

CONSIDERATIONS FOR SELECTING A PRINTED CIRCUIT BOARD SURFACE FINISH

Randy Schueller, Ph.D.
DfR Solutions
Minneapolis, MN, USA
rschueller@dfrsolutions.com

ABSTRACT

The selection of the surface finish to be used on the PCB could be the most important material decision made for the electronic assembly. The dominant surface finish prior to the implementation of RoHS was SnPb HASL. With the elimination of Pb and the need to attach components with finer and finer pitch, a number of planar surface finishes have recently gained in market share. These include ImAg, OSP, ENIG, ImSn along with some newer options to be discussed. The surface finish influences the process yield, the amount of rework necessary, the field failure rate, the ability to test, and of course the cost. One can be lead astray by selecting the lowest cost surface finish only to find that the total cost is much higher. The selection of a surface finish should be done with a holistic approach that considers all important aspects of the assembly. This paper will weigh the attributes of the various finishes and make suggestions of which fits best with various product applications. In situations where there is no clear fit, various engineering tradeoffs can be made.

Key words: Surface Finish, Selection, ENIG, ENEPIG, HASL, ImAg, OSP.

INTRODUCTION

The surface finish selected for use on the printed circuit board (PCB) is perhaps the most important material decision made in the construction of an electronic assembly. The surface finish heavily influences the cost, manufacturability, quality and reliability of the final product. There are about half a dozen commonly used surface finishes today with a number of new finishes recently introduced. As little as 7 years ago over 70% of electronics used SnPb hot air solder level (HASL) as the finish. Since then the usage has declined to near 15%. The lost market share has been distributed among the other finish options. Two trends are responsible for this major industry transition, the most obvious being the RoHS restriction of Pb. The other is the continual trend toward placement of finer pitch components (making assembly with HASL more difficult).

There are many surface finishes currently sharing the market with no clear winner coming forward. Each surface finish has attributes that make it attractive for certain applications; however, this also implies that important tradeoffs are being made. An obstacle in

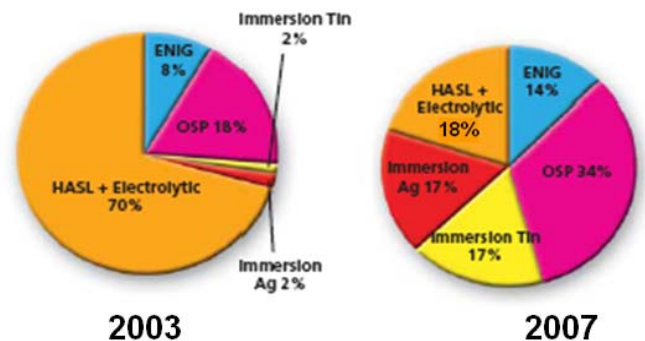


Figure 1. Market share of surface finishes (measured in board area).¹

the selection of the proper surface finish is the fact that this decision directly impacts many functional groups within an organization. For example if the procurement group is responsible for selecting the finish they will likely proceed with whatever is lowest cost at the PCB level. A PCB component engineer might select the finish that is easiest for the PCB supplier to produce with high yield. The assembly or quality engineer would likely be most attracted to the finish that provides the largest process window for assembly and test. The sustaining engineer and reliability engineer are interested in the finish that provides the most robust end product. In the end, the best surface finish for your application is the one that considers the impact to all functions and provides the lowest overall cost. This paper will present each finish option with the key product considerations in mind.

KEY PRODUCT CONSIDERATIONS

Electronic products are often grouped according to their industry. For example some of these might include:

- Medical
- Aerospace/Defense
- Consumer
- Consumer hand held
- Telecommunications
- Testing equipment

However, there are subsets within each group and peripheral products used in these products that have differing requirements. Therefore, grouping by industry is usually inadequate. It can be helpful to break it down further and list the primary attributes of the product that

are heavily influenced by the surface finish type. For example, questions pertaining to product attributes might include:

- What is the cost sensitivity of the product (how important is the surface finish cost)?
- What are the product volume requirements (high, medium, or low)?
- Is it a SnPb or Pb-free process?
- If Pb-free, is shock/drop a concern for your product?
- Is fine pitch assembly required (how fine)?
- What is the user environment (is corrosion a concern)?
- Is wave solder required and if so, how thick are the boards?
- Is high yield in-circuit testing required?
- Is wire bonding to the PCB SF required?

Understanding how the surface finishes influence each of these product attributes will provide guidance on which may be optimal for your product, regardless of the industry class.

SURFACE FINISH CHARACTERISTICS

Matching the surface finish to the product attributes will provide the best chance for selecting the optimal finish for your application. The important aspects of each surface finish will be discussed and a diagram will be shown that summarizes which product attributes fit the characteristics of the finish and which are not critical to the product application.

