As originally published in the IPC APEX EXPO Conference Proceedings.

Selecting Stencil Technologies to Optimize Print Performance

Chrys Shea
Shea Engineering Services
Burlington, NJ USA

Abstract
The SMT stencil is a key factor in the solder paste printing process. It has been shown repeatedly that print quality has the largest impact on end-of-line quality, and a good print process can make or break the profitability of building a PCB assembly. A good print process relies on a good stencil.

Much research has been performed to identify individual key factors in stencil performance; this paper and presentation discuss the real-world application of numerous findings. They review the numerous considerations in design, material, manufacturing and coating considerations, and how to best choose them based on PCB layout.

Introduction

The stencil design optimization process begins with a review of the PCB layout. Component type and location, population density and PTH presence all factor into selecting the appropriate technologies to produce the best possible results in the solder paste printing process. The first choice is stencil thickness, and the determination if multiple thicknesses are required. Some devices, such as uBGAs, BTCs, or through-hole components will also require attention to aperture design. The stencil thickness(es), aperture size and aperture density will determine the best material, and the material drives the manufacturing process.

The manufacturing process holds many keys to success, as good dimensional accuracy and cut quality are critical to producing repeatable, quality solder paste deposits. Performance-enhancing coatings, which were once considered only for challenging processes, have demonstrated improvements in nearly every print process to which they are introduced.

Reviewing designs prior to production and selecting stencil technologies based on PCB layout will improve yields, throughput and reliability. Understanding and considering the influence of PCB layout on stencil design during layout will not only improve assembly performance, it will help lower overall product cost before designs they are locked in. Figure 1 illustrates the process.

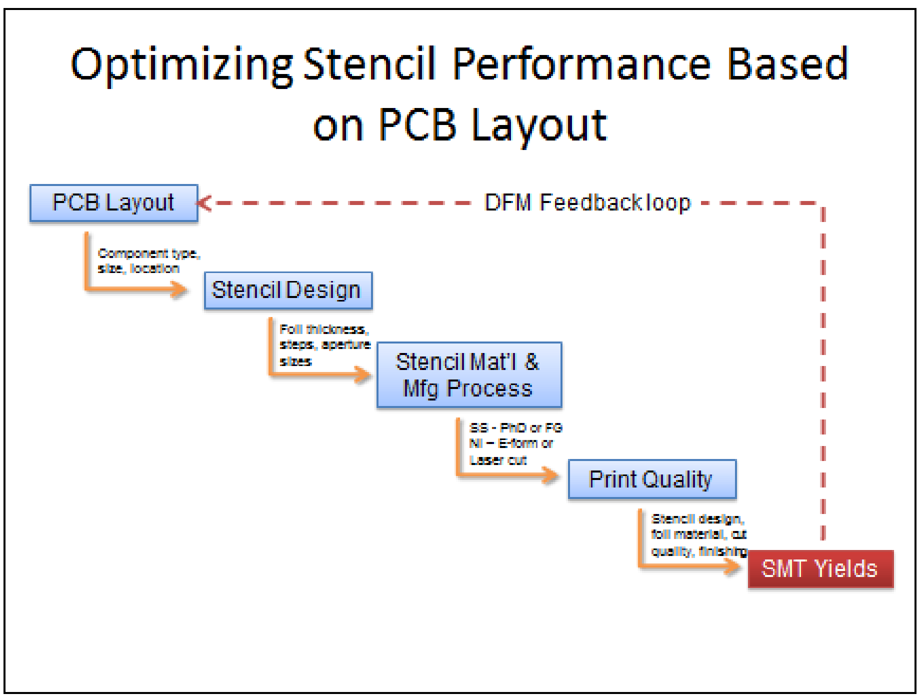


Figure 1. Overview of stencil optimization process[1]

Design Review

PCB layout drives the primary considerations of stencil design: foil thickness and aperture sizes. Smaller components with finer pitch I/Os require thinner foils in the 3-5mil range; larger components, through-hole connectors or devices prone to warpage or coplanarity problems require thicker stencil foils in the 6-8mil (or more) range. When both appear in the same layout, they can be accommodated by a number of options.

