Densification Techniques

Getting ten pounds of circuits in a five-pound bag.

Jerry Murray, West Coast Editor

One of the oldest ways to jam more circuitry into less space is still among the most effective. The discrete wiring process developed 25 years ago by Kollmorgen, now highly refined and still called Multiwire®, can convert a 22-layer MLB design with 12 signal layers to a six-layer board with just two signal (wiring) layers.

After prepreg is laminated on a standard voltage/ground two-sided board, a 3.5-mil thick B-stage epoxy wiring film is added to hold the very fine round wire precisely applied and ultrasonically bonded in place by a machine the size of a four-station driller. The wires, whose copper cores are as small as 2.5 mil in diameter, are coated with clear polyimide, which serves as insulation as the wires cross over each other in the X, Y, and diagonal planes. Additional layers are added as needed. Holes are drilled through the pads and the wires intersecting them. The layers are then electrically connected by etching back the insulation on the cut ends of the buried wires and copper plating the holes. Outerlayers and finishes for the Multiwire boards are manufactured using standard methods. The end result is a very MLB-looking board configured for anything from SMT to fine-pitch BGAs. But inside the board, each layer holds as much as 232 in/sq. in. of wiring.

According to Rob Kunkle, president of I-Con Industries Inc. (Euless, TX), "Multiwire can't economically compete with the standard MLB process for the average board, but the MLB can't compete in terms of density and high-speed signal transmission. And the process allows some boards to be built that simply can't be manufactured any other way."

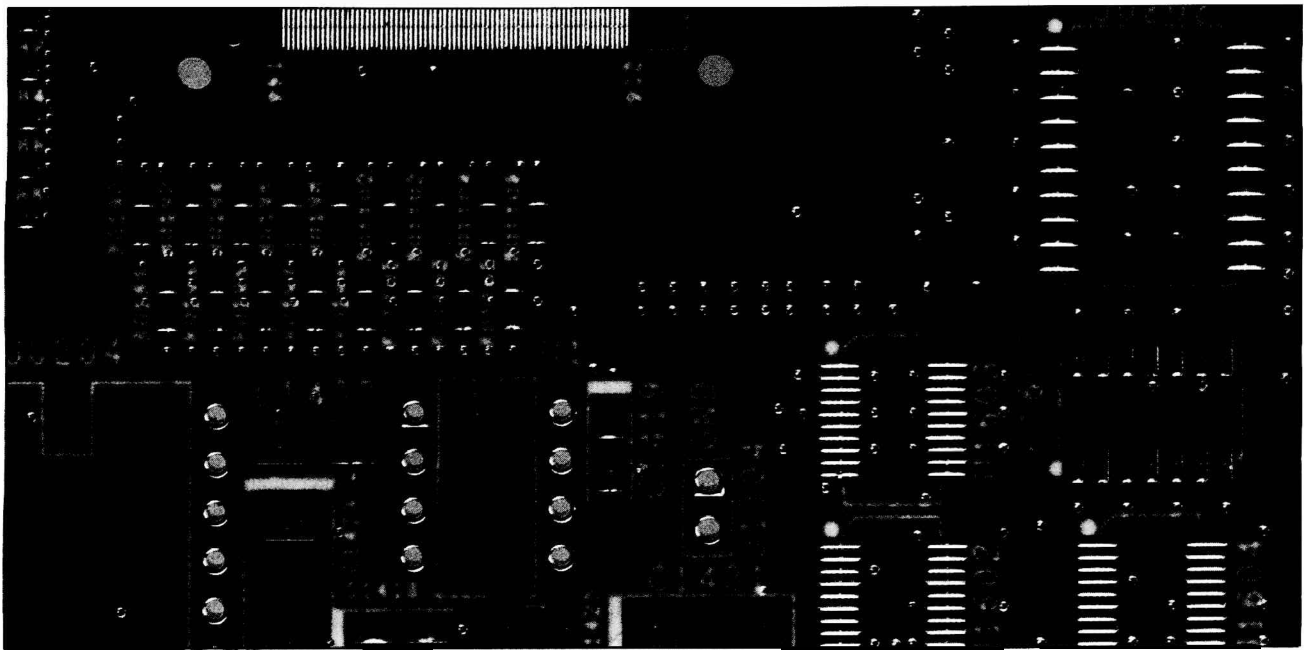
Hitachi bought the Multiwire process from Kollmorgen in 1990 and licenses the process through Advanced Interconnection Technology (Islip, NY), which also manufactures the wiring machines, the wire and some boards. AIT, Hitachi, and I-Con Industries are U.S Multiwire manufacturers, and additional licensees manufacture in the U. K., Europe, and Japan.