ORGANIC SOLDERABILITY PRESERVATIVE (OSP)

We'll begin with OSP and include both the low temperature SnPb version along with the higher temperature version developed for Pb-free applications. OSP is the low cost leader at the circuit board level. It is an organic coating that is deposited with a wet in-line panel process. Many PCB manufacturers have the equipment to handle high volumes of boards. OSP is the most common finish with a market share of about 35%. For many less complex assemblies, OSP is an excellent surface finish selection. Where it sometimes falls short is with double sided boards where wave soldering is required. The surface mount thermal exposures can break down the film and allow oxidation of the copper in the barrels, thus reducing the solderability of the through-hole vias. Boards with a thickness of 0.062" or less have a reasonable chance of achieving sufficient hole-fill with OSP, however, thicker boards have proven difficult with inconsistent results.

In circuit testing is another drawback with OSP since it is a non-conductive coating. Probing through the coating is not recommended so the strategy is typically to print solder paste onto test points. With no-clean fluxes this

can be a problem since the flux residue can gum up the probe tips. This is particularly troublesome when the test points are vias, since printing paste in vias results in a dimple that collects the flux, as illustrated in Figure 2.

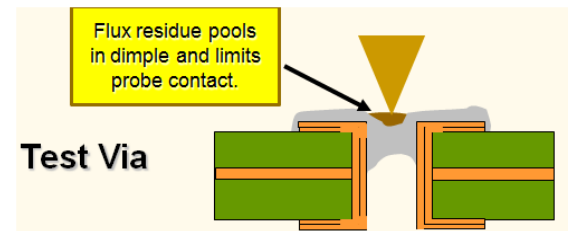


Figure 2. Test vias prove difficult since solder paste deposited onto them tend to create a dimple that collects the flux residue.

As one can imagine OSP is very good for fine pitch assembly (as are the other planar finishes) since the stencil is allowed to seat firmly against the surface of the copper pads (creating good gasketing). Since nearly all PCB shops have OSP capability and the throughput tends to be high, this is a good finish when high volumes are required. If your product is Pb-free and there is a possibility for high strain rate to occur on the PCB (from board deflection or drop/shock events) then any surface finish that allows for a tin to copper bond is favored (as with OSP, ImAg, ImSn, or ImPd).

The diagram shown in Figure 3 illustrates how OSP fits with the important product attributes. In this example, OSP is a favorable finish if the product being built does not require Pb-free wave soldering for thick boards, high yield ICT is not required, and wire bonding to the surface finish is not required.

IMMERSION SILVER (ImAg)

ImAg is a thin film of silver (typically 6-12 μ "") deposited directly on copper. In 2003, before RoHS took effect, the market share of ImAg as a surface finish was below 2%. By 2008 this market share had quickly risen to near 17%.² Silver has many favorable attributes but chief among these is the favorable solderability along with the ability to easily probe directly to the finish during ICT. The deposition equipment can be costly but the through-put is high enough to bring the cost per sqft to just above that of OSP. The popularity of this finish received a boost in 2006 when Underwriters Laboratory reconfirmed ImAg as an acceptable finish after performing temperature/humidity/bias testing with favorable results (no electrochemical migration took place).

When scaled up to high volume for use in commercial electronic products, a number of weaknesses were discovered with ImAg. One of these was the tendency to cause micro-voids (or champagne voids) along the surface of the PCB pad (see Figure 4). This resulted in early field failures from thermal cycling. Eventually the root cause