A thorough design review will identify conflicting requirements that need special attention to stencil design, and also help determine the best material, manufacturing process and coating. It starts with an automated review of a PCBs layout by a design checker software system. These systems read the Gerber file and calculate all the area ratios, flagging those that fall below a specific threshold, typically 0.66. Many stencil suppliers have this capability.

Stepped Stencils

Stepping stencils, or locally varying their thicknesses, can provide the optimum solution to conflicting thickness requirements. Stencils can be stepped a number of ways:

- **Step Up:** Thickens stencil locally
- **Step Down:** Thins stencil locally
- **Top or Bottom side steps,** or both
- **"Stepless" steps:** Smooth the transition (used with enclosed print heads)
- **Angled steps:** Reduce squeegee damage (also used with enclosed print heads)
- **Cavity relief:** on the PCB side of the stencil to accommodate labels or other topographical features

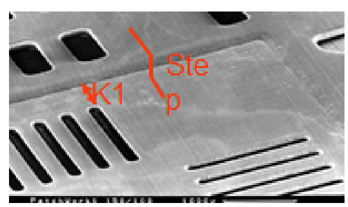
Steps can be created by:

- **Chemical etching**
- **Milling**
- **Welding**

General guidelines for stencil stepping vary by resource. The "official" IPC-7525 specification offers a method of calculating the required keepout zone based on step depth to optimize print quality, as shown in Figure 2.

IPC 7525 stencil guidelines

As a general design guide **K1** should be **0.9mm** [35.4mil] for **every 0.025mm** [0.98mil] of step- down thickness.



Step Depth	K1 is distance form the step edge to the nearest aperture in stepped down area
0.010mm, 0.4mil	0.36mm, 14mil
0.025mm, 1mil	0.90mm, 35 mil
0.030mm, 1.2mil	1.08mm, 42mil
0.050mm, 2mil	1.80mm, 71mil
0.080mm, 3 mil	2.88mm, 113mil
0.100mm, 4mil	3.60mm, 142mil

Courtesy of LaserJob GmbH

Figure 2. IPC-7525 Stencil Step Down Design Guidelines[2]

Very often, the PCB layout itself precludes the use of optimally designed keepout zones; therefore, alternate guidance has been developed, as shown in Figure 3.

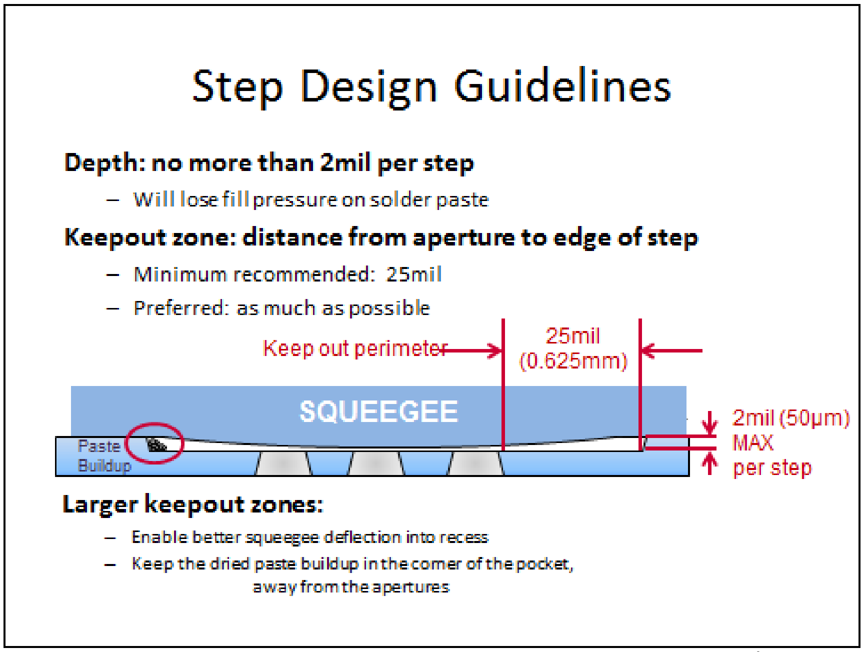


Figure 3. Alternate Stencil Step Down Design Guidelines[4]

