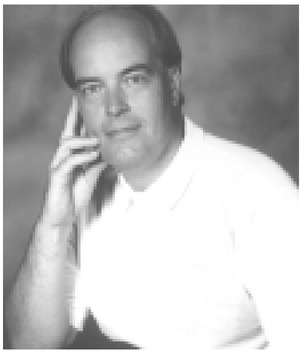


Printed Board Alternative Solder Finish

Bob Willis

The following is a brief review of the solderable and protective finishes which are available and being used in the industry. The finishes are all currently used for printed boards which will contain both conventional and surface mount components. Although a mixed technology process exposes boards to heating during reflow and glue curing double sided reflow exposes the board to two high temperature cycles, care needs to be taken that the intended soldering process is compatible with the solder finish.

Engineers should understand the rate that solderability on PCBs changes, there are two ways the solder joint formation is impacted. A surface effect normally associated with oxide formation and the effect between the coating and the base material. Normally temperature or stronger fluxes can overcome surface reactions but not between the base material you are soldering to and the coating.

The storage of and packing of PCB for solderability protection and for protections from mechanical damage. Any coating or masking must be checked for its effect on the solderability of the PCB finish. Many making agents and tapes can cause solderability problems on alternative finishes. Care needs to be taken with peelable masks during packing, thick layers and tightly packed boards can cause warpage of the boards to the shape of the mask.

When boards are packed proper protection must be provided, often as the size of the board goes up or multi-panels are used they must be protected from twisting. This can cause bow and twist or break out of routed and scored panels, this will not allow them to be used in an automated process.

Here is a guide to the solderability life of different PCB finishes based on the correct processing, packing etc:

Tin/Lead reflowed	***12 months minimum***
Hot air levelled	***12 months minimum***
Nickel/Gold	***12 months minimum***
Silver	***<12 months***
Roller tin	***< 3-6 months***
Immersion tin	***< 3 months***
Protective lacquer	***< 6 months***
Copper protective (OSP)	***< 12 months***

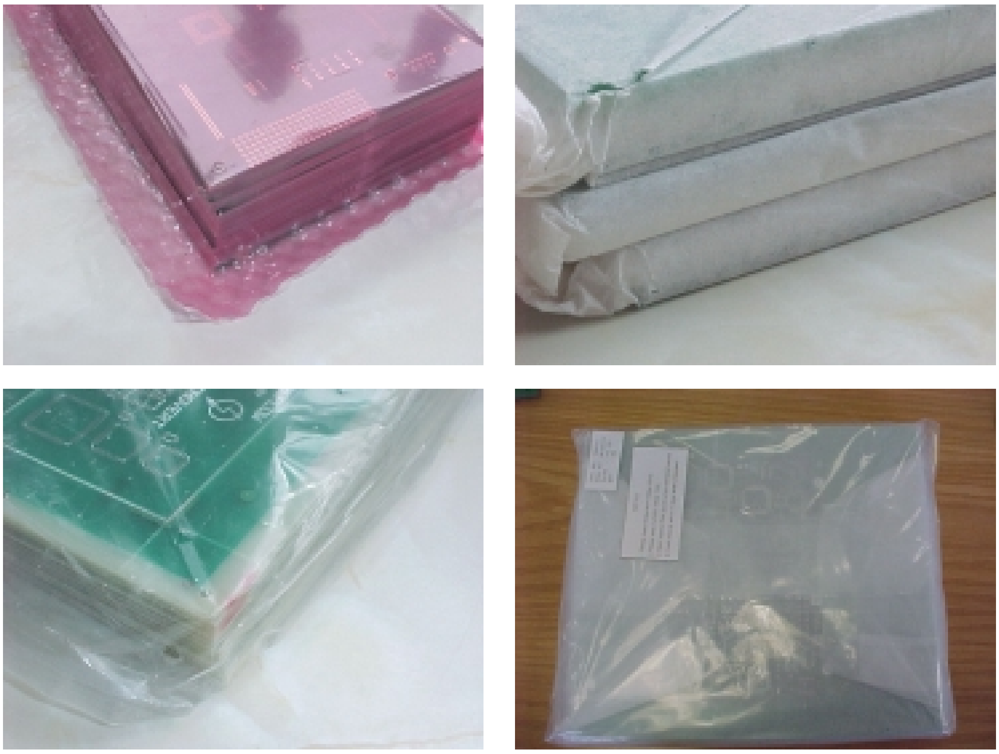
Storage varies from factory to factory and the type of packaging can also vary from supplier to supplier and must be defined in the PCB specification or the life expectancy of the solder finish defined rather than the packing. Any packing material be it plastic bags, shrink wrap, paper or tissue should not cause deteriation of the surface finish and proof should be requested on the packing materials protective performance. Sulfur free materials are often quoted but this may not be the only reason why solderability changes. Having a sealed bag or open bag does not necessarily provide better performance as the bags are not gas tight. Adding silica gel can absorb moisture but again this has only a small impact. Oxide absorbing papers have been used in the industry and this can be beneficial for a limited time.

