production process, vulnerable to Murphy's Law.

You may argue that we're comparing apples to jetliners because the complexity of flex manufacture is far greater than the production of nuts and bolts, and you'd be correct. Nuts and bolts are hogged out of stable, naturally-occurring raw materials with a BOM calling for two items—a base metal and a surface finish, if any. FPC manufacture involves many more materials including highly unnatural films and adhesives that are produced by film makers and compounders from polymers, which are in turn produced by other chemical companies. This is a much more complex supply chain with multiple layers of manufacturers. Latent defects are the rule. And that's just for the raw materials; making nuts and bolts is primarily an automated machine-shop enterprise, while FPC production takes multiple interacting processes using thermal, chemical and mechanical forces and process equipment all influenced by both operator skill and latent defects in those complex raw materials. There is no comparison in degree-of-difficulty and the horrors of compounding yield losses, but this only adds to the argument: If the situation were reversed and the complex product was a commodity and the easy-to-build product was custom made, these problems would disappear. But FPC is the tough one to make and can't be sold from a catalog because it's a custom product. You sell something that you have; you contract to make something you don't have. Selling something you don't have is pretty close to fraud.

Competition, Second Sources and Partner Selection

What about competition? PB&S makes a catalog product, but so do all of the other bolt and screw makers of the world. If your project needs 1,000, 1/4-20 round head

1" brass screws per day and Paradise can't supply them, there's always somebody else with an identical and fully interchangeable part that can. In the case of an FPC buy, a hiccup at the factory—your personally selected, custom-contracted factory—means that you start production all over again: there's no backup supplier ready to ship on demand. You selected a pro-

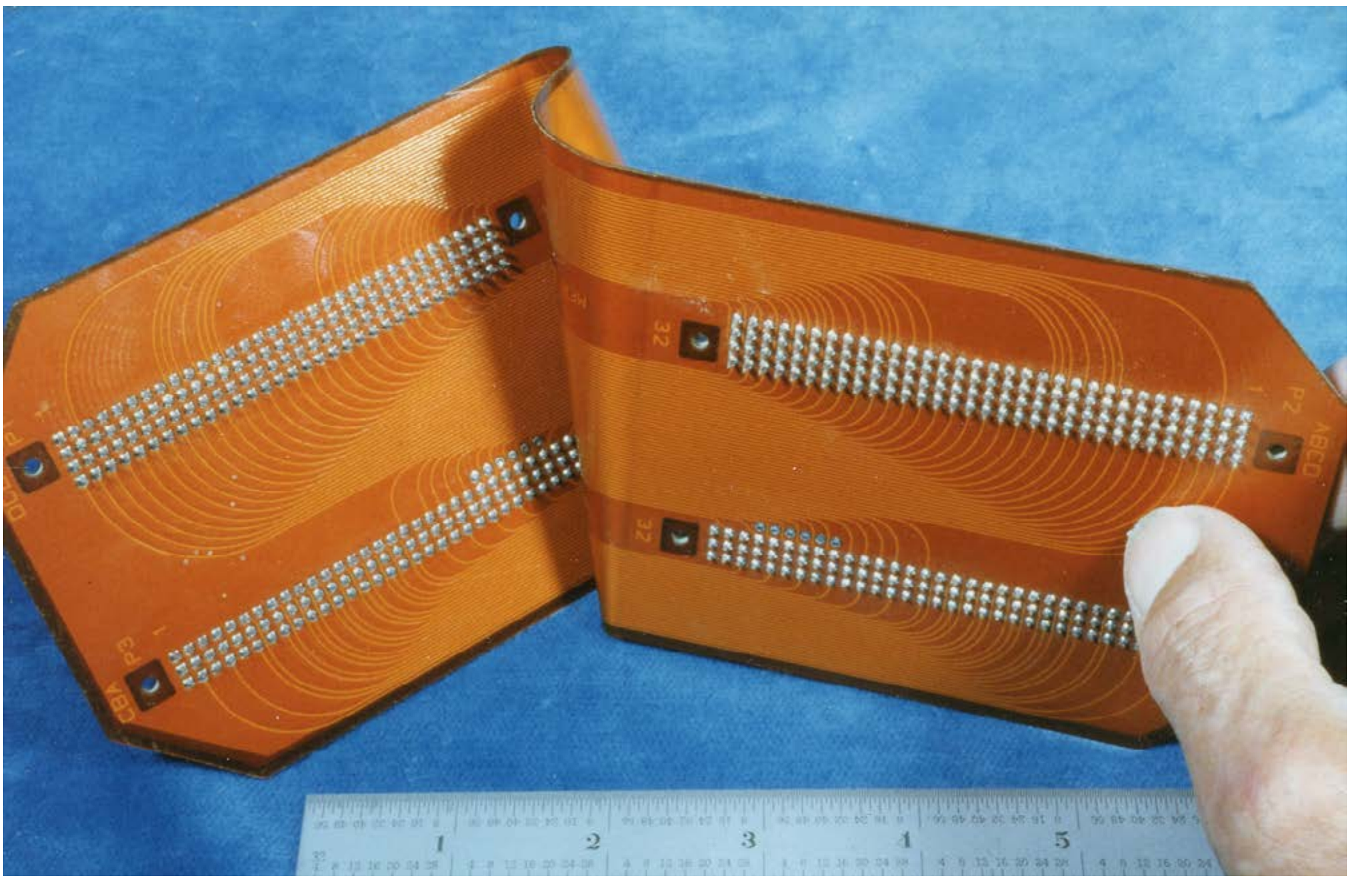


Figure 4: Four connectors: A three-layer FPC with central shield. Terminations are plated through and have soldered connectors.

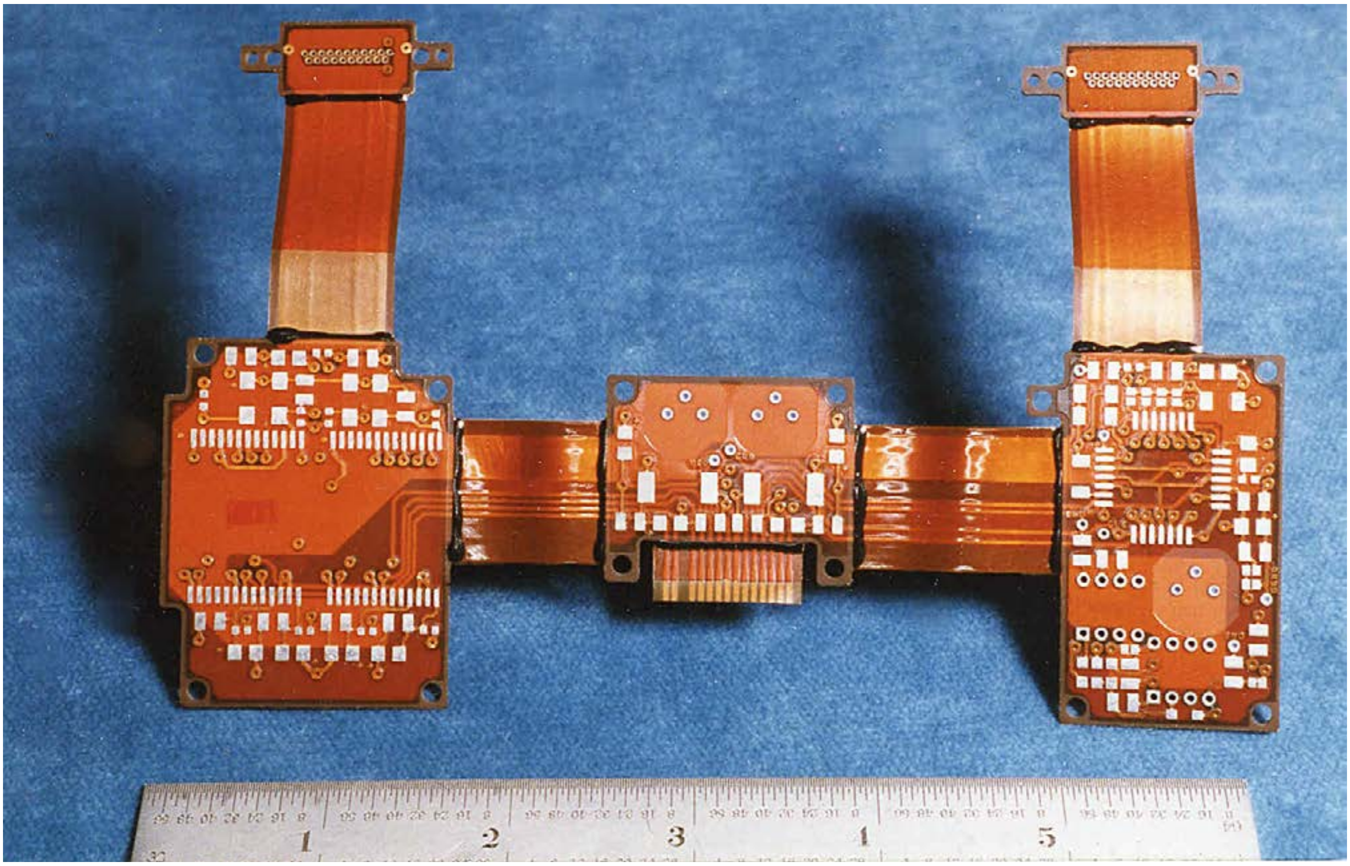


Figure 5: Rigid-flex circuitry provides the most space and weight efficient interconnection. This circuit has five rigid terminal areas with plated through-hole interconnections. Black material along the rigid edges is semi-hard and gives strain relief.

duction partner and that partner has just let you down. You could leave in a huff and start finding a new partner, but that's more time wasted and you would be even further from production with no greater assurance of success.

This is why selecting a vendor for an FPC requirement is a contract negotiation and partner-selection, and not just a price comparison.

In most cases, the Number 1 factor in choosing an FPC source is delivery and the reliability of same. Piece part cost may seem important at the negotiating table but your production line will shut down if you don't get the FPC you need for your assembly. FPC is usually the starting point in product manufacture, so regardless of its cost position on the BOM, and it is usually a trivial cost item, if the FPC isn't at your factory you will produce nothing. At this point the FPC will assume major importance: Remember the old adage, "For want of a nail the shoe was lost...." Tiny problems can stop big enterprises!

The only way to choose between vendors is by history (i.e., past performance). The negotiating chatter may be about NREs, tooling and lot charges, but in the back of the (knowledgeable) buyer's mind the big issue is: if he says four weeks, will I see parts by the end of June? And if he doesn't deliver, what do I do? You can stare cold-eyed at the IGF salesman, trying to read his intentions, but your best guide is how IGF performed in the past. Do they recover quickly from disaster? Do they have the necessary materials on hand, ready to re-release to the factory? And so forth.

Repeat Orders and High-Volume Production

You might expect that custom manufacture is used only for prototypes, but since there are almost no catalog FPCs, it is the rule in the FPC universe. A second order should run much more

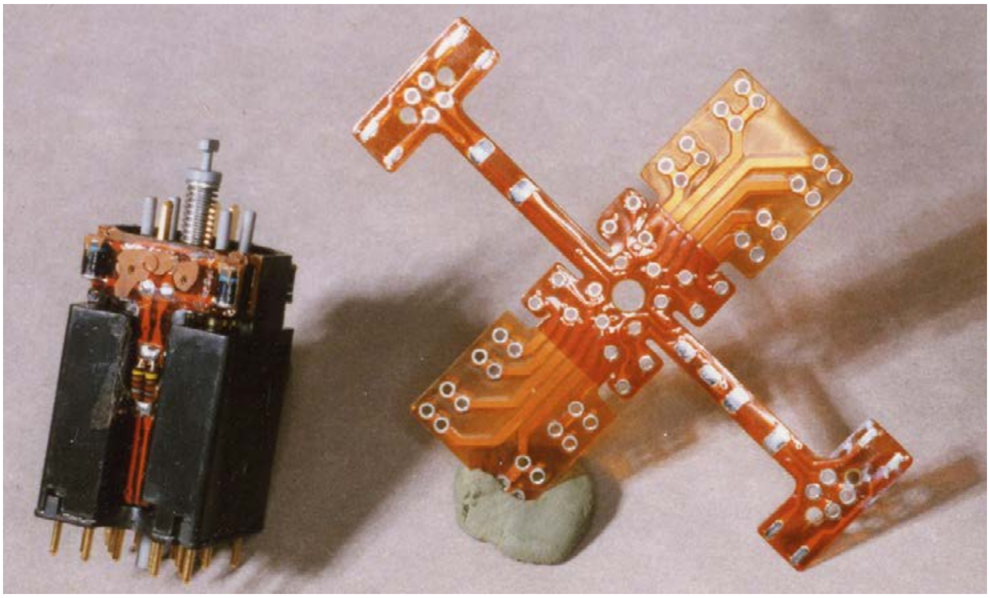


Figure 6: Relay circuit: FPC enables dense packaging. Relay on the left shows the FPC as folded/potted. The circuit is shown as manufactured on the right.

smoothly than the first order; if it doesn't, start looking for another vendor because good vendors learn from mistakes and all others should be avoided. Much of the high cost of first-run FPC comes from the engineering supervision that is needed to assure a decent yield on a new design; this extra oversight goes away when an acceptable yield is achieved and unique problems have been seen and neutralized. This brings us to the high-volume situation.

Because FPC is so good at reducing production cost in dense electronic packages, it frequently appears in high-volume production. What happens there? Given a good, fully productionized design (i.e., one with rational tolerances and adequate room for uncontrollable variation in films and adhesives; well-set process conditions with functional limits; mature quality control procedures with their limits) FPC can be produced reliably at competitive cost.