Alternate design guidelines for steps include a maximum step height or depth of 2mil (50µm) per step to maintain good fill pressure, and a minimum keepout perimeter of 25mil (625 µm) around the apertures. The farther away from the apertures the step can be located, the better. It will allow for better squeegee blade deflection into the step, and keep the paste that always builds up and dries out near the step wall farther away from the apertures. Components that do not necessarily require steps but can accept them are often included in the stepped area to maintain the keepout zone. Other layout options include clustering components that require steps to create fewer, larger stepped areas instead of many smaller ones.

If the desired step depth is only 1mil (25µm), then an incrementally-sized foil may provide the ideal solution. Electroformed nickel foils are available in half-mil (12.5µm) increments: 3.5, 4.5, 5.5 or 6.5mils thick, because they are “grown” in plating tanks. Nickel foils not only offer these sub-1mil incremental thicknesses; they also offer high durability for processes that must run excessive print pressures.

QFNs or other bottom termination components are often the driver behind stepped stencils, because many of them require small apertures on 0.5mm pitch. These components are becoming increasingly popular because they are economical and reliable.[3] In addition to sometimes requiring stepped down stencil areas, they also need special attention to the center pad, often used for heat sinking or grounding. If insufficient paste is applied to the pad, its efficiency is reduced. If too much solder paste is applied, the component can tilt or float, creating opens or unreliable solder joints. If thermal vias are in the pad, they can rob paste from the bond and cause voiding. Additionally, the flux in the solder paste will also cause voiding on this pad. Therefore, it is important for the stencil designer to divide the center pad apertures to ensure proper standoff (2-3mils preferred), maintain outgassing paths to limit voiding, and avoid printing over thermal vias. Some examples of BTC aperture design are shown in Figure 4.

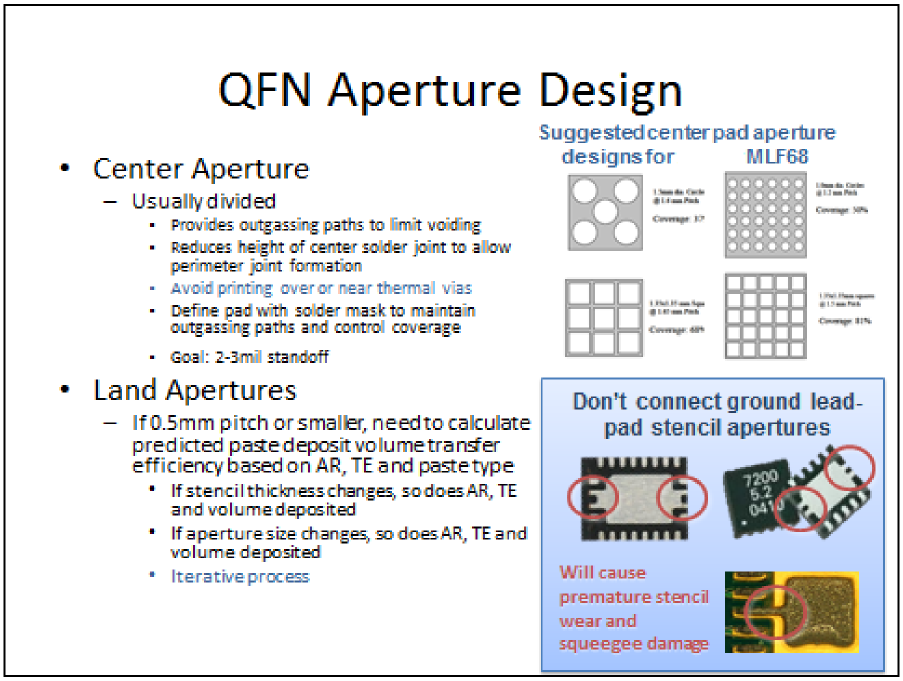


Figure 4. BTC aperture design considerations[4]

