

HASL

WHAT A HASSLE, or HASL'd AGAIN

And

The Best Alternative Solder Termination Finishes

Earl Moon (POD)

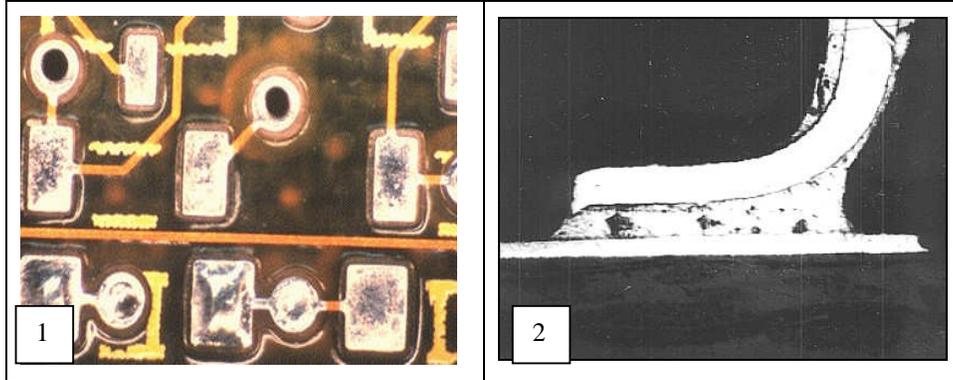
INTRODUCTION

This article updates one I wrote for Printed Circuit Fabrication Magazine in December 1991. Actually, this article trashes that - and about time. I was much too kind then talking about how to control the HASL process. I really couldn't say it was uncontrollable, as no one, at that time, would have believed such an "advanced", save us all technology could be flawed. Well, it's time to dispel that bunk though some still love the stuff, and there are many staunch defenders selling equipment and process elements as well as those using its effects.

Again, I say now as then, the least controlled/controllable process in PCB manufacturing is the solder coating/hot-air-solder-leveling (HASL) process. This lack of control definitely is as due to the process itself as the boards entering it. First, solder must be "wetted" to the PCB surface and this is not a process normal to board fabrication operations as, traditionally, solder was always plated and often "fused" thereafter rendering a highly solderable surface. Second, boards received into the process often are oxidized so as not to allow acceptable solder wetting/coating.

Controllability (I now call it manageability) is important because, during the HASL process, solder must wet to all exposed metal surfaces - especially solder termination areas as SMT pads. The thing is, it doesn't well - or without quite a struggle resembling nothing manageable. Because examples of non-wetted conductors are prolific, we all know something is not right with the HASL process. That something is discussed in the following:

Figure 1 is a common example of non-wetted SMT pads. We've all seen such examples. Notwithstanding surface topography, this condition obviously is unacceptable. This is because no amount of effort or futile attempts at process management, during assembly soldering operations, allows initially acceptable solder joint quality or, of course, long-term reliability. Simply, if it doesn't wet during HASL, it won't wet during assembly. Figure 2 is an example of problems associated with this condition as a solder joint cross sectioned to show just a part of the excessive voiding effected by flux remaining in an attempt to continuing fighting excessive pad surface oxidation.



Another factor deleterious to solder joint formation and quality is excessive intermetallic compound (IMC) formation. All proponents of the process understand and admit an IMC layer is formed involving the copper surface and the eutectic solder coated onto it. Though part of any "normal" soldering process, it is formed during solder coating operations and can become too thick for subsequent assembly soldering operations to be effective. It is known that high temperature and liquid solder's intimate contact with the basis metal enhances IMC formation rate though enhance might mean too much, too quickly - as well as growth over time. Plus, after solder coating, another thermal excursion is experienced as the board travels through the hot air knives. All this adds up to a recipe for failure (created in an unmanageable process) during and after assembly operations.

It is well known that IMC is required for solder joint formation though the IMC layer itself is not a solderable surface. Therefore, say experts, there must be sufficient additional eutectic solder thickness to assure acceptable solder joint formation during assembly. Again, the conflict comes as how thick both the IMC and solder? If the IMC is too thick, and the solder is too thin (as is often the case with HASL coating), acceptable solder joints are not possible. If the IMC continues growth, becoming "extremely" thick - well, you know the outcome no matter the solder thickness. So, how much is too much and how does one control it? One doesn't.

Intermetallic compound thickness can be minimized depending on process parameters used. It is said by some that typical tin/copper IMC thickness is 6 to 12 micro-inches (.15 to .3 microns). In other hot air leveling processes, the IMC layer can exceed 100 micro-inches (2.5 microns). This is pretty contradictory and confusing stuff as "other" may mean those processes not so well managed as in different machine types and processes. That's a whole other concern because so many choices are available as vertical, horizontal, features, capabilities, cost, etc - making it hard for board shops and their customers to make "safe" decisions. How would the average board buyer begin to evaluate a board shop based on all the above?

Though difficult to measure HASL finish thickness, some findings show the range to be 80 to 1000 micro-inches (2 to 25 microns) on the same board. That's a wide range indeed. Again, this points to compromises, variables, and concerns facing all involved and affected. It comes down to how many compromises are acceptable and how each affects solder joint formation, quality, and reliability over time.