Bob Willis Process Guides

Even if the boards were packed in a gas tight condition this may protect the surfaces from oxide formation but not control the natural aging of the base coating or base metal.

The bottom line that every engineer should understand there is a limit and you should work within the limits of the coatings. It poor practice to have to store boards for long periods of time your soldering yields will suffer.

A more detailed introduction to evaluating solderable finishes and the introduction to a company is covered in the SMART Group/Shipley Europe reports on the video on Guide Alternative PCB Solder Finishes or on the interactive CD-ROM on "PCB Design for Manufacture". A workshop on solderability and alternative solder finishes is also available.



Different packaging used in the industry for printed boards, shrink wrap, tissue paper interleaved, plastic bags and combination of tissue and plastic bags

Tin/Lead Reflow

Tin/lead has been the standard finish in the industry for many years due to its use as an etch resist for the production of plated through hole boards when subtractive processes have been adopted. It has provided an ideal production solution to protecting the copper surfaces during the final copper etching process. It has also proved useful as it provides a solderable finish for the protection of the copper pads and tracking for subsequent soldering operations.

With the increase in modern manufacturing methods using wave soldering and reflow soldering, the finish has proved unacceptable due to the circuit's exposure to high temperatures during assembly which causes reflow of the tin/lead coating under any solder resist coatings. This has led to lifting and de-lamination of the resist if the tin/lead coating is too thick.

It has also been necessary for the tin/lead plate to be reflowed during PCB manufacture prior to solder resist coating. The reason for reflowing the tin/lead has been undertaken for two reasons:

During the etching stage tin/lead slivers are left due to the undercutting which takes place during etching. If not removed the slivers are trapped under the solder resist coating and during the assembly process shorts between tracking.

The second reason or benefit for reflowing the tin/lead plate has been the improvement in long term solderability of the circuit. The tin/lead plate has a short solderability life, but if reflowed the surface is no longer porous and provides a longer shelf life. A minimum of one year's shelf life should be obtained from a surface coating of five microns or more after it has been reflowed.

Unfortunately all tin/lead coatings which are reflowed or are applied to the circuit in a liquid form will tend to form a convex meniscus of solder on the circuitry. This is generally of no consequence to conventional assembly processes apart from affecting hole size, but has led to poor yields on screen print, glue dispense and component placement during surface mount assembly if the coating is inconsistent.

With the proposed ban on lead tin/lead plating will have to change to tin plate as an alternative or one of many other coatings as a final solder finish. Recent work on lead free solder levelling has provided some very good results.

Brushed Tin/Lead Plate

The brushed tin/lead process has been offered over the years by PCB manufacturers as a compromise for surface mount assembly as it provides a flat surface for component mounting and for screen printing. It probably became popular as it was a simple solution to the reflow process which resulted in uneven pad geometry for component placement. It prevented the distortion of the laminate due to exposure to the high temperatures associated with reflow. A further benefit was that printed board manufacturers avoided the investment in solder levelling equipment and could still offer a "Surface Mount Finish".