SMT Stencil Foil Material Selection

Stainless steel (SS) is the material of choice, except when special circumstances dictate nickel, and emerging SS alloys may soon rival nickel in hardness and durability.[5] Standard SS is the least expensive option, but can be prone to thickness variations, inclusions or other flaws in the material, and warping or bowing in reaction to the heat generated by laser cutting. Premium SS manufactured specifically for SMT stencils is typically precision-rolled to maintain very tight thickness tolerances and stress relieved to prevent distortion from the heat of cutting. For higher performance, Fine Grain (FG) SS is also precision rolled and stress relieved, but it reduces the typical grain size by an order of magnitude (figure 4). The finer grains produce smoother stencil walls and crisper steps. Many high precision stencil printing processes depend upon it.

In four independent tests over four consecutive years, Fine Grain SS outperformed every other stencil material it was tested against: standard SS, electropolished standard SS, premium SS, electroformed nickel, laser cut nickel, and nickel-plated stainless. Figure 5 summarizes the results of the studies.[6-9]

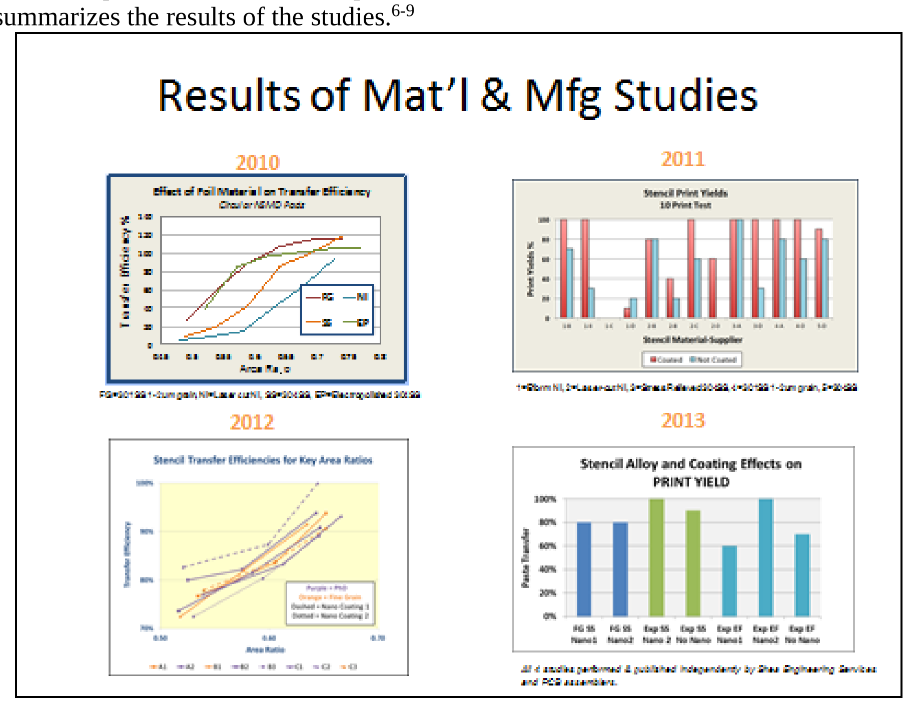


Figure 5. Results of print tests comparing foil materials

Manufacturing Process

Well-tuned, modern laser cutters produce high accuracy stencils. If nickel foil material is required, laser cutting the apertures into a formed nickel "blank" will likely produce a more accurate stencil than most electroforming processes. It will also save lead time and cost. Regardless of the foil material, the overall performance of a stencil is heavily dependent on the quality of the aperture wall, and many studies have correlated wall roughness to print performance. The smoother the wall, the better the print performance.

Wall quality is heavily dependent on laser cutting parameters and machine calibration. Users should inquire with their suppliers about equipment age and history, and ask if they have performed any cut optimization studies.