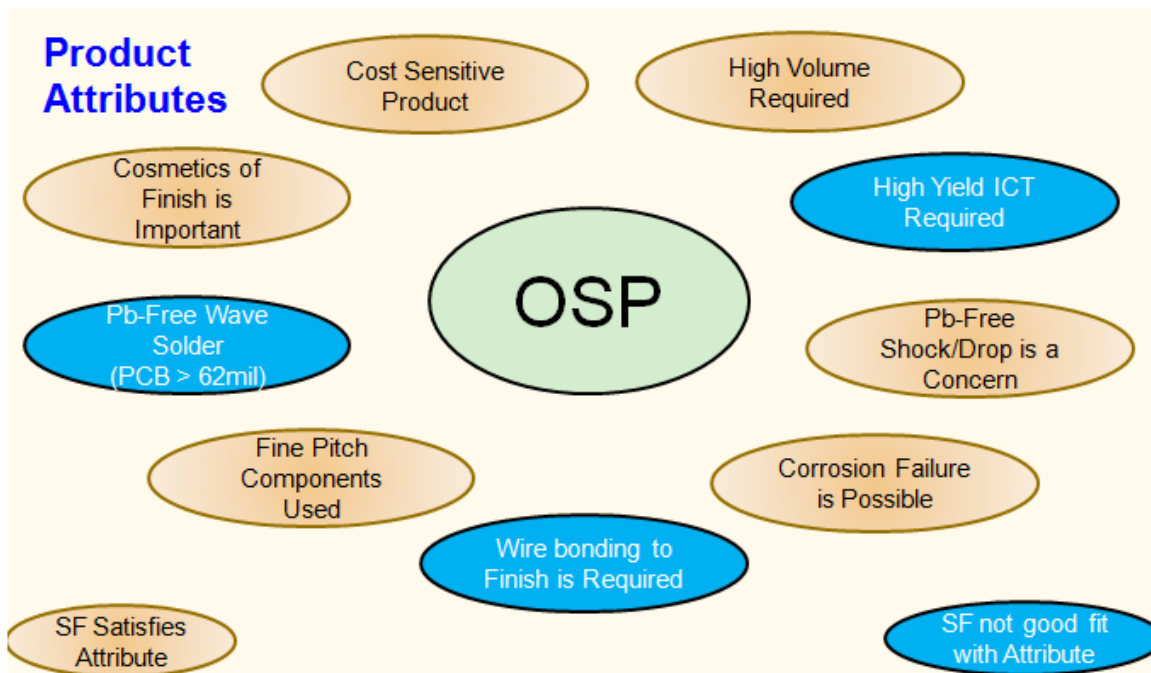


Figure 3. A list of product attributes, color coded to show how well they fit with OSP capabilities.

of these voids was discovered and the plating process improved to eliminate this issue.³

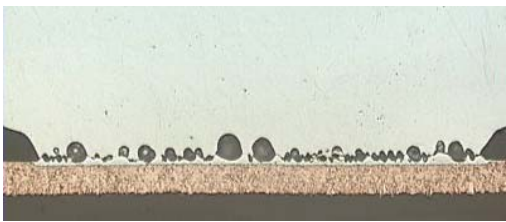


Figure 4. Example of microvoids due to improper silver plating.

Another issue was tarnishing of any exposed silver after the assembly has been in the field for a period of time. In

extreme cases the tarnish can become almost black in color. Such tarnishing does not typically result in failure, however, if the PCB is visible to the user the result can be a poor perception of quality (for example the exposed PCB on a hard drive). Poor cosmetics of the PCB can result in more frequent replacement of PCBs in the field.

The most significant issue that has arisen from use of ImAg has been creep corrosion when products are used in an environment high in air-borne sulfur (and exacerbated with high humidity). A typical example is shown in Figure 5.⁴ The UL testing mentioned earlier did not include any gaseous contaminants and so this issue was not discovered until products were deployed into factories

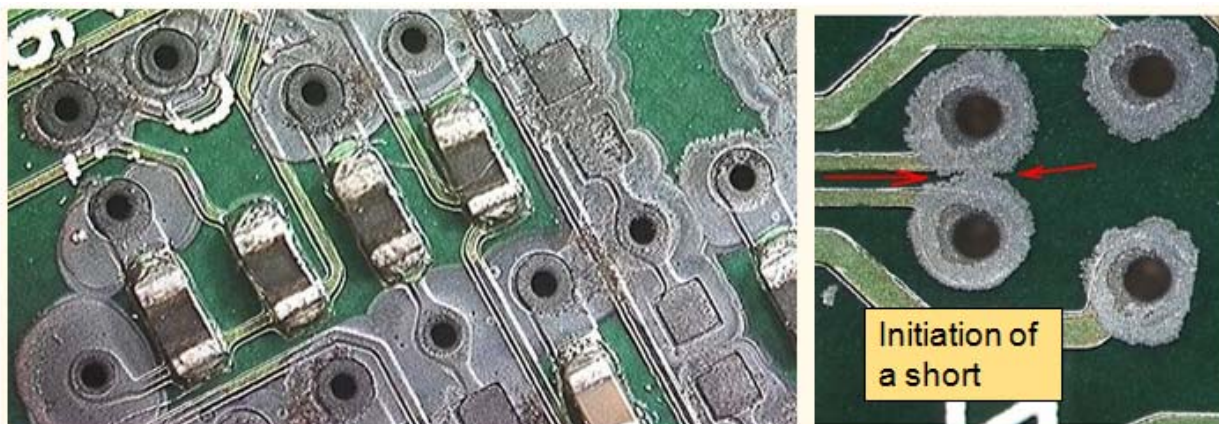


Figure 5. Example of creep corrosion failure of ImAg when exposed to sulfur.

or regions of the world where elevated levels of sulfur were present. Industries high in sulfur include: paper mills, rubber/tire manufacturers, fertilizer, waste water treatment, mining/smelting, asphalt, petrochemical, clay modeling studios, and others. Locations near these industries were also affected. These locations had no problems with previous generations of electronics where SnPb HASL or OSP were the dominant surface finishes. However, upon switching to ImAg (with the Pb-free transition) they saw rapid failure in as little as 3 weeks. Failure rates were over 100% since replacement units would fail just as quickly. There has been activity to improve the ImAg finish to reduce the effects of creep corrosion; however, the only guaranteed solution has been to replace this finish when PCBs have the possibility of being exposed to air-born sulfur.

The challenge with sulfur induced creep corrosion is that the user environment (or shipping environment) is not always known. 98% of the end users may not have a problem, but for the 2% that do, the replacement costs can be very large. One observation that may be helpful is that airflow is a large contributor to the corrosion rate. Electronics that draw in air for cooling often fail fairly quickly, however electronics that are self contained will survive much longer and may not fail at all in a high sulfur environment.