Summary

Keep in mind that vendor choice is more important in FPC applications than anywhere else. Building FPC involves chemical, thermal and mechanical processes, which means the FPC vendor has to be several times more skilled than other vendors with a grasp of much more technology.

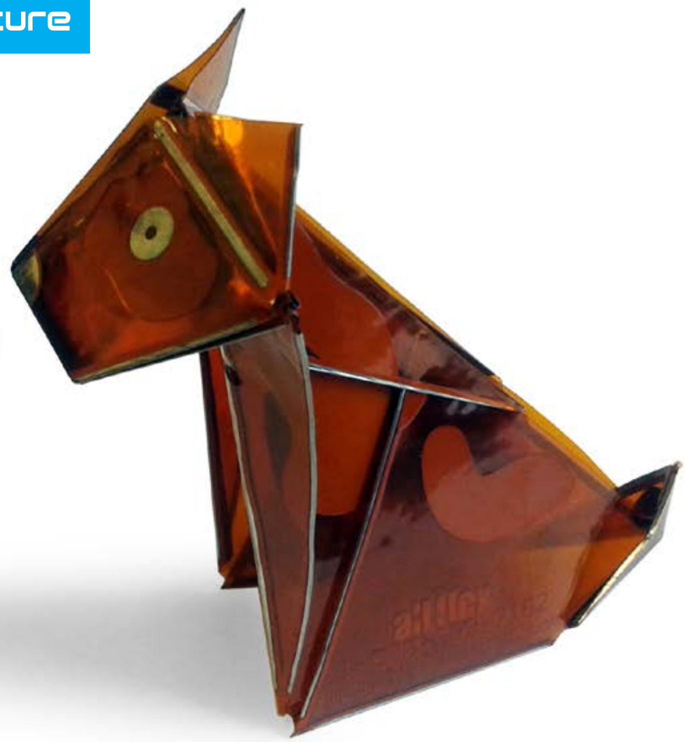
The raw materials of FPC production are manufactured, not natural. This brings in another layer of potential variation and out-of-spec products, along with latent defects like varying shrinkage in polymer films, out-of-date adhesives and other reactive chemicals, and so forth. Your vendor not only has his hands full controlling his own factory, but is exposed to variations anywhere upstream in his supply chain.

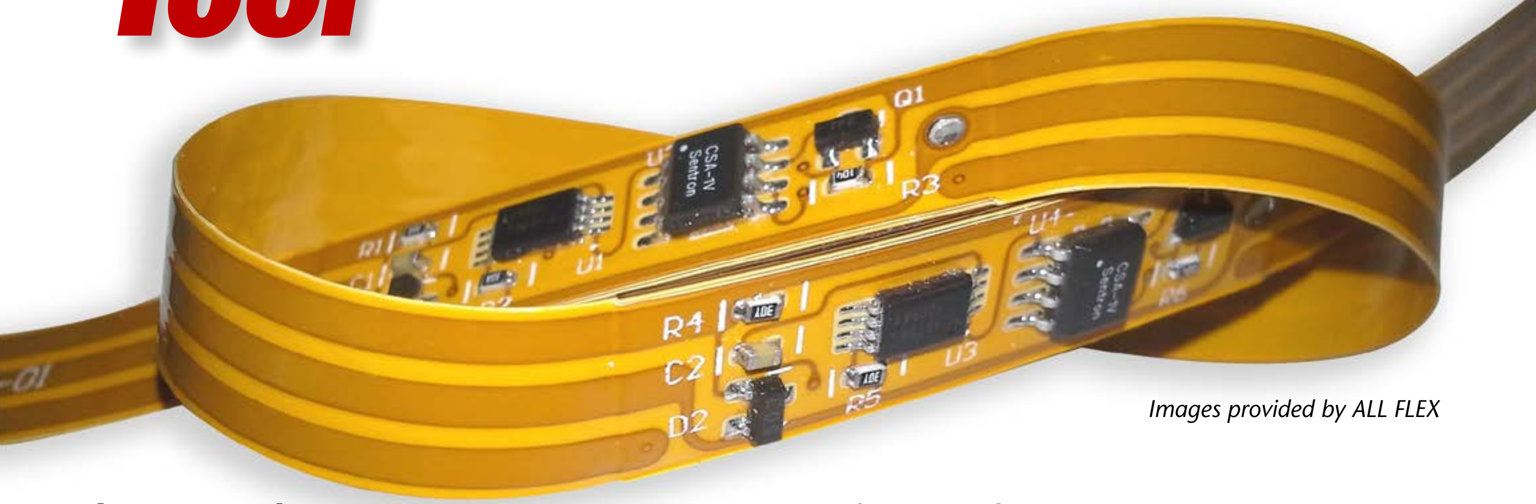
FPC has been historically overspecified because it started out as a military component. Overspecification, (i.e., specs that force the production to be "on the edge of state-of-the-art" will rebound on you when yield drops. Your vendor is a partner and his problems are your problems. Set reasonable tolerances and inspection limits; it will help you.

Ever had a house built? Remember dealing with your contractor? FPC is also a custom-built product and this means you don't "buy" it, but contract for its manufacture sometime in the future. Price and delivery may be stated on the PO and embellished with boilerplate, but reality rules on the production line. Therefore, choose your vendor carefully. **PCBDESIGN**

Thomas Stearns is a 57-year veteran of the FPC business and the holder of 23 patents. Stearns is the president of Brander Int'l Consultants, providing design, process and quality support in flex and rigid-flex circuitry. He may be reached at thomasstearns@comcast.net.

Flexible Circuitry... a 3D Packaging Tool





Images provided by ALL FLEX

by Dave Becker
ALL FLEX

The Market

The global market for flexible circuits is currently estimated to exceed $13 billion, with an estimated annual growth rate exceeding 10% as electronic devices continue to become smaller, lighter and more personal. Markets include medical, avionics, industrial, cell phones, military, and consumer among others. It is a technology an electronics designer should understand…and know when to consider adoption of the technology. Borrowing the words of Humphrey Bogart from the movie Casablanca, “Maybe not today, maybe not tomorrow, but soon and for the rest of your life.”

Moving Parts?

When most people think of flexible circuits, they imagine component parts moving within an electronic application. Clamshell cell phones, notebook computers, printers and hard disc drives are often mentioned as examples of products requiring moving parts with flex circuit interconnection. Questions such as “How many times can a flex circuit flex?” are best answered by “It depends.” Moreover, it depends on a lot of things. This political reply should be followed by a comment about several well-known design guidelines to be adopted when designing circuits for use in a “dynamic” flex application. In most cases, the fabricator will incorporate design practices intended to maximize flex life. Feature and construction recommendations include:

ventec
8 1/2 Reasons You WIN with Ventec
#8½
Show Me
#1
#2
#3
#4
#5
#6
#7
#8
#8½
Ventec Europe
ventec-europe.com
Ventec USA
ventec-usa.com
Ventec International Group
venteclaminates.com

• Copper traces should be in the "neutral" axis (i.e., equivalent amounts of dielectric on each side of the traces). This prevents the copper from being in either a tension or compression mode during flexing.

• Thinner circuits are better than thicker circuits. A good design will use .5 or 1 ounce copper with either .5 or 1 mil polyimide.

• Rolled annealed copper should be used for dynamic applications (the rigid PCB world uses electrodeposited copper) with a consideration for grain direction. The flexing motion of the circuit should be parallel to the copper grain direction. This is a critical consideration when the fabricator does their panelization.

• A gentle bend radius is best. An accepted guideline for dynamic flexing is the bend radius should be at least 10x the thickness of the material. Since single-sided circuits are quite readily designed at .005" thickness, a flexing radius of .050" is safe. In static applications, a guideline for single-sided circuits is that it can be folded on itself three times with a radius equal to its thickness if rolled annealed copper is used.

• When double-sided circuits are required, the circuits should be fabricated without electroplated copper plated on the surface of traces in bend regions. This practice is known in the industry as "pads only plating." Bend regions should be single layer copper to allow neutral axis flexing.

• Copper features are usually modified to "flexize" the circuit design. This includes radiusing traces through corners and adding fillets to solder pads. These feature upgrades not only improve flex life, they also improve the overall reliability of the design. Expect the flex supplier to provide these recommendations and file upgrades as part of their service.

The above guidelines are good rules of thumb. Some applications are only required to flex a few times, others thousands, while some might require millions. It is always a good practice to flex test an individual circuit design with prototype parts with an assembly that mimics the final application. And make sure the circuit supplier knows the product is going to be used in a dynamic application so proper grain direction is chosen during panelization!

Flex to Install

So while it is certainly true flex circuits are used successfully in a wide range of dynamic

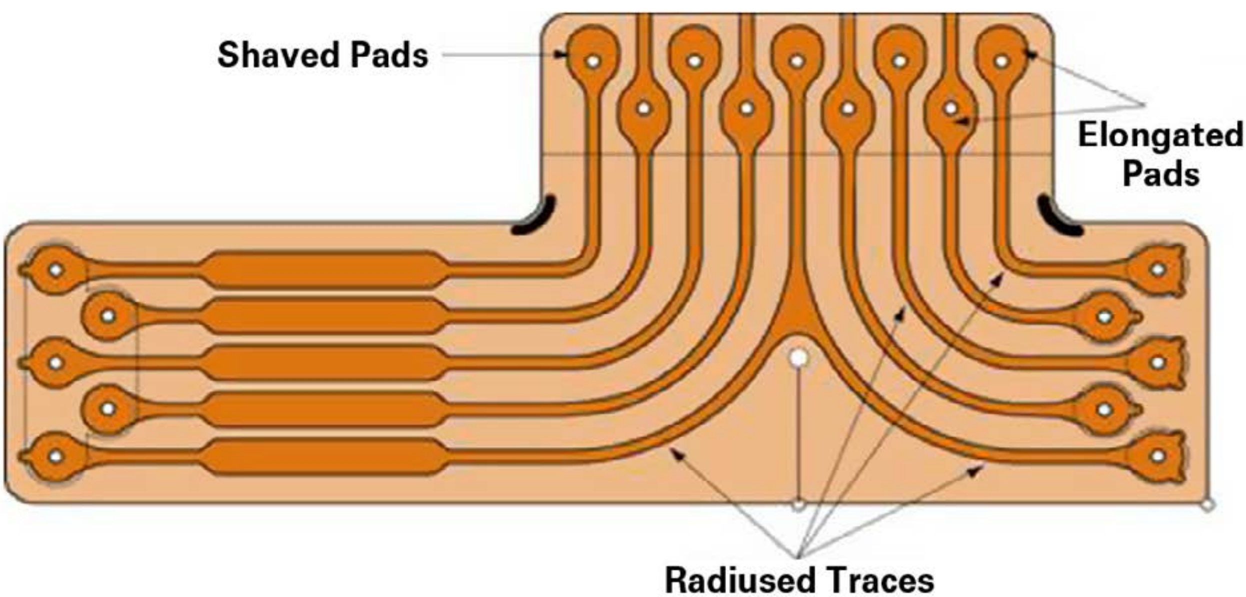


Figure 1: Flexized design.

flexing applications, the vast majority of flex circuits do not bend, fold, or flex once they are installed. This obviously begs a few questions like, "what are all those other applications?"; "why do they use flexcircuits?"; and "when should a flex circuit be used rather than a rigid PCB?"

The answers vary as diversely as the applications themselves. Answering these questions requires an examination of why past products have incorporated flex circuits. What utility is offered by this novel interconnection packaging option? Answers include space savings, weight reduction, high-performance cable design, multi-planar interconnection, and cost.

Cost vs. Hardboard

One of the questions I field most frequently is, "How does flex circuitry compare in cost to a rigid printed circuit?" And my honest answer is generally is that it is more expensive. The specific premium paid for flex circuitry is a multi-dimensional factor and a function of material costs, handling requirements, applications engineering support, volume, and the construction. I generally advise that, if a design doesn't take advantage of the bending characteristics or the thinness of a flex circuit, it is likely that a rigid PCB will be a lower-cost solution. But taking advantage of a flexible circuit's characteristics involve considerations for multiplanar interconnections while eliminating connectors or hand soldered wires, simplified assembly, improved reliability, or providing a pre-tested component/subassembly. Applications adopting flex circuits have cleared the hurdle of looking at total cost of ownership and decided the technology is cost effective. In fact, the market for flex circuits is estimated to be about 10% of the overall printed circuit market. It is a pervasive technology.

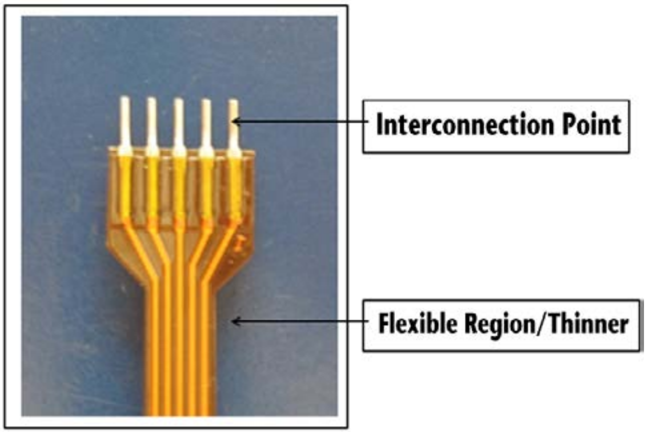


Figure 3: Circuit with integrated pins.