The process has the disadvantages that the tin/lead remains in a plated state and can become unsolderable within six months; it also still has the problems of tin/lead slivers. This is probably how the finish got its name of brushed tin/lead as all surfaces were mechanically brushed to remove any tin/lead slivers from the surface of the board prior to solder resist coating. Some PCB suppliers offered this coating as an alternative to purchasing a solder levelling system in the late 80's.

Solder Levelled

The solder levelling process became popular in the early 80's and is still the most commonly specified finish for surface mount boards. Results from the author's last survey on solderable finishes trends is included in this report for reference. Eliminating the solder coating under the resist reduced the possibility of the resist lifting during the assembly soldering operation. It provided a guaranteed solderable surface from the PCB manufacture. It also provides a further benefit to the assembler of stressing the board. If the solder resist coating was poor or the lamination of a multilayer circuit was questionable then it would generally show up during exposure to the molten solder bath prior to shipment to the customer. Originally the coating was more expensive than traditional tin/lead plated finish but this is not now true.

The solder levelling process also eliminated significant mismatch between circuitry and the resist apertures. This was due to only limited temperature being applied to the laminate prior to resist application. With the tin/lead reflow process the laminate is exposed to soldering temperatures which exceed the laminate's glass transition temperature. This causes expansion and contraction due to the stress in the laminate which is no longer held by the copper foil.

The quality and consistency of the solder levelling process could be far better than some of the examples in the industry. Often the variation on coatings is down to the time spent on setting up the process for different designs. The PCB industry could do better even with vertical levelling systems. The levelling process has been demonstrated to produce satisfactory coatings for lead free coatings in recent trials on tin/copper and tin/nickel.

The key with levelling is making sure the supplier sets up his process to get the correct coverage.

As a guide the following specification may be used for tin/lead:

If specified all exposed outer copper surfaces shall be coated with solderable finish of tin/lead on the surface of mounting pads, test points and plated through hole. The tin/lead coating should provide a minimum of 12 months shelf life and meet the solderability requirements of the IPC. The coating thickness should be between 10 – 15um. The tin/lead coverage in the plated through hole should be with a minimum of 3um on the knee of the plated through hole. The solder levelling process should not affect the minimum hole size requirements.

Gold & Nickel

Gold is a traditional finish used in the industry due to excellent electrical finish, corrosion resistance and, when required, good solderability.

There has been some resistance to the use of gold, originally in Europe and still in the USA, due to concerns of reliability of the final solder fillet. In the past gold has been widely used for connectors; it was also used in the 1970's for a solderable coating on boards. In both cases the ill-informed use of thick gold > 1um coatings led to the formation of gold/tin intermettalics which in turn led to weak and fragile solder joints.

Ever since, soldering to gold has been avoided particularly in high reliability application like military and aerospace. Many existing standards relating to assembly and soldering require all gold coatings to be removed prior to the final soldering operations. It is a pity that standards are not re-examined every few years as many are just not relevant in today's technology.

Over the last six years gold over nickel have become popular finishes for surface mount boards particularly for fine pitch and mobile communication products like mobile phones. They have provided an ideal assembly surface, highly solderable and an aid to inspection due to the contrasting colour between component leads, solder and the solder paste. When wire bonding is required for chip on board applications gold over nickel has been the finish of choice when bonding and soldering is required. The cost is generally the same as solder levelled boards in medium to high volume.

The use of this finish has however raised questions with joint failures related to the plating process between gold and nickel with the black pad problems. Issues have also been seen with incorrect profiling in combination with gold and BGA boards.
As a guide the following specification may be used as a reference:

Exposed outer copper surfaces shall be coated with solderable finish of high phosphorous electroless nickel, 3 microns minimum to 7 microns maximum with finish of immersion electroless gold to a thickness of 0.03 microns minimum to 0.07 microns maximum. The coating shall be homogenous and completely cover the conductors and maintain the solderability for a minimum of 12 months.

