MACDERMID INCORPORATED • 245 FREIGHT ST. • WATERBURY, CT USA 06702-9984 • TEL (203) 575-5700 • US FAX (203) 575-5630 • INTERNATIONAL FAX (203) 575-7900

Assembly Guidelines
Sterling™ Silver & MacStan™ Immersion Tin Coated PCB's

♦ Flat solderable surface finishes are required for the increasingly dense PCB designs.

♦ Smaller features and SMT placement have driven the conversion of HASL to flat coatings.

The HASL coating can vary greatly, causing incomplete soldering of component leads. While protecting the underlying copper circuitry, Sterling™ Immersion Silver and MacStan™ Immersion Tin provide a planar surface for subsequent assembly operations. This allows for level placement of components for reflow and reduced paste misprints. The resultant solder joint forms a tin-copper intermetallic identical to a HASL finished board. In addition, the Sterling™ Silver and MACStan™ Immersion Tin solderability preservative provide a final finish compatible with both Tin/Lead and Lead-free alloys. Many assemblers have been successful in improving yield on fine pitch product once an alternative surface finish replaced HASL.

This document reviews critical aspects of thickness specification, handling and storage of Sterling™ and MacStan™ coated PCBs. For component assembly engineers, critical aspects of SMT that relate to PCB finish are discussed. Recommendations to optimize these aspects are provided.

Specification, Handling and Storage

Sterling™ Silver

Sterling™ Silver's thickness range is 6-25 microinches, and can withstand up to 5 assembly operations such as adhesive cure, reflow and wave soldering. Details on thickness specification are given in MacDermid's TR-205.

PCB's shipped from the board supplier shall be wrapped in acid/sulfur-free packaging materials.

Visually, circuits should be uniform and silver in color.

Wrapped packages should be stored below 86°F (30°C). When properly stored, the Sterling™ Silver finish shall maintain all functional and cosmetic properties for a minimum of 12 months. Details on handling Sterling™ Silver finished boards are provided in the Sterling™ Operating Guide.

Once the packages are opened for assembly, the parts should be kept in an environment not to exceed 86°F (30°C) and 75% RH.

Boards should be assembled within 1 week of opening or re-wrapped in acid/sulfur-free packaging material.

Sterling™ Silver coated boards can be baked. Heat and humidity alone will not harm the Sterling™ Silver surface or have any impact on functional performance. However, sulfur and chloride in the baking atmosphere can discolor the Sterling™ Silver surface and impose a deterioration of solderability performance. If Sterling™ Silver plated boards must be baked for stress relief & moisture reduction, a dedicated oven should be used. If a dedicated oven is not available, the boards should be tightly wrapped in aluminum foil to prevent possible tarnish of the silver.

