BOARD LEVEL SIMULATION SPECIALISTS

ICD Stackup Planner - offers engineers/PCB designers unprecedented simulation speed, ease of use and accuracy at an affordable price

- 2D (BEM) field solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Unique field solver computation of multiple differential technologies per stackup
- Heads-up impedance plots of signal and dielectric layers
- User defined dielectric materials library - over 16,250 materials up to 100GHz

ICD PDN Planner - analyze multiple power supplies to maintain low impedance over entire frequency range dramatically improving product performance

- Fast AC impedance analysis with plane resonance
- Definition of plane size/shape, dielectric constant & plane separation for each on-board power supply
- Extraction of plane data from the integrated Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- Frequency range up to 100GHz
- Extensive Capacitor Library – over 5,250 capacitors derived from SPICE models

LEARN MORE
www.icd.com.au
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 with 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, which 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:

- Lead-free hot air solder leveling (HASL)
- 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 affect. 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.

ENIG plating thickness is specified by the IPC-4552 standard for ENIG plating for printed circuit boards. The gold plating is typically very thin, 0.075–0.125 µm. On the other hand, nickel plating, which is used to stop copper migration to the gold, normally requires 3–6 µm 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 too 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 µm 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.

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).

Figure 1: Microstrip electric fields and return current distribution.
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, placing holes very close to SMD pads is 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
1. Barry Olney: Beyond Design: Ground Pours, Beyond Design: Electromagnetic Fields, Part 1, Beyond Design: Electromagnetic Fields, Beyond Design: Mythbusting - There are no One-way Trips!
2. IPC-4552 – Specification for ENIG Plating for Printed Circuit Boards
3. Henry Ott: Electromagnetic Compatibility Engineering
4. Howard Johnson: High-Speed Signal Propagation
5. John Coonrod, Rogers Corporation: “Understanding PCBs for High-Frequency Applications,” Printed Circuit Design & Fabrication
6. Yuriy Shlepnev, Simberian Electromagnetic Solutions, Simbeor Application Notes
7. Dr. Al Horn, Rogers Corporation, “Increased Circuit Loss due to Ni/Au”

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

Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD), Australia. This PCB design service bureau specializes in board-level simulation, and has developed the ICD Stackup Planner and ICD PDN Planner software. To read past columns, or to contact Olney, click here.

2D Transistor Paves Way to Faster Electronics

2D Transistor Paves Way to Faster Electronics

Faster electronic device architectures are in the offing with the unveiling of the world’s first fully two-dimensional field-effect transistor (FET) by researchers with Lawrence Berkeley National Laboratory (Berkeley Lab). Unlike conventional FETs made from silicon, these 2D FETs suffer no performance drop-off under high voltages and provide high electron mobility, even when scaled to a monolayer in thickness.

Ali Javey, a faculty scientist in Berkeley Lab’s Materials Sciences Division and a UC Berkeley professor of electrical engineering and computer science, led this research in which 2D heterostructures were fabricated from layers of a transition metal dichalcogenide, hexagonal boron nitride and graphene stacked via van der Waals interactions.

“In constructing our 2D FETs so that each component is made from layered materials with van der Waals interfaces, we provide a unique device structure in which the thickness of each component is well-defined without any surface roughness, not even at the atomic level,” Javey says.

For the 2D FETs produced in this study, mechanical exfoliation was used to create the layered components. In the future, Javey and his team will look into growing these heterogeneous layers directly on a substrate. They will also look to scale down the thickness of individual components to a monolayer.