Immersion Silver

This is a relatively new finish which was developed to provide a solderable and wire bondable coating providing all the benefits of traditional tin/lead coatings. Basically the coating is an immersion silver coating of between 0.08-0.1um which also incorporates an organic layer as part of the process. The silver "Alpha Level" coating is maintained in a highly solderable state by the organic coating. Although Cookson were the first to offer this coating other suppliers have also provided silver solutions.

The surface coating has all the benefits of any alternative finish and also resembles the tin coating when soldered. In the case of unsoldered holes or test pads there is no visible gold or copper, which to some engineers is an emotive subject. The coating cost in medium to high volume is equal to nickel/gold, but may become more cost effective as the material is further established in the market place.

Like any alternative coating, provided it is processed correctly by the circuit board manufacturer, the surface will remain solderable even after multiple heating cycles. Hence it is compatible with double sided reflow. Issues have been seen with washed off boards and the use of temp solder mask coatings. The basic coating process must be reviewed with the supplier to make sure he is using guidelines from the chemical supplier.

Flux Lacquer

The protection of the copper pads during storage and assembly prior to soldering are of prime importance However, the cost of the printed circuit is also an important issue particularly in consumer electronics. Surface mount technology is being used in all sectors of the electronics industry, inevitably it is being used in the consumer industry.

The use of flux lacquer as a protective coating which is applied to a copper pad is particularly widely used in the high volume TV/VCR industry dominated by the Japanese and Korean companies. Its use is generally confined to single sided boards.

The flux lacquer materials are supplied by a wide range of suppliers, particularly those companies who existingly supply soldering fluxes to the PCB industry. The coating is generally applied by dip, spray or roller coating. Unfortunately all coating methods provide an inconsistent coating to the board surface, with spray coating method probably being preferred. Thickness variations can lead to solder skips during wave soldering and the loss of surface mount components due to adhesive adhesion failures.

The coating provides a limited life expectancy due to the porosity of the coating and to its inconsistent coating thickness. The material is now being used by selected companies as part of a two part process. The lacquer is used as a secondary coating after the copper surface has been chemically treated and protected by a proprietary treatment.

Limitations of the coating have been its short shelf life, inconsistent coating thickness and incompatibility with Low Residue/No Clean fluxes. Significant residues are still left on the surface of the board after soldering. The coating is not really compatible with solder paste and reflow soldering so it would not be a coating of choice for double sided reflow.

A further problem has been seen when using the coating on boards which are to be flux soldered. In the case of flow soldering the components are held in place on the underside of the board with adhesive. In cases where the coating is thick the bond between the adhesive and the component is with the lacquer coating and not the printed board. During the fluxing and soldering operation the bond strength between the adhesive and lacquer can drop, causing components to be lost during contact with the solder bath.

It is common for the soldering process to be blamed for this loss of adhesion, but only limited force is applied to components during wave contact. For example, measurements of as little as 10-20 grams have been recorded acting on SOICs during contact with the wave.

The use of flux lacquers is undoubtedly a cheap option for providing a limited shelf life protection to the bare board. It does, however, suffer from the same problems as other protective coatings. Multiple high temperature exposures affects the solderability of the remaining pads thus causing soldering problems.

Protective Coatings

The protective coatings are generally defined as organic coatings referred to as OSP, (Organic Solderable Protector). The most common coatings are benzotriazole and imidazole; both are organic nitrogen compounds. Benzotriazole has long been recognised as an anti-tarnish coating used in the general metal finishing industry. Inhibitor coatings are extremely thin and essentially invisible on the copper surface.

The coatings protect the copper by chemically bonding to the surface and prevent the reaction between the copper and oxygen. The coating may be applied by dip or spray coating and followed by a rinse operation to remove any residues remaining on the solder mask surface. If required, the coating may be removed and re-applied to rejuvenate a surface which has become solderable. If required the surfaces would need to be re-cleaned with an acid etch and rinse prior to re-treatment.

