Phil Zarrow

SSD: The Other Surface Finish

With lead-free and via-in-pad designs, solid solder deposit may finally find its niche.

"There's a time for us, a special place for us…"
Steven Sondheim, *West Side Story*

Many assemblers are not familiar with a concept called solid solder deposit. Solid solder deposit (SSD) is a method whereby solder material is deposited on the PCB substrate by the fabricator. The assembler has no need to print solder paste; the solder deposit is already there. Typically flux is added and the PCB passed to component placement. The tackiness of the flux keeps the components affixed to the board. Following component placement, boards are reflowed in the usual manner.

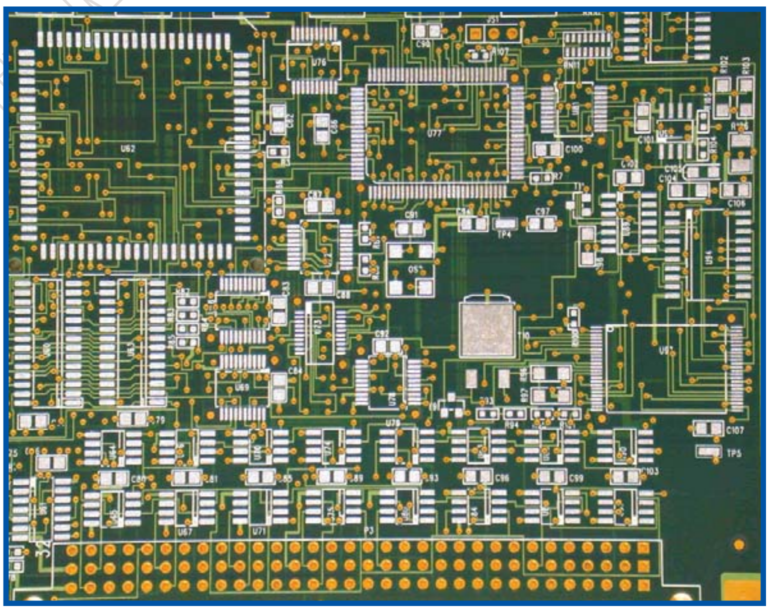
Solid solder deposit process technology has been around for some time, 18 years to be exact. A number of such processes exist, but one of particular interest is the original technique. Called Sipad, it was invented by Siemens in 1986 (not all material and component technologies have their roots in IBM and Bell Labs/AT&T in the mid 1960s). It remains one of the most economical and viable SSD techniques and is downright cool!

Sipad begins with the application of a special soldermask. Originally, the process used dry-film mask (remember, this was 18 years ago). Today's process uses LPI (though dry-film can be used; either way guidelines must be followed). The thickness is typically about 100 µm. The mask is applied to the PCB and exposed, developed and cured, similar to any soldermask material.

Now comes the fun part. Solder paste is stencil-printed onto the PCB – by the fabricator. The soldermask, in conjunction with the stencil thickness, defines the volume of solder applied to the land. The board is then passed through a reflow oven, where the solder is brought to reflow temperature and then cooled. Applied over bare copper, gold, silver, HASL or virtually any solderable surface finish, OA (water-soluble) fluxed solder paste is used. After reflow, the PCB passes through an aqueous cleaner. The next step is a flattening operation that uses two heat zones and a cold press plate to flatten the solder deposit flush with the top of the mask. For a double-sided assembly, the solder deposit and reflow steps are repeated for the second side prior to flattening. Finally, an adhesive no-clean flux is applied to the solid solder deposit, cured to a tacky finish, covered with protective paper and shipped to the assembler.

The first step in assembly is removing the protective paper and exposing the tacky flux. This replaces the printing operation since the solder and flux are already in place. (Note: peeling the protective paper requires somewhat less skill than printing paste.) The boards are then populated and reflowed in the usual manner. That's it.

This technology is worth examining for a number of reasons. First, solder paste deposition is eliminated from the assembly operation (recall that the combination of paste characteristics and printer parameters typically account for approximately 50% of assembly



SIPAD Systems

FIGURE 1: Solid solder deposit may get a boost as a lead-free surface finish. Here, the Sipad process of paste on pad is shown.

***Phil Zarrow** is president and SMT process consultant with ITM Consulting, (ITMConsulting.org); phil_zarrow@itmconsulting.org.*

defects). Paste alignment, paste work life, deposition precision, low humidity: all become non-issues. Also, post-print inspection is not necessary. Paste height is controlled and consistent, ideal for area arrays like BGAs, CSPs and flip chips. And yes, the precision of the photomask film lends itself nicely to fine pitches. It works well on QFPs with 0.4 mm (0.16") lead pitches and with 0402s.

If the concept is so good, why don't we see more SSD? Many of the other SSD techniques I've seen require specialized equipment and processes. A board fabricator has to see the demand for the technology before investing in the capability; most chose not to. Sipad is not immune; it requires a special flattener and it is a proprietary process that must be licensed. Beyond that, it is straightforward: to a PCB fabricator soldermask technology is not something new and scary.

SSD is a good process to consider for two other key reasons. First is that old nemesis to assemblers: via-in-pad syndrome. Perpetrated upon us by the design folks, the situation gets worse as component density and pin-counts increase. Via-in-pad accounts for insufficients and opens in many joints, and is extremely deadly (and prevalent) in area arrays. This design attribute contributes a great deal to the presence of voids in solder joints as well. Tenting is problematic and not always possible. SSD eliminates the problem because the solder deposit has already dealt with that pesky via.

Then there's another nemesis, lead-free, which is coming at us like a train (T-19 months and counting!). With the debate and concern regarding the cost of silver, embrittlement, tin whiskers and the like, SSD of a lead-free alloy (e.g., SAC 305) is a viable lead-free surface finish. Depending upon the application, virtually any alloy could be used, including higher temperature blends. It even appears to enjoy a longer shelf-life than OSPs (and without the exposed copper in the corner of the pad).

But don't consign the printer to storage yet, since SSD is not for everyone. Sipad lends itself nicely to single-sided assemblies. For double-sided SMT boards, the second side will require a flux application. This can be printed or even sprayed on. The technology is application driven. In this case, though, it appears to be a widening niche. This technology will continue to evolve as it has a lot going for it. ■

Special thanks to Matt Kehoe of SIPAD Systems Inc. for the history and background of SSD, and Don Johnson of DuPont, who taught me more than I ever wanted to know about dry-film soldermask.