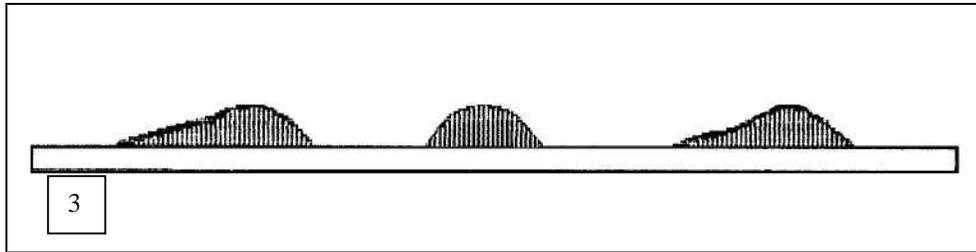
I've consulted with many operations wherein are found varying degrees of ineffective process management concerning HASL (there are none truly effective, nor can there be), and all going

into it before attempting operations. Often, deleterious causes consist of poor flux management, improper or missing tools and procedures, air knives not adjusted or operating properly, solder pot dwell times and temperatures improperly set or operating, and poor control of boards introduced to the process. The latter often is the main problem as it is impossible to keep and assure copper board surfaces oxidation free. Therefore, the process often is "tweaked" mercilessly in futile attempts to overcome the main problem. This article now focuses on the process itself.

HOT AIR SOLDER LEVELING - THE PROCESS

The PCB fabrication HASL process is analog to the PCB assembly wave solder process. It requires all the same elements and sub-processes - once the process is started. I emphasize once the process is started because only a few things are fundamentally wrong with it - by itself (to be discussed). It just can't do its job mostly because of what needs processing in it.

If it weren't for often found non-wetting and excessive intermetallic compound (IMC) formations, the process, and the product effected in it, would almost be acceptable. The only other problem would be solder thickness and topography inconsistencies, or SMT pad (solder termination area) flatness and uniformity. We've all seen and complained about the "puddling" effects rendered, as in figure 3 below and figure 1 previous, and how they wreak havoc on fine pitch SMT stencil printing and placement operations.



Again, the HASL process's biggest problem is what happens to boards before submission to solder leveling. This requires understanding a bit more (how much more can assemblers take) about board fabrication concerning printing, etching, and plating.

After a PCB is printed and etched (talking outers now), its surface must be coated with something to keep it from oxidizing. We all know the major reason for this is to prevent copper oxide formation. Oxides, and other contaminates, prevent solder wetting. Whatever is coated, plated, or otherwise applied to a board's conductive surfaces must keep oxidation from occurring. That's it. Otherwise, no coating is required.

Most of us know many coatings, or plating types, are available. Plating (as in electroplating), however, is the only truly effective process available to ensure contamination free copper surfaces and those effected after the process is finished. Tin/lead plating, that has been "fused" (using hot oil or convection air - though never focused IR), provides the most solderable surface yet devised. Its drawback, mimicking HASL'd surfaces, is topography. After fusing, there is a very homogeneous (very tight shiny grain structure that is very solderable) surface. However, it is one with a "dome" that can be .003" thick at its center line and no less than .001" around its edges. This, as HASL, doesn't readily invite SMT printing and placement.

Note: Some die hard advocates remind that HASL, after all, is pure eutectic solder having that advantage over tin/lead plated surfaces. To that I say, so what in light of all else negative?

I will say, that if tin/lead plated boards were HASL'd, there would be fewer solderability problems though the irregular surface condition would remain as an irritant to many, or serious problem to some. This will never happen because of associated costs and environmental issues.

Getting down to it: The pre-HASL board surface is bare copper. What's wrong with that you ask? Everything, I answer - because that surface cannot avoid becoming oxide contaminated before solder coating.

In the print and etch process, the board moves through a series of chemical solutions to final plate. This is when and where problems arise. With the specified plating thickness ensured (.001" - .0012" minimum in hole walls), the board is imaged and etched (panel plating) or the specified plating thickness is plated up along the resist walls. Walls resemble tiny canyons about .0015" high after imaging, resist polymerizing, and removal (where the conductor pattern is to be). After the imaged resist is stripped and the etching process is effected (while using tin/lead as the final etch resist), a conductor pattern is left as bare copper (after the final etch resist is stripped).

After the final tin/lead etch resist stripping operation, a final rinse is applied to the bare copper surfaces. The board can go two ways from here. In the first, it is dried, racked, and sent to HASL. The second introduces the board to a micro-etch process (usually a relatively mild acid solution to remove oxidation) followed by rinse, drying, and racking processes. In either case, drying and racking defines the problem. Boards are dried, because they cannot be introduced to HASL with wet surfaces or holes. Drying is burning, and burning is oxidation.

Note: The copper surface cleaning and micro-etch process may be carried out during the HASL process, in some machines, therefore reducing time between processes. Though more effective, a drying process still is required. This is the heart of the problem.

Depending on how long each process step takes, and how long it is before the board is introduced to solder coating, the amount of oxidation correlates directly. It matters little, however, as copper oxidizes rapidly and any amount can negatively affect the outcome.

Ok! That problem has been addressed. Now, what to do about it? There really isn't much that can be done, but an effort must be made in what remains a process very dependent on what enters it. That effort consists of what we all experience as qualified wave solder process, manufacturing support, and quality engineers - plus operators. For this, I have broken down each HASL sub-process as follows (after an overview) - keeping in mind most board shops are not soldering or solderability experts:

HASL PROCESS OVERVIEW

To effect solder coating, a board is loaded and cycled into the machine (figure 4) and process. After flux application, board preheating, and flux activation, the board is conveyed becoming totally immersed in molten solder for a specified time. To effect solder leveling, the board is

conveyed out of the molten solder, through a set of air knives where hot, pressure/volume controlled air is directed at both board surfaces removing molten solder as the board passes through them. Solder is leveled, or blown off board surfaces as is hole solder.

