



Optimizing SMT stencil design based on PCB layout

Ben Scott, Datum Alloys; Chrys Shea, Shea Engineering Services; and Carol Wood, Alpha Stencils

With so many different stencil technologies out there, how is a stencil designer to know which will work best on any given product? Just like almost everything else related to SMT assembly, it all depends on the board layout. Component type and location, population density, and PTH presence all factor into selecting the best materials, manufacturing processes, and performance-enhancing coatings to get the best possible results in the solder paste printing process.

ARTE – Area Ratio and Transfer Efficiency Calculator

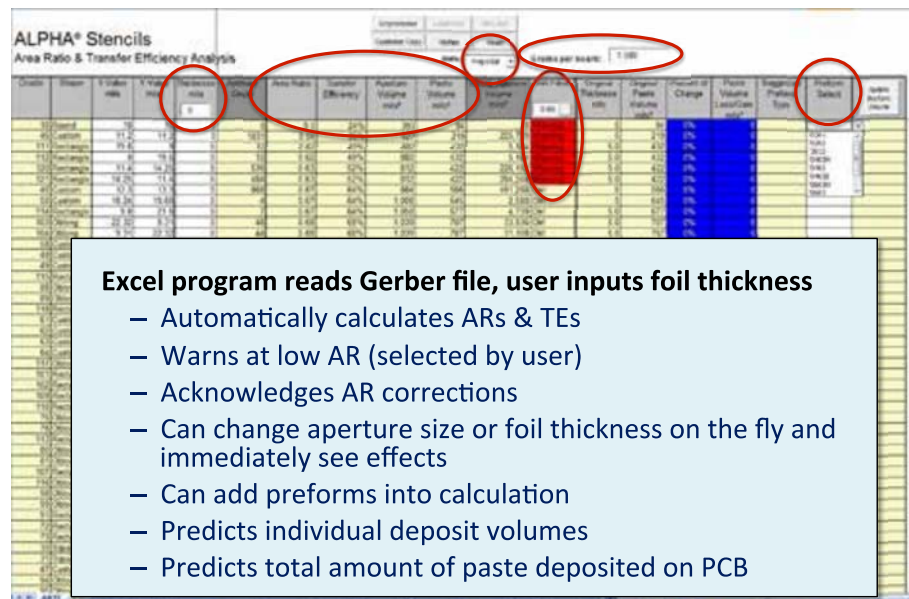


Figure 1. Screenshot of ARTE.

Design review

PCB layout drives the primary considerations of stencil design: foil thickness and aperture sizes. Larger components, through-hole connectors, and components with known coplanarity or warpage issues require larger paste deposits from thicker stencil foils, whereas smaller components with finer pitch I/Os push the area ratio limits and drive the use of thinner foils. When both appear in the same layout, they clash, but there are a number of options for accommodating the conflicting requirements.

The first step in resolving the conflicts is identifying them. Automated design-checker software like Alpha’s ARTE system reads a stencil’s Gerber file and calculates all the area ratios, issuing warnings about those below a threshold set by the user, often in the 0.60 to 0.66 range. In addition to basic design rule checking, ARTE also

When both large and smaller/fine pitch components appear in the same layout, they clash, but there are a number of options for accommodating the conflicting requirements.

uses transfer efficiency equations developed in Alpha’s solder paste labs to predict transfer efficiencies and deposit volumes. This unique capability enables the user to quickly investigate numerous aspects of

This article was presented at IMAPS New England 2014 Symposium and Expo held on May 6 in Boxboro, MA

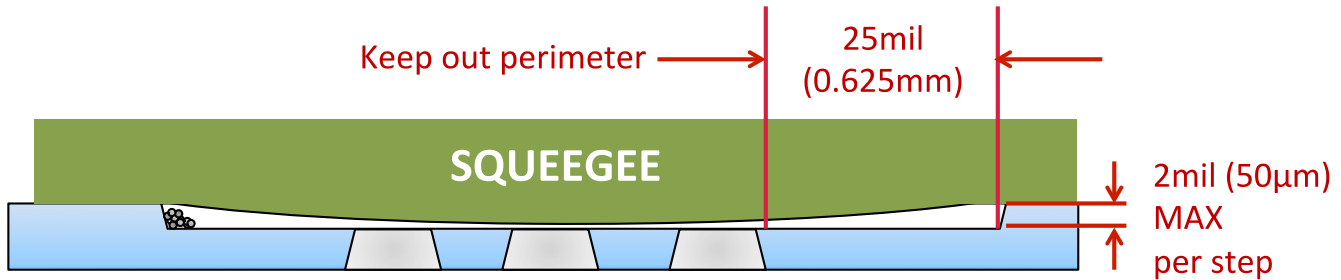


Figure 2. Larger keepout zones keep dried paste buildup in the corner of the pocket, away from the apertures.