The Product Requirement illustration for ImAg is shown in Figure 6. From this chart it is seen that ImAg is a good surface finish if one is confident that the product will not be exposed to sulfur during shipping or use of the product. It is a favorable surface finish for most other attributes.

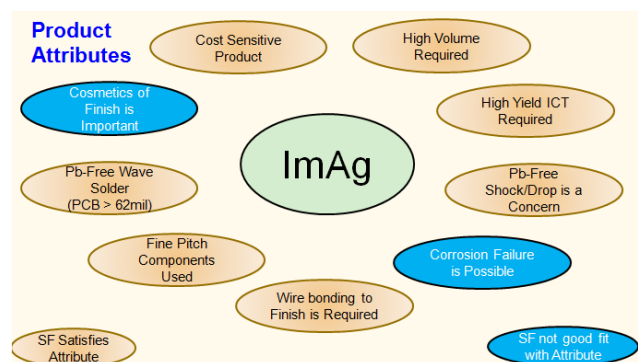


Figure 6. Product requirements and how they fit with ImAg characteristics.

ENIG and ENEPIG

Electroless nickel immersion gold (ENIG) consists of a thin layer of gold over a thick layer of nickel. Plating gold directly onto copper is not effective since copper will quickly diffuse through the gold and oxidize on the surface (reducing solderability). Nickel serves as a barrier layer to copper, though Ni can eventually also diffuse to the surface of gold and cause the same

solderability issue (it just takes place at a slower rate than copper). Typical ENIG specifications are defined by IPC-4552 Specification for Electroless Nickel/Immersion Gold. The nickel thickness must be in the range of 3-6 μm which is sufficient to prevent porosity through to the base copper. The nickel is plated with an autocatalytic process with phosphorus reducing agents. This phosphorus gets incorporated into the deposit in the range of 8-12%. The gold thickness is specified as 0.05 μm minimum (2 μinches) at 4 sigma standard deviation below the mean. This often results in a target mean of 3-5 μinches .

ENIG is a versatile surface finish that provides the many advantages listed below:

- Excellent flatness (easier to assemble fine pitch components)
- Robust for multiple reflow cycles.
- Supports alternate connections (wirebond, edge card connection, push buttons, etc.).
- Good for in-circuit testing.
- Good for press fit connectors.
- Gold readily dissolves into solder and does not tarnish or oxidize
- Nickel strengthens the PTH and increases thermal cycle life.
- The Nickel barrier prevents copper dissolution during wave solder & rework

These advantages come at a cost since this is the most expensive surface finish (estimated at almost \$4.00 per square foot of PCB due to the high gold prices). This cost typically prohibits use for high volume applications which in-turn limits the installed capacity in place at PCB shops.

Black pad remains a potential problem with ENIG when the gold plating process is not well controlled. Black pad results from dissolution of the surface of the Ni layer upon submersion in the acidic gold plating bath. Removal of too much nickel (especially along grain boundaries) leaves behind a phosphorus rich layer. Deposition of gold then takes place over the phosphorous layer. When wetted with solder, the gold quickly dissolves and the tin is left to react with the phosphorous rich layer. The resulting poor bond strength can cause failure in the field when the solder joint is stressed. A cross section example of a black pad is shown in Figure 7. This figure also shows the preferential etching of the nickel along the grain boundaries, as observed when the top gold layer is stripped away.

What makes the black pad defect so troubling is that it is invisible to the naked eye since the gold covers the underlying problem. Furthermore, for various reasons it can occur on some regions of the board and not others. A great deal of effort has gone into resolving this defect. Most would argue that the incident rate has gone down

over time; however, it still occasionally shows up with costly consequences.

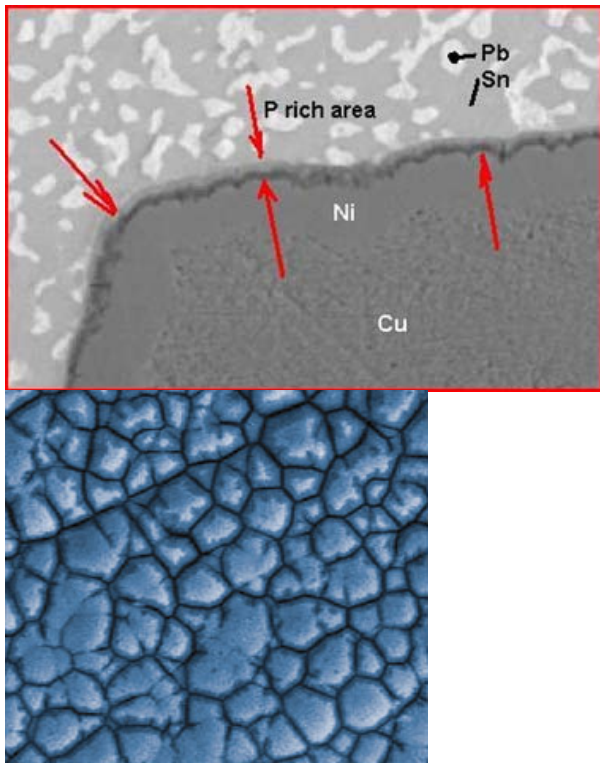


Figure 7. Cross section showing high phosphorous layer that is a sign of black pad. If the gold top layer is etched away, the nickel layer with high phosphor regions at the grain boundaries can be detected.