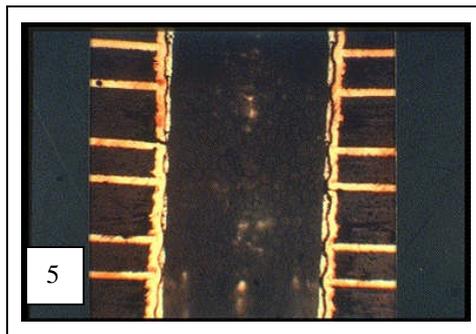


Molten solder is kept in motion as it is continuously pumped through the pot and over the weir at its surface. A high solder movement rate provides a scrubbing action over board surfaces while preventing any solder pot problems.

Critical process parameters are solder pot temperature and board dwell time during immersion. Other control factors are just as those in wave soldering.

Eutectic solder, as 63/37% tin/lead, typically is used. Its melting temperature is about 187 degrees C. However, the minimum required wetting temperature is near 220 C. with 235 C. + providing better assurance for complete wetting - depending on thermal mass and processing continuance. No matter, solder termination areas must be contamination and oxide free for any wetting to occur.

For effective process management solder temperature is maintained between 235 C. and 240 Centigrade. Within this temperature range, boards must only be subjected to solder immersion times less than 5 seconds as epoxy/glass resin system formulation Tg's typically are in the 120 C. to 180 C range. If the specified time is exceeded, thermal stress and/or shock is imparted to the board with effects as delamination, hole wall failure due to excessive Z axis expansion over extended periods, pad lifting, and associated problems. Figure 5 shows a cracked hole wall in X-Section - not a pretty picture.



When boards are transported from the solder pot, excess solder is removed from surfaces and holes. Air knives, set to ensure correct positioning, angles, air temperatures, and air pressures and volumes, are used to "blow" away the excess leaving "acceptably leveled" surfaces and hole wall conditions. One important tradeoff may be encountered however, as it is not possible to produce a maximum surface coating thickness while maintaining specified hole diameters. Therefore, only one condition may take priority.

FLUX MANAGEMENT

Flux management, as all HASL sub-processes, must be effected just as in wave soldering. Flux must be selected for the job to be done. Usually, a mild organic type is used that may be cleaned easily in a water wash process. In some cases, in shops really out of control, new flux types are tried everyday doing to a board what shouldn't be done to anything.

All rules apply concerning flux composition and maintenance. What is specified, must be maintained. Using a standard hygrometer works to determine specific gravity, or flux density. Using titration provides more insight to pH, as an example of a more thorough composition analysis. At a minimum, these requirements must be regularly determined and assured as meeting specified requirements. It should be noted some high-end machines have automated flux management processes.

As some board shops still know little about the soldering process, flux composition and maintenance gets them in lots of trouble - especially with customers. I still see operations in which little attention is paid to this critical process element. No hygrometers, certainly no titration kits present, and often flux resembling very thick honey capable only of never being removed from any surface. The stuff has no affect on solder wetting or quality except to absolutely prevent it.

So, we're mismanaging flux and its application. Why continue? Let's pretend this sub-process is being managed, as it should, so we can move on.

PREHEATING AND FLUX ACTIVATION

All fluxes have to be activated to work. Some HASL equipment types have preheat capabilities. Some do not. We all know what this is supposed to do. However, it is very difficult to run a thermocouple through most HASL equipment to verify "top side" board temperature and correlate it with flux activation temperature requirements. Therefore, this vital sub-process often is left to more chance than skill. Enough said here.

CONVEYOR SPEED

Whether horizontal or vertical, boards subjected to this process must be conveyed through each sub-process to completion. Again, this is no different than wave soldering but if either of the former sub-processes is not managed well, you know the outcome.

It's really a shock to see boards running slower than the flux applied to them. I've seen examples running so slow, movement barely was perceptible. You all know why. Some board shops do not. The outcome again is not good as boards emerging on the very well done side.

SOLDER COMPOSITION, TEMPERATURE, AND DWELL TIME

Solder composition can suffer even more in some board shops than in most assembly houses. Very large quantities may be run through the coating process in short periods. The problem

usually is smaller solder pots, often containing less solder than many modern wave machines, providing a vehicle for faster solder contamination.

Some board shops are very good at performing solder analysis. Others don't do such good jobs often relying on tools as "solderability testing" on bare boards for an indicator. This is foolish as well.

HASL solder temperatures are specified exactly as those for wave soldering. As previously discussed, 500 degrees F. falls in the normal temperature range while considering the entire board is immersed to effect solder coating on both sides. If adequate preheating is not effected, thermal shock occurs and that can be disastrous.

Solder immersion dwell times are critical as well. It is important for boards to get through the solder coating process as quickly as possible, while effecting acceptable coating, or thermal stress becomes excessive with effects similar to thermal shock. Conveyor speeds have to be properly set and maintained. This gets back to all other sub-process management requirements.

On the wave, a board has a fighting chance for survival. Its bottom side surface makes minimal contact (about one or two inches at a time) only, and only for about three seconds - maximum. In the wave, HASL sees most all the board acting as a submersible, at a given time, under the solder's surface for five or more seconds.

In the HASL process, it is critical flux application, conveyor speed, preheat condition, flux activation, and solder immersion factors all are precisely managed. If not, no wetting occurs, no board survives the ordeal, intermetallics form excessively, and unacceptable solder conditions are realized.

