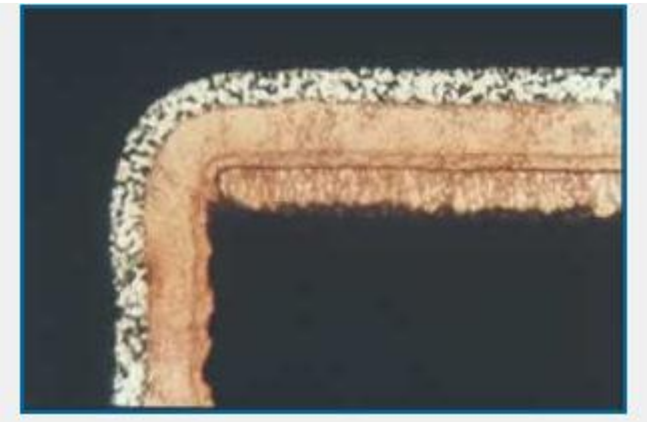
Printed circuit boards-Conductor finishes: tin-lead

Introduction

Tin-lead has been a traditional track finish for many years because it protects the underlying copper from attack during etching and, if correctly applied, has good solderability even after a long shelf life.

A coating containing both tin and lead in the right proportions can be produced by plating the elements simultaneously.



Microsection of a plated through-hole showing a flat tin-lead surface

However, this is only a co-deposited finish, the actual solder alloy being formed only when the plating has been fused, and the unfused plating remains highly solderable only for a short time. Reflowed tin-lead plating is, however, sufficiently dense to remain solderable for up to a year.

Where boards have been made using electroplated tin-lead as an etch resist, this can be prepared for solder assembly by reflowing using hot oil or infrared systems. However, the etch resist plating is usually insufficiently thick, and finishing is almost always carried out as a separate operation.

Apart from plating a thicker layer, there are two main ways of applying a tin-lead coating:

by dipping in hot solder

by printing solder paste, and then fusing it by infrared or convection reflow.

Whether plated, dipped or pasted, there is a major drawback with reflowed tin-lead, that the fused surface has a convex shape due to surface tension in the molten solder.

Microsection of pad showing the domed appearance of a reflowed tin/lead finish

Achieving a flatter solder finish

A very widely-used method of producing a flatter surface finish on boards with devices down to 0.6mm lead pitch is ‘Hot Air Solder Levelling’ (HASL). The board, previously patterned with solder mask, is first fluxed and then immersed in molten solder. Whilst the board is being withdrawn from the solder pot, jets of hot gas are directed at both sides of the board through angled nozzles usually referred to as ‘air knives’. These clear solder from the holes and blow away excess solder from the pads, a process referred to as ‘levelling’. So as not to ‘freeze’ the solder coat, the temperature of the air used is usually a little lower than that of the solder bath, at around 240°C.

The process can be carried out with the board held either vertically or horizontally: vertical HASL (shown schematically in Figure 1) tends to give better hole coverage, whereas horizontal HASL gives flatter surface mount pads.