Power Circuits

A lesser known segment of the flex circuit market is found in high-power applications. Most applications utilize .5 or 1 ounce copper (.0014" = one ounce) but some products require considerably more current-carrying capacity than can be reasonably designed in with these copper thicknesses. One option is to consider thicker copper. Copper thicknesses of .010"–.015" thick are not uncommon but flexibility is dramatically reduced with these materials. Using polyimide thicknesses of .001" (dielectric strength at 6000volts/mil per ASTM D149) does allow a considerable thickness reduction in applica-

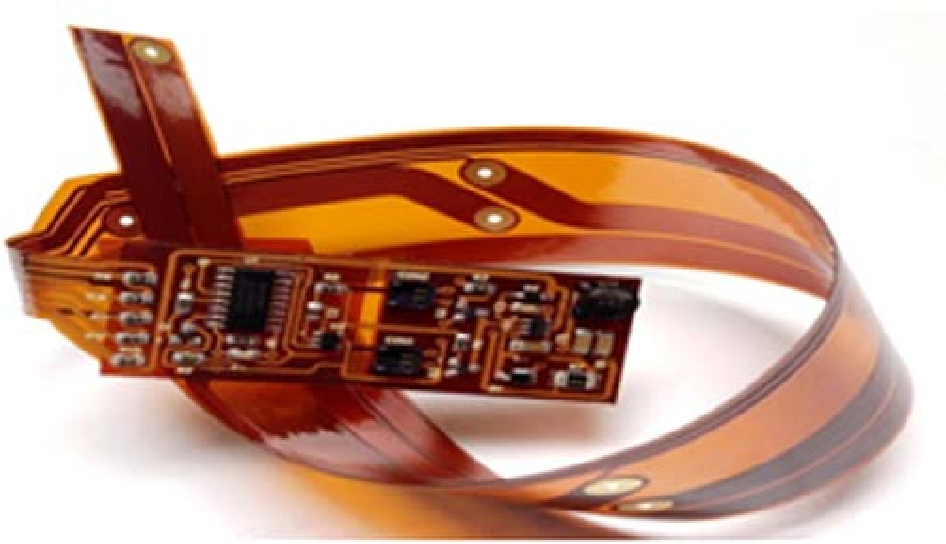


Figure 2: Folded circuit with components.

CURRENT CARRYING CAPABILITIES FOR EXTERNAL TRACES IN AIR

	COPPER WEIGHT							
AMPS	1/2 OZ. (.0007")	1 OZ. (.0014")	2 OZ. (.0028")	3 OZ. (.0042")	4 OZ. (.0056")	5 OZ. (.007")	6 OZ. (.0084")	8 OZ. (.0112)
	CONDUCTOR WIDTH IN INCHES							
0.5	.013"	.008"	N/A	N/A	N/A	N/A	N/A	N/A
1.0	.028"	.017"	.013"	N/A	N/A	N/A	N/A	N/A
1.5	.040"	.027"	.020"	N/A	.012"	.005"	.003"	N/A
2.0	.053"	.040"	.030"	.0235"	.020"	.018"	.016"	.013"
2.5	.080"	.060"	.042"	.0325"	.0285"	.024"	.023"	.018"
3.0	.100"	.083"	.057"	.045"	.0387"	.035"	.030"	.024"
4.0	.160"	.120"	.088"	.066"	.055"	.048"	.043"	.037"
5.0	.225"	.158"	.118"	.09"	.074"	.065"	.059"	.048"
6.0	.285" (off chart)	.195"	.153"	.117"	.094"	.082"	.074"	.062"
7.0	N/A (off chart)	.250"	.187"	.145"	.124"	.105"	.0905"	.075"
8.0	N/A (off chart)	.307"	.232"	.180"	.1485"	.130"	.122"	.095"

This chart gives recommendations for width of conductor needed to carry current on different copper thicknesses. For informational purposes only.
PCB Trace Width Calculator

Table1.

tions with space constraints. And the materials do remain bendable if necessary.

Current carrying capacity is a function of a copper traces cross sectional area and the following chart (Table 1) provides a quick reference for design considerations.

Circuit features with integrated pins are sometimes favored in applications with high current requirements. The thicker copper creates robust pins that are not easily bent or damaged during handling. These integrated pins are often used in high reliability applications serving as jumpers from one PCB to another, eliminating the need and cost of connectors. Circuits with this feature require a multi-step etching process to produce a flexible circuit having thinner and more flexible finished conductors in regions that are bent, but with thicker copper pins for the soldered connections.

High-Speed Cables

High-performance cables with controlled impedance requirements can also be created with flexible printed circuits. These cables are designed for use in high-speed digital applications. A common microstrip circuit, easily constructed with PCB materials and requiring controlled impedance, uses a flat conductor over a ground plane. The conductor and ground plane are separated by a dielectric—usually polyimide and adhesive. The surface microstrip transmission line also has free space (air) as the dielectric above the conductor. Stripline designs with two reference planes are also easily designed with circuit board materials and processes. There are several options on the Internet for finding design assistance when calculating controlled impedance. One option is located here.

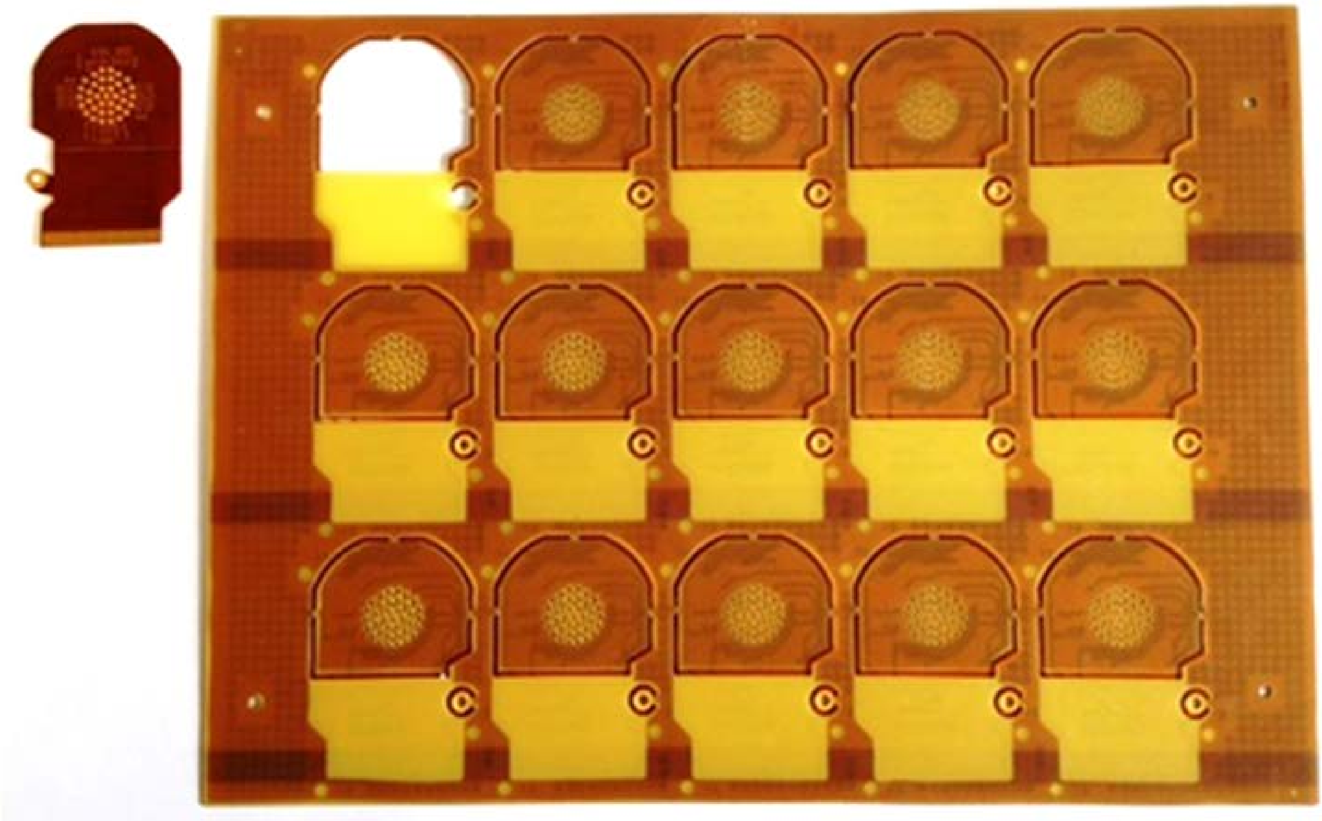


Figure 4: Circuits with stiffener carrier.

Component Assembly

Component assembly on flex circuitry is common and similar to methods used for rigid PCBs. More than 50% of flex circuits are sold with value-added component attachment. This can be done with hand soldering techniques or hand placement of SMT components for low volume. Both traditional tin/lead solders and lead-free solders (used to meet RoHS requirements) are common as polyimide constructions are very robust at high temperatures. With higher-volume parts there is custom fixturing used which can get complicated as it is adopted to accommodate the handling nuance characteristic of thin, flexible materials. Both high-volume wave soldering and surface mount soldering are common assembly techniques. Selective stiffeners are attached with either thermoset or high temperature pressure sensitive adhesives on regions of the circuits supporting components. The stiffeners are often glass epoxy materials and prevent bending the circuit at a solder joint. Fracturing traces can occur absent adequate mechanical support from a stiffener.

Additionally there is some process preconditioning recommended when soldering flex circuits, especially if they are stored in a modestly humid environment. Polyimide is considered quite hydroscopic as it absorbs up to 2% of its weight in moisture. This absorption can cause circuitry to delaminate or blister when exposed to solder temperatures. Prebaking flex circuitry at 105°C for an hour eliminates this issue and is a good general recommendation prior to solder temperature exposure.

As volumes increase beyond prototype quantities, fixturing becomes a requirement for both wave soldering and SMT soldering. This fixturing will be custom and design depends heavily on the individual parts and assembly required. With wave soldering, FR-4 glass epoxy is often used to mechanically support a soldered connector. This glass epoxy can also be used as a carrier fixture with individual parts broken from the panel after component soldering. Carrier plates are also used with cavity openings allowing access to reflowed solder in a wave soldering machine.

Fixturing for SMT assembly is often done with vacuum assist or by bonding the flex panel to a low tack carrier plate. Both methods allow the panel to be held in place through processes including solder stencil, SMT placement and reflow. Pre-cutting or routing the individual parts prior to assembly allows them to be easily removed from the panel post assembly. The myriad of individual parts and variety of assembly requirements leads to a wide variety of fixture options. There are companies specializing in this type of tooling that can also assist with ideas and custom fixtures.

Summary

Flexible PCBs are used extensively throughout the electronic market for a wide variety of reasons, including their ability to move in an application, their light weight, and extreme thinness. They can also be used in high current-carrying applications and are often populated with components with assembly processes typical of rigid printed circuits. There are certain important design, processing, and handling considerations that separate flex circuits from their rigid cousins. Many suppliers offer printed design guides and early involvement of a flex supplier is recommended as a design unfolds. This early involvement will help insure proper tolerances, costs, and constructions are considered for a robust and reliable adaptation of the technology. **PCBDESIGN**

Dave Becker is the V.P. of sales and marketing at All Flex Flexible Circuits and Heaters, which specializes in low to medium volume flexible circuits and flex circuit assembly.

PCB007 News Highlights

Viasystems' Sales Up 8.4%; Q2 Growth Expected

"While the markets we serve did not demonstrate any significant strength, and in some instances declined both year-over-year and sequentially, we were able to generate net sales and earnings in line with our first quarter expectations," said CEO David M. Sindelar. "In terms of our end markets, our automotive, telecommunications, and military and aerospace sectors performed as expected."

Speedy Circuits Looks West for More Business Prospects

Speedy Circuits has seen a marked increase in output of prototype PCBs at its board fabrication facility in Taiwan lately due to strong demand from numerous ICT companies. Demand for HDI PCBs in particular has been strong since it beefed up its board fabricating capabilities two years ago with the acquisition of several advanced processing equipment that include Orbotech InCAM systems, Mitsubishi laser drill, Orbotech LDIs, and more.

IPC: N.A. PCB Book-to-Bill Shows Positive Growth

"The book-to-bill ratio's climb and positive year-on-year sales growth in March are encouraging signs of a recovery ahead," said Sharon Starr, IPC's director of market research. She cautioned, however, that "the book-to-bill ratio has just reached positive territory after six consecutive months below 1.00. Economic indicators are positive for 2014, but the PCB industry's recovery is developing slowly."

HEI Suffers 53% Q1 Sales Drop

HEI, Inc. has announced unaudited financial results for the first quarter of 2014, which ended March 29, 2014. Sales in the first quarter of 2014 were $5,944,000, compared to $12,516,000 in the first quarter of 2013. The company generated a net loss of $896,000 in the first quarter of 2014, compared to a net income of $636,000 in the first quarter of 2013.

Graphic PLC Ups North American Presence

Scott Kohno, president of Graphic USA announced that Graphic PLC has begun its strategic penetration of the North American market. Graphic PLC bought CalFlex USA last summer with plans to increase its market reach into North America.

Advanced Circuits Marks 25 Years in PCB Industry

"Advanced Circuits has experienced consistent growth since its establishment in 1989 and currently occupies a dominant market position with the industry's largest customer database," said President and CEO John Yacoub.

MFLEX Sees Net Sales Drop 32%

Commenting on the company's business outlook, Reza Meshgin, CEO, noted, "Based on the timing and breadth of new programs expected to ramp across our customer base, we continue to anticipate a significant increase in net sales and a return to profitability, excluding impairment and restructuring, in the fiscal fourth quarter."

Despite Loss, TTM Meets Guidance; Sees Demand in Q2

Net sales for the first quarter of 2014 were $291.9 million compared to $366.1 million in the fourth quarter of 2013, and $325.4 million in the first quarter of 2013. First quarter 2013 included $23 million of revenue from TTM's SYE plant, in which TTM sold its controlling equity interest during the second quarter of 2013.