SOLDER COATING

With all else considered and properly managed, this is where the board meets the solder. It's amazing how hot or cool a pot can be run and see not such amazing effects. If all else is as required for process management, the solder coating process renders an "acceptable" board, ready for hot air solder leveling.

SOLDER LEVELING

If everything else could be controlled, and it could if not for board solder termination area conditions, this sub-process still renders everything null and void. If a board could be effectively coated and made solderable, without copper oxidation, this sub-process makes pad surface conditions unacceptable to many.

What other way could there be to level solder to a state smooth and flat, with a solderable surface - with a consistent, measurable thickness capable of being specified on the master drawing? Asking a lot eh? "Puddles" form, as in figure 3, because air makes waves on the molten solder's surface. When the solder solidifies, waves remain. Still, there must be a way to prevent copper oxidation (don't believe that for a minute), and make smaller waves (equally doubtful). Much detail was covered in this article's introduction.

ALTERNATIVE COATINGS (CAN YOU NAME A BETTER ONE?)

Is it just me or is everyone in this business searching for alternatives guaranteed to make life good/perfect? Dumb question, what? Of course everyone is looking for that elusive panacea type solder termination area coating. How's it go again?

- 1) It has to be free of oxidation - forever.
- 2) It has to solder wet, no matter the shelf life.
- 3) It has to be incredibly flat.
- 4) It has to smell good?
- 5) It has to have a long shelf life - how long for everyone to be satisfied?
- 6) It has to have a long shelf life under a variety of storage and use conditions.
- 7) It has to be washable after many misprints - anyone considering this but me?
- 8) It has to be compatible with over 1 million flux and solder paste types.
- 9) It has to support initial solder joint quality as well as long term reliability.
- 10) It has to be cheap, yes cheap - not just inexpensive.
- 11) It has to be available from a wide supplier base.
- 12) It has to be lead free (?) - something else to ponder but what about other environmental considerations as what happens in board shops processing nickel, gold, palladium, etc. - how about that for equally nasty stuff?
- 13) It has to be all things to everyone.
- 14) I'm going back to tin/lead plating that I can HASL and still have flat surfaces.

SUMMARY AND REVIEW

The following requirements, as absolute minimum pre-process requirements, must be met to ensure effective HASL process management:

- 1) PCB solder termination areas, as bare copper, must be oxide free.
- 2) PCB solder termination areas must be free of all other contamination.
- 3) PCB surfaces must be dry, as must their holes, before introduction to the HASL process.
- 4) Staging time, after PCB drying and before HASL introduction, must be kept to a minimum to ensure minimum copper oxidation.
- 5) No solder mask may be present on any areas to be solder coated and leveled.

The following requirements must be met to ensure effective HASL process management before submitting boards, with the above conditions, to the process.

- 1) Machine must be maintained and calibrated to ensure all functions operate as specified.
- 2) Flux composition and condition must meet specified requirements.
- 3) Solder composition and condition must meet specified requirements.
- 4) Conveyor speed must be set as specified.
- 5) Preheat temperatures and times must meet specified requirements to ensure proper flux activation and board thermal conditioning to avoid excess thermal stress or shock.
- 6) Solder temperature must be set as specified.
- 7) Board immersion times must be short and still effect acceptable solder wetting
- 8) Air knife positioning, air pressure, and temperature must be set as specified.

With all the above requirements met, the process is initiated. The board travels through the fluxer, through the preheat system, into the solder pot, and through the air knives. With proper settings, compositions, temperatures, pressures, and dwell times, the process effect is a board meeting few acceptable requirements - or wide spread customer satisfaction.

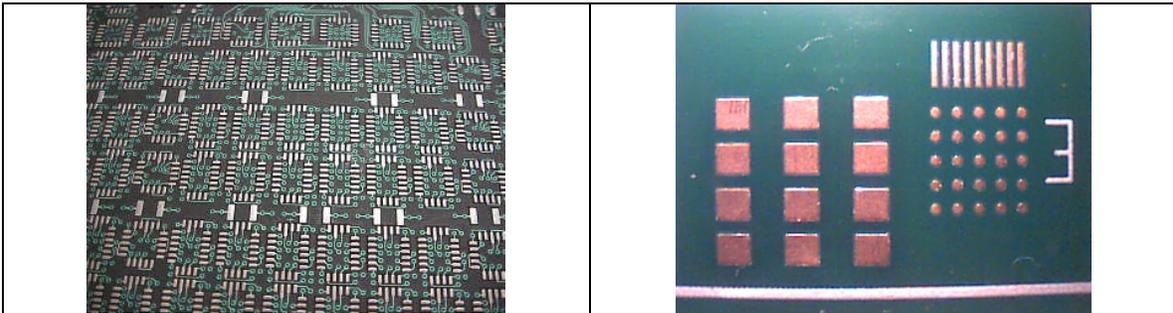
I still vote for tin/lead plating followed by HASLing yielding flat surface conditions. That would almost work, but it's not going to happen. How about bringing "flash" electroplated gold, instead of immersion, to a thickness of 7 u" - like H-P did when gold prices soared? And on it goes, as HASL perpetuates its namesake.