The protective coatings have been used for many years by large volume manufacturers for surface mount products, an example of which is IBM. The limitations of the coating was its general inability to stand up to multiple soldering operations. The coatings are degraded by exposure to high temperature and become unsolderable with mildly activated soldering fluxes. The use of high activity water soluble fluxes have often been used on second side wave soldering processes requiring thorough cleaning.

The coatings have in the past also been susceptible to damage by high humidity storage which can degrade the solderability. Incorrect handling by assembly staff has also been seen to affect the coating due to the introduction of handling soils. A training video covering each of the different solderable finishes is available from the SMART Group to provide guidance on the correct use of these finishes.

The new generation of alternative copper protective finishes have been demonstrated to protect the surface during multiple reflow and high temperature storage. They have also been shown to withstand the handling issues during assembly and storage and are destined to provide the best vehicle for the future due to their competitive cost which is much less than any other finish.

Recent trials have indicated that cooling rates after reflow should be improved to reduce the effects on copper coatings. Cooling the board surface directly after reflow below 80oC can prolong the solderability life of the OSP coating. The use of nitrogen during first side reflow with a oxygen level of 100ppm has also provided improved performance during second sided yields. Generally reflow engineers strive to reduce peak board temperature as it exits the reflow oven to reduce the chance of component misplacement, reduce intermetallic formation and, of course, the board needs to be cool for second pass printing in a high volume operation.

OSP is currently used by divisions of IBM, Siemens, Motorola, AT&T, Olivetti, Compaq and Dell. It is also a common process offered by many printed board suppliers. OSP coated boards were recently shown by Motorola to provide better joint reliability than gold or tin/lead. If correctly processed it can provide a 12 month shelf life.

Further Alternative Technology Finishes

There have been a number of new processes introduced into the industry specifically designed for Fine Pitch SMT. The three new process finishes still use tin/lead coatings but it is their method of application which has caused them to be considered new techniques.

SIPAD - developed by Siemens, SIPAD is used on the finished board after solder mask coating. The solder mask coating must be relatively thick as it is required to define the position and height of the solder coating. The process requires solder paste to be stencil printed into the solder mask openings and onto the copper pads. As cleaning is easy on an unassembled board activated pastes may be used without difficulty. After the solder paste is reflowed the domed molten meniscus is then flattened using a hot metal plate. As the metal plate cools to below the melting point of the solder a flat surface is achieved with the solder filling the solder mask opening.

OPTIPAD - in the OPTIPAD process a temporary solder mask is used to create a mould around the land areas on the circuit board with the molten solder forced under pressure into the apertures. A membrane then seals the board surface during the solder solidification. After removal from the process equipment the temporary solder mask is stripped. The result is a well defined solder deposit of a controlled height and surface flatness.

OPTIPAD is a registered trademark of SMW Elecktronik, Germany

PPT (Precision Pad Technology) - the PPT process is a variation on the previous two processes. As in the SIPAD process, solder paste is used to provide the tin/lead coating. A dry film resist is used to create a mould around the individual pads on the circuit board. The solder paste is applied by stencil printing and reflowed in a standard reflow oven.

In the case of PPT a fixture is used to help control the position of the solder paste and to create a dimple effect on the surface of the pad surface. By using different fixtures the surface topography of the pad may be changed to aid coverage and adhesion of the sticky flux which is required for component positioning and to aid soldering.

PPT (Precision Pad Technology) is a registered trademark of Mask Technology

All of the above is based on the solder finish being correctly processed by the supplier and the assembly company doing nothing to effect the coating. These are often the main areas were problems arise not the actual storage time

Bob Willis is a process engineer providing engineering support in conventional and surface mount assembly processes. He runs production lines for suppliers at exhibitions and also provides seminar and workshops world wide. Bob has one of the largest collection of training videos, interactive CD-ROMs and training material in the industry. Bob will be presenting four Master Classes at APEX Exhibition in California this year, he will also be presenting Master Classes at SMT Nuremberg in Germany for those engineers visiting the exhibition. For further information on how Bob may be able to support your staff contact him via his web site **www.bobwillis.co.uk** or by email bob@bobwillis.co.uk