Hand Placing Leadless and Fine Pitch Leaded Components with SIPAD Solid Solder Deposition

Matthew J. Kehoe
SIPAD Systems Inc.
Alpharetta, GA.
mkehoe@sipad.net

"There is nothing more difficult to take in hand, more perilous to conduct, or more uncertain in its success, than to take the lead in the introduction of a new order of things. Because the innovator has for enemies all those who have done well under the old conditions and lukewarm defenders in those who may do well under the new".
Niccolo Machiavelli

Abstract
As the electronics industry continues to evolve and change, companies are being forced to adapt to maintain their current business, while attempting to develop new customers. Adaptation comes in different forms. It could be a larger building, a new piece of equipment, or a shift in philosophy, but the bottom line is, sometimes success depends on "a new order of things".

In the manufacturing of surface mount electronics, much has changed over the last 10 years. Components that were considered leading edge in 1993 don't even exist any more. Devices 1/10 the size with 10 x the horsepower have become common place. Ultimately the new devices may meet the same fate as "the new order" evolves.

How can CEM's and OEM's keep up with the ever changing requirements demanded by their companies or customers? It is easy to throw money at a problem, buying better, more capable equipment to manufacture and verify your assemblies but in a time when money is tight and business slow, is that an option? Does your budget for 2003 include an x-ray machine, new stencil printer or more sophisticated chip shooter?

Even if it does, will you be able to purchase this new equipment, get people trained on it, and pay for it while the industry is in a slump? Maybe? Maybe not.

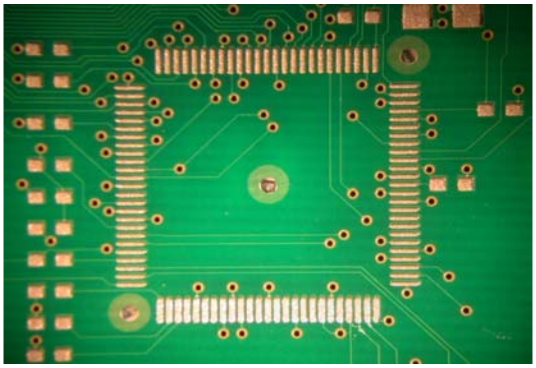
A shrewd planner must be an innovator. Some call them a MacGyver. A company has to have a person that sorts through the sales pitches to find true innovation. Things that work better than the *"old conditions"* they are used to. Even if a company has an innovator, he or she may have to battle "*enemies, all those who have done well under the old conditions"*. They may be co workers, bosses, or even sales people offering advice based on selling their product versus solving your challenges.

Solid solder deposit has been around for 16 years, in the United States for 5 years. Some in our industry have even called it the "greatest idea that no body has ever heard of". SSD has proven successful as a "new order of things" for more than 50 companies that know about it and have sent their own *innovators* to the shop floor to see if it works. Yes you can place BGA and flip chip components. Yes you can hand place .4mm QFP, 0402, 0201 components. Rigid boards, flex cables, thin boards, thick boards, lead free, high temp, low temp. Assemblies that are cleaner, higher quality, and have better throughput than those produced by traditional hand paste and place methods.

This paper will explore solid solder deposition for BGA, LLP, and fine pitch leaded parts. Who is using ssd, how they are using it, and how it can be used by companies that are willing to consider the "new order of things" as an immediate, affordable, and easy to implement pathway to success.

Old Conditions
Traditional methods for hand assembly vary however, one fact remains constant, the smaller the pitch size the harder it is to hand place and solder the components. There are numerous challenges including solder volume, placement accuracy, cleaning, and solder joint integrity. In addition, the throughput experience with hand assembly is normally very low.

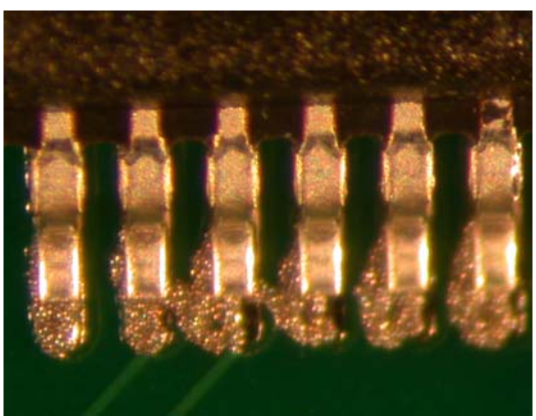
No matter what size the component lead pitch is, it is necessary to add solder to the surface mount pads in order to attach the components. No surface finish provides enough solder to attach component leads straight from the box.