Nickel Plating

Secondary processes like nickel plating over SS or electropolishing the SS are sometimes used in conjunction with laser cutting. Plating nickel over SS is supposed to add the durability of nickel to the precision of SS to combine the best qualities of both. In recent tests, it did not fare as well as laser-cut premium SS in print performance; the nickel plating lowered area ratios both by increasing the foil thickness by and reducing the aperture sizes.[8] Differences as large as 0.4mils in aperture size and foil thickness were noted.

Electropolishing

Electropolishing was very popular in the early days of laser cutting SS because it removed the scalloped peaks in the walls produced by the wider laser beams of the original cutting equipment. In contrast to the nickel plating process that adds material to the original stencil, electropolishing removes small amounts of it. In addition to smoothing the walls, the electropolishing process actually opens up the apertures and thins the stencil a bit, giving it a slight area ratio advantage, helping it demonstrate better transfer efficiency than non-electropolished stencils. Unfortunately, it has historically tended to round the corners of the apertures to compromise gasketing and induce more print volume variation[6] (figure 6).

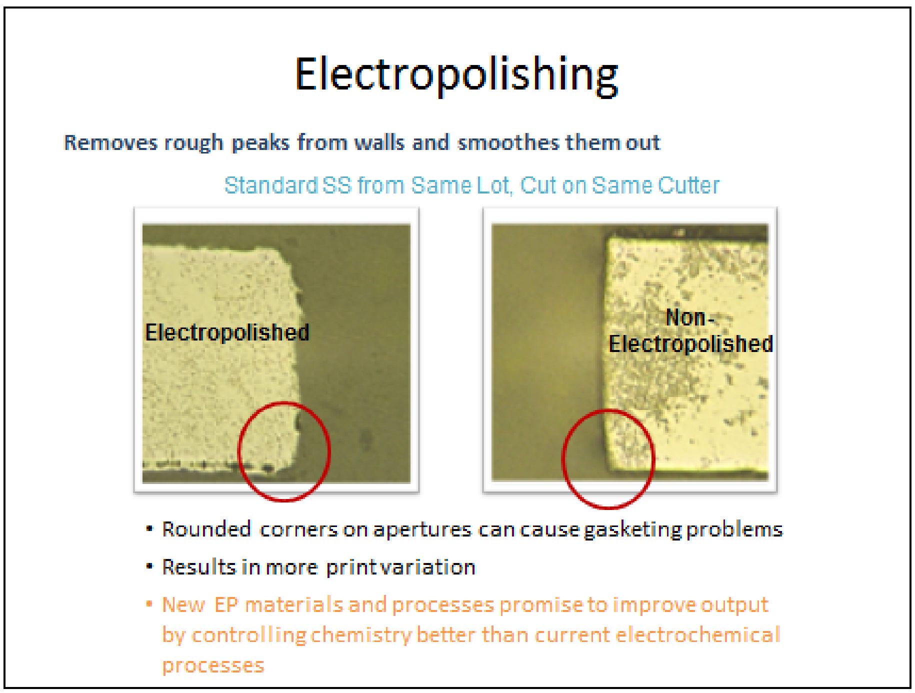


Figure 6. Cross sections of electropolished stainless steel apertures

Traditional electropolishing is still sometimes used, although it is often considered unnecessary on stencils cut with newer, well run equipment. Because modern lasers can cut cleaner, smoother walls and more consistent, laser-friendly materials are used, electropolishing is not as effective as it once was (or needed to be). New electropolishing processes and chemistries are under development that can remove the smaller peaks of the laser cuts without compromising corner quality.

Nanocoating

Nanocoating can substantially boost productivity. This special repellency treatment is applied to the finished stencil, and prevents the flux from spreading on the bottom surface, keeping it cleaner for longer.[10] The cleaner stencil bottom:

- Produces crisper prints
- Extends under wipe intervals
- Cleans more easily
- Is more forgiving when gasketing is bad
- Saves money on wiper paper, and (sometimes) solvent and cycle time

Figure 7 shows the stencil apertures for QFNs after 10 prints with and without a second-generation SAMP-based nanocoating. Figure 8 shows the resultant prints. The difference in print definition is visible in the deposits for the thermal pad and the wet-bridged 0201s.