It can take up to six weeks to lay out a very difficult Multiwire board and another week to manufacture it, and for some boards the cost may exceed that of the standard MLB. Using conventional MLB technology, the same board would spend eight or more weeks in design and layout and another two weeks in manufacturing. Kunkle says that only about 10% of the $6 billion in boards made in the U.S. have applicability to I-Con's technology in terms of density, signal speed, thickness and electrical properties. And in order to more widely disseminate the use of the technology, Kunkle added, AIT is currently licensing the Multiwire design software at no charge.

SLC

A much newer and still evolving technology, developed by IBM Yasu, Japan in 1989, surface laminar circuit (SLC) technology densifies PCBs by emulating some of the steps used in manufacturing ICs.

The SLC board starts out as a standard two-sided



PCB, finely circuitized as a power or signal layer. As opposed to being constructed of prepreg, dielectric layers on each side consist of closely controlled thicknesses of a photoimageable epoxy, such as Ciba's Probimer 52. Circuitry and via holes are photoimaged on the epoxy, and full-panel plating is done, which provides connectivity to the layer below through the vias. The liquid prepreg process continues until the sufficiently layered board is drilled for through-holes, plated, and suitably finished.

IBM Yasu uses SLC boards in conjunction with the company's flip-chip attachment technology. They also license the technologies to other PCB manufacturers. Variations of the Yasu process are used in IBM's plants at Endicott and Austin, and are also licensable to anyone interested in making compact boards suitable for BGA and other very dense applications. The interest to date has come from automotive and mobile communications industries, both of which are in the SLC test-and-approval process. IBM Austin's Jack Fisher feels that the thinness of the SLC board as well as its density could qualify it for use as an advanced PCMCIA substrate, in spite of a cost of about 1.5X that of an FR-4 board.

The High-Density Image

The wiring density of the finished SLC board (more than 800 in./sq. in.) can be traced back to IBM's imaging process that enables its fabrication shop to produce lines and spaces down to 2 roils and vias as small as 4 roils. IBM uses collimated light in a Class 10,000 cleanroom. According to a survey reported in ***PC FAB*** just a year ago, cleanrooms were in use at 6696 of the fabrication shops surveyed, 63% of which said their rooms were Class 10,000. Bill Soules had this to say about the numbers: "Maybe 75% have what they call a cleanroom. But cleanrooms mean different things to different people."

After 30 years in contamination control with Kodak and several more as Soules Consulting (Rochester, NY), he sounds like he knows what he's talking about when he says, "People like to think they have a Class 10,000 cleanroom, but most of these are rated by a particle counter in the as-built, at-rest condition as opposed to the operational mode. This leads to a false sense of security. A company can pay up to $1,000 per square foot for a semiconductor-quality cleanroom and walk off expecting it to stay clean without properly training and auditing its personnel. PCB shops don't need a semiconductor cleanroom, but they can't survive without a sensible cleanroom approach."

When I asked him what he meant by sensible, he said, "First determine the need for a cleanroom by looking at the product and whet-e it is being exposed to a potentially damaging environment. Then define cleanroom practices and techniques, like laminar unidirectional airflow, conditioning cabinets, garmenting and housekeeping. The overall approach consists of three factors-the facility, the process, and the personnel. You can't ignore any of them. You can take care of these matters yourself or use a consultant to help."