Sunstone Named Among Healthiest Oregon Employers

The company earned this award by launching a formal wellness program to encourage healthy lifestyles. By implementing a health incentive, employees who obtain biometric readings, undergo a health risk assessment, and participate in the "Wellness BINGO" challenge have the ability to receive a reduction in their health insurance premiums each month.

WUS, Schweizer Collaborate on HF PCB Production

WUS Printed Circuits Co., Ltd. and Schweizer Electronic AG concluded an exclusive long-term agreement on the cooperation for the production of high-frequency PCBs for the global automotive and industrial market.

Ninja Flex Circuits

by Tom Woznicki
FLEX CIRCUIT DESIGN COMPANY

One thing that I've always liked about flex circuits is that they look cool. They're shiny, orange, and clear—you can see those beautiful curved copper traces unlike their green rigid-board counterparts.

But if you've looked inside a smartphone or read an iPhone teardown article, you don't see beautiful shiny flex circuits; you see flex circuits that look like they were spray painted with black primer.

Do not be deceived by their drab appearance—these are Ninja flex circuits! They have special cover films that give them powers to suppress EMI, eliminate glare, control impedance, and reduce cost. They also disguise and protect intellectual property; it takes a great deal of effort to reverse-engineer them, and you literally destroy the circuit in the process.

Figure 1 shows an iPhone 5s teardown by iFixit. Figure 2 is a close-up view of the camera module. You can see many different flex circuits all covered by either black cover film or shielding film.

Black cover films are exactly as the name implies: Cover film made with black polyimide. It is widely used on flex circuits for LEDs and cameras. Just punch openings and laminate it to the base laminate as you would any normal cover film. DuPont offers Pyralux LF-B and halogen-free Pyralux HXC. Korean companies Innox and Doosan also make black cover film. Other companies, such as CEN in China, offer black polyimide which can be used with free-film adhesive to create cover film.

A subset of black cover films are used to protect components from static; the polyimide is electrically conductive, but with a very high resistance. These materials are not readily available off-the-shelf as cover film. They are applied using free-film adhesive or by having a custom materials company make cover film from the polyimide sheets. DuPont has Kapton XC and CEN's antistatic polyimide is called BY.

Figure 1: An iPhone teardown by iFixit. Used with permission.

Figure 2: Close-up of iPhone camera module. Used with permission.

Shielding films have a very thin metal coating (approximately 0.1 micron), and an electrically conductive anisotropic (conducts in z axis) adhesive. They provide an extra layer of shielding without the expense and thickness of an extra copper layer. Some are black, some silver, and are applied after cover film lamination and require cover film openings that expose copper connected to ground.

Ninja Flex Design Time

I recently designed a flex with shielding film, a simple ZIF jumper (Figure 3). The circuit features two copper layers: Signal and ground plane. The customer wanted complete shielding, so rather than adding another copper layer, we specified shielding film. Along each edge of the flex is a wide copper trace tied with vias to the ground plane below. Round cover film openings exposed these traces so the anisotropic conductive film could make the electrical connection. In Figure 4 you can see the depressions where the lamination process pushed the shielding film down into the cover film openings. This particular shielding film is only 8 microns thick, so it really conforms to the surface of the flex. Figure 4 also demonstrates how the shielding film completely hides the trace layout.

It's possible to create a fully-shielded flex circuit with only one copper layer by using shielding film. Just apply an oversized piece of shielding film to both sides of a single-layer flex after the circuit is cut from the panel. The shielding films will bond to one another in the area past the flex circuit's outline. Trim back the bonded shielding film close to the flex outline and you have shielding all the way around!

Many companies are currently making shielding film: Tatsuta, Toyo Ink, Doosan, and Innox are major suppliers.

So, if your flex application has special requirements, look to black cover film, antistatic cover film, or shielding film to give it some ninja powers!

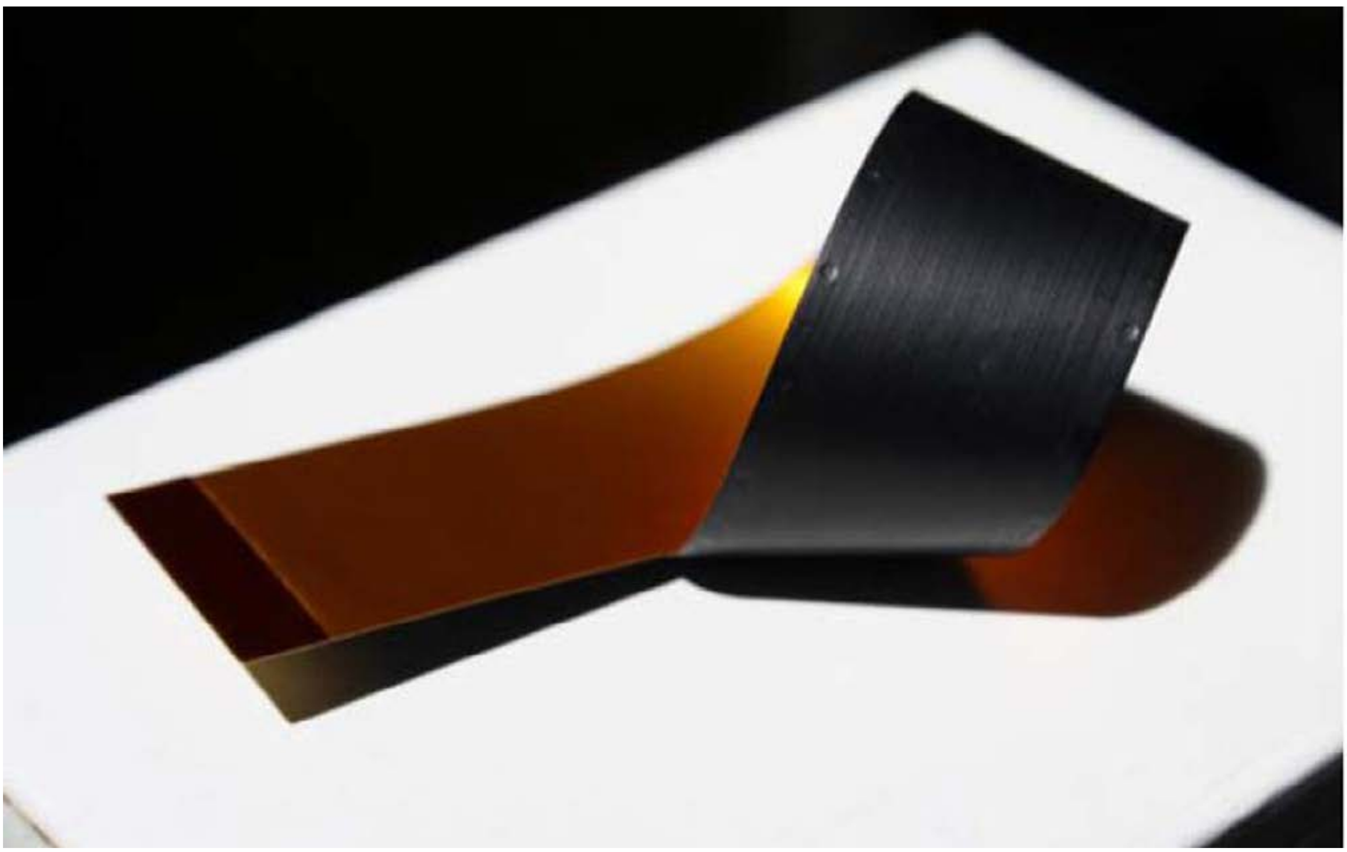


Figure 3: A ZIF flex circuit with two copper layers and a shielding film.

Figure 4: Depressions showing where cover films allow electrical connection to the shielding film.

Figure 5: A nearsighted ninja designer with ninja flex circuits.

Big thanks to iFixit for permission to use the iPhone 5s teardown images and to Robert Jung at Altaflex for his help with sourcing info. **PCBDESIGN**

References

1. DuPont
2. CEN
3. Innox
4. Doosan
5. iFixit
6. Altaflex

Tom Woznicki is the president of Flex Circuit Design Company, a consulting company in San Jose, California, that specializes in designing flexible printed circuits for OEMs and flex circuit manufacturers. For more information, visit www.flexdude.com.

Surface Finishes for High-Speed PCBs

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD AUSTRALIA

The Nickel Doesn't Make Cents!

PCB surface finishes vary in type, price, availability, shelf life, assembly process and reliability. While each treatment has its own merits, electroless nickel immersion gold (ENIG) finish has traditionally been the best fine pitch (flat) surface and lead-free option for SMT boards over recent years. But unfortunately, nickel is a poor conductor with only one third the conductivity of copper. Also, nickel has a ferromagnetic property that can adversely affect electromagnetic fields in the high-frequency domain.

The PCB industry has addressed the issue of the ferromagnetic properties of nickel by introducing a nickel/gold (NiAu) alloy. Gold is slightly less conductive than copper, and has no ferromagnetic properties, so it has relatively little impact on the conductor's loss characteristics at high frequencies.

Microstrip (outer) layers of a multilayer PCB suffer from wide variations in both trace width and thickness. This is due to the additional fabrication process of electroplating the through-holes. Copper barrel thickness is generally specified as a minimum of 1 mil (25.4 µm), and so extra copper plating is applied to the surface in order to produce the correct barrel wall thickness. This, unfortunately, is also added to the traces. But as the thickness and width varies, so does the impedance. This is one of the reasons why routing controlled impedance signals, on the microstrip layers, should be avoided.

It is also very important not to pour copper fills on the signal layers of the board, as these will dramatically change the impedance of the traces rendering the impedance control ineffective.

A surface finish can be defined as a coating, either metallic or organic in nature, which is applied to a PCB in order to assure solderability of the metal underneath after a certain time in storage and under different environmental conditions. The surface finish protects the copper mounting pads from corrosion in order to ensure good soldering. There are several choices of protective coatings:

- Lead-free hot air solder leveling (HASL)
- Organic solderability preservative (OSP) such as Entec 106 or Shikoku
- Electroless nickel immersion gold (ENIG)
- Electroplated gold over nickel (NiAu)
- Immersion silver (IAg)
- Immersion tin (ISn)

Electroplated NiAu can be applied as a full body finish, prior to solder mask application. This is because for that finish, the NiAu layer is typically used as a resist to etch the outer layers. Therefore, nickel ends up covering the entire length of the outer layer traces. But as mentioned, nickel is a ferromagnetic material and is not desirable from a loss perspective.

Alternatively, ENIG can be applied as a full body finish, or the more common (and recommended) way as a selective finish, because it is

applied after solder mask. In the second case, nickel is not plated on the traces. However, the lands are protected by the plating, which are exposed to the environment prior to assembly. This is described as solder mask over bare copper (SMOBC) processing and is best for high-speed design. Plus, given the cost of gold, liquid photoimageable solder mask (LPISM) should be applied before the ENIG process in order to limit the plating area.

In a microstrip configuration (Figure 1), electric fields (blue) exist between the traces and the reference plane. At high frequencies (> 1 GHz), the current density (red) tends to build up on the edges and surface of the plating due to the skin affect. Therefore, special consideration should be incorporated for any edge-coupled structures on the outer layers. This impact is more prevalent when ENIG is applied over all copper features, such as application of ENIG before solder mask. But, this effect can be minimized when only the pads (not the traces) are plated with the finish.

ENIG plating thickness is specified by the IPC-4552 standard for ENIG plating for printed circuit boards. The gold plating is typically very thin, 0.075–0.125 µm. On the other hand, nickel plating, which is used to stop copper migration to the gold, normally requires 3–6 µm thickness. These thicknesses can vary between fab shops and processes. Higher gold thickness requires extended solution dwell time or increased solution temperature.

Plating finishes can impact the overall loss of the transmission line due to lower conductivity metallization that covers the surface of the trace. Silver plating is an exception to this increased loss effect due to its similar conductivity to copper. This finish, however, is not as preferred as other finishes like ENIG or immersion tin.

As electronic devices have miniaturized, so too have their chip package sizes, along with the size and clearance of lands. As a result, short-circuit risk has increased. The plating of fine patterns (15 µm spacing) clearly favors palladium/gold (Pd/Au) or EPIG. This is also a solution to the lossy transmission lines, since this finish contains no nickel. EPIG is also ideal for wire bonding applications.

Generally, most of the surface treatment dissolves into the solder paste or wave solder during the soldering process and the solder joint is formed between the solder and the copper. One exception is ENIG, where the solder dissolves the thin layer of gold and forms a joint with the underlying nickel alloy.

The immersion systems process (Figure 2) uses a chemical displacement reaction to deposit a metal layer onto the exposed copper surface of the PCB. The base metal (copper) donates the electrons that reduce the positively charged metal ions present in solution. The immersion layer will continue to grow however, as the thickness of deposit increases, the rate of deposition falls. Therefore, the process is self-limiting.

There are four design concerns associated with ENIG plating:

1. Solder mask defined BGA pads should be avoided, due to the risk of:

a) Brittle joints (the inter-metallic compound that is formed when soldering against nickel).

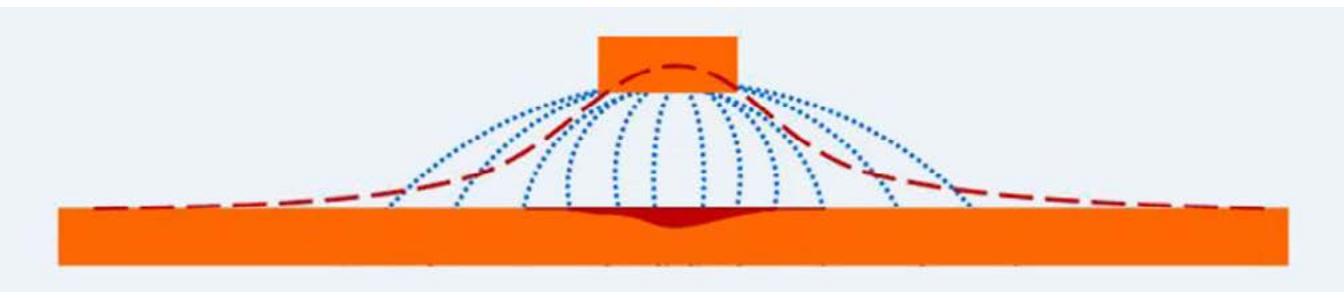


Figure 1: Microstrip electric fields and return current distribution.