ALTERNATIVE SOLDER TERMINATION AREA FINISHES

As no one is an expert or even nearly perfect, neither are surface finishes - including component leads (most component suppliers have been heading in this direction for years). HASL, even though still used by many, has its problems and I have addressed them many times since the beginning of SMT. No longer is that necessary. Most everyone now is searching for alternatives or has already found and employed them with success. Still, compromises exist as they do with all things PCB or anything else. This book section attempts to present various alternatives to HASL.

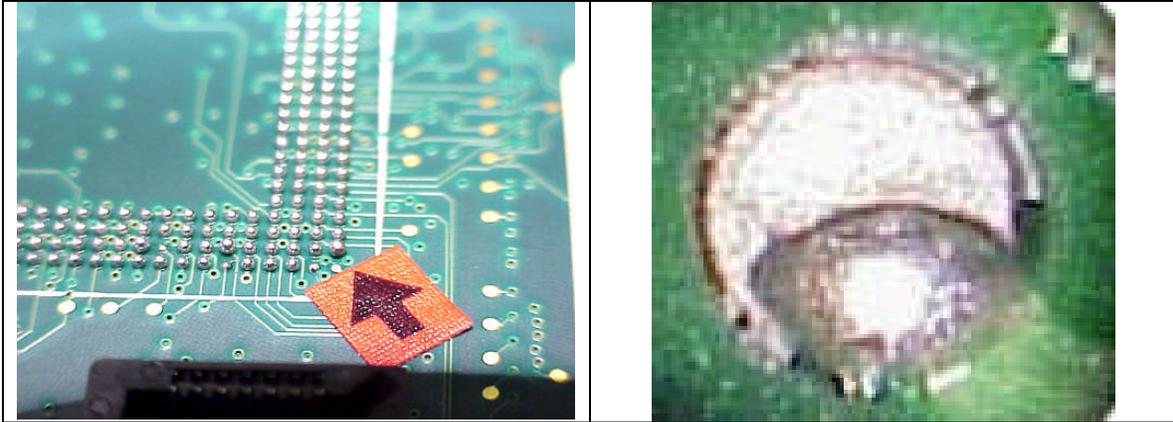
1) ENIG

I have used ENIG since its inception and first PCB use back in the early 1990's. Before that time, many other finishes were available, including tin-lead plating with or without fusing (still my favorite solder termination finish but now incapable of working for SMT applications unless not fused). The first image represents one of the earliest/serious MLB's (HAC LANTIRN program in the early 1980's) with tin-lead plating without fusing. The second shows an ENIG coated surface finish on a test coupon:



Of course, as we all know, the ENIG finish has its problems but few limitations when properly processed in well managed PCB operations. Black pad is widely known and "feared" as it should be - especially when UNQUALIFIED board suppliers cannot "prove" they manage process well enough to prevent its effect. For the most part, this no longer is an automotive type plating process. It has been refined for electronics use and rarely does anyone report black pad - as in the following figures:

NOTE: AGAIN, THE PADS SHOWN ARE BLACK. HIGH INTENSITY LIGHTING NEGATED THIS ATTRIBUTE.



The following is one of my notes used on PCB master drawings stating exact ENIG requirements. This note was derived working concurrently with major PCB suppliers and provides no possibility of mis-understanding or interpretation - or so it says somewhere.

4.3 Solder termination area (surface mount lands and plated through holes) coating/plating shall be immersion gold (0.000002" – 0.000010") over electroless nickel (0.000110" minimum).

Gold plated problems (solder joint embrittlement) exist when the metal becomes a high percentage of the termination area metallization and subsequent solder joint. This issue is non-existent with this type finish.

NOTE: WHEN I WORKED AT XEROX PRINTED CIRCUITS, IN THE EARLY 1970'S, WE DID TONS OF HEWLETT-PACKARD ELECTRO-PLATED GOLD PLATED BOARDS. THEY WORKED VERY WELL FOR THROUGH HOLE COMPONENTS - WHEN GOLD COST "ONLY" \$16/OZ. OR SO. ALSO, THEIR MLB'S WERE DESIGNED AND FABRICATED AS LANDLESS PLATED THROUGH HOLES MAKING IT ALL VERY INTERESTING AT THAT TIME. OF COURSE, 1.5 MILS OF COPPER WAS SPECIFIED AS A MINIMUM HOLE PLATING THICKNESS PROVIDING A VIRTUAL ANNULAR RING OF 1.5 MILS.

I also add that during my latest far-eastern excursion, in search of qualified board shops in China, only one of about twelve could provide specified requirements. Most didn't offer ENIG. Out of those that could, supplied non-solderable surfaces and, in two incidents, supplied black pads determined after solder testing samples provided as part of the initial evaluation process. Most shops offered tin as another provided silver.

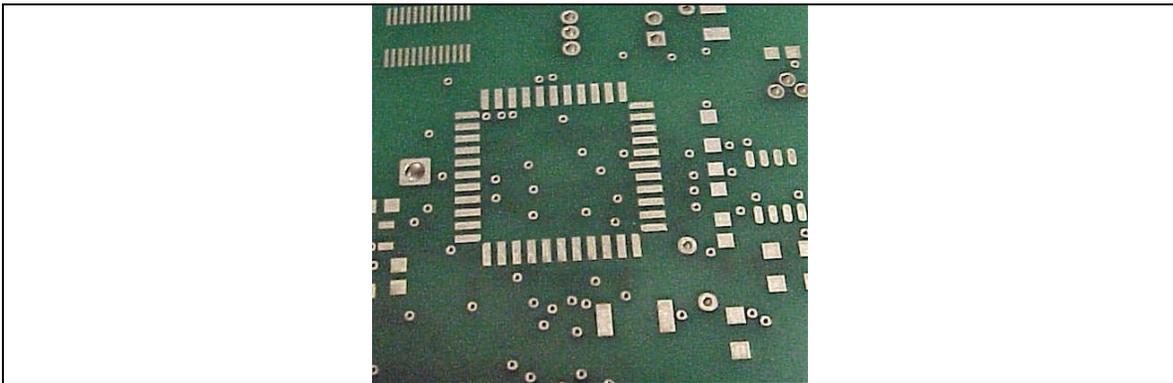
Topsearch was the only Chinese (Hong Kong based but moving inland soon) PCB fabricator proven capable after an exhaustive evaluation and qualification process. They now are doing all my client's major production requirements.