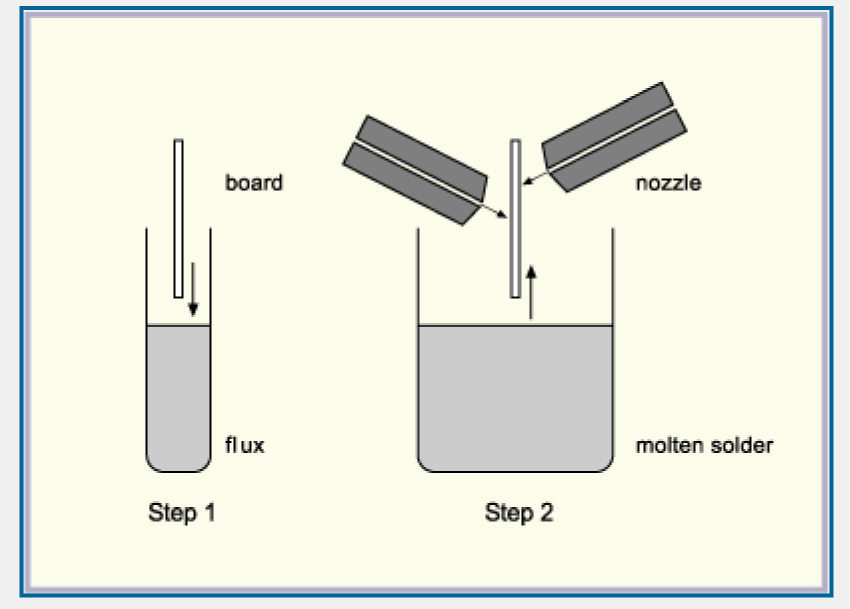


Figure 1: The process of hot air solder levelling

The soldering process parameters are determined by the amount of copper to be coated – thick boards, and boards with many plated holes, need a longer immersion.

The levelling process is straightforward, depending only on automatic control of parameters such as dwell times, withdrawal speeds, air pressure and temperatures.

Vertical HASL machine

However, the fabricator has to take extreme care to keep equipment clean from flux and solder, as debris can adversely affect the process, and the HASL process is expensive to run.

The resulting finish is adequately thin and flat for many applications, but the process must avoid the two extremes of leaving either too much or too little solder on the lands:

Too much solder leaves non-flat solder lands, causing problems in the application of solder paste, particularly for fine-pitch components.

Too little solder results in too thin a solder layer to provide good solderability.

The surface thickness range is 2–25μm, while the thickness of coating in a hole can range from 5–75μm, and at the corner of a hole it may be <1μm. The thickness is determined by the speed of withdrawal, the hot air temperature and the air flow rate.

One of the problems with the vertical process is that one end of the panel receives a longer dwell time than the other, resulting in differences in the growth of the intermetallic which grows between tin and copper – there is more about this aspect in How joints are made. The horizontal hot air levelling systems (Figure 2) which have been developed not only overcome this problem, but they also help integrate the pre-cleaning and post-cleaning modules. These operations have to be close to the fluxing and dipping parts of the process in order to minimise contamination before solder dip, and prevent the solder discolouring afterwards.

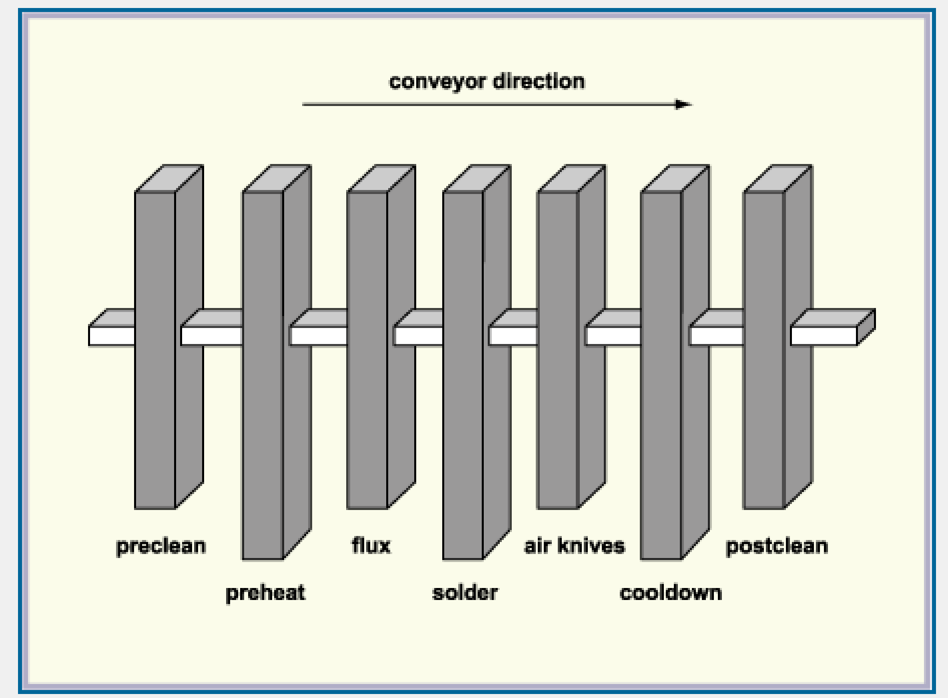


Figure 2: Schematic for the horizontal HASL process

Getting an even coat

Driven by the surface tension of the molten solder, HASL will always produce a slightly domed surface (Figure 3). Nor can HASL provide a perfectly even coating thickness, since the surface tension has more effect on small pads than large ones, and the air knives can also remove too much solder from large pad areas. Also with the vertical HASL process, there is a tendency for the coating to be slightly wedge-shaped in section.