(Paste printed on .4mm pitch footprint)

Adding solder can be done in different ways. A solder paste stencil can be used to print paste or solder paste is available in a syringe, or solid core solder wicks melted over the component leads is also used

Printed paste, while the most accurate application method, is extremely difficult to use in a hand placing operation. Even though the solder is placed accurately onto the pads, the possibility of



(Z axis pressure causes squeeze out)

introducing defects in the placements of the components negates any accuracy gained from printing the paste. Many operators are very skilled at placing component leads into wet solder paste but the introduction of z axis pressure into a liquid solder paste deposit can ultimately produce squeeze out and shorting which creates re work.

Applying the solder with a syringe is another option but the Z axis issues still loom large as operators try and seat components without creating shorts.

Placing component leads onto traditional surface finishes like HASL or gold and flowing solder over them using solid core solder wick has proved successful in the past but this method is slow and impossible for leadless components like BGA and LLP.

A new order of things

Solid solder deposit has been in existence for many years. SIPAD, invented by a research scientist working for Siemens in Munich Germany, is the original ssd and has been around for 17 years although only available in the United States for 6 years. Numerous variations were attempted over the years, Optipad, V-Pad, and Precision Pad Technology (PPT) however only SIPAD and PPT actually made it past the development stage and into the industry. Solid solder works because it eliminates the basic challenges of liquid solder. Squeeze out, smearing, and shorting introduced by Z axis pressure are all taken out of the picture by solidifying the solder before the placement of the components. If the squeeze out is eliminated and the pads retain their shapes during placement then the reflow and subsequent attachment becomes predictable and repeatable.

Placing components by hand is something that has been done for at least the 20+ years that I have been in printed circuits. It used to be all through hole which was much more forgiving than the tiny surface mount pads people try to assembly today. Some companies end up sending their surface mount boards to a CEM versus attempting it in house even though they may have all of the equipment they need to assemble these boards sitting on the shop floor. This keeps their people working and usually proves to be the most cost effective.

There are other companies that hand place and solder their own smd part except for BGA's and LLP's at a cost of $50.00 each. Cost effective? Maybe however ssd gives the an alternative

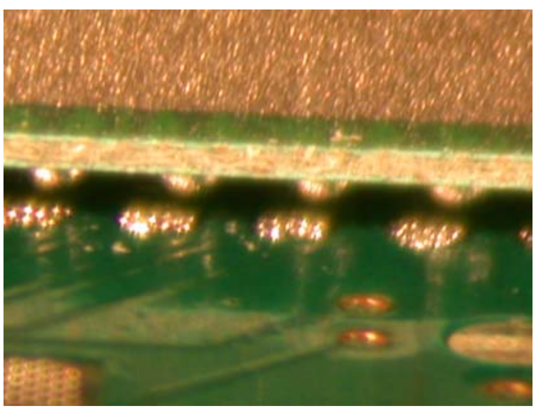
BGA's & LLP's

Placing leadless components such as ball grid array (BGA) or leadless lead frame packages (LLP) is considered by most to be impossible. Even if the component is placed perfectly it is still likely that a defect will occur without the consistent accuracy of a pick and place system. The human hand is just not repeatable enough to accomplish this especially as pitch sizes shrink.

The BGA and LLP component package share a common thread. The leads are contained under the body of the component so even if a hand operator can place the components with exacting Z axis pressure and alignment, into perfectly printed wet solder paste, there is no way to tell if this was successful until the component is soldered on to the board, usually with a lot of other components as well. At this point it is too late to correct it without expensive and elaborate re-work tools and procedures. EXPENSIVE being the key word as any activity associated with re-work is expensive.

Typical solid solder deposit BGA and LLP pads are slightly higher than the surface of the solder mask especially when liquid photo image-able (LPI) mask is used. The smaller pitch sizes can be flush but most of what is being seen is 1mm pitch and these pads are finished with a lightly higher surface than the solder mask.

This slightly raised surface creates a channel or trough between rows of pads.