ENEPIG

The best solution to the black pad defect has been to deposit a layer of electroless palladium over the nickel layer. The Pd is not etched away during the gold plating process so the potential for black pad is eliminated. For this reason, ENEPIG has become a popular choice for electronic products that require high reliability (i.e. medical and aerospace).

Board Strain Sensitivity

On circuit boards with finishes such as OSP, ImAg, HASL or ImSn, the tin based solder bonds directly to copper with formation of Sn-Cu intermetallics. In the case of ENIG and ENEPIG the tin forms intermetallics with the nickel layer. In this case the intermetallic is mostly Ni_6Sn_5 which is more dense and brittle than the copper based IMCs. With the use of eutectic SnPb solder, the elastic modulus of SnPb was rather low and this allowed the solder to deform and absorb stress on the joint. With Pb-free solder, such as SnAgCu alloys (especially SAC305), the elastic modulus is 25% higher. When stress is applied, the solder does not deform as easily and the stress concentrates on the intermetallic interface. As a result, when a Pb-free PCB with ENIG or ENEPIG finish undergoes board strain due to deflection or a drop/shock event, solder joint failure occurs at lower strain levels than with the other surface finishes. This

layer can be so fragile that cracks can sometimes be found at this interface on Pb-free assemblies that had not even seen mechanical testing, as shown in Figure 8.



Figure 8. A brittle crack at the Ni/IMC interface in a Pb-free solder joint. This was an as-assembled board after random cross section.

The most significant effect of the brittle interface between Ni and Pb-free solder is the reduction in fracture toughness that is apparent when performing drop/shock testing. The following graph in Figure 9 compares drop results for SnPb and Pb-free solder with both an OSP and an ENIG surface finish (note that OSP results in a Cu to Sn bond). Two trends are observed, 1) SnPb does better in a shock/drop environment than Pb-free SAC305 alloy, and 2) the ENIG surface finish significantly reduces the drop strength – especially when combined with the SAC305 solder.

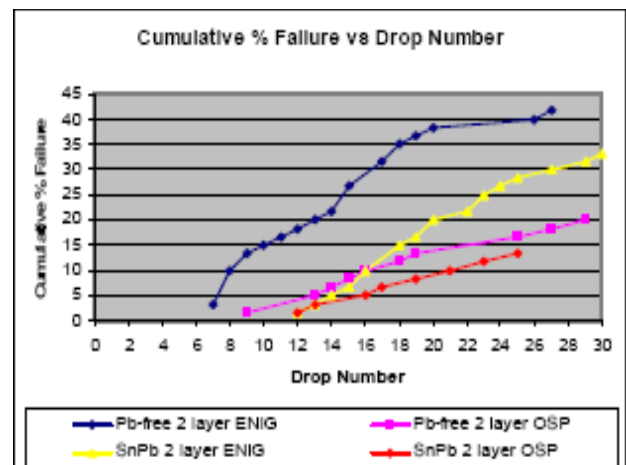


Figure 9. Cumulative drop to failure vs. drop number. Showing relative strength of alloy/surface finish combination.⁵

Similar results were gathered at DfR Solutions where we tested Pb-free BGAs (with SAC305 solder) on OSP and ENIG. A SnPb solder control was also tested and the results are shown in Figure 10. This data, along with a number of other industry studies, would suggest that ENIG or ENEPIG surface finish be discouraged for use in any application where high or frequent shock force is a possibility. Furthermore, one needs to be concerned with the reliability of this material set surviving the manufacturing and shipping process. Board strain can occur during manual handling, the stuffing of PTH components, installing press fit connectors, in-circuit testing or attaching the PCB to the chassis.

How ENIG performs with respect to the critical product requirements is illustrated in Figure 11. In summary, this is an excellent finish unless your application is cost sensitive, is high volume, or is Pb-free and susceptible to board strain or shock events.

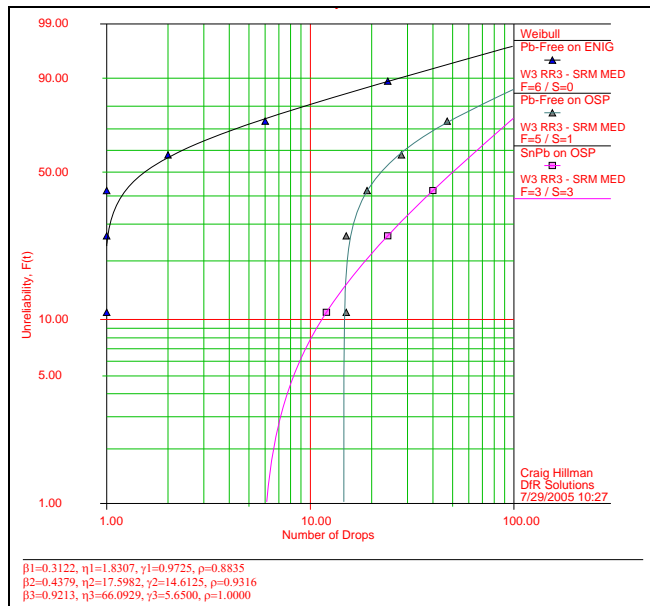


Figure 10. Results of drop testing for BGA components. The combination of SAC305 and ENIG surface finish resulted in very rapid fracture (some at only 1 drop).⁶