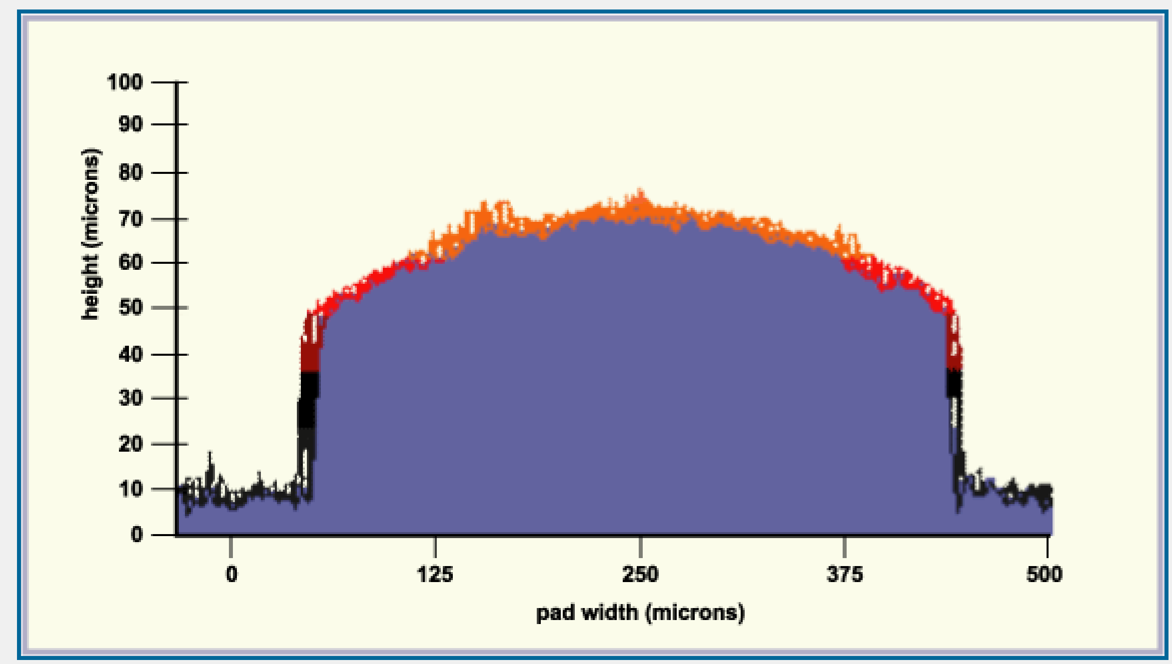
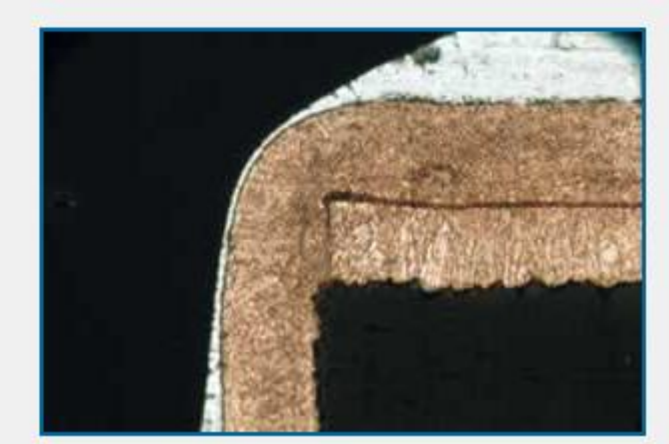


Figure 3: Typical profile of a trace with a HASL finish

In an attempt to achieve improved flatness for SMT pads, air knife pressure can be increased, but this tends to blow too much solder from through-holes. This exposes the copper-tin intermetallic layer at the ‘knee’ between hole and wall, which impairs joint quality in subsequent wave-solder operations.



