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.