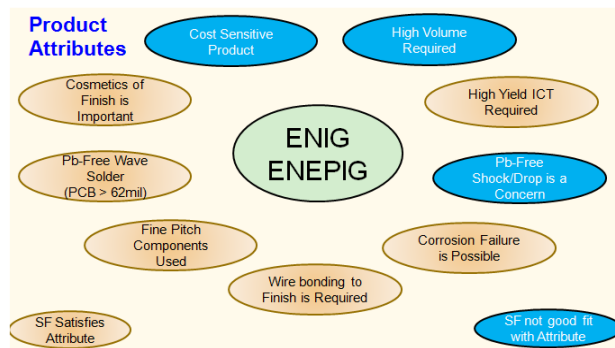


Figure 11. Product requirements and how they fit with ENIG and ENEPIG characteristics

PB-FREE HASL

Pb-free hot air solder level finish continues to gain momentum as a viable surface finish option. Although SnPb HASL was the most common surface finish prior to RoHS, Pb-free HASL wasn't given serious consideration when the transition to Pb-free occurred. There was concern with finish planarity, copper dissolution, and heat damage to the circuit boards. These issues have since been largely overcome when solder alloys such as SnCuNi, SnAgCuNi or SnCuCo have been employed. The solder temperature is only 10°C higher than what was used for SnPb HASL and since PCB laminates used for Pb-free products tend to have higher temperature

capabilities, the thermal excursions are not typically a significant concern.

The planarity of the Pb-free HASL coating is actually reported to be better than with SnPb and it is currently being used on products with component pitch as low as 0.5 mm.⁷ One might expect pitches finer than this to have screen printing challenges. The most significant advantage of HASL is excellent solderability which is useful when one needs to achieve complete hole-fill on a complex double sided PCB. Its corrosion resistance and testability are also excellent.

The biggest challenge with this finish is simply finding a PCB shop that offers it as an option. More suppliers are making this finish available however; they are purchasing vertical HASL machines that are not capable of running high volume. The other challenge is making sure that the coating is thick enough. The previous mindset was to blow as much solder off the board as possible to achieve a thin finish, however, with alloys such as SnCuNi, the solder blows off easily and one can get a coating that can actually be too thin. A coating < 2 µm can convert to intermetallic and result in poor solderability. The attribute chart for Pb-free HASL is shown in Figure 12.

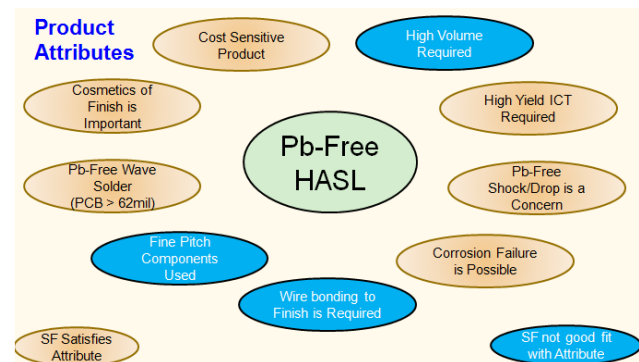


Figure 12. Product fit with Pb-free HASL.

Immersion Tin (ImSn)

ImSn surface finish has seen a growth rate similar to ImAg since 2003 (from 2% to 17%). The growth of ImSn has primarily been in Europe. It is a good finish for ICT, the cost is relatively low, and it is a planar finish so assembly of fine pitch components is possible. The risk of tin whiskers is often brought up as a concern, but generally the finish is too thin to grow long whiskers. ImSn requires a rather complex plating process with a bath consisting of stannous halide and thiourea. The bath can become rich in copper if not well controlled.

The product requirement chart is shown in Figure 13. Wave soldering after assembly is listed as a primary difficulty with this finish. When the finish is exposed to elevated temperature, the thin tin layer can become almost completely converted to SnCu intermetallic (leaving little tin for soldering). Solderability can become a problem after the first reflow cycle (or even after long term storage

of the PCBs). High volume availability can also be an issue (unless the PCBs are being produced in Europe). Many PCB shops in the Americas or in Asia do not have high volume capacity with this finish.