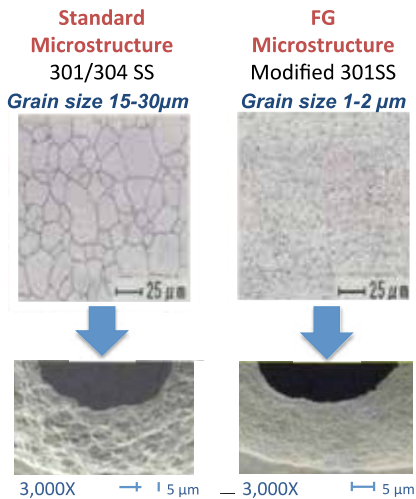


Figure 3. Fine grain stainless steel reduces the typical grain size by an order of magnitude.

the stencil’s design, including the effects of changing foil thickness or aperture size and stepping stencils. It can also suggest the best solder preform to use to compensate for thinning foils and calculate the solder volume difference based on its suggestion or on the user’s own selection. A screenshot of ARTE is shown in Figure 1.

Stepped Stencils

Often times stepping stencils provides 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

Design guidelines for steps include a maximum step height or depth of 2 mil

	PhD Stress-relieved 304SS	FG Stress-relieved 301SS
Miniaturized or high-density assembly		✓
Area ratios <0.66		✓
General SMT, lead pitches ≥ 0.5 mm, leadless	✓	
Stepped stencil for μBGA, CSP, QFN, BTC		✓
Uniform foil thickness ≥ 150 μm	✓	
Powder size Type 4,5,6		✓
Powder size Type 3	✓	✓

Table 1. When to select PhD or FG.

(50 μm) per step to maintain good fill pressure, and a minimum keepout perimeter of 25 mil (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 (Figure 2). 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 1 mil (25 μm), then an incrementally-sized foil may provide the ideal solution. Alpha provides nickel foils that are grown in their electroforming tanks in half-mil (12.5μm) increments: 3.5, 4.5, 5.5 or 6.5 mils thick. Nickel foils not only offer these sub-1 mil incremental thicknesses; they also offer high durability for processes that must run excessive print pressures.

Foil thickness, material selection and manufacturing process

Stainless steel (SS) is the material of choice, except when special circumstances dictate nickel. Standard SS is the least expen-

sive 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 by Datum Alloys is precision rolled to maintain very tight thickness tolerances and is stress relieved to prevent distortion from the heat of cutting. In addition to the popular stress-relieved SS alloy known as PhD, Datum also offers a fine grain (FG) SS that reduces the typical grain size by an order of magnitude (Figure 3). The finer grains produce smoother stencil walls and crisper steps. Many high precision stencil printing processes depend upon it.

When should a designer choose PhD or FG? The considerations are shown in Table 1.

Well-tuned modern laser cutters produce high accuracy stencils. If nickel is required, laser cutting the apertures into a formed nickel “blank” will likely produce a more accurate stencil than most electroforming process. 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.