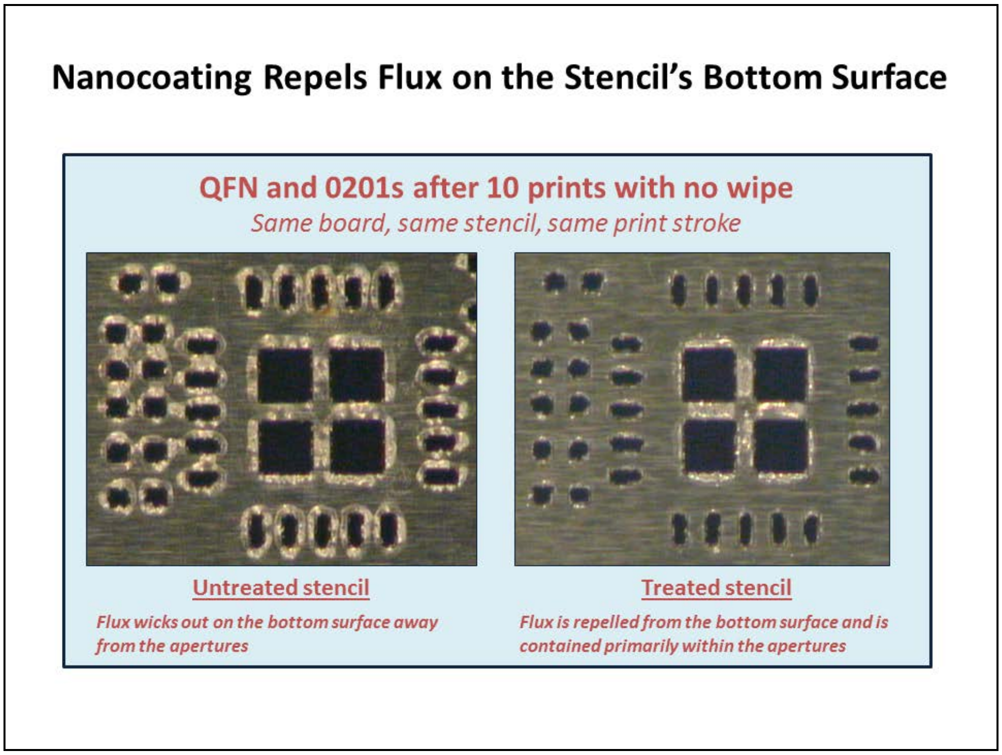


Figure 7. Flux spread on stencil bottom after 10 prints

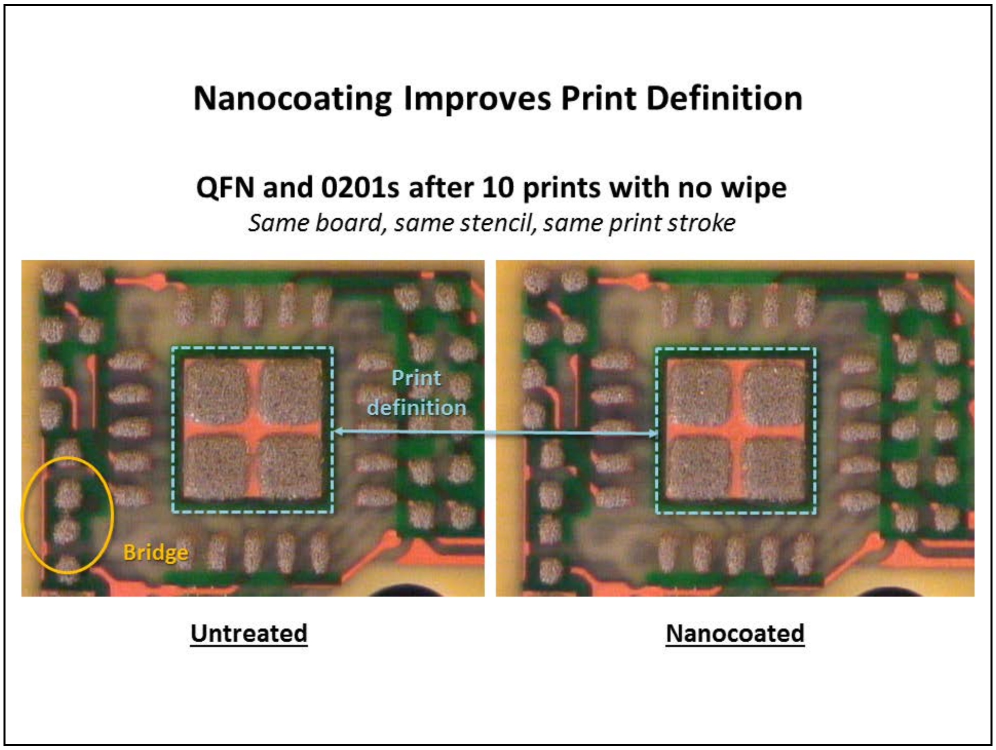


Figure 8. Print definition with and without nanocoating

When nanocoatings were first introduced, their utilization was focused on improving fine feature printing processes. Continued research on their application has revealed that they will enhance just about any print process, regardless of PCB layout. Over the past year, their cost has been reduced and their availability improved, and as users continue to document the increases in quality and productivity, their popularity will continue to grow.

Figure 9 compares some of the different nanocoatings available on the market at the time this document was created.

TYPE	SOL-GEL POLYMER	SAMP – Gen1	SAMP – Gen2	POLYMER
Year Launched	2009	2011	2013	TBD
Application method	Vacuum	Wipe	Wipe	Spray
Cure Required?	Yes	No	No	Yes
Application/cure cycle time	2 hrs	10 min	10 min	45 min
Commercially Available?	Yes	Yes	Yes	No
Thickness	up to 2000 nm	3-5 nm	3-5 nm	2000-4000 nm
Truly a Nanocoating? (<100nm)	No	Yes	Yes	No
Thickness Variation	?	+/- 1 nm	+/- 1 nm	+/- 2000 nm
Aperture redesign required?	Sometimes	No	No	Yes
Stencils treated	One mfr only	Any metal	Any metal	One mfr only
Applied by	One mfr only	Any mfr or user	Any mfr or user	One mfr only
Proven in Production	Yes	Yes	Yes	No
Cost	$650 incl stencil	varies	$25	TBD

Figure 9. Comparison of SMT stencil nanocoatings available as of November 2014[11]

Conclusion

Optimizing stencil performance based on a PCB layout is a straightforward process, but requires a number of decisions based on the features of the layout. The first, and most critical, choice is on foil thickness. Typical SMT processes use 5mil (125um) foils, but some components require larger deposits that drive thicker foils and some require smaller, more precise deposits that require thinner foils. The stencil designer needs to make sure the foil thickness and aperture sizes do not violate area ratio rules, and stencil design analysis software speeds the calculation process while preventing errors.

Sometimes foils must be stepped to accommodate multiple thicknesses. Stepping guidelines are available to help insure the best possible print quality; if the guidelines are compromised, the print quality is likely to suffer. Steps of 1mil or less may be addressed by using an incremental-size nickel foil; steps or 2mil or greater should use Fine Grain SS. Other considerations for using Fine Grain SS include foil thicknesses of 5mil (125um) or less, devices with pitches of 20mil (0.5mm) or less, high density or highly miniaturized layouts, or area ratios less than 0.66.

Secondary processes like nickel plating over SS or electropolishing have not recently been shown to improve overall print performance and typically should not be a factor in stencil design or manufacturing decisions. Rather, the cut quality that a supplier is capable of providing should be a larger consideration. The smoother walls created by the combination of specialized SS and modern laser cutters have shown to produce the best print quality in successive tests, consistently outperforming every other stencil fabrication technology available. The finer the feature, the more important cut quality becomes – 1206s are more forgiving than 0201s; as are QFPs compared to QFNs. If secondary processes are employed, users should understand why they are required and what specific impact they have on the process.

Nanocoatings improve quality and cost by keeping the PCB side of the stencil clean, reducing underwipe frequency and improving print definition. They can positively impact any solder paste printing process, regardless of PCB layout.

