

Beyond Design - Surface Finishes for High-speed PCB's

by Barry Olney | In-Circuit Design Pty Ltd | Australia

The nickel doesn't make cents!



PCB surface finishes vary in type, price, availability, shelf-life, assembly process and reliability. While each treatment has its own merits, Electroless Nickel/Immersion Gold (ENIG) finish has traditionally been the best fine pitch (flat) surface and lead-free option for SMT boards over recent years. But unfortunately, nickel is a poor conductor having only one-third the conductivity of copper. Also, nickel has a ferromagnetic property that can adversely affect electromagnetic fields in the high frequency domain.

The PCB industry has addressed the issue, of the ferromagnetic properties of nickel, by introducing a nickel/gold (NiAu) alloy. Gold is slightly less conductive than copper, and has no ferromagnetic properties, so it has relatively little impact on the conductor's loss characteristics at high frequencies.

Microstrip (outer) layers of a multilayer PCB, suffer from wide variations in both trace width and thickness. This is due to the additional fabrication process of electroplating the through-holes. Copper barrel thickness is generally specified as a minimum of 1 mil (25.4 μm) and so extra copper plating is applied to the surface in order to produce the correct barrel wall thickness. This unfortunately, is also added to the traces. But, as the thickness and width varies – so does the impedance. This is one of the reasons why routing controlled impedance signals, on the microstrip layers, should be avoided.

It is also very important not to pour copper fills on the signal layers, of the board, as these will dramatically change the impedance of the traces rendering the impedance control ineffective.

A surface finish can be defined as a coating, either metallic or organic in nature, that is applied to a PCB in order to assure solderability of the metal underneath after a certain time in storage and under different environmental conditions. The surface finish protects the copper mounting pads from corrosion in order to ensure good soldering. There are several choices of protective coatings:

- Hot Air Solder Leveling (HASL) – Lead free
- Organic Solderability Preservative (OSP) such as Entec 106 or Shikoku
- Electroless Nickel-Immersion Gold (ENIG)
- Electroplated gold over nickel (NiAu)
- Immersion silver (IAG)
- Immersion tin (ISn)

Electroplated NiAu can be applied as a full body finish, prior to solder mask application. This is because for that finish, the NiAu layer is typically used as a resist to etch the outer layers. Therefore, nickel ends up covering the entire length of the outer layer traces. But as mentioned, nickel is a ferromagnetic material and is not desirable from a loss perspective.

Alternatively, ENIG can be applied as a full body finish, or the more common (and recommended) way as a selective finish, because it is applied after solder mask. In the second case, nickel is not plated on the traces. However, the lands are protected by the plating which are exposed to the environment prior to assembly. This is described as Solder Mask Over Bare Copper (SMOBC) processing and is best for high-speed design. Plus, given the cost of gold, Liquid Photoimageable Solder Mask (LPISM) should be applied before the ENIG process in order to limit the plating area.

In a microstrip configuration (Figure 1), electric fields (blue) exist between the traces and the reference plane. At high frequencies (> 1 GHz), the current density (red) tends to build up on the edges and surface of the plating due to the skin effect. Therefore, special consideration should be incorporated for any edge-coupled structures on the outer layers. This impact is more prevalent when ENIG is applied over all copper features, such as application of ENIG before solder mask. But, this effect can be minimized when only the pads (not the traces) are plated with the finish.

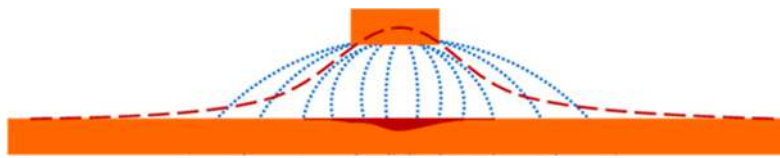


Figure 1 – Microstrip electric fields and return current distribution

ENIG plating thickness is specified by the IPC-4552 standard for ENIG plating for printed circuit boards. The gold plating is typically very thin, $0.075\text{--}0.125\ \mu\text{m}$. On the other hand nickel plating, which is used to stop copper migration to the gold, normally requires $3\text{--}6\ \mu\text{m}$ thickness. These thicknesses can vary between fab shops and processes. Higher gold thickness requires extended solution dwell time or increased solution temperature.

Plating finishes can impact the overall loss of the transmission line due to lower conductivity metallization that covers the surface of the trace. Silver plating is an exception, to this increased loss effect, due to its similar conductivity to copper. This finish however, is not as preferred as other finishes like ENIG or immersion tin.

As electronic devices have miniaturized, so to have their chip package sizes, along with the size and clearance of lands. As a result, short-circuit risk has increased. The plating of fine patterns ($15\ \mu\text{m}$ spacing) clearly favors Palladium/gold (Pd/Au) or EPIG. This is also a solution to the lossy transmission lines, since this finish contains no nickel. EPIG is also ideal for wire bonding applications.

Generally, most of the surface treatment dissolves into the solder paste or wave solder during the soldering process and the solder joint is formed between the solder and the copper. One exception is ENIG, where the solder dissolves the thin layer of gold and forms a joint with the underlying nickel alloy.

