

THE ^{pcb} design MAGAZINE

January 2015

an IConnect007 publication

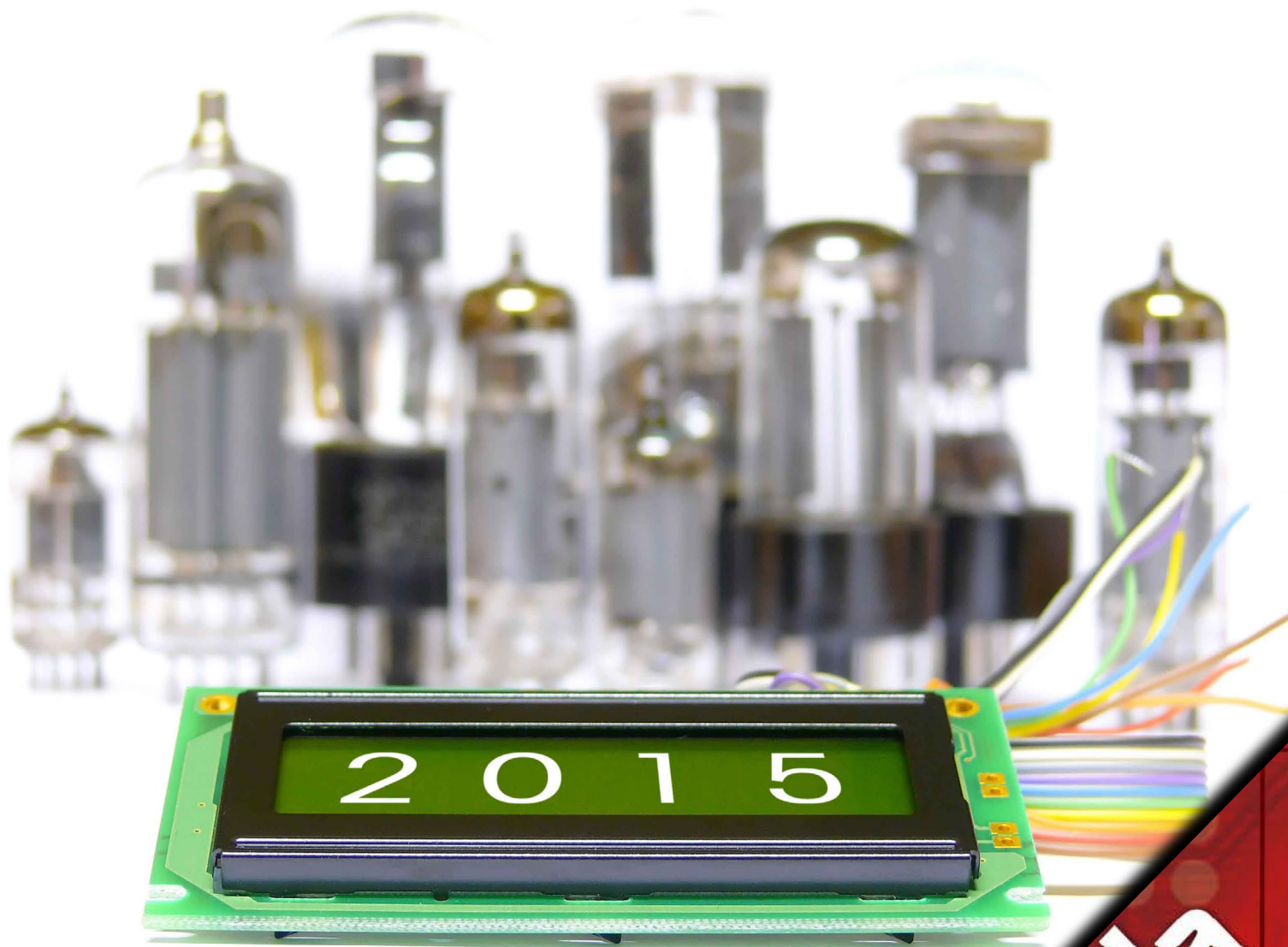
**N. American PCB Opportunities:
Investment is Crucial**
by Sharon Starr p.18

The Road Ahead: 2015 and Beyond
by Martyn Gaudion p.22

2015: A Look at Moving Trends
by Abby Monaco p.26

What Will 2015 Bring?
by Mark Thompson p.32

WHAT'S AHEAD FOR 2015?