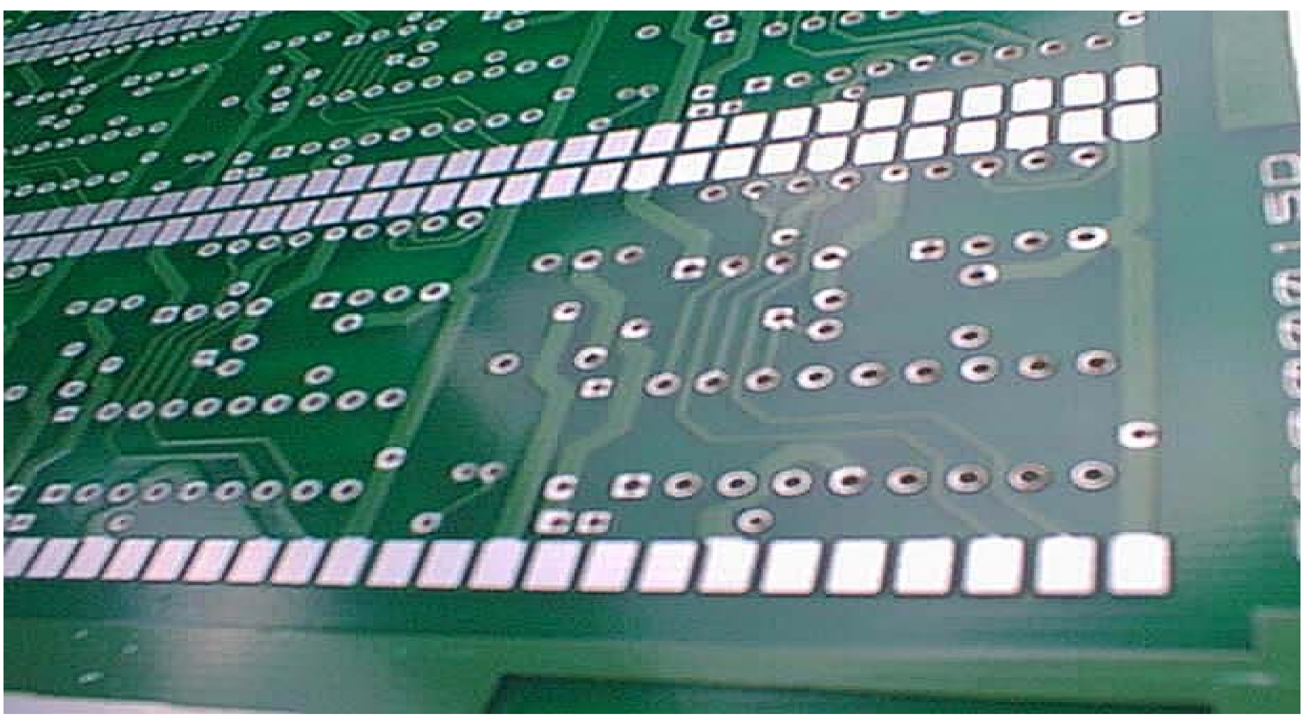


Fig. 1. Sterling™ Silver coated PCB

MacStan™ Tin

The MacStan™ Tin thickness should be greater that 0.65micron (25 microinches) pure tin in order to withstand 3 passes through assembly heat excursions, such as reflow and solder wave.

X-ray Fluorescence (XRF) determines total tin while Sequential Electrochemical Reduction Analysis (SERA) can determine pure tin thickness.

MacStan™ Tin coated PCB's may be packaged in standard packaging material.

Visually, circuits should be uniform white/gray in color.

Details for thickness and handling of MacStan™ Tin are provided in TR-305.

Wrapped packages should be stored below 86°F (30°C). When properly stored, the MacStan™ Tin finish shall maintain all functional and cosmetic properties for 6 months. Once the packages are opened for assembly, the parts should be kept in an environment not to exceed 86°F (30°C) and 75% RH.

All Tin deposits will form an intermetallic compound with copper. The intermetallic forms as a square root function of time and temperature. Once the intermetallic compound consumes the pure tin, solderability performance will be compromised. For this reason MacDermid recommends against baking of MacStan boards. However, if baking is necessary, the temperature should be kept below 110°C and not to exceed 60 minutes. Heat excursions should be kept to a minimum. Baking the panels drives the growth of copper-tin intermetallic. An oxidized surface intermetallic resists soldering.

Board Design Considerations

There are two options for forming the land pads:

Etched copper defined pads: Figure 2, A

Etched pads allow for wetting by the solder around the sides of the pads, giving a strong solder joint. The quality of the PCB etch process is important. Oversized features will "rob" critical solder mass from the component. While the thickness of the surface copper is not directly related to reliability of the assembly, it is difficult to maintain tight tolerances when etching through copper >1oz/ft^2. The soldermask clearance from the pad should be a minimum of 50µm to allow space for solder to flow around the edges. Soldermask registration becomes critical.