The process involves laying-down a layer of phosphorous-enriched micro-crystalline nickel on copper. The copper then is catalyzed with palladium or ruthenium. The nickel base metal is deposited with two millionths inches of gold, or so. The gold "layer" provides protection from the easily oxidized nickel and ensures a very solderable, flat surface for SMT with a very long shelf life.

2) Silver

We all know what a great conductor silver is. That has little relevance considering the type silver used as solder termination finishes today but some tout this as a plus. Naturally, back in the good old days, everyone tried to make silver work as a plating or, in my case at Sperry Flight Systems, as a conductive layer in hybrid thick film circuits. You all also know that didn't work because of migration issues/problems inherent with the medium. Now, silver is used in a different formulation and way.

I have used silver solder termination area finishes, off and on, for many years (1995 using Cookson's formula). The image below shows a one sample of about twenty plus boards I kept to test solderability over time:



Immersion silver is a direct chemical process. This process nearly is similar to those used to process OSP/OCC solder termination coatings - just as is some tin coatings. Silver is part of the mix. Chemical control and analysis is relatively simple provided all process management rules and procedures are followed. The bath nearly has indefinite life - again, when the process is carefully managed as need be with all processes. Drag-out of the low chemical concentrations quickly becoming relatively benign and silver migration is not a factor.

Tarnish can be a concern as silver surfaces oxidize over a short time, as does the copper it is supposed to protect. However, as I have shown the image above and have proven over many years of solderability testing, it keeps on soldering. It seems reasonable that any board designed for soldering during assembly operations should not exhibit tarnish or any other apparently negative attributes, as in this case, but mostly for subjective, appearance reasons. Silver should be used because of its objectively proven positive characteristics. Solderability is the most important one.

Much analysis and many studies have been done on the effects of tarnish and whether it negatively affects solderability. Mine is the least expansive but non the less important because, from the beginning, I've shown how well it works. Very little "yellow" tarnish may be visible, but solderability is not affected until it really gets black and "ugly." I've not seen it reach that stage over many years now. I would specify it more but gold has not proven deleterious to solder joint quality and reliability and it doesn't cost that much when fabricators manage their processes as they should.

3) OSP/OCC

When I was a young (relatively) process engineer at Xerox Printed Circuits in Pomona in the early 1970's, we used a process called "Seal-Brite" or some such name. It merely was a flux composition designed only to protect copper plated surfaces through the soldering process much as organic chemical coatings (OCC), or organic surface protectants, or organic solder preservatives (OSP) today, or yesterday I should say.

Of course, SMT was not around then nor was the acronym. None the less, it worked reasonably well but did not receive wide acceptance or attention - too bad. Plus there was no such thing as several soldering operations as wave, reflow, and hand types. It was all through hole way back then.

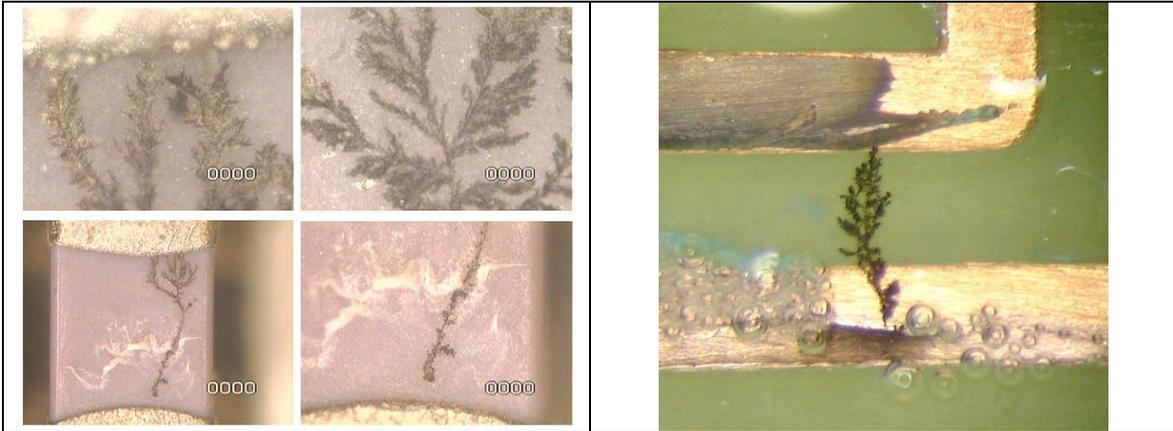
Shelf life was/is not an issue provided boards were/are soldered within a reasonable period depending on packaging, environmental, and usage conditions as today. Also, testing was not an issue because little of it was done especially in the manner it is today. There were no test probes to "gum-up" or become ineffective as is the case with OCC/OSP's used for SMT. There was/is a simple "beauty" surrounding these finishes as shown in the following images:



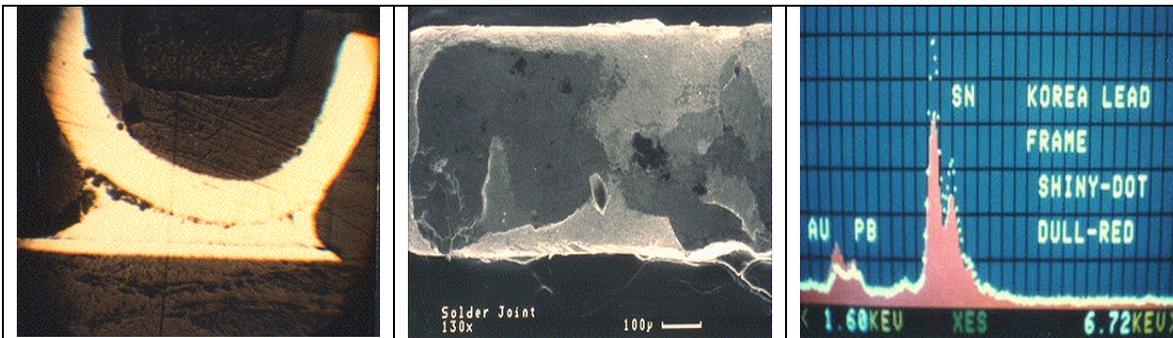
There are other issues, or problems, with OCC's. One is no contact capabilities are possible without selective plating processing. Handling can be a problems as well. Another is the cleaning process used today. We're all trying to use none and this is not always possible with this technology. However, it is a cheap way to go but that's another compromise we face in all facets of the electronics world.

4) Tin

I am not a tin kind of guy just as I was not a silver advocate many years ago as I learned early that silver migrates and tin grows whiskers. Whiskers are a bit pretty to see, don't you think, as they just don't look like anything but flowery, fern or tree-like growths called dendrites. Maybe they aren't so pretty after all:



As I've never experienced or used any tin processed boards, I cannot offer any other information. I can say there are many who can. I can, however, talk reasonably well about tin plated or coated component leads. The following images clearly show one HUGE problem with this trend that's been going on for many years but now seems to be a popular issue and point of discussion again and again:



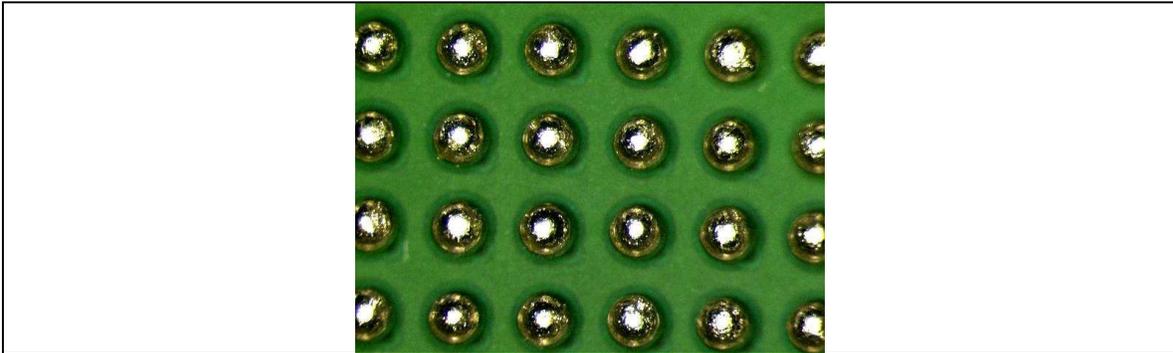
The rest of the story concerning this little J-lead revolves around NCR and its first venture into the tin neverland ranch. It was in the mid-1980's when commercial SMT was really getting off the ground. One of the companies, for which I consulted at the time, had PLCC's literally falling off all their boards.

It turns out, as seen in the third image above showing results of SEM/EDX analysis, that very high tin amounts were present on the leads. No kidding, as these little things also had an 18:1 tin-oxide to metal ratio preventing any type solder wetting using any known soldering method. It put my client out of business. As it turns out, NCR went to many off-shore lead frame operations with no solder wetting specifications or testing - AT ALL. Sounds familiar today, eh?

I also would add, that before this cheap tin trick was put in place, as it continues today, there was one last good and solderable lead finish provided in about 1985. Teledyne always tin-lead plated and fused its lead frames and they never did anything but solder very well. Then, they were bought and SMT took off, and tin became the cheap route to take for most component suppliers. Isn't it great?

Other finishes, over the years have been tried and used. Palladium, palladium/silver (to mitigate silver migration issues), gold combinations, and rhodium though primarily used on high wear surfaces as contact fingers, etc.. I'm sure there are many more and many more types will be invented over time.

At Celestica, working with SGI, we even used a high bismuth content solder paste. It was used to make solder balls to "attach" some devices with high physical masses and weight just to transport them from Wisconsin to California where they were removed and replaced. The thinking was that only temporary solder joints were required before testing and re-attachment could be performed. The following, though not an exciting image, shows some of these things:



5) SIPAD

I've never used or been exposed to this surface finish requirement. Many folks keep asking others and me about it, so the following questions and answers are provided by one supplier:

SIPAD FAQ's Midwest Printed Circuit Services

Below are many of the frequently asked questions about the SIPAD solid solder deposit process. There are still many things to learn about how SIPAD can be a benefit to the user so, if you do not see your question here or you have additional questions about the topics discussed on this page please email your questions or comments to mpcs@midwestpcb.com

What kind of solder paste can be used with SIPAD?