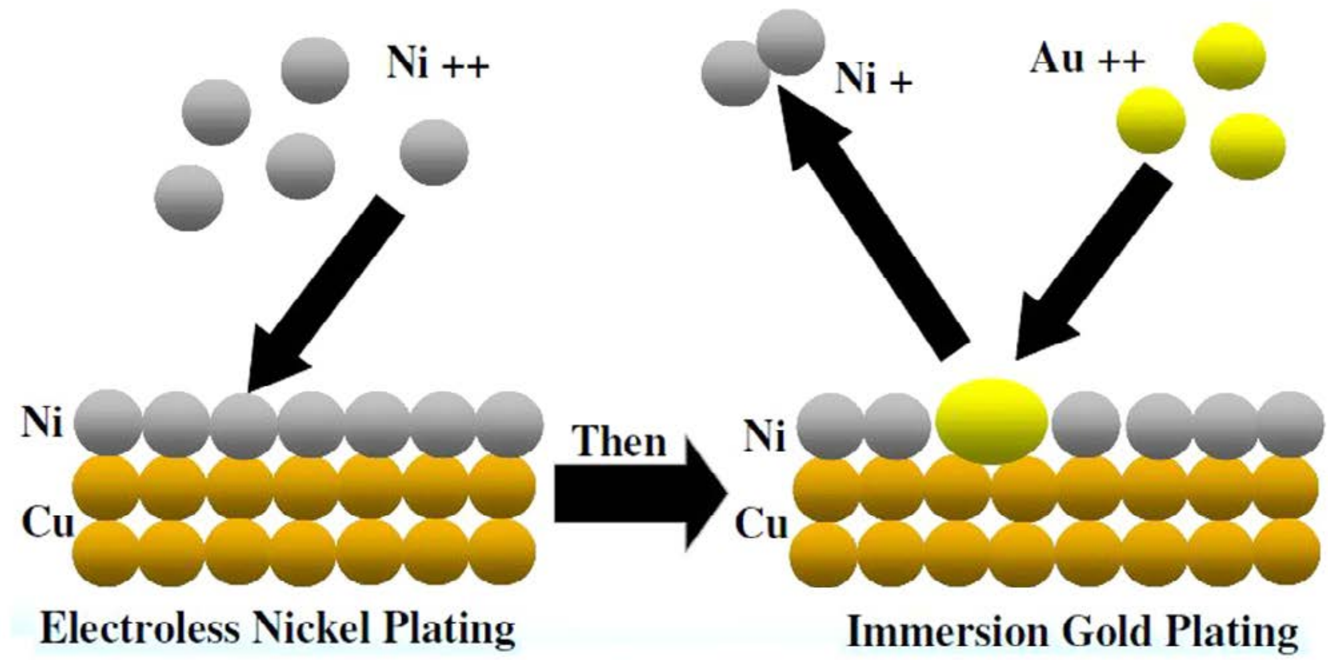


Figure 2: Immersion systems process (courtesy of Multitek).

b) The risk of black pads (lack of balance within the ENIG plating chemistry) especially on smaller BGA pads.

2. Single-sided plugged via holes:

As with most finishes, via holes plugged from one side partially plugged—are not recommended. Also, placing holes very close to SMD pads is not recommended, since the plating solution can become trapped inside and may contaminate or reduce the solderability of the joint.

3. Solder mask bridges between SMD pads:

As with immersion tin, this treatment is aggressive towards the solder mask. Therefore, larger solder mask bridges may be necessary, on fine pitch SMT components, at some fab shops.

4. The impact on the conductor's loss characteristics at high frequencies:

Selection of low-loss plating is just as critical as the selection of dielectric material when designing high frequency circuits.

At approximately 2.7 GHz, the resonant behavior of the nickel component in ENIG increases insertion loss. This resonance is attributed to the ferromagnetic properties of the nickel layer. It is therefore wise to avoid using full body ENIG coating of microstrip traces at high frequencies. In fact, it may just be an odd 3rd or 5th harmonic that falls on this particular lossy region and causes radiation with much lower fundamental frequencies. Therefore, solder mask over bare copper (SMOBC) processing should be considered for all high-speed designs.

Points to Remember

- Electroless nickel/immersion gold (ENIG) finish has traditionally been the best fine pitch (flat) surface and lead-free option over recent years.
- Nickel has a ferromagnetic property that can adversely affect electromagnetic fields in the high frequency domain.
- The issue of the ferromagnetic properties has been addressed by introducing a nickel/gold (NiAu) alloy.
- Microstrip (outer) layers of a multilayer PCB suffer from wide variations in both trace width and thickness, hence impedance variations.
- It is very important not to pour copper fills on the signal layers of the board.

• ENIG can be applied as a full body finish, or the more common (and recommended) way as a selective finish, because it is applied after solder mask.

• The plating of fine patterns (15 µm spacing) clearly favors palladium/gold (Pd/Au) or EPIG.

• Solder mask over bare copper (SMOBC) processing is best for high-speed design.

• There are four design concerns associated with ENIG plating: solder mask defined BGA pads, single sided plugged vias, solder mask bridges and the impact on the conductor's loss characteristics at high frequencies.

• Selection of low-loss plating is just as critical as the selection of dielectric material when designing high-frequency circuits.

• The resonant behavior of the nickel component in ENIG increases insertion loss at 2.7 GHz. Avoid using full body ENIG coating of microstrip traces at high frequencies—SMOBC processing should be considered for all high-speed designs. **PCBDESIGN**

References

1. Barry Olney: Beyond Design: Ground Pours, Beyond Design: Electromagnetic Fields, Part 1, Beyond Design: Electromagnetic Fields, Part 2, Beyond Design: Mythbusting - There are no One-way Trips!

2. IPC-4552 – Specification for ENIG Plating for Printed Circuit Boards

3. Henry Ott: Electromagnetic Compatibility Engineering

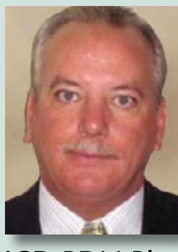
4. Howard Johnson: High-Speed Signal Propagation

5. John Coonrod, Rogers Corporation: "Understanding PCBs for High-Frequency Applications," Printed Circuit Design & Fabrication

6. Yuriy Shlepnev, Simberian Electromagnetic Solutions, Simbeor Application Notes

7. Dr. Al Horn, Rogers Corporation, "Increased Circuit Loss due to Ni/Au"

The ICD Stackup Planner and PDN Planner can be downloaded from www.icd.com.au



Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD), Australia. This PCB design service bureau specializes in board-level simulation, and has developed the ICD Stackup Planner and ICD PDN Planner software. To read past columns, or to contact Olney, click here.

2D Transistor Paves Way to Faster Electronics

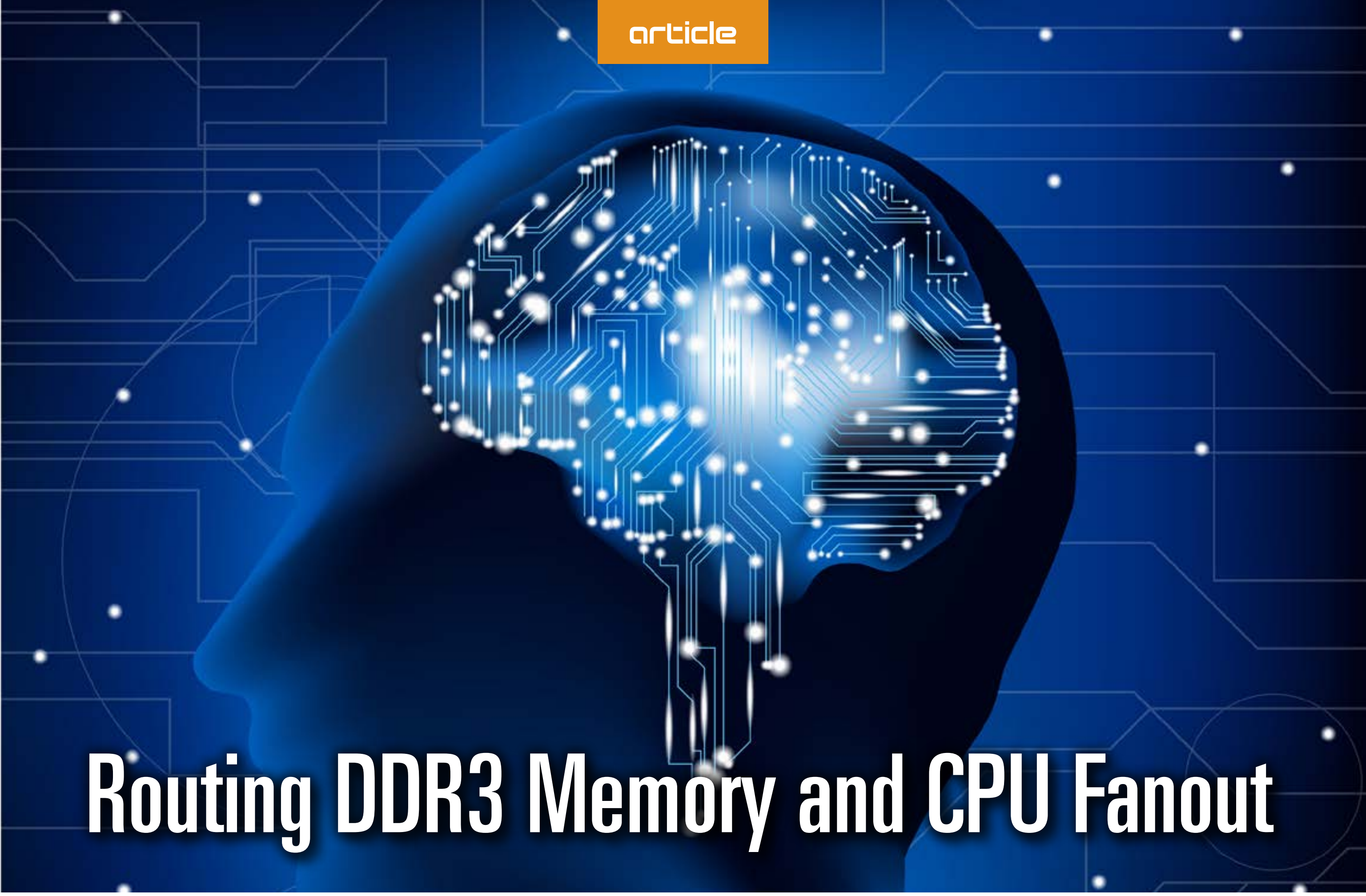
2D Transistor Paves Way to Faster Electronics

Faster electronic device architectures are in the offing with the unveiling of the world's first fully two-dimensional field-effect transistor (FET) by researchers with Lawrence Berkeley National Laboratory (Berkeley Lab). Unlike conventional FETs made from silicon, these 2D FETs suffer no performance drop-off under high voltages and provide high electron mobility, even when scaled to a monolayer in thickness.

Ali Javey, a faculty scientist in Berkeley Lab's Materials Sciences Division and a UC Berkeley professor of electrical engineering and computer science, led this research in which 2D heterostructures were fabricated from layers of a transition metal dichalcogenide, hexagonal boron nitride and graphene stacked via van der Waals interactions.

"In constructing our 2D FETs so that each component is made from layered materials with van der Waals interfaces, we provide a unique device structure in which the thickness of each component is well-defined without any surface roughness, not even at the atomic level," Javey says.

For the 2D FETs produced in this study, mechanical exfoliation was used to create the layered components. In the future, Javey and his team will look into growing these heterogeneous layers directly on a substrate. They will also look to scale down the thickness of individual components to a monolayer.



Routing DDR3 Memory and CPU Fanout

by Robert Feranec
FEDEVEL

DDR3 memory is so pervasive, it's almost inevitable that professional PCB designers will use it. This article advises how to properly fanout and route DDR3 interfaces, even in very high-density and tightly packed board designs.

DDR3 Design Rules and Signal Groups

Everything starts with the recommended high-speed design rules for routing DDR3 in groups. During DDR3 layout, the interface is split into the command group, the control group, the address group, as well as data banks 0/1/2/3/4/5/6/7, clocks, and others. It's recommended that all the signals belonging to the same group be routed "the same way," i.e., using the same topology and layer transitions.

As an example, consider the routing sequence shown in Figure 1. All the DATA 6 group signals go from layer 1 to layer 10, then to layer 11, and after that to layer 12. Every signal within the group makes the same layer transi-

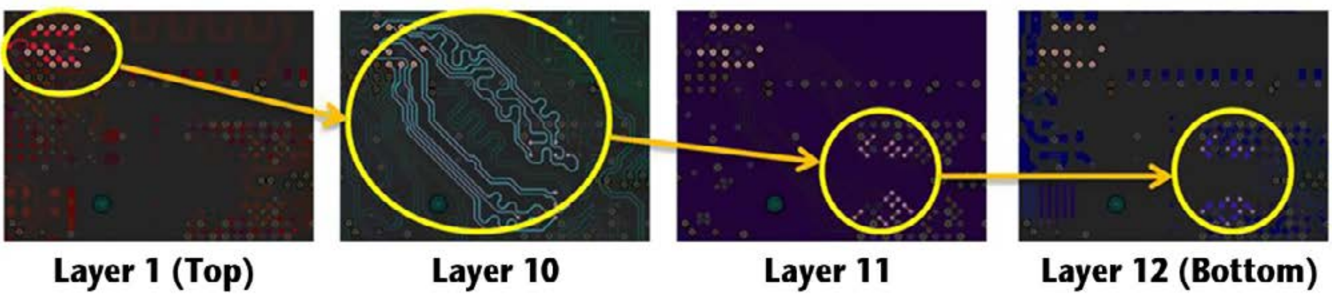


Figure 1: All signals in the DATA 6 group are routed "the same way," using the same topology and layer transitions.

tions and generally takes on the same routing distance and topology.