Reduction of tin/lead finish thickness at
through-hole ‘knee’

This problem is made worse by long storage, as the growth of copper-tin intermetallic can easily consume the available tin at the ‘weak knees’, destroying solderability.

With surface mount components it is critical that the pad surface is flat and uniform, as this will affect the component placement process: there is data to support a reduction in defect rate when the solder coating thickness is consistent from pad to pad within a component type.

Fortunately, pads of a given size are likely to receive similar treatment and be of the same thickness, even if different types of component have more or less solder. Solder thickness requirements should be set according to the most critical area of the board, which will usually be in high density areas. Heavy solder deposits should be avoided, because thick solder deposits may flow and cause bridges.The thickness of the solder mask used has to be considered in conjunction with the solder specification requirement. Where a thick solder mask is used, for example, the 100µm mask typical of dry film resist, a 'well' may be formed between the mask and the surface to be soldered. This well can reduce the ability of the air blast to level the solder, resulting in thicker deposits. Conversely, the pad may lie above a thin solder mask, when the solder deposit may become very thin indeed. This effect is illustrated in Figure 4.

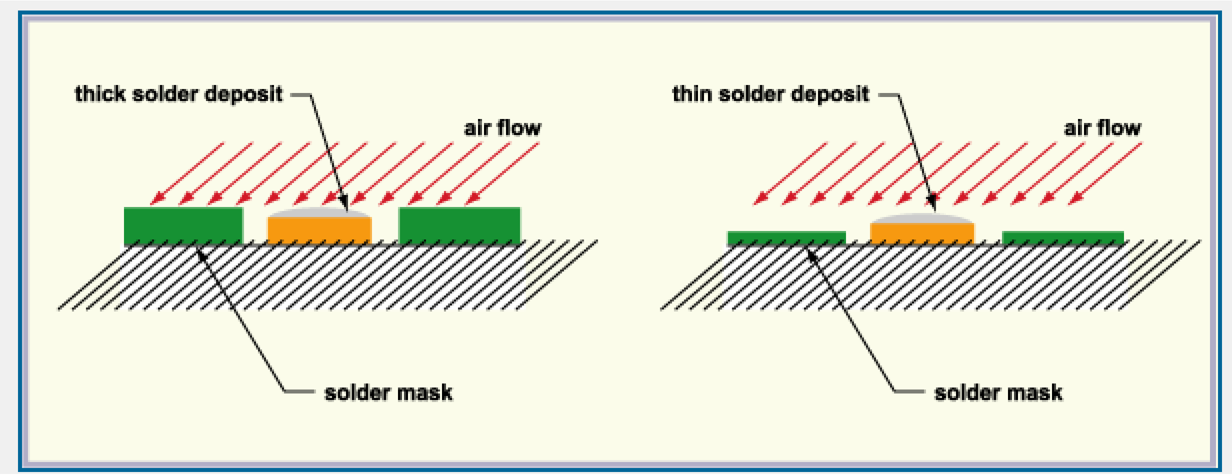
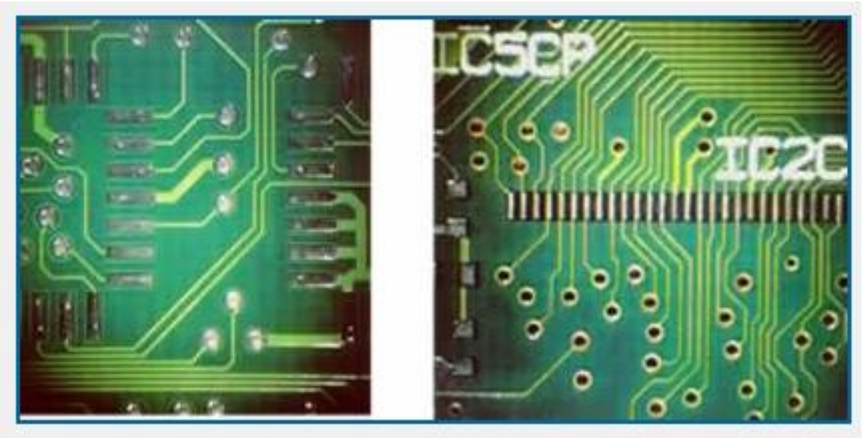


Figure 4: Effect of solder resist thickness on HASL thickness

Typical capability guidelines for thickness are a nominal thickness of 6–13µm on critical features. Most other component features will then vary in thickness from 2.5–15µm, and the total process window for all pads and traces will be 1.8–20µm. This seems fairly wide, which helps explain why manufacturers looking for an even coat have tended to prefer other coatings, especially for fine pitch applications.