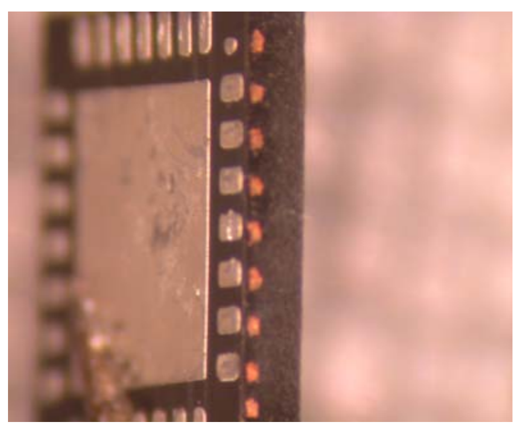
(BGA's slide ont SSD's without smearing)

This trough can be used as an alignment tool. Because the pads are solid solder, the balls of the BGA can be placed in between the rows between ssd's and slid diagonally up and on to the surface of the pad. This motion is very easy to repeat and operators' report that it is very easily learned with only a few tries. Additionally, with a SSD pad, the balls need only be touching the pads to be pulled into place by surface tension during the soldering process. Even miss-registered BGA's solder without shorts because there is no smear or squeeze out from the miss-alignment.

(Component placed close to final position)

(BGA component slid into position on SSD's)

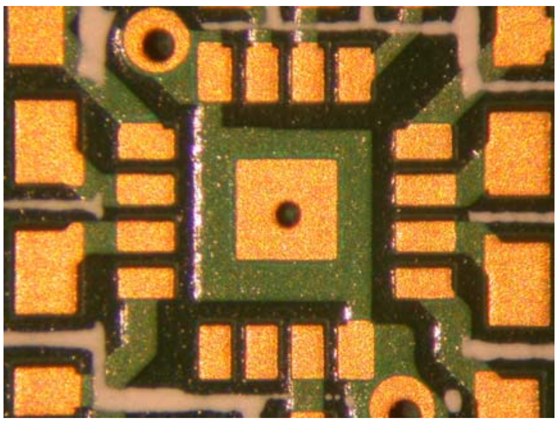
LLP components do not have balls under them however; the leads are still under the body of the component so visual alignment is impossible with wet solder paste.



(Leadless Leadframe package)

Again, the raised pads of the SSD/LLP footprint come in handy. The components can be placed onto the ssd's and slid into place. Once close, the entire assembly can be tilted and viewed from the side to make sure leads and ssd pads line up. If not, the LLP can be nudged into position for soldering. Like BGA's, the LLP pads do not have to be perfectly aligned to solder successfully. While specific testing for the minimum percentage of alignment has not been conducted the estimates from users are 50%-60% of the leads lined up are enough. Surface tension will take care of the rest, pulling the pad into place during reflow.

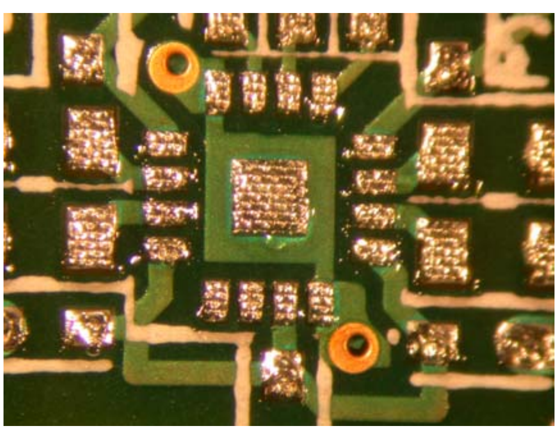
Many LLP components utilize via in pad technology.



(SSD/VIP, Via in pad, LLP technology)

Via in pad (VIP) allows power to be transferred to inner layers of the board and distributed to other area without fanning out the vias wasting valuable real estate. VIP technology presents numerous challenges for solder paste printing including

- Paste volume
- Printing definition
- Cleaning
- Voids



(Finished LLP/VIP with flux applied)

Paste Volume
Paste volume is something that is addressed every day when dealing with printed circuit assembly. The volume of amount of paste to achieve predictable, repeatable and economical throughput is critical to making (CEM's) or Saving (OEM's) money.

Too much paste and you get shorts or bridges that need to be removed to pass test. Shorts = rework=cost.

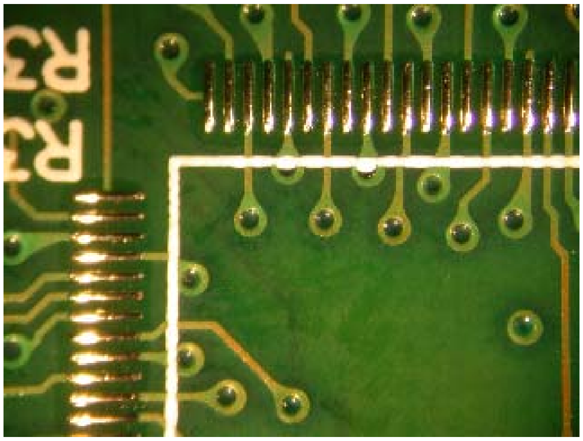
Too little paste and you get starved pads or lean solder joints that require rework to add paste. Again, defects = rework = cost.