One of the advantages of routing the signals this way is that during length tuning (a.k.a. delay or phase tuning), the Z-axis length in the vias can be ignored. This is because all the signals routed "the same way" will have exactly the same via transitions and lengths through

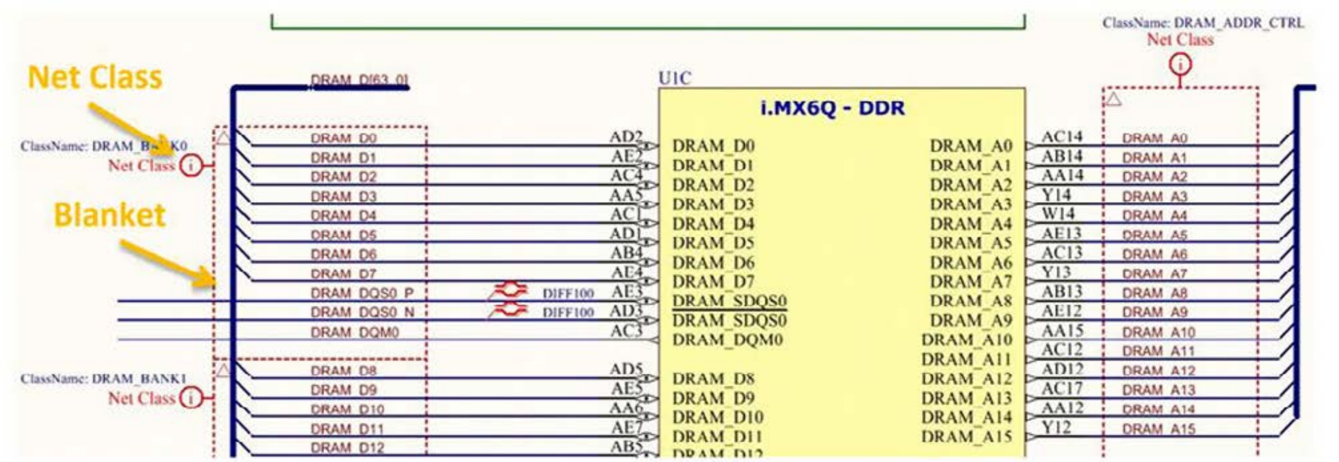


Figure 2: Signal groups are created on the project's schematic for DDR3 routing.

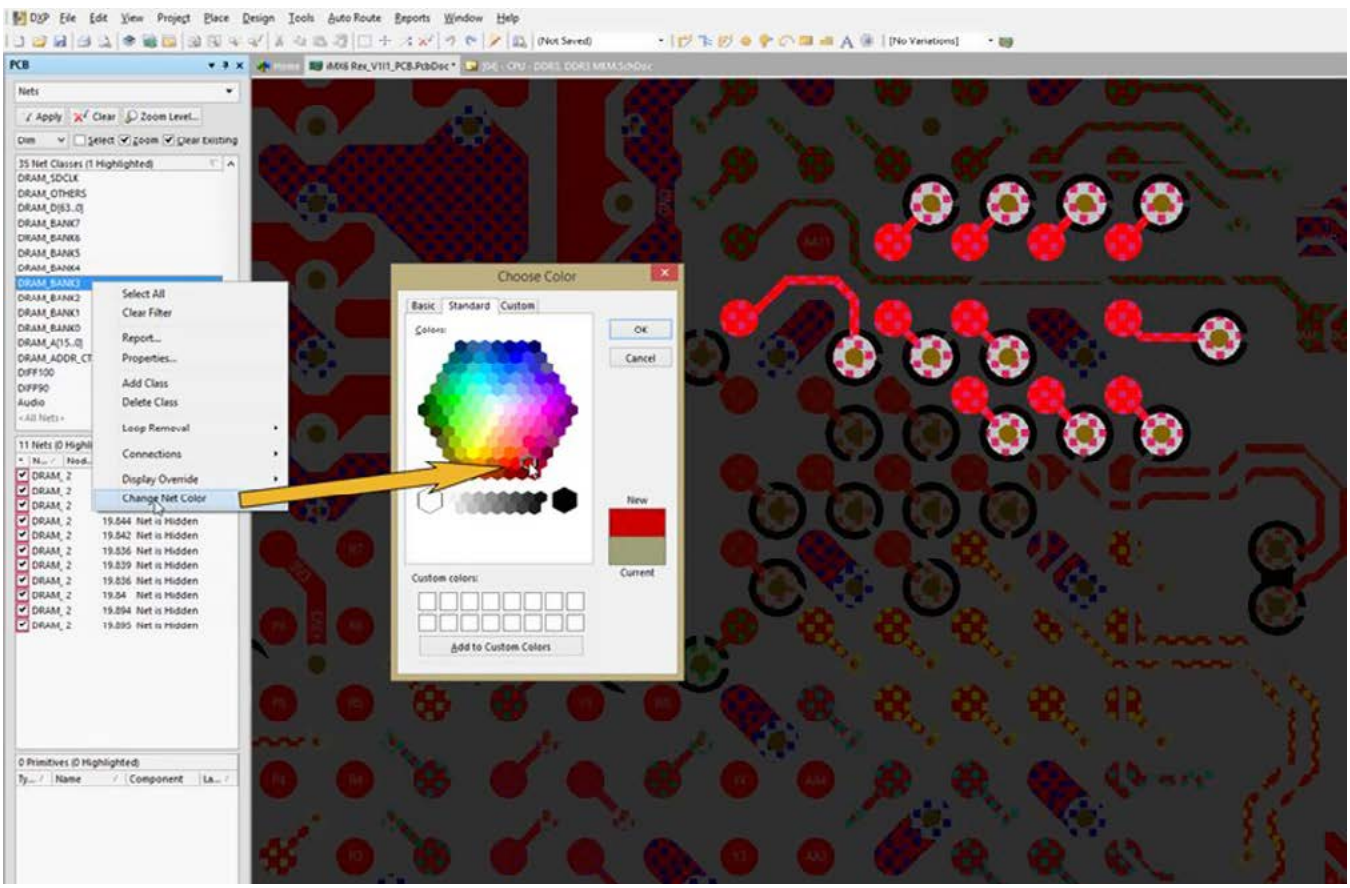


Figure 3: Assigning different colors to each group can make routing easier to mentally follow.

vias. (You can be more liberal with vias as long as their lengths are taken into consideration for length matching).

Creating DDR3 Groups

The first step before routing is to create the necessary signal groups, which can be done from the project's schematic. The specific method for doing this will depend on how constraints are managed in the software, but nets that are constrained and grouped will follow certain rules during routing. An example of a system for signal grouping on the schematic is shown in Figure 2. Most ECAD tools have a "net class" feature for this kind of net grouping.

After bringing the signal groups to the PCB, it's very useful to assign different colors to each group, to make routing easier to mentally follow, as shown in Figure 3. Once a distinct color has been chosen for each signal group (net class), it's time to fanout the DDR3 interface of the CPU.

Choosing the correct via style for a particular memory group and deciding how PCB layers

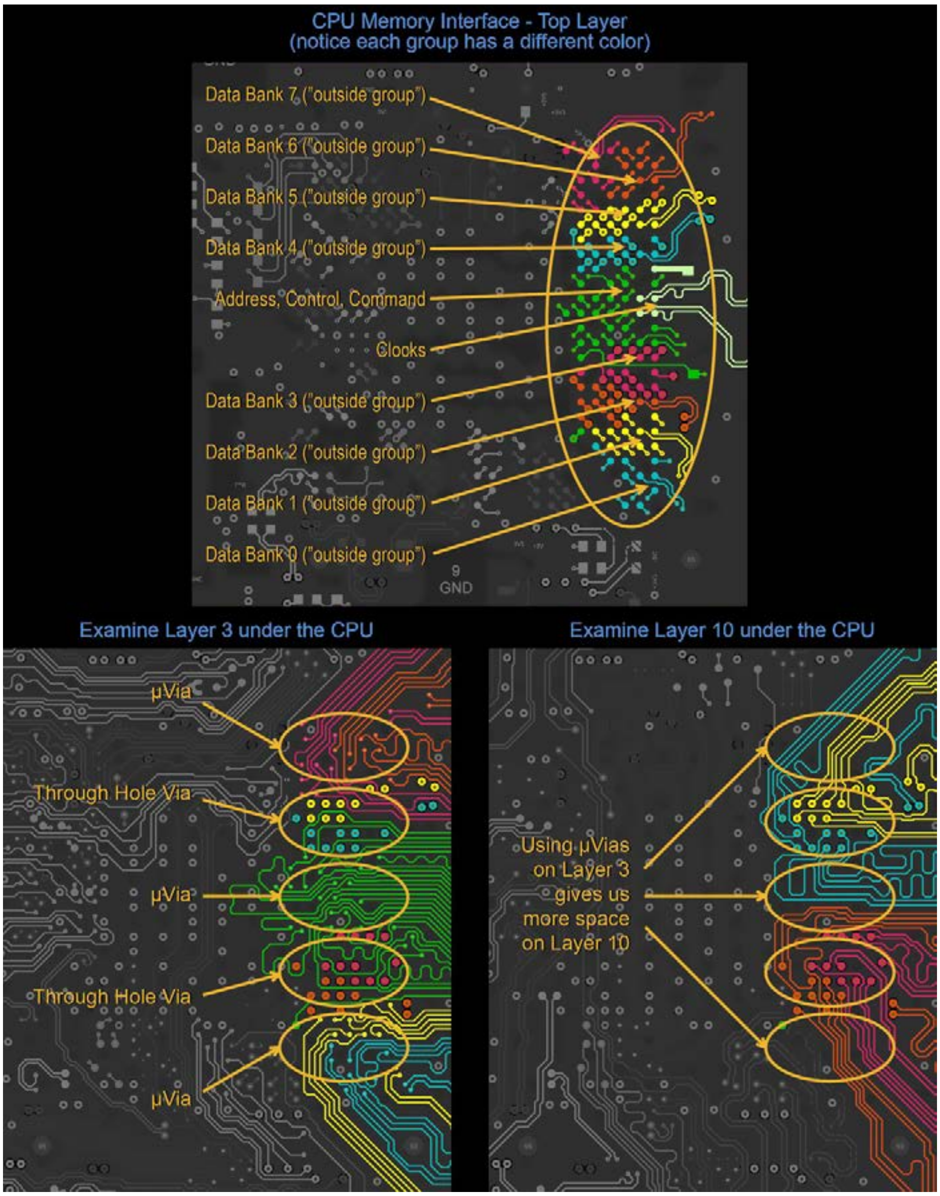


Figure 4: Choosing a correct via size can help save space for more tracks.

will be used can make DDR3 layout much easier. Assigning different colors to each memory group helps to visually define the interface. A microvia (µvia) takes less space than a through-hole via, allowing for fanout of more tracks in the same area. Microvias also save space on other layers, and the extra free space there can then be used for routing.

Why use Microvias for Address, Command, and Control Signals?

The address, command, and control groups have the highest number of signals out of the memory groups. If through-hole vias are chosen when routing these groups, a lot of space will be wasted on every layer. By instead choosing

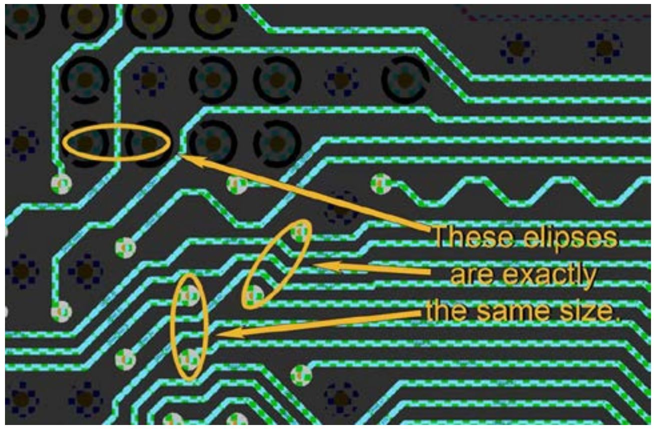


Figure 5: Two or three tracks can fit between microvias, in the same space needed for one track between through-hole vias.