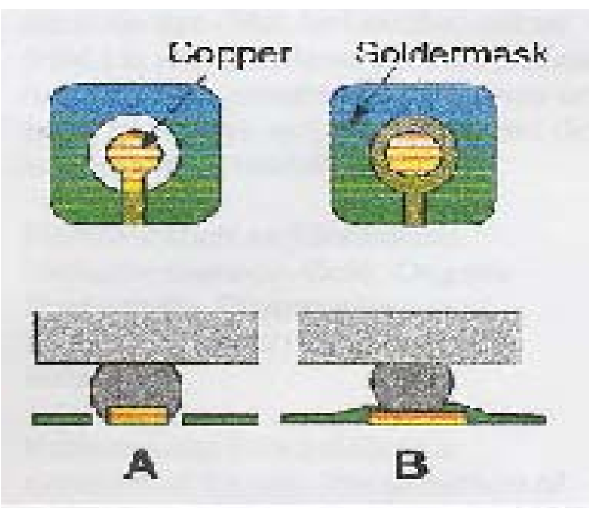


Figure 2 - Pad Styles

Solder mask defined pads: Figure 2, B

The diameter of the pad will be directly related to good mask registration. In addition, it must be ensured that no washout or developer residues are present on the copper area. Thin layers of mask around the edges of the aperture will prevent full wetting of the solder and lead to a less reliable joint. Some industry reports have indicated that the distortion formed between mask, copper and solder may form an area more susceptible to thermal fatigue than found in a solder joint with etched pads and solder mask clearance.

Assembly Processing: Reflow Process

Selecting a solder paste to use with Sterling™ Silver or MacStan™ Tin.

Most paste manufacturers have selections of solder pastes formulated specifically for use with immersion silver or tin (see attachment A for a list of commonly used flux and pastes). Acid levels and oxide level in a solder paste can have an effect on the ease of assembly. Typically no-clean and low activity solder paste meet these requirements. High activity pastes also work well for solderability and wetting, however the high level of acid in these pastes may degrade the silver or tin coating on adjacent pads, and result in a less than optimum solder joint in latter assembly operations. High level of oxides in the solder powder will impede solderability. In general, the level of oxidation of the powder increases as the particle (mesh) size decreases. Ultra fine pitch powders will have higher levels of oxide. Also the degree of sphericity affects the oxide level. Irregular shaped particles in the paste will exhibit higher oxide levels. As a general reference:

parts with larger than 20 mil pitch	-type 2 size
16-20 mil pitch	-type 3 size
12-16 mil pitch	-type 4 size
less than 12 mil pitch	-type 5 or 6 size

Paste Application

1. HASL finished PCBs arrive at assembly with a thick solid layer of solder on all surface mount and through hole features
2. At assembly, the volume of solder from the HASL coating will readily provide much of the volume of the solder joint.
3. Sterling™ and MacStan™ are extremely thin and flat finishes. Since no solder is present on the board incoming to assembly, stencils may need to be adjusted for EMS providers accustomed to assembling HASL parts.
4. Sterling™ and MacStan™ coated PCBs may require stencil apertures with a ratio of 1:1.
   - Current assembly operations use an aperture of 1:1 on feature sizes greater than 25 mils.
   - An aperture ratio of 0.9:1 is used for features 25 mils and smaller.
5. Alternately, a thick stencil may be used to supply the required bulk of solder needed for flat finishes. Of course, a chief benefit of flat finishes is the dramatic reduction in paste misprints.

For initial start up in manual or automatic mode, visual inspection is recommended to insure complete paste coverage of all pads. Some of the defects caused by insufficient volume of solder paste on a pad include cold solder joints, dewetting, non-wetting, tomb-stoning and electrical opens (due to insufficient solder joints).

Always check with the paste manufacturer for exact solder paste handling and storage. Engineers should pay careful attention to paste suppliers' requirements for viscosity, temperature, shelf-life, humidity, time on the stencils and stability of the paste during the assembly operation.

Reflow Profile Optimization:

There is no one best reflow profile for all board assemblies. Ideally, a reflow profile must be characterized for each board assembly, using thermocouples at multiple locations, on and around the device. The solder paste type, component and board thermal sensitivity must be considered in reflow profile development.

The paste supplier will have specific recommendations for the heat profile that works best with their product. Customized applications should not deviate in any significant area from the supplier's recommendations.

For more uniform heating, convection and hybrid ovens rather than straight IR ovens are recommended. Packages with large devices require fine tuned ovens that can provide uniform temperatures to all devices on the board. Use an oven with many top and bottom heating elements, to minimize temperature variations and to heat board areas that may be shielded.