Past and Future Trends in PCB Design

by David Wiens, page 10

isola
The base for innovation

Tips Every Designer Should Know

by **Amit Bahl**

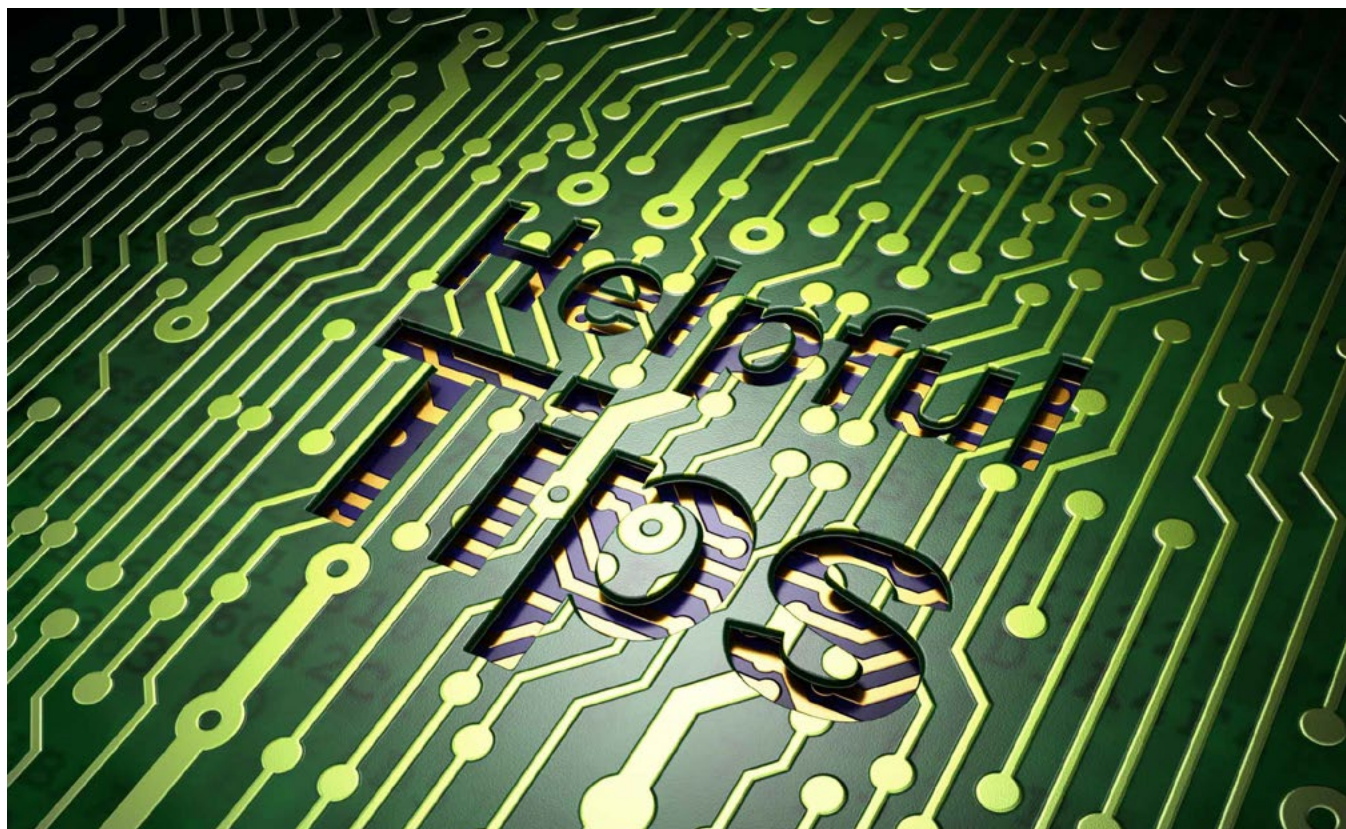
SIERRA CIRCUITS

As the chief PCB designer resident within a company that manufactures circuit boards, Greg Albers has unusual insight about how fabrication and assembly processes ought to influence design. In his position at Sierra Circuits, he's often asked by customers to recast designs that would be tricky to build as submitted, in addition to taking projects from schematics forward to achieve optimum yield.

Albers' career spans 30 years, including stints within several major OEMs—among them a network card supplier and a manufacturer of computers and smartphones—before joining the company. He's designed thousands of boards. I asked him to pass along some basic PCB design tips, simple reminders based on the path he treads to ensure the best outcome from a manufacturing perspective.

A Few Words to the Wise

First of all, Albers emphasizes, engineers and designers should understand the unique relationship between the fabrication process, design rules, and the costs of different design approaches. The easiest and definitely the cheapest way to accomplish that is through consultation with the intended manufacturer at the stackup stage. Although practically any design can be built, determining up-front the architecture that will result in the best yield should be the objective, certainly for volume production. Designers who take into account ahead of time how boards are built can save a good deal of money, and that economy extends beyond just fabrication charges, including fast manufacturing turnaround at the front-end and heightened product reliability in the field.



TIPS EVERY DESIGNER SHOULD KNOW *continues***Footprint Development:
The Foundation of PCB Design**

Needless to say, every project starts with a schematic, a mechanical description of the board, a stackup, and a complete bill of materials, or BOM. The bedrock of PCB design, the foundation, is footprint development, Albers advises. He refers to the data sheet for virtually every part on every BOM to leave no doubt about exact package descriptions and create proper footprints. In fact, footprint development is the most time-consuming aspect of nearly every project.