Comparative results show HASL unsuited to 0.5mm pitch or below

Nevertheless, the process provides excellent protection of solderable surfaces from corrosion and contamination.

Concerns about HASL

We have already looked at the principal concern about HASL, that it may produce an uneven coating, leading to problems with surface mount component placement and reflow, and mentioned the ‘weak knees’ experience. There are, however, some other quite significant concerns relating to yield and reliability:

If the setting is incorrect, some holes may be plugged, rather than just reduced in size. Typically a HASL hole reduces in diameter by 25-50µm, and 75µm reduction is regarded as a maximum acceptable. Hole reductions are a function of the thermal preparation of the panel, through-hole drying, proper air knife set-ups and air pressures

The high temperatures involved in the process, combined with any moisture trapped in the board, can result in delamination. Although this effect happens at the HASL process, the cause lies in earlier processing. Especially on thick back planes, baking will prevent the defect, but at other times the problem stems from under-cured laminate or panel layout, and needs to be eliminated at source

The Cu_6Sn_5 intermetallic layer formed during the HASL operation: around 0.8µm thick when formed, this continues to grow with time, and increasing the temperature increases the growth rate

The potential for corrosion caused by residues of the relatively aggressive fluxes used for HASL.

There also difficulties in applying HASL to boards which are either very thick or ultra thin:

On thick boards, such as those used on backplanes, applying the substantial amount of heat needed to ensure wetting creates the possibility of delamination

On very thin boards, the air knives buffet the panels, which tend to oscillate between the two air knives on withdrawal and cause mechanical damage to the solder mask.

Controlling HASL

The uniformity of coating is affected not only by the solder composition chosen, by the type of equipment used, and by the parameters set, but also by:

the surface condition of the copper

the properties of the flux

the concentration of tin in the solder pot

any copper contamination in the solder pot.

The copper surface has to be clean, free both of copper oxides and any organic contamination. Normally an alkaline degreasing chemical bath is followed by an acid microetch, which both cleans the surface and gives it a slightly rough surface to which the solder can adhere easily. The microetched board has to be rinsed, dried, fluxed and hot air levelled as soon as possible, in order to ensure good solderability.

Poor pre-clean procedures will result in defects such as exposed copper, which are normally due to non-wetting. Goodell suggests that a good indication of contamination on copper is an oxide test, where clean solderable copper areas show a uniform coating of oxide.

Solder mask residues on the pads will impact on the HASL process, since residues are quite difficult to remove once they have been deposited on copper pads.

Copper contamination has a significant effect on the fluidity of solder, often resulting in dull, thin and uneven deposits with a grainy appearance. Merely increasing the pot temperature, although alleviating the grainy appearance, in fact only makes matters worse since the copper dissolves at an even higher rate in the solder.

The Hot air levelling Users Group (HUG) has carried out a great deal of work on the issue of uniformity, using a standard test vehicle which is representative of board designs that are known to be difficult to process through HASL equipment. Boards of varied thickness were produced and sent to HUG member companies for processing. The thickness goals were a minimum of 2.5µm and a maximum of 25µm for all locations on all features, including large chips. Coplanarity, the difference between minimum and maximum thickness on QFPs, was targeted to be 500µm maximum.

The analysis showed that values for capability index1 (CP) above 4.0, which indicates the process is ‘robust’ and adequate to meet the specifications; average values for CPK were similarly above 1.33. It was noted that:

The mean thickness and standard deviations were significantly higher (>10%) for the bottom side of the panel

Average standard deviations were significantly lower for pads orientated parallel to the direction of travel, indicating that the data was tighter and more consistent for this direction.