To define a profile for a specific electronic device and solder paste, place thermocouples on the leads and bodies of the largest components. Monitor lead temperature to ensure good solder joints; also monitor device body temperature to protect the devices. The delta between the lead and device body temperatures can be as high as 15°C; therefore, solder joint quality can be compromised if the temperature is only monitored in one location.

Thermocouples should not be applied to heat sinks, but to alternative surfaces such as the package substrate or the PCB dielectric adjacent to the package. Adjust the dwell time in the reflow section to create good solder joints without raising the device body temperature beyond 220°C.

Reflow Heat Zones:

In the reflow process, the solder paste must be heated above its melting point, and be completely molten. This is required for the solder to fuse with the plating on the component lead, and to form the desired heel and foot fillets, or to melt the ball and cause it to collapse and form the desired joint. The solder joint formation depends on temperature and time, which are reflected in the reflow profile. Diversity in devices, solder paste, and circuit boards must be accommodated by modifications in the reflow profile. The profile can be broken down into four zones: **pre-heat, pre-flow or soak, reflow, and cool down**.

Each of these sections requires a significant temperature differential. In the reflow profile, it is essential to monitor the slopes (rate of change) of these deltas. A maximum slope in the range of 2-4°C/second is a common recommendation for PCB finishes with HASL alternatives such as Sterling™ and MacStan™.

- During the **pre-heat** phase, the solvents evaporate from the solder paste. If the temperature rises too rapidly during the pre-heat, two problems can occur. First, solder balls can be spread when the solvents burst through the flux surface membrane. This is called solder balling. Second, the solder paste can slump, because too rapid a temperature rise changes the viscosity of the solder paste. This will result in bridging. A Typical preheat phase has a ΔT of 0.5–1.0°C/second.

- The **pre-flow** or **soak** section, also called the flux activation stage, brings the entire assembly device up to the temperature at which the paste changes from solid to liquid, and becomes active. During this phase, the flux activators begin to chemically scrub oxidation from the solderable surfaces of the paste and the PCB. This soak time should be long enough to allow the flux to clean the bonding surfaces, but not so long that the flux is evaporated prematurely. If the activation temperature of the flux is not reached in the soak zone, only partial cleaning of the areas to be soldered is achieved, and poor wetting will result. While in the soak zone, the temperature of the PCB and its components is equalized. If the temperature on the PCB is not equalized, cold joints, or tombstoning, can occur. Dwell time in the soak zone will depend on solder paste type, population, and heat sink of both PCB and components to be attached.

- During the **reflow** phase, the temperature increases to melt the solder paste alloy and subsequently form the solder joints; the typical solder temperature is 30 – 40°C above the solder paste melting point. For solder paste composed of Sn62Pb36Ag2, this means 209 - 219°C, while for the eutectic Sn63Pb37 solder alloy, 213 - 223°C. However, in order to secure that all solder joints are soldered correctly, in practice a slightly higher temperature around 215 - 225°C is often used. An excessive peak temperature will result in a gritty and wrinkled surface of the solder joint, due to high oxidation. A high temperature will also result in delamination of the PCB laminate, and the electrical characteristics will be altered. A third failure caused by excessive peak temperature can be charring of the flux. It is important to monitor the time at peak reflow temperature in several areas of the PCB and components.

  During the reflow phase, the solder and the copper begin the formation of intermetallic. PCBs coated with either Sterling™ or MacStan™ result in a copper-tin intermetallic identical to the solder joint of a HASL coated PCB.

- The often-overlooked **cool down** phase is a critical step in the formation of a reliable solder joint. For the solder joint to perform as a strong bond between the solder pad and the component terminal, the cooling should be as fast as possible. A slower rate may increase the grain size of the intermetallic compounds making the solder joint brittle and weak. The components can crack if the temperature drops too rapidly. The cooling should therefore be 3-4°C/second down to around 130°C. Below 130°C the cooling rate is not critical to the solder joint quality.

Conveyor speed is a critical component of the reflow profile speeding up or slowing the conveyor results in altered temperature profiles.