"As a rule, the spacing I use between parts is very large," Albers points out. "When I build a footprint, I make a reference line for the maximum size of the part, extend outside that by 15 mils for the silkscreen, then 5 mils more for an assembly line, and then a minimum of 15 mils beyond that for a package keep-out. That's 35 mils between the body of one part to the edge of the next part footprint, which has the same 35-mil spacing, so that's 70 mils in total from the body of one part to another."

"Discretes ordinarily are less than that," he continued. "For a connector that I know you have to get your fingers around, I may enlarge the keep-out boundary by 40 mils. If it's a right-angle connector, the keep-out accounts for clearance needed for the cable at the edge of the board, and so forth: Each part has its own keep-out value. Keeping the boundaries large certainly helps via placement, eases routing, and is better for the silkscreen, but it leaves enough latitude for me to violate the rule when necessary without jeopardizing assembly, if I need to place two parts 10 mils closer."

Via size, spacing, and location are the most challenging tasks in design, not routing or part placement. Through-hole vias impact where you can put parts on the opposite side of the

board; of course, that's a principal advantage of using blind and buried vias. There's a point at which an HDI approach becomes unavoidable (generally the result of tight BGA pitch) or more economical in terms of yielding more boards per panel to counteract the increased processing cost for multiple laminations. A board designer who understands fabrication can make a reasoned analysis.

“

If a board will be used merely on a benchtop as a test or development tool and only four or five are needed, then the size is irrelevant for manufacture, provided it fits on a panel, coupon width included.

A design that will be produced in any quantity, however, had better take into account how it can be replicated per panel with minimum waste.