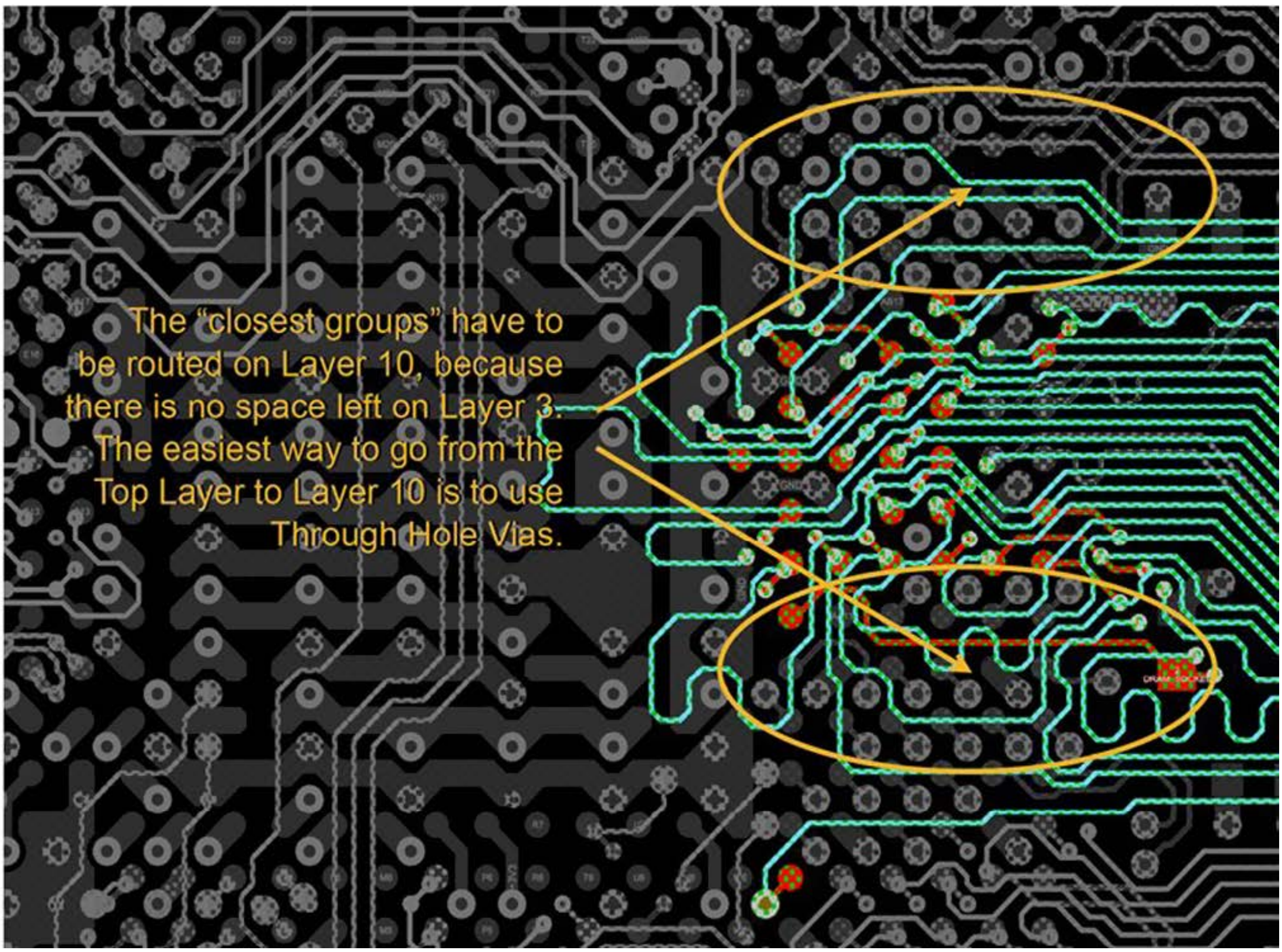


Figure 6: Some of the address, command, and control tracks have to be routed under the pads of the closest groups.

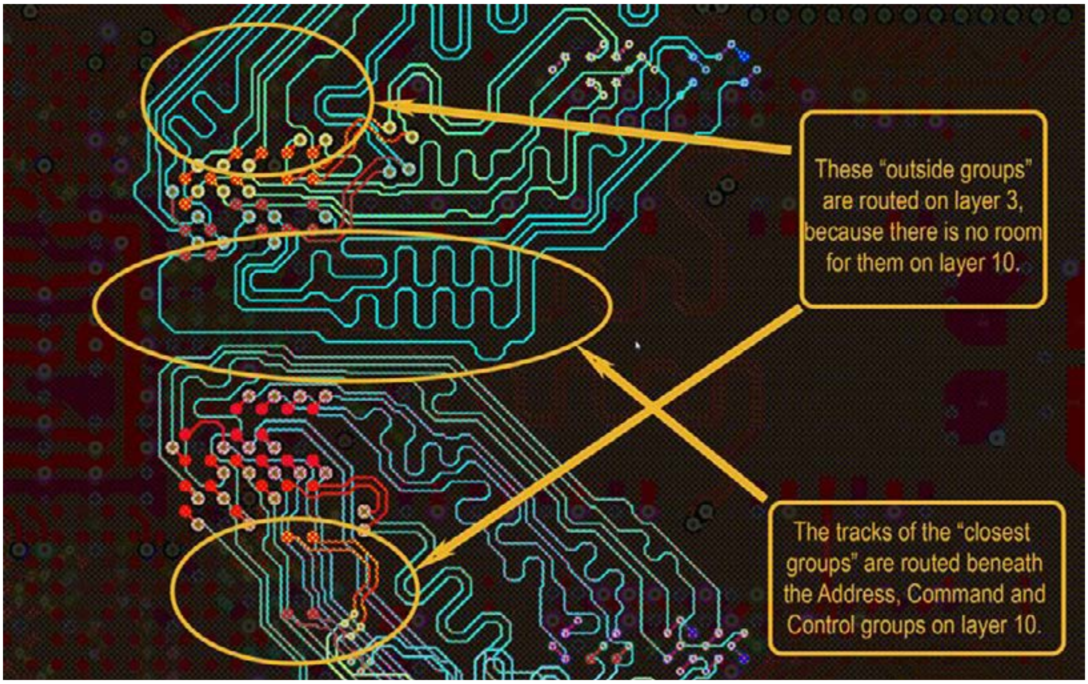


Figure 7: Free space left under groups routed with microvias can be used to fanout the closest groups.

microvias, only the space on layer 3 is needed, and because microvias have smaller diameter holes, there is more space to fanout the signals on layer 3.

Why use Through-Hole vias for Closest Groups to Address, Command, and Control Group?

Signals from the data banks can be organized into "closest groups" and "outside groups," making it easier to distinguish between routing in the center of the board (on the X-Y plane), and the outer edge of the board. Some signals of the address, command, and control group will need the space below the closest groups.

When a signal from the address, command, or control group is routed through a microvia on layer 3, there will be free space left under this group on layer 10. This space can be used to fanout the closest groups.

Why do Outside Groups use Microvias?

From Figure 7, it's clear that there won't be space left on layer 10 to fanout the outside groups. So placing the outside groups on layer 3 and using microvias is the result of the fanout planning. Note: The same layer/via/microvia fanout planning technique can also be applied to the other interfaces (e.g., PCI, ISA, etc.). This way, even really packed and very dense designs can be routed.

Conclusion

With a bit of care and planning ahead, routing and length tuning DDR3 fanouts can be a stress-free process, even on the most compact and densely packed designs. The iMX6 Rex is a terrific example of this care and planning, designed in part as a tool for showing how it's done. By following these steps, any DDR3 design can be completed in far less time and with much greater likelihood of designing it right the first time. The complete project reference design including schematics and PCB documents with the DDR3 layout from the screenshots can be downloaded at www.imx6rex.com/. **PCBDESIGN**

Robert Feranec is a contract design engineer and training instructor. He can reached through his website, www.fedevel.com.

Design Rule Checks: For Your Protection

by Tim Haag
INTERCEPT TECHNOLOGY

I have designed multitudes of printed circuit boards over the years, but I have a confession to make: It can be hard for me to run that final design rule check.

I know that it is important, but at the end of a long design cycle, I just want to be done. I don't want to redo anything, and I sure don't want to look at my own errors. Do any of you feel that way? As the manager of customer support for my company, I have helped users who have run into problems because they didn't run their DRCs. So I'm guessing that I'm not the only one who has contemplated skipping this particular step.

DRCs exist as a barrier of protection for designers, and whenever I contemplate skipping some of those protections, I think back to the story of one of my heroes. I have a great love of aviation, and through the years I have studied its different aspects from aircraft specifications to the exploits of courageous pilots. My heroes have included pilots like Eddie Rickenbacker, Wiley Post, Amelia Earhart and Chuck Yeager, just to name just a few. One of the pilots who most intrigued me was America's No. 2 air ace of WWII, Major Thomas B. McGuire.

Tommy McGuire amassed 38 confirmed victories while flying the P-38 Lightning, the big twin-tail, twin-engine fighter built by Lockheed. McGuire was said to have "the pilot's touch" and those who flew with him claimed that he could do things with the P-38 that were impossible for ordinary pilots. He was known as "the best of the best," and in addition to being a phenomenal pilot, he was also a great teacher. He taught other pilots from his own experiences, not only in the skills of aerial attack, but how to be prepared for the unexpected.


SHAX Engineering
Delivering peace of mind to the industry since 1998.
hp
LOCKHEED MARTIN
maxim integrated
GENERAL DYNAMICS
intel
cadence
These are just some of the major corporations that have placed their trust in us.
SATISFACTION GUARANTEED
100%
SATISFACTION GUARANTEED
VIEW SHAX SERVICES
PCB Design. Fabrication. Assembly.
www.shax-eng.com | 800.814.3680

Above all else, Tommy McGuire had three cardinal rules for P-38 pilots:

1. Never attempt combat at low altitude.
2. Never let your airspeed fall below 300 miles per hour.
3. Never keep your wing drop-tanks in a fight.

The P-38 was a big, heavy airplane that performed poorly when going low and slow. Therefore, these three rules were designed to protect pilots from getting into a situation that would limit their productivity, or in a worst-case scenario, cost them their lives.

So why would McGuire's story come to mind when I would consider skipping a DRC?

McGuire's first rule of never attempting combat at low altitude could be translated as, "Are we positioned to be the most successful?" Are we being pushed by a schedule to finish a PCB design before it is really ready to finished? Are we allowing another agenda to force us into a bad decision regarding doing a final thorough check of our design? Obviously schedules are important, but there are usually more creative ways around a schedule problem than rushing a design to completion that is not ready. Have we become trapped? Do we need to reposition ourselves for success?

McGuire's second rule was to ensure that his pilots kept their combat air speeds at a peak level so that they would be prepared. How about PCB designers: Are we prepared? When we are tempted to skip that DRC, is it because we have become distracted from the job at hand? Do we want to just get through it so that we can move on to the next project or head home for the weekend? New projects and weekends are all fine. But is our attention where it really needs to be—on completely finishing the project in front of us? Are we prepared?

The third rule was to always drop external fuel tanks during combat. Although they provided extra fuel, those tanks hurt the plane's performance because they were an aerodynamic drag on the plane. How about PCB designers? Is there anything dragging us down and getting in the way of us performing at our best? Is it annoying to run those DRCs because we don't really understand the results? Are we tempted to avoid the DRC because the corrections they highlight will cause us to undo what took a lot of effort to do initially? Does it hurt our pride just a little to find out that we've made some errors along the way?

My hand is up here. I know that I've run into all of these emotions. But it is important to put aside annoyance, frustrations, and pride in order to do what's best for the design. I've had to take the time to understand how to best work with DRCs and correctly interpret what the reports are telling me. Sometimes we just need to jettison those attitudes that drag us down so that we can be at our best.

So I think that we could all agree that these are important rules. But are they really so serious? To McGuire they were.

Towards the end of the war, McGuire and three other pilots were on a mission deep into enemy territory when an enemy plane was sighted. The P-38s were configured for distance on this mission: low and slow and carrying drop tanks. This mission was important to McGuire; with a few more victories, he would become the leading American air ace of the war. He would return home a hero with the Medal of Honor hung around his neck.

Maybe the pressure of success clouded his thinking. After all, here were four of the best pilots taking on a lone enemy plane. Perhaps he may have thought that with all of these factors in his favor, the odds were with them and he could afford to bend his own rules this one time. So he gave the order to attack, at low altitude, at low speed, and with the drop tanks still attached.

But the supposedly helpless enemy pilot was just as good as McGuire, if not better. With 80 eventual confirmed victories, Warrant Officer Akira Sugimoto was an instructor pilot with over 3,000 hours in his aircraft, and he was also the best of the best. And what had started out as an easy victory for McGuire's group now turned into a totally unexpected fight for survival.

Before anyone could react, Sugimoto destroyed one P-38 and then flicked through a roll and was after a second one with guns hammering. The big P-38s, low, heavy and flying slower than they should have been, were in perfect position for the enemy ace. The radio exploded with shouts of help from the next victim in the enemy's sights.

And then it happened.

McGuire had already violated not one but all three of his cardinal rules of combat. Now eager to help, he pulled his plane around in a maneuver that tried to beat the invisible laws of aerodynamics. But the big Lightning simply could not turn that tight at such a slow speed with the extra weight of the drop tanks still attached, and it aerodynamically stalled. Plunging out of control towards the ground below he almost pulled it out, but there simply wasn't enough room. A blinding sheet of flame in a lonely jungle setting marked the end of Tommy McGuire.

He wasn't engaged with the enemy, and no one was shooting at him. But still McGuire died because he thought that he could just this one time break those very rules that he instituted for protection. He died because he didn't believe that there was anything out there that could beat him.

How about us? Do we understand that there is always something out there that is waiting to snare us if we give it opportunity? Do we understand that DRCs are designed to protect us and not to frustrate us? CAD vendors build those rules into our tools so that we can be prepared for the unexpected; we just have to take the time to use them. McGuire's story has always served as an example to me when I am tempted to rush a job to completion as to what can happen if I disregard those very rules that are designed to protect me.

Now, I have a couple of DRCs that I need to run. **PCBDESIGN**

Tim Haag is customer support and training manager for Intercept Technology.

---

video interview

Supplying Unusual Flex to Meet Customers' Needs

by Real Time with...
IPC APEX EXPO

Pete Uka, director of technology at Q-FLEX Inc., joins Guest Editor Joe Fjelstad to discuss a few of the unusual flex solutions his company provides to various customers. In business since 1998, Q-FLEX is moving into specialty products says Uka, including a 15-foot flex board, an 8-foot, 8-layer flex board, heat-sensing products, and extremely tiny flex products smaller than a grain of rice.