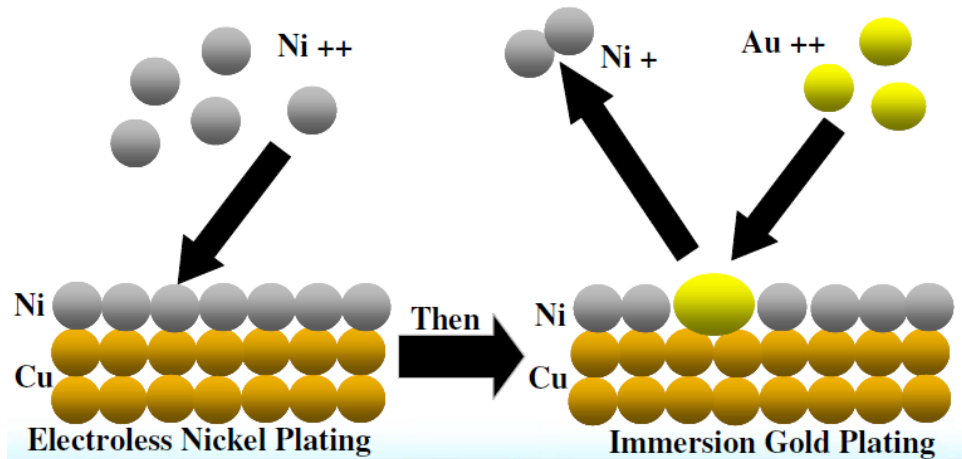


Figure 2 – Immersion systems process – courtesy Multitek

The immersion systems process (figure 2) uses a chemical displacement reaction to deposit a metal layer onto the exposed copper surface of the PCB. The base metal (copper) donates the electrons that reduce the positively charged metal ions present in solution. The immersion layer will continue to grow however, as the thickness of deposit increases, the rate of deposition falls. Therefore, the process is self-limiting.

There are four design concerns associated with ENIG plating:

1. Solder mask defined BGA pads should be avoided, due to the risk of:
 - a) Brittle joints (the inter-metallic compound that is formed when soldering against nickel).
 - b) The risk of black pads (lack of balance within the ENIG plating chemistry) especially on smaller BGA pads.
2. Single sided plugged via holes:

As with most finishes, via holes plugged from one side – partially plugged – are not recommended. Also, holes very close to SMD pads are not recommended, since the plating solution can become trapped inside and may contaminate or reduce the solderability of the joint.
3. Solder mask bridges between SMD pads:

As with immersion tin, this treatment is aggressive towards the solder mask. Therefore, larger solder mask bridges may be necessary, on fine pitch SMT components, at some fab shops.
4. The impact on the conductor's loss characteristics at high frequencies:

Selection of low loss plating is just as critical as the selection of dielectric material when designing high frequency circuits.

At approximately 2.7 GHz, the resonant behavior of the nickel component in ENIG increases insertion loss. This resonance is attributed to the ferromagnetic properties of the nickel layer. It is therefore wise to avoid using full body ENIG coating of microstrip traces at high frequencies. In fact, it may just be an odd 3rd or 5th harmonic that falls on this particular lossy region and causes radiation with much lower fundamental frequencies. Therefore, solder mask over bare copper (SMOBC) processing should be considered for all high-speed designs.

Points to Remember:

- Electroless Nickel/Immersion Gold (ENIG) finish has traditionally been the best fine pitch (flat) surface and lead-free option over recent years.
- Nickel has a ferromagnetic property that can adversely affect electromagnetic fields in the high frequency domain.
- The issue of the ferromagnetic properties has been addressed by introducing a nickel/gold (NiAu) alloy.
- Microstrip (outer) layers of a multilayer PCB suffer from wide variations in both trace width and thickness – hence impedance variations.
- It is very important not to pour copper fills on the signal layers of the board.
- ENIG can be applied as a full body finish, or the more common (and recommended) way as a selective finish, because it is applied after solder mask.
- The plating of fine patterns (15 μm spacing) clearly favors Palladium/gold (Pd/Au) or EPIG.
- Solder mask over bare copper (SMOBC) processing is best for high-speed design.
- There are four design concerns associated with ENIG plating: solder mask defined BGA pads, single sided plugged vias, Solder mask bridges and the impact on the conductor's loss characteristics at high frequencies.
- Selection of low loss plating is just as critical as the selection of dielectric material when designing high frequency circuits.
- The resonant behavior of the nickel component in ENIG increases insertion loss at 2.7 GHz. Avoid using full body ENIG coating of microstrip traces at high frequencies – SMOBC processing should be considered for all high-speed designs.

References:

[Beyond Design: Ground Pours](#) – Barry Olney

[Beyond Design: Electromagnetic Fields, Part 1](#) – Barry Olney

[Beyond Design: Electromagnetic Fields, Part 2](#) – Barry Olney

[Beyond Design: Mythbusting - There are no One-way Trips!](#) – Barry Olney

IPC-4552 – Specification for ENIG Plating for Printed Circuit Boards

Electromagnetic Compatibility Engineering – Henry Ott

High-speed Signal Propagation – Howard Johnson

Understanding PCBs for High-Frequency Applications – John Coonrod, Rogers Corp

Simbeor Application Notes – Y. Shlepnev, Simberian Electromagnetic Solutions

Increased Circuit Loss due to Ni/Au – Dr. Al Horn, Rogers Corp

The ICD Stackup Planner and PDN Planner can be downloaded from www.icd.com.au

All trademarks are the property of their respective owners

Bio - Barry Olney is CEO of In-Circuit Design Pty Ltd (ICD), Australia. The company developed the ICD Stackup Planner and ICD PDN Planner software, is a PCB Design Service Bureau and specializes in high-speed, board level simulation.