”

Fab Complexity vs. Yield

The balance between fabrication complexity and yield per panel is not the only economic concern. Sometimes PCB designers need to push back, forcefully, against decisions by their engineering department or procurement. Albers recalls working at a computer company and encountering a BOM that called for a large through-hole capacitor for which there was a surface-mount substitute that would consume much less territory. The capacitor in the call-out cost \$0.17 and the substitute cost \$2.40 each. Procurement was adamant his group had to use the \$0.17 capacitor, not the substitute. So much room was required for the part that just one board would fit on a panel, instead of two if the substitute had been acceptable. It cost the company \$7.40 more per board to use the cheaper part instead of the substitute.

If a board will be used merely on a benchtop as a test or development tool and only four or five are needed, then the size is irrelevant for manufacture, provided it fits on a panel, coupon width included. A design that will be produced in any quantity, however, had better take into account how it can be replicated per panel with minimum waste. If you contemplate using large panels for economy, you need to consider minimum hole and pad sizes because the tolerance across the panel may not support using 6-mil drills and small annular rings, for exam-

ple. In any event, unless a board involves a BGA with a very tight pitch or one with a great many connections, there's seldom any need for a drill size smaller than 8 mils.

Many designers don't realize the soldermask is what determines design rules, not copper-to-copper or pad-to-pad minimums, Albers points out.

"The smallest spoke of a solder mask is 4 mils, period. You can't go below that whether or not you use standard LPI soldermask or laser-defined soldermask or there will be registration issues," says Albers. "You can decide to use a laser-defined solder mask to provide the better precision than LPI, which is unavoidable if there's a BGA with tight spacing, but that costs more. Getting the most for your money should be the objective when you need production quantities. However, unless you really know fabrication, collaborate with your manufacturer before you design."

Albers notes that engineers often pick BGAs with the finest pitch available, figuring the smallest packages will conserve board territory. However, the breakout for larger packages in many cases will actually result in less area con-

sumed. Package selection for complex devices ought to involve both the circuit designer and the PCB layout designer.

"As a rule of thumb, to estimate the number of layers a board will include, count the number of ball rows in the most complex BGA. You'll need a power plane per supply value for the part and associated ground planes. That package guides the project," Albers says.

Albers places and routes the power section of designs first, keeping most of the active devices on one side of the board and discretes on the other with assembly in mind. He can almost always accomplish a design without turning to fine trace and space widths.

Albers' advice in a nutshell? Think big.

PCBDESIGN



Amit Bahl directs sales and marketing at Sierra Circuits, a PCB manufacturer in Sunnyvale, California. He can be reached by [clicking here](#).

Sensors Enable Further Flexing of Flex Displays

Crocus Technology's new magnetic logic unit-based (MLU) solution can detect the position and shape of flexible two dimensional surfaces. Wearable devices, curved panel displays, flexible solar panels, and, in the future, mobile phones will integrate flexible shape sensor foils.

Crocus' magnetic sensors aim to provide an efficient solution for shape sensing in flexible surfaces and foils to overcome deficiencies occurring in other solutions, such as piezoelectric sensors. Unlike other solutions, Crocus' MLU sensors exhibit high sensitivity and directional capabilities. This means that only a minimal number of MLU sensors need to be embedded in flexible shape



sensor foils. In its prototype, Crocus only uses 0.25 sensors per square centimeter, making its solution extremely cost-effective.

In addition, Crocus' MLU sensors offer advantages in low power consumption and high-speed detection. They provide strong signals without active components. Crocus' 20cm x 20cm prototype consumes less than 10mA during the sensing cycle that lasts less than 1ms.

"Crocus has created a new IP-based on magnetic sensors for flexible surface position detection. This enables equipment makers to gain in the added performance of flexible shape devices, while reducing costs," said Bertrand Cambou, chairman and CEO of Crocus Technology.

As flexible displays are light, thin and unbreakable, they are expected to replace conventional displays. According to Crocus' press release the market for flexible displays is expected to reach USD \$3.89 billion by 2020.