Soldering atmosphere for both Sterling™ and MacStan™ can be air or Nitrogen environment. Sterling™ silver relies on optimized exhaust to prevent the accumulation of sulfide in the reflow environment. MacStan™ tin is less impacted by sulfur in the atmosphere.

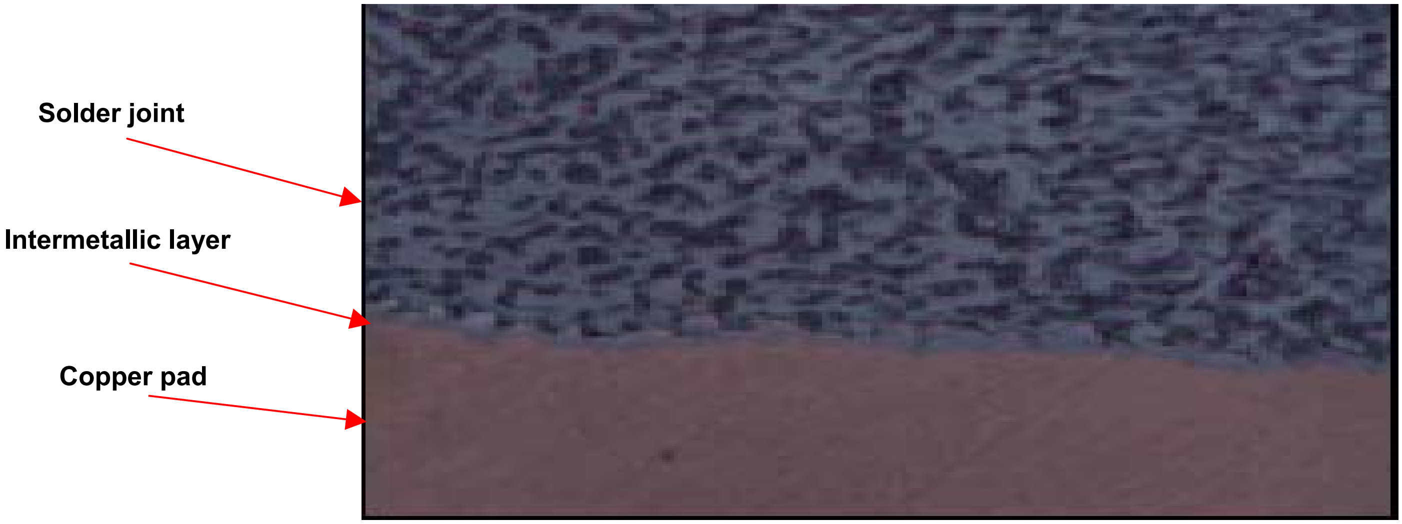
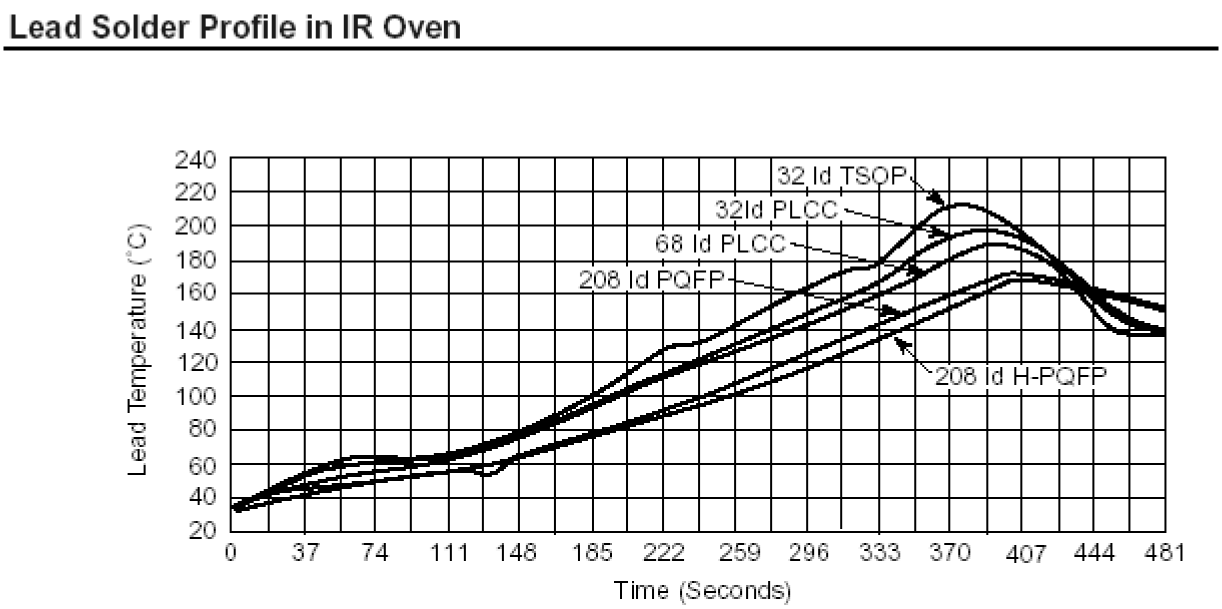