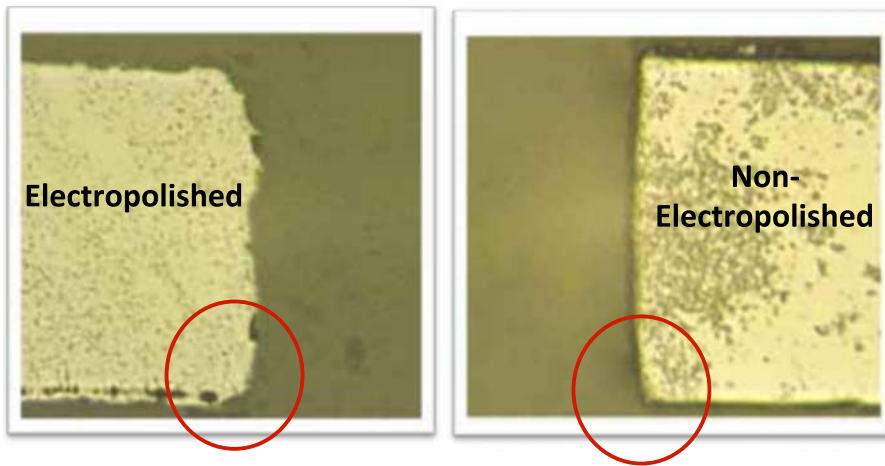
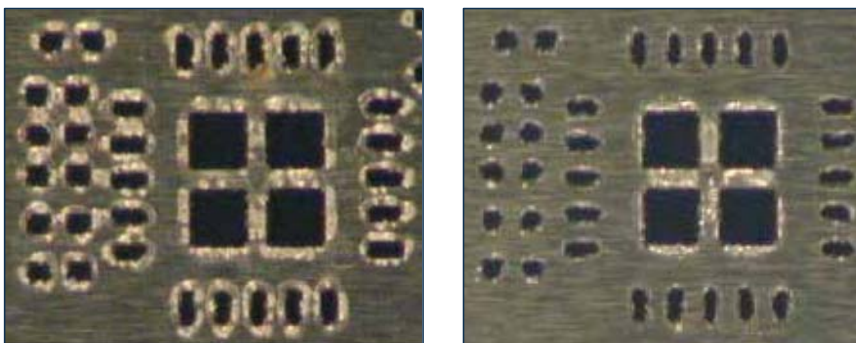


Figure 4. Effect of electropolishing on aperture geometry.

QFN and 0201s after 10 prints with no wipe

Same board, same stencil, same print stroke



Untreated stencil

Flux wicks out on the bottom surface away from the apertures

Treated stencil

Flux is repelled from the bottom surface and is contained primarily within the apertures

Figure 5. Nanocoating repels flux on stencil's bottom surface.

Nickel plating and electropolishing

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. Differences as large as 0.4 mils in aperture size and foil thickness were noted.

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 also tends to round the corners of the apertures to compromise gasketing and induce more print volume variation (Figure 4). It is not often used anymore—modern lasers can cut cleaner, smoother walls from more consistent, laser-friendly materials and electropolishing is not necessary. A good laser cut achieved with a well calibrated and maintained cutter will produce walls smooth enough to provide optimal paste transfer performance without requiring any secondary processes.

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. The cleaner stencil bottom:

- Gaskets better
- Produces crisper prints

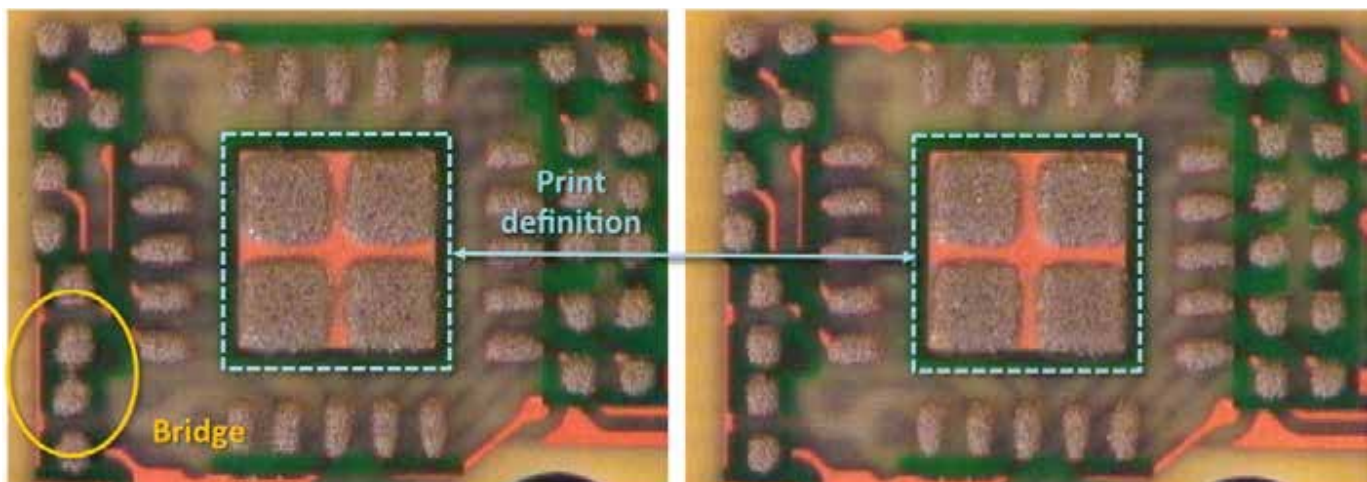


Figure 6. QFN and 0201s after 10 prints with no wipe: same board, same stencil, same print stroke. Untreated left, nanocoated right.