Soules contends that 60'% of "lint-free" wipes aren't free of lint and that depending solely on tacky rollers is like depending on aspirin. But he quickly adds that every cleanroom tool is a step in the right direction. Soules scoffs at some of the soft-wall mini-enclosures he's seen, and says that others, mm-e thoughtfully designed, could provide a clean environment on a sandy beach.

Systems Division Inc. (Mission Viejo, CA) manufactures soft-wall mini-enclosures and related products. Ann Ashburn, sales manager, surveyed the following users of cleanrooms:

- Van Pham, photo supervisor at Polyclad (Santa Ana, CA), said, "A cleanroom is critical within the photo areas to reduce failures and also critical for resist. Higher yields are attributable to a cleanroom environment."
- Pamela Shultz, senior production engineer at Unisys (Roseville, MN), said, "A Class 10,000 environment is critical for the lamination area because of the prepreg and in the photo area because of the film. Humidity and temperature control, along with the HEPA filters, are individually installed for each area."
- Everett Simons, senior staff engineer at Motorola (Schaumberg, IL), feels that a Class 1000 area within a Class 10,000 environment is the most economical environmental control method.

Although their views on cleanrooms differ, all are in agreement with Soules' firm opinion that an educated cleanroom approach has become an indispensable tool in the manufacture of circuit boards.

High-Density Processing

From phototooling to finished product, there are a great many steps involved in manufacturing a high-density PCB. Consultant Jim Bryan looked at these steps collectively in his April 1994 PC ***FAB*** article titled "PCB Manufacturing Magic ?" "The processes we use for manufacturing PCBs are based on chemistry and physics; there is no magic involved . . . The keys to successful execution of the manufacturing process are:

1) an in-depth understanding of each process element and how it interacts with other elements, and

2) managing the variability within the boundaries necessary to produce the attribute, set that comprises the most complex circuit in the company's portfolio. "

The limiting Factor?

With cleanroom conditions in place and interacting processes in full control, the most complex circuit produced is essentially worthless until it is proven functional. In short, the limiting factors in high-density PCBs are inspection and testing.

AOI is a wonderful tool for high-volume inspections. But as line widths are reduced, pixel size and throughputs diminish accordingly, and AOI is never enough to prove functionality. Many products have already exceeded the capabilities of universal-grid testers. Higher-technology testing systems are available. Many of them exceed the PCB manufacturer's budget, but they are nevertheless needed by high-density producers.

Consultant Dave Shier of Shier Systems and Software (Westlake Village, CA) provided his views on ET systems that take up where the universal grid is left behind. "A universal-grid tester is the cheapest and fastest way, but it can't be relied on for anything smaller than 16-mil work. Buckling beam technology extends the use of the universal-grid tester by significantly increasing testing density through reduction of the diameter of the probes. The technology consists of a wire that buckles or bends to the side under pressure. These testers have typically been quite expensive, but an innovation now being patented by CK Technologies (Newbury Park, CA) reduces their cost by a factor of five and also extends the short life of the probes.

"The flying-probe tester is excellent for high-density prototypes and very short runs, but the equipment is very costly and it can take hours to test one board. Another alternative is some type of dedicated head. With its spring probes put directly in place on the board, its advantage is that the number of probes is reduced from tens of thousands to as few as one thousand. Again, the disadvantage is cost; just the opposite from the flying-probe tester, the dedicated-head ester is high in fixturing cost and low in equipment cost.

"No single tester suffices for every board. And as boards get denser and denser) the only valid testing answer I see is using a combination of testing technologies. Very few shops that can afford a million-dollar tester are those that use it a lot. So the viable, valid answer is outsourcing testing to a testing specialist that can keep all their machines at work amortizing themselves. With very necessary help from Merix (Forest Grove, OR), Zero Defects Testing (Forest Grove, OR) is providing multiple testing services suitable for just about any board made. Merix helps ZDT finance any tester needed for their most difficult boards. ZDT not only tests the boards in the Merix facility; it also tests boards from smaller PCB shops that could never hope to afford a line of multiple testers. I think this approach is the wave of the future."

Conclusion

To keep pace with growing high-density demands, keep it clean, keep it under control, and don't forget testing. FAB