Figure 3. Solder joint - intermetallic layer



. Lead Peak Temperatures for Large Mass/Low Thermal Resistant and SMC

Package	Device	θ_{Jc}	Mass	Span	Peak Temp
32-lead TSOP	28F010	20° C/W	.37 gm	.79 in	220° C
32-lead PLCC	28F010	27° C/W	1.1 gm	.59 in	208° C
68-lead PLCC	80C196KB	12° C/W	4.8 gm	.99 in	194° C
208-lead PQFP	82425EX	8.5° C/W	6.2 gm	2.21 in	191° C
208-lead H-PQFP	80486 DX4	1° C/W	11.0 gm	1.21 in	187° C

Figure 4. Optimized Reflow Profile

Attachment A

The following table lists flux and paste products commonly used with Sterling™ and MacStan™.

MANUFACTURER & PRODUCT NAME	TYPE	APPLICATION
AIM		
293+	No Clean	SMT Stencil printing – Solder paste
WS483	Water soluble	SMT Stencil printing – Solder paste
ALPHA FLUX		
IF2005M	Active solvent complex (alcohol blend)	Solder float, no preheat required
PF2	No clean alcohol based	Wave soldering
RF 800.	No Clean	Wave soldering
BRITISH STD.		
	Rosin based	Solder float, use preheat to dry
ENTHONE		
NCF 8655	No clean, VOC free	Wave solder
NCF 874OS.	No clean, low residue	Wave solder
ESP		
6Sn63-575-D	No clean	SMT Stencil printing – Solder paste
KESTER		
1585	Rosin	Wave soldering
2235	Water soluble	Wave solder, std preheat
922-CX	Low residue	Wetting balance, no preheat required
951	Alcohol based no clean	Wave soldering
958	Low residue	Wave soldering
959	Low residue, no clean	Wave solder, std preheat
R244	No clean	SMT Stencil printing – Solder paste
R560	Water soluble	SMT Stencil printing – Solder paste
LONCO		
26F	No clean	Wave soldering
MULTICORE (Loctite)		
CR36	No clean	SMT Stencil printing – Solder paste
Hydro X20	No clean	Wave soldering
MP200	No clean	SMT Stencil printing – Solder paste
NR3S-05 Ecosol	VOC free, No Clean	Wave soldering, std preheat
RP15	No clean	SMT Stencil printing – Solder paste
X3-07I	No clean	Wave soldering
X3-08i	No clean	Solder float, no preheat required, wave std preheat
X32-10M	No clean	Wave solder, std preheat
X1248	RMA No clean	Wave soldering
QUALITEK		
302	No clean	Wave solder, std preheat
737	Water soluble	Wave solder, std preheat
Tamradio		
NC7-SS-200	No Clean	SMT Stencil printing – Solder paste

Consult the suppliers' Technical Data Sheet for each product. The TDS will contain important information on specific applications, recommendations for handling, printer set up, paste application, and reflow profile data. The supplier will also have information on their best product to use with either silver or tin finished PCBs.