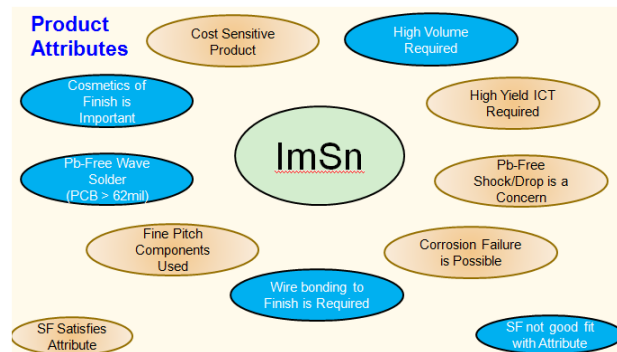


Figure 13. Product requirements and how they fit with immersion tin characteristics.

NEWLY DEVELOPED FINISHES

The lack of a satisfactory surface finish to fit all applications has led to the development of new finishes. Two leading candidates for new finishes that could gain market share in the future are Electroless palladium and a PTFE-type film that is plasma deposited onto the board.

Immersion Pd

Electroless Pd (or direct Pd) is being developed by OMGI and is being called Palla Guard 46. The target thickness is 10 microinches (similar to ImAg). A thin coating is desirable due to the high cost of Pd. The cost of this coating is approximately 25% of ENIG but almost 2x the cost of ImAg. The Pd dissolves into solder at a similar rate as silver, so upon soldering all the Pd is dissolved and Sn-Cu intermetallic bonding is achieved. The advantage of Pd is that it will not tarnish like Ag. However its electromotive force (EMF) value is +0.83V which is similar to Ag (+0.80V). Since galvanic interaction with Cu helped drive copper sulfide creep corrosion in the presence of ImAg, it is possible the same could occur with EPd (corrosion testing is needed). The Pd finish would have all the benefits of silver (flatness, good testability, solderability and reasonable cost), but would not tarnish. This could be an attractive finish for many applications and is summarized in Figure 14.

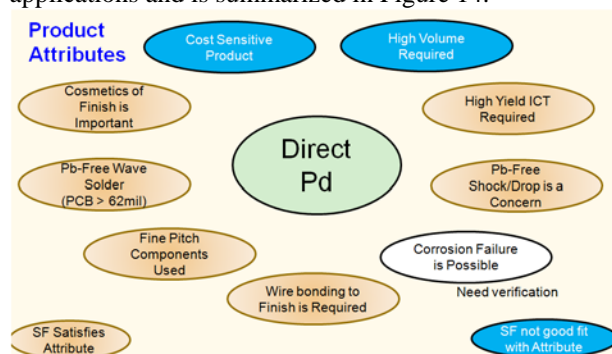


Figure 14. Product requirements and how they fit with electroless palladium characteristics.

Plasma Coated Finish

Semblant is a recent start-up company that is commercializing a unique new surface finish that is plasma deposited over the surface of the PCB.⁸ The film is a PTFE-type coating that is extremely thin (60 nm is the target thickness). This film is very hydrophobic so it repels water and other contaminants, thus presumably preventing corrosion. The coating withstands temperatures over 300°C but is designed to dissolve when in contact with flux at reflow temperatures. An example of the film is shown in Figure 15. Solderability testing has been very favorable and the corrosion protection of the film in mixed flowing gas has been remarkable. It can be deposited before or after the solder mask process. The thinness of the film enables low resistance contact during in-circuit testing. Wire bonding through it has also been demonstrated. This surface finish has the potential to be a good substitute for ENIG/ENEPIG with the potential cost being considerably less. The nature of this film enables it to replace the solder mask altogether for some applications. Note that this coating is also very promising as a conformal coating that could be applied after assembly of components.

How well this finish satisfies product attributes is summarized in Figure 16. The film is deposited rather quickly in a batch process (with many panels at once). The throughput is anticipated to be quite high but this cannot be proven until PCB fabricators get equipment installed. Getting an installed base will enable the cost to rapidly come down.