CLICK TO VIEW

realtimewith.com

Mil/Aero007 News Highlights

Saab to Acquire ThyssenKrupp, Inks MoU

ThyssenKrupp Industrial Solutions AG, a subsidiary of ThyssenKrupp AG, and Saab AB have signed a non-binding Memorandum of Understanding concerning the sale of the Swedish shipyard ThyssenKrupp Marine Systems AB (formerly named Kockums) with operations in Malmö, Karlskrona and Muskö to Saab AB.

Celestica Earns Supplier Award from Rockwell Collins

The company was presented with the 2014 Assemblies Supplier of the Year Award from Rockwell Collins during the recent Annual Supplier Conference held in Cedar Rapids, Iowa. The award is an acknowledgement of significant contributions made during the year by suppliers and is based upon quality, delivery, total cost of ownership, lead time, and customer service.

Sypris' Electronics Group: 16% Revenue Increase in Q1

The company reported revenue of $84.2 million for the first quarter compared to $78.4 million for the first quarter of 2013. Revenue for the Electronics Group expanded 16%, to $8.4 million in the first quarter, an increase of $1.1 million from $7.3 million in the comparable prior year quarter.

Ducommun Posts 2.2% Sales Increase in Q1

"The first quarter of 2014 showed the diversity of Ducommun's product portfolio and strength of our operating leverage," said Anthony J. Reardon, chairman and CEO. "Top line growth was driven by gains across our commercial aerospace business, including a further pickup in revenue with Airbus and on Boeing's 787 platform. Operating margins expanded year-over-year reflecting higher revenues and an improved product mix."

Digicom Electronics Earns ITAR Registration

"Our attention to quality and detail and our ability to handle low-volume, high-end products has enabled us to prototype and manufacture many military, aerospace, and government-related projects," explained GM Mo Ohady. "Receiving ITAR certification expands the base of projects we can accept and gives more flexibility to our customers."

OnCore Implements E2open's Supplier Collaboration Solution

OnCore Manufacturing LLC, a global supplier of EMS, has implemented E2open's supplier collaboration solution. E2open enables OnCore to offer more effective customer service, given the real world of unpredictable demand.

Cal Quality to Support Boeing's CSEL Program

The company announced a multi-year, multi-million dollar contract signing with The Boeing Co. to support the Combat Survivor Evader Locater (CSEL), a major government program that is the U.S. Department of Defense (DoD) Program of Record for Joint Search and Rescue.

API Nets $1.6M Filter Solutions Order

"Our customers in the electronic warfare arena demand products that address a broad set of technical requirements with unparalleled performance and reliability," said Bel Lazar, president and CEO, API Technologies. "As a supplier to both emerging and existing EW platforms, our products are helping save the lives of warfighters from our armed forces and allies around the world every day."

Probe Order Backlog Rises to $3M; To Meet Target

Kam Mahdi, CEO, commented, "Our order backlog puts us in excellent position to meet our revenue target of 20% year-over-year growth by the end of 2014. We strongly believe that manufacturing for small- to medium-sized businesses is coming back to the U.S. We have witnessed businesses of this size become adversely affected by the subsequent increase in costs, unmanageable processes, and intellectual property protection concern of manufacturing abroad."

Nortech: Modest Growth in Q1; Continues Improvements

"We're pleased to see our pretax profits rise 13% on a slight sales increase," said Rich Wasielewski, president and CEO. "While the mixed economic recovery is impacting each of our customers differently, we continue focusing on improving our operating performance by managing the areas under our control and actively working on building a qualified sales pipeline."

Prototron Circuits

Quality printed circuit boards

America's FASTEST Board Shop

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

"I have been using Prototron for more than two decades. This includes working for four different companies and a few more as a consultant. Throughout this time period, Prototron has never let me down and has always made me look good as an employee and a consultant. I have never gotten a single board error in the hundreds of different boards I have ordered. This includes simple two layer boards, complicated 12 layer boards and everything in between. They are extremely consistent in delivery times. They can handle large quantity jobs and extremely fast turn around if you need it.

Over the years I have had a variety of difficult and unusual mechanical issues on my boards. Recently I have had a board set with unusual hole patterns that needed to be routed in half. They were very helpful with DFM consultation on this project and have again been very consistent with perfect quality. Many board houses would not even quote this job and the one time I ordered the identical board from another "production-only" vendor, it was a disaster. Late delivery and poor quality.

The entire staff is very friendly and even though they do a very large volume of business, you feel as if it is the "little board house around the corner". My current company does a fair amount of business in Denver and I was impressed when I was talking to an assembly shop in Denver and without knowing anything about me or where I lived, they waxed on about the unique abilities of Prototron.

Great attitude, good prices, fantastic service, 100% perfect quality...what's not to like?

Bill Barnard
Principal Engineer and All Round Good Guy
dBMEDx Inc.

dBMEDx

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

When is Controlled Impedance Important?

by John Coonrod
ROGERS CORPORATION, ADVANCED CIRCUIT MATERIALS DIVISION

The importance of controlled impedance hinges upon many variables, such as the PCB's characteristics and how it is to be used. A PCB designed for digital applications will often have different impedance requirements than a circuit designed for RF applications. Within both of these categories, however, there are sub-categories of specific types of applications.

Digital applications, especially high-speed digital applications, will require consistent and controlled impedance values for signal integrity purposes. There are many different methods for checking a PCB for good signal integrity: eye diagrams, pulse distortion, bit error rate, and skew. A critical trace on a digital board with impedance variation can impact these quality measurements of the PCB.

Typically, when energy of a varying signal goes from one impedance environment to another, there will be some amount of reflection in the transition. A digital pulse going from one area of a PCB with 40 ohms impedance to another area with 50 ohms will have some reflected energy at that transition. Along with that, the pulse amplitude and shape may vary as well, depending on the significance of the reflection. The pulse distortion will impact the previously mentioned signal integrity concerns for a high-speed digital system.

Another issue with an unmatched impedance area of a PCB for digital systems is electromagnetic interference (EMI). The reflection associated with the impedance mismatch will cause electromagnetic radiation in the localized area of the transition. That radiation can couple its energy to neighboring traces or components on the board, causing electrical performance distortion of those items.

Impedance matching for RF applications sometimes share similar issues associated with high-speed digital applications. Many times in RF applications, there is a need for efficient energy transfer from one module to another. A simple example would be to consider the energy generated within the transmitter portion of a radio and how to transmit that energy to the antenna efficiently. If the feed line from the transmitter to the antenna is not well matched for impedance, some energy will be lost before it can be transmitted. The transmitting portion of the radio will not function as well as it should, possibly shrinking the distance at which the signal can be received decreasing the clarity of the signal received.

There is a large study within the RF/microwave and millimeter-wave design community devoted solely to impedance matching. Besides the simple radio transmitting example given, there are other issues with power amplifiers, radar, low noise amplifiers (LNA), and many more. Within each of these applications there may be multiple areas where impedance matching is critical.

In the case of power amplifier circuits, the PCB typically has many different areas of functional-

WHEN IS CONTROLLED IMPEDANCE IMPORTANT? *continues*

ity. There are also multiple areas of impedance matching, and the impedance values can purposely change greatly from one area to another on the circuit. The power amplifier IC itself will often have input impedance of much less than 10 ohms, whereas the traces on the PCB will be 50 ohms. To get the energy to efficiently transition from 50 ohms down to less than 10 ohms, it is critical to have clean signal energy input into the power amplifier IC.

There are several different types of impedance. The most common type of controlled impedance on a PCB is referred to as characteristic impedance. Other types include input impedance, wave impedance, image impedance, and others. Many of the impedance issues are frequency-dependent, and this can be an issue for wideband power amplifiers. The impedance matching network that is necessary for the design of a power amplifier can only tune the impedance transitions over a range of frequencies. Many times the actual power amplifier IC can be used over a wider range of frequencies (wideband) than the tuning networks that are used to match the impedance for traces going into and out of the chip.

The impedance value of a PCB is dependent on many variables. In order of their significant impact on impedance, these variables include: thickness of the substrate, dielectric constant, conductor width and copper thickness. High-frequency circuit materials are formulated to have a tight thickness control as well as dielectric constant control. The same can be said for the laminate's copper thickness. But remember that this thickness can vary on the final circuit because of differences in PCB fabrication techniques. **PCBDESIGN**

John Coonrod is a market development engineer for Rogers Corporation, Advanced Circuit Materials Division. To reach Coonrod, click here.

video interview

PCB Auditing at the Next Level

by Real Time with... IPC APEX EXPO

What capabilities should you look for when you're selecting a new fabricator? Is a survey good enough, or should you require a full audit? Rob Scott of the auditing company Next Level PCB explains the criteria that he assesses when auditing a PCB supplier.

realtimewith.com


Why did you choose THAT fabricator?
Designers: when you choose a fabricator, what guarantee do you have that they won't deliver scrap?
We can connect you with manufacturers that we have audited and are qualified to deliver the product that you need, on time and on budget.
How we qualify PCB manufacturers
CLICK TO VIEW
NEXTLEVEL
A PCB AUDITING COMPANY
(360) 531-3495
www.nextlevelpcb.com

TOP TEN

News Highlights from PCBDesign007 this Month

1 Altium Designer Provides Improved Support, New Features

This update will bring Altium users improved Component Variant Support, which will give them the ability to design the schematic and PCB for drop-in replacement and alternative parts. This provides sizeable productivity gains for product variants, while also making it easy to source and manage multiple alternative parts from the supply chain.

2 Intercept Intros New RF Design Flow with Agilent's ABL

Intercept Technology, Inc., a leader in PCB/hybrid/RF electrical engineering applications, has launched an enhanced bidirectional link between its Mozaix schematic and Pantheon layout design applications and Agilent's ADS software.

3 IPC Releases Standards Committee Reports, Part I

These standards committee reports from IPC APEX EXPO 2014 have been compiled to help keep the industry up to date on committee activities. This is the first in a series of reports from the association, which include testing, cleaning and coating, product assurance, and PCB design.

4 T-Tech, Geek Group Partner on PCB Design, Prototyping

T-Tech, Inc. and The Geek Group have announced a joint agreement to offer a state-of-the-art solution and facility for rapid PCB design, verification, and fabrication. The collaboration will combine T-Tech's QCJ5 series Quick Circuit System and IsoPro Software for PCB prototyping with The Geek Group's expertise and Grand Rapids laboratory.

5 Mentor Exceeds Q1 Guidance; Expects Increased Demand

"First quarter results were better than our guidance, driven largely by strong revenue growth in scalable verification, particularly emulation," said Walden C. Rhines, chairman and CEO. For the second quarter of fiscal 2015, the company expects revenues of approximately $250 million.

6 Agilent Webinar to Focus on Packaging Reliability

Agilent Technologies has announced immediate online access to a highly informative webinar tailored specifically for scientists and engineers working in the semiconductor and electronic packaging industry.

7 Intercept Names Hankuk Valence Distributor in Korea

Intercept Technology Inc. announces its newest authorized reseller, Hankuk Valence Co. With over 25 years of experience in the electronics industry, Hankuk Valence plans to increase its customer base by expanding sales with Intercept's EDA software solutions.

8 Mentor Graphics Acquires Nimbic Inc.

"Nimbic's recognized solutions enable the industry to cope with increasingly higher-end complexity. We see joining Mentor Graphics as a natural fit with its own leadership in PCB and package systems design, global footprint, and extensive network of enterprise customers," said Raul Camposano, Nimbic CEO.

9 IEEE Connected Vehicles Conference Calls for Papers

Accepted papers will be presented at ICCVE 2014 in Vienna, Austria, November 3-7. Global experts, practitioners, and policymakers will present the latest innovations and breakthroughs on connected vehicles, while sharing insights, discussing economic, policy and social implications, and forecasting trends and opportunities.

10 Agilent Intros BGA Interposer for Probing DDR4 Designs

Agilent Technologies Inc. introduced two new interposer solutions for testing DDR4 and DDR3 DRAM designs with a logic analyzer. Both interposer solutions provide fast, accurate capture of address, command and data signals for debugging designs and making validation measurements.

events

For the IPC Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For a complete listing, check out *The PCB Design Magazine's* event calendar.

RAPID Conference & Exposition
June 9–12, 2014
Detroit, Michigan, USA

IPC SE Asia Workshop on Soldering of Electronics Assemblies
June 9, 2014
Penang, Malaysia

MedTech MD&M East
June 9–12, 2014
New York, New York, USA

IEEE ICC 2014
June 10–14, 2014
Sydney, Australia

CES Unveiled Warsaw
June 17, 2014
Warsaw, Poland

Upper Midwest Expo & Tech Forum
June 18, 2014
Bloomington, Minnesota, USA

CE Week
June 23–27, 2014
New York, New York, USA

Symposium on Counterfeit Electronic Parts and Electronic Supply Chain
June 24–26, 2014
College Park, Maryland, USA

Ohio Expo & Tech Forum
July 17, 2014
Cleveland, Ohio, USA

SusTech 2014
July 24–26, 2014
Portland, Oregon, USA

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

PUBLISHER: **RAY RASMUSSEN**
(916) 337-4402; ray@iconnect007.com

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

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

EDITORIAL:
GROUP EDITORIAL DIRECTOR: **RAY RASMUSSEN**
(916) 337-4402; ray@iconnect007.com

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

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

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **MIKE RADOGNA**
mike@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER ART: **RON MEOGROSSI**

THE pcb design MAGAZINE

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

June 2014, Volume 3, Number 6 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc

ADVERTISER INDEX

Coming Soon to *The PCB Design Magazine:*

July:
Concurrent Design

August:
Materials

September:
Data Transfer: The State of Standards