Only water soluble paste is available at this time. This is because in the SIPAD process, the flux and carriers that are released during the initial reflow are washed off prior to flattening.

How thick is the SIPAD ssd?

The normal SIPAD ssd is approximately .0025 - .003 thick. This is calculated by subtracting the copper thickness and other surface finish thickness from .004, the thickness of the SIMASK.

What is the shelf life for a SIPAD board?

This depends on whether the flux is applied or not. A non fluxed SIPAD board will have the same shelf life as any solder coated board. Because the paste is in solid form, there is no reason a SIPAD board could not be stored for months or years if kept in a clean and controlled environment.

Fluxed SIPAD boards also need to be stored in a temperature and humidity controlled area to remain tacky. The paper applied to the fluxed side of the SIPAD board must not be removed in storage. The manufacturer of the flux applied by MPCS Atlanta specifies a 6-7 month shelf life however, there are many documented cases of fluxed SIPAD boards remaining tacky for 12 - 18 months in both controlled and non controlled environments. If the flux loses tack, it can be re-applied and papered.

How Are SIPAD Boards Assembled?**1) Single Sided Surface Mount**

Single sided surface mount boards offer a few options for assembly. If the boards are to be hand placed and soldered, it is recommended to use a hot air soldering tool vs. a hot tip. This allows the SIPAD to flow freely, returning to the initial reflowed meniscus. The paper protecting the flux can be torn off to expose small areas of smd pads for selective soldering or completely removed and all of the parts placed at once. Because the solder paste is solid, there is no smearing during the component placement. The tacky flux is non conductive and holds even large components in place very well.

The loaded board can be run through a conventional reflow oven using a standard profile. The SIPAD ssd's will "pop" into a predictable and repeatable meniscus, attaching the component lead in the process with no shorting.

2) Two Sided Surface Mount

Two sided surface mount boards are a different story. A lot of the success depends on the process that is currently being used before SIPAD is introduced. Because solid solder deposits can only be reflowed once, placing the second side becomes the challenge as it returns to a meniscus when the first side is reflowed, eliminating the flat finish. There are several companies still seeing a lot of benefits from SIPAD even on two sided smd. Some companies are currently processing boards with mixed technology, through hole and smd on the same board.

Because there are through holes on the board, a wave solder operation must be used to attach these through hole parts. Many times, with two sided smd/PTH assemblies, the bottom side smd components are specified to be glued and fused during the wave solder process. If this is the case, the SIPAD finish need only be applied to the first side of the assembly. The components are placed on the first side using adhesive flux and SIPAD then reflowed. The board is flipped, glued, and placed. Again the board is flipped, through hole components inserted, and all bottom side smd and pth components are fused during the wave solder process.

If the boards are 100% smd, then all the components can be placed onto the SIPAD and held in place by the flux adhesive. A single pass reflow then attaches all components at once. MPCS is developing a solution for two pass reflow by using different temperature solder paste on either side opening a 30-40 degree window to allow one side to reflow at a time.

What About BGA?

Absolutely perfect for the SIPAD process. BGA footprints are very easy to print and reflow as they are not affected by the direction of the squeegee as in QFP footprints using rectangular shapes. BGA components can be placed into the flux adhesive and dragged into place without smearing any conductive paste. Because of the natural springing action of the SIPAD, even a slight misregistration is usually corrected during the reflow process. Because SIPAD uses dry film, tenting vias under BGA components becomes routine as dry film is well known for its excellent hole tenting properties.

How Much Does SIPAD Cost?

The price for the SIPAD finish is very simple to understand. The additional cost cannot be specified as a percentage of board cost because SIPAD cost the same for all boards whether 2 or 20 layers. Only the outer layers are coated so the price is only affected by the following factors.

One side or two side smd**Type of paste specified****Order Quantity****Delivery Time****Board Density**

As you can see, the SIPAD price would affect the cost of inexpensive boards more than the higher tech, more expensive boards. In most cases the extra cost of the SIPAD has to be made up in the assembly savings for the process to make sense. Some of the cost is compensated by combining the purchase orders traditionally sent to two vendors (one for the boards, one for the stencil) into one purchase order from one vendor.

Contact Midwest Printed Circuit Services for exact pricing.

Midwest Printed Circuit Services, Inc.
1741 Circuit Drive
Round Lake Beach, IL 60073
Fon (847) 740-4120
Fax (847) 740-4187
E-Mail mpcs@midwestpcb.com

NOTE: LOOK FOR OTHER SOLDER TERMINATION AREA SURFACE FINISH ALTERNATIVES ON Daan Terstaggi's FINE AND TOTALLY INVALUABLE SMT INFO SITE (http://www.smtinfo.net/Db/_Conductor%20Finishing.html)

6) Summary

I really haven't offered much, if anything, new here. However, I can say that ENIG and silver are my favorite solder termination area finishes - **AT THIS TIME**. They have served all my clients and me well, as we've never experienced any problems with either - after my first and last black pad episode at H-P in the 1990's.

I am sure there will be some new concoctions, at some time, and there will be future replacements because all things printed circuit related always are compromises and subject to change. As long as that change is positive, it is improvement. There is another word for that which is negative.

Earl Moon (Proof Of Design)

Email: moonshine@provide.net

Web: <http://moonmanondarkside.com>