The Inevitable Demise of SMT

The printed circuit board (PCB) process has been around since just the start of WWII and has only been improved upon since the conception of etching away copper traces on a single side. Significant improvements have been made which include:

- Etching of both sides and interconnect with eyelets realizing twice the interconnect capability

- Introduction of the plated through hole (PTH) eliminating the eyelets

- Multiple layer construction (multilayer) creating several etched inner cores, laminating with presses then forming PTH interconnection

- SMOBC: Solder Mask over bare copper, eliminating reflow of the tin/lead, allowing smaller traces

- SMT: Surface Mount Technology

- A multitude of packaging techniques: Buried vias, where the inner layers are interconnected with PTH, BGA's, Blind vias where small holes are laser formed for interconnection etc.

All these refinements are evolutionary with the basic principles of chemical etching and metal finishing still requiring to be performed. Some of the other "improvements" have simply complicated the manufacturing process to the point where yields are a big factor. In many of the high technology boards, it's really hit n' miss, sort out the good versus the bad.

With printed electronic circuits, the product is reduced to four simple repeatable steps. At every fourth step, the product can be tested and re-worked eliminating the huge scrap percentage of the old methods. Taking out the complexity reduces the capital investment required by 80 percent, yet, the process should be considered GREEN because of the elimination of the PTH, thus the end product should prove to be much more reliable as the interconnect is a solid metal plug compared to a thin layer of copper plating.

The Printed Electronic Circuit (PEC) can also be incorporated with a revolutionary assembly process called OCCAM, where components are pre-positioned and the circuit built atop. This combination would allow the making of electronic cards totally green, under one roof and potentially in-line.

Applications range from the most "impossible" designs to the lowest and simplest print only designs, starting from RFID to forming a circuit trace on a hypodermic needle

probe. Sub mil lines with equal spaces, buried and blind via formation with nothing more than a silk screen press.

As packaging density increases, the potential for more reliable end products increases with bonuses for economic and environmental benefits. Therefore, PEC may very well become the next generation interconnect packaging method of choice for a very large portion of electronics. Unfortunately, the existing PCB manufacturers either get on board or new start up could very well overtake the interconnect market.

The professional technology developing board manufacturers have diminished with the demise of the captive manufacturers. These were the board makers who at one time pushed the PCB making envelope by introducing new processes and increasing reliability. Not much remains of these today, so those who are left are converters. This is rather sad because the innovation isn't coming from the board shops but rather from sometimes outlandish customer requirements with board shops struggling to make the product. The latest innovations in printed electronics are now involving product designers, and not the layout guys, and the process is very simple.

Basic conductive inks are available now. This includes a very low resistance silver (4x resistance of copper) and an LPI (photo imageable dielectric) which we have to date done 1 mil with UV exposure and expect sub mil lines with LDI equipment.

Silver conductive for now, and we expect that with the R&D efforts going on worldwide at the university level, as well as commercial alliances formed in search of, a conductive nano particle copper ink will satisfy commercial needs. In many of the processes, many consumable products are required: for example, a drill bit is used to make 3,000 holes only so a common 25,000 hole PCB, 8 bits are used up at a cost of $16.00. A lot of chemistry is used and must be disposed of such as dry film, excess copper, excess tin, plastic sheeting, release films, press pads, router bits.

By comparison, whatever is used in the PEC process remains on the board. While the inks are more expensive than those used currently in the PCB process yet overall, the total costs are less because a multitude of skill level requirements are eliminated, including the required floor area and it's related costs (HVAC, lighting etc.), power requirements to heat and cool the multiple processes and on and on.

The PEC process is reworkable in every step and so there is no scrap. The old process banks on the hoped for integrity of the inner traces such as plated hole formation to be 100 percent perfect until final testing is performed. The shame is that a bad board from pressing onward is totally finished alongside a good one and that is a big waste of good money.

At the end user level, the reliability of the board could even be better because there is no weakness in solid metal via formation. The PTH on the other hand gets worse in reliability as the holes are smaller and metalizing them difficult. With the advent of cooler solders and conductive adhesives, the PEC stands to be a better product.

Is it 100% ready? Certain suppliers such as Caledon Controls provide the materials and process for anyone to make product for evaluation purposes for the more complex assemblies. Other less critical applications abound where the process can be applied.

Is this the beginning of the end for SMT? The answer is probably yes although the timeline is undetermined.

My thanks to Mike DuBois of Caledon Controls for much of the content of this article.