1 For an explanation of what this means, refer to any good textbook on statistics for manufacturing. We like John Oakland’s Statistical Process Control, published by Butterworth-Heinemann, though this may not be easily available to purchase.

Comparing the HUG analysis with the review carried out in 1994 by Banerji of Motorola, which reported the consistency of HASL solder thickness on BGA pads from various sources, indicates that substantial improvements in process control had been made over the intervening five years.

HASL is a process which gives hassle! However, there are a number of users who have combined over the years to get the process to a stage where is it reliable and cost-effective. For continuing information, visit the HASL User Group website at www.huggroup.com.

Solder Mask Over Bare Copper

Particularly if thick, a solder coating will remelt when the assembly is soldered, causing solder mask to wrinkle over the track areas.

Solder mask crinkling over reflowed tin-lead finish

Subsequently, the mask can flake off causing unsightly boards and a potential hazard at contact areas.

This problem can be circumvented by selective plating to produce an ‘SMOBC’ board (Solder Mask Over Bare Copper).

With no solder coating to flow under the solder mask during soldering, the mask remains smooth and unwrinkled

Because flux removal is easier, it is less likely that flux residues will be trapped, reducing the chance of low surface insulation resistance and potential breakdown

There can be no solder bridging under the solder mask, such shorts being hard to find and repair

The adhesion of the solder mask is better because it is applied to a flat even surface.

The most widely used method of producing SMOBC is to use tin2 as an etch resist in a conventional subtractive process, and then strip the etch resist before cleaning and applying solder mask. Solder coating is then applied just to the areas where it is needed. Figure 5 shows the process stages for solder levelling subtracted boards.

2 Tin-lead was traditionally used, because the plating could be reflowed to form a solderable surface. As that finish is now only seen on a few old military-style products, the use of tin as a replacement etch resist plating is almost universal.

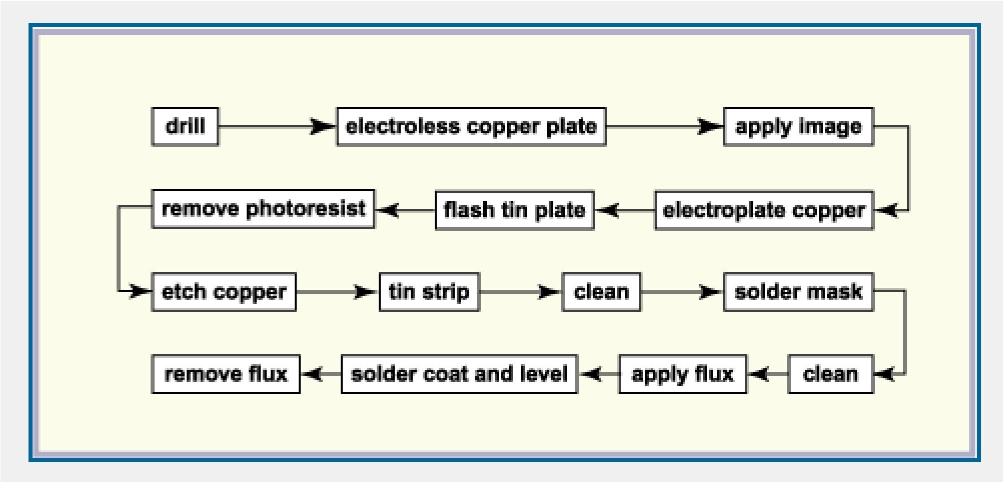


Figure 5: Schematic of the SMOBC process

Figure 5: Schematic of the SMOBC process

Solder mask over nickel (SMON) is a similar process, except that the flash plate used is nickel instead of tin. Nickel has a much higher melting point than tin, so does not have to be stripped off the tracks, there being no risk of it reflowing under the solder mask during either the solder levelling process or subsequent soldering.

SMON finish is, however, prone to a specific manufacturing problem. When the nickel is brushed to assist solder mask adhesion, slivers can become partially detached from the nickel plating, ‘overhang’ on each side of the tracks and other copper features that is created during the etching process, and thus cause short circuits. This is especially evident when boards are over-etched.

Author: Martin Tarr

Source: http://www.ami.ac.uk/courses/topics/0142_cftl/index.html