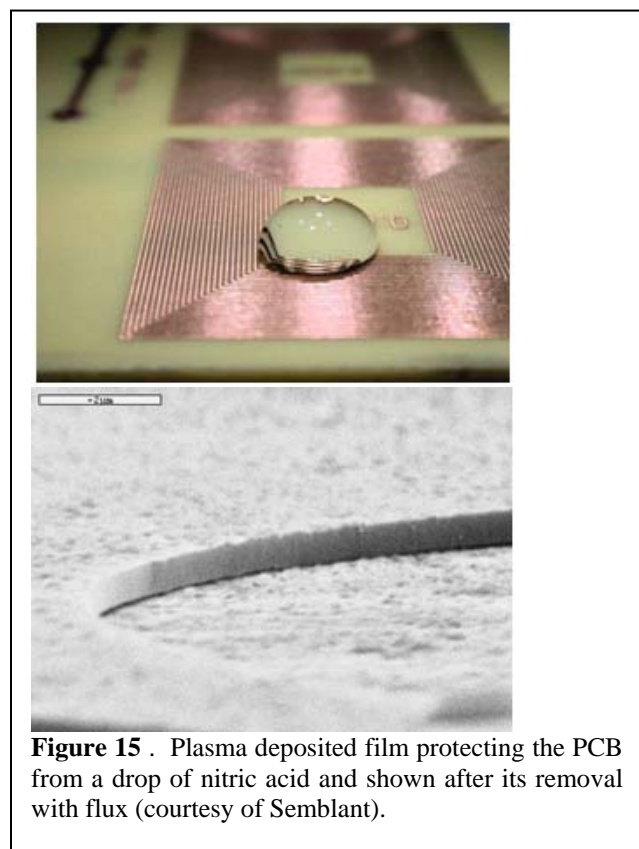


Figure 15 . Plasma deposited film protecting the PCB from a drop of nitric acid and shown after its removal with flux (courtesy of Semblant).

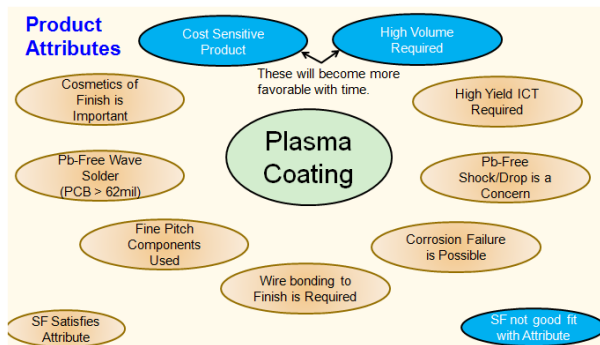


Figure 16. Product requirements and how they fit with the characteristics of the plasma coated finish.

BEST FIT APPLICATIONS

Each surface finish tends to have industrial applications where it fits best. For example OSP tends to work well for hand held electronics, notebook computers, basic desktop computers, and consumer electronics. ENIG/ENEPIG works well for medical equipment and aerospace applications (with SnPb solder) or small specialty electronics. ImAg is a good fit for hand-held electronics or consumer electronics that use enclosures that restrict airflow. Immersion tin works well for consumer electronics that might not be fully enclosed or simple electronics with components on a single side. Lead-free HASL is being used on thicker lead-free boards where wave soldering is required. These include work stations and servers. The newer finishes are currently being evaluated to determine the applications where there is best fit.

ADJUSTMENTS

As luck would have it, there are often situations where the product requirements do not fit well with the characteristics of the available surface finishes. In those situations there are other industry tricks that can be used to compensate for the shortcomings of a surface finish.

For example, the application may require a finish that is low cost, high volume, corrosion resistant with good ICT properties. One solution might be to use ImAg but design the board to mitigate the risk of corrosion by filling the vias with solder mask and moving any exposed silver regions far apart. Another possible solution might be to use OSP with water soluble flux and perform cleaning prior to ICT (however both these solutions do add some cost).

Another common example is a situation where the product characteristics require ENIG or ENEPIG with use of SAC305 solder and there is the potential for drop/shock. One solution might be to reinforce the sensitive components using underfill or edge bonding. Another solution may be to dampen the shock force with better design of the product enclosure. Use of the newer

finishes (Pd or plasma deposited film) may also satisfy the requirement.

A third example might be a situation where OSP finish is desired but wave soldering of a thick board is required. Solderability can be improved (to achieve hole-fill) by using more aggressive flux or using nitrogen during assembly to reduce oxidation and improve wetting (these options will add cost).

SUMMARY

The surface finish impacts the PCB cost, the manufacturing process, the quality, and the reliability of the product. There is currently no surface finish that satisfies the needs of all products so sacrifices are often being made with whatever finish is selected. Selecting the optimum finish for a product application can be best done by determining what problems are most important to solve and making sure they are satisfied. This often involves gathering input from multiple functional groups within the organization to make sure everyone's point of view is taken into account. The object should be to select a finish that minimizes the overall cost (after considering the component cost, the assembly cost and the warranty cost).

REFERENCES

- ¹ J. Beers, Gold Circuits, 2008.
- ² Van der Pas, "Selecting the Right Final Finish for RoHS PCBs", Printed Circuit Design and FAB, March, 2007.
- ³ M. Mukadam, "Planar Microvoiding in Second Leel Interconnect Joints", SMTA Conf Proceedings, 2006.
- ⁴ R. Schueller, "Creep Corrosion of LF PCBs in High Sulfur Environments", SMTA Int'l Proceedings, 2007.
- ⁵ Chai TC, Sharon Quek, Hnin WY, Wong EH, Julian Chia, Wang YY, Tan YM, Lim CT, Board Level Drop Test Reliability of IC Packages, Institute of Microelectronics
- ⁶ C. Hillman, DfR Solutions.
- ⁷ R. Schueller, W. Ables, J. Fitch, "Case Study for the Transition of Server Motherboards to LF", SMTA Int'l Proceedings, 2008.
- ⁸ F. Ferdinandi and T. VonWerne, "Introduction of a New PCB Surface Finish for the Electronics Industry", SMTA Int'l Proceedings, 2009.