- Extends under wipe intervals
- Cleans easier
- Saves money on wiper paper, and (sometimes) solvent and cycle time

Figure 5 shows the stencil apertures for QFNs after 10 prints with and without Aculon's NanoClear nanocoating. Figure 6 shows the resultant prints. The difference in print definition is visible in the deposits for the thermal pad and the wet-bridged 0201s.

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.

Summary

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 5 mil (125 μm) 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 like ARTE 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 2 mil or greater should use Fine Grain SS. Other considerations for using Fine Grain SS include foil thicknesses of 5 mil (125 μm) or less, devices with pitches of 20 mil (0.5 mm) 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 been shown to improve overall print performance in recent studies 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

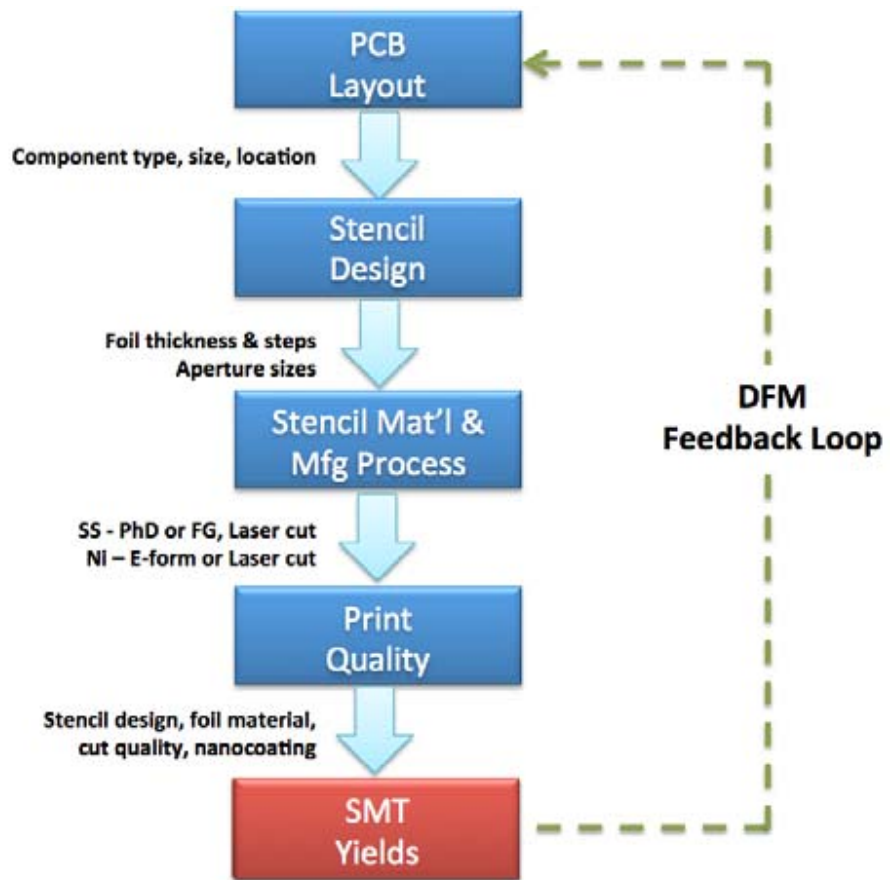


Figure 7. PCB layout drives the stencil print process.

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

Figure 7 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.

Ben Scott is the CEO of Datum Alloys, the #1 worldwide supplier of SMT stencil materials; ben@datumalloys.com

Chrys Shea is the president of Shea Engineering Services, an electronics manufacturing consulting firm; chrys@sheaengineering.com

Carol Wood is the North American Operations Manager for Alpha stencils, combining Alpha's global knowledge, experience and service to produce over half a million stencils worldwide; cjwood@alcent.com