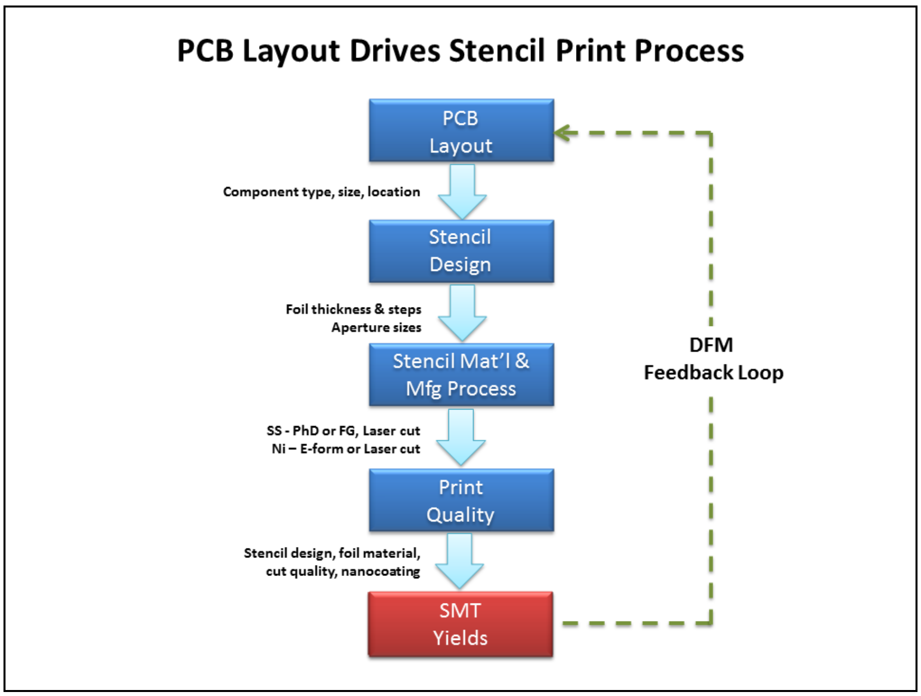


Figure 10. Influence of PCB layout on SMT printing process

Figure 10 depicts the decision path and its factors, and indicates a feedback loop for Design for Manufacturability inputs. Understanding how the PCB layout affects the entire print process – from stencil design to production yields – enables development teams to incorporate cost-conscious manufacturing decisions in early stages of product design, where they have the greatest impact.

References

1. "Optimizing SMT Stencil Design Based on PCB Layout," B. Scott, C. Shea and C. Wood, *Global SMT and Packaging*, May 2014
2. Images courtesy of LaserJob
3. "BTC: Bottom Termination Component or Biggest Technical Challenge?" C. Tulkoff, & G. Caswell, *Proceedings of SMTA International*, October, 2014
4. "Enabling Technologies for Broadband Printing," C. Shea, Proceedings of IPC Conference on Soldering and Reliability, November, 2013
5. "The Effects of Stencil Alloy and Cut Quality on Solder Paste Print Performance," C. Shea and R. Whittier, *Proceedings of SMTA International*, October, 2014
6. "Quantitative Evaluation of New SMT Stencil Materials," C. Shea and Q. Chu, et al, *Proceedings of IPC APEX/EXPO*, March, 2011

7. “Evaluation of Stencil Foil Materials, Suppliers and Coatings,” C. Shea, and R. Whittier, *Proceedings of SMTA International*, October 2011
8. “Stencil and solder paste inspection Evaluation for Miniaturized SMT Components,” R. Farrell and C. Shea, *Proceedings of SMTA International*, September, 2013
9. “Fine Tuning the Stencil, Manufacturing Process and Other Stencil Printing Experiments,” C. Shea, and R. Whittier, *Proceedings of SMTA International*, September, 2013
10. “Development, Testing and Implementation of SAMP-Based Stencil Nano Coatings,” C. Shea, R. Whittier, and E. Hanson, *Proceedings of IPC APEX/EXPO*, March, 2014
11. www.comparenanocoatings.com