Many of today's surface mount designs contain very fine pitch areas as well as large high power areas that are

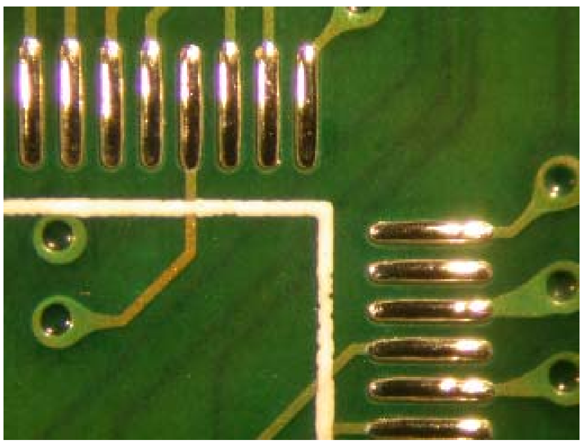
With SSD the amount of solder paste is easier to control because of the forgiving nature of the application. Because the paste is reflowed without applying any Z axis pressure, the results of the reflow are much more predictable. Natural flowing solder will pull to the center of the feature it is being applied to and conform itself to the shape of that feature every time.

Print Definition
This makes overprinting of features, even very fine pitch features, possible because the solder will pull onto what ever pads it is touching, even when printed on solder mask. That means that openings in the stencil can be cut longer than the openings in the mask to add paste without causing shorts. Additionally, BGA and LLP pads can be over printed in one or both directions to add paste to accommodate a via hole. The paste will flow freely into the holes creating a plug.

Cleaning
SSD boards require less cleaner because of the the contamination from the solder paste, usually trapped under the components, is removed during the application of the SSD. Because the boards are initially reflowed without components the contamination is released on to the bare board surface. This means that during the wash cycle, all the contamination is removed. It is always difficult to remove contamination from printed circuit assemblies that are initially reflowed with the components in place as contamination is trapped under the components, especially in the case of BGA and LLP packages.



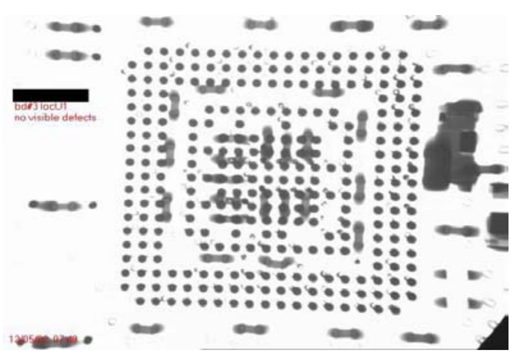
(Contamination released during initial reflow)



(Contamination completely removed)

Voids

The VIP-SSD pad is reheated during the flattening stage the pressed gently with a cold press to create that flat surface. Any potential voids are being forced out during this step. It is believed that there are no voids to be forced out. Voids are caused by many things but the failure to properly release contamination is a major contributor.



(BGA X-Ray. No voids)

History shows that SSD BGA exhibits little to no voids. Although documented testing still needs to be done, the fact remains that the experiences of those companies using SSD for BGA attachment have shown that voids do not occur with SSD technology.

It is believed that because the BGA pads are initially formed without interference from component placement, the resulting solid solder deposit is less likely to form a void. Additionally, if a void is formed during the initial reflow the subsequent flattening process would actually squeeze the void out.

The flux

After the SSD pads are flattened they are coated using the stencil, with adhesive no clean flux. This flux is specially formulated to be dried to a super tacky finish with a shelf life of over 6 months as well as a tackiness un matched by any soder paste. Most components, when placed onto the flux finish, can be viewed from the side as the board can be tilted without disturbing the placement.

The flux is only applied to the surface mount pads and burns off during the final reflow process. The tacky flux is protected by a cut to size release paper until ready to assembly.

Summary

Hand assembly is here to stay. There will always be a need to be able to put components onto a circuit board without setting up an entire line to do so. Hand assembly requires a unique individual. This person will have one hand for the component, one hand for the soldering iron, and a third hand for the solder. Employees with 3 hands are very hard to find. SSD becomes that third hand providing the right amount of paste, in the right spot, and holding power that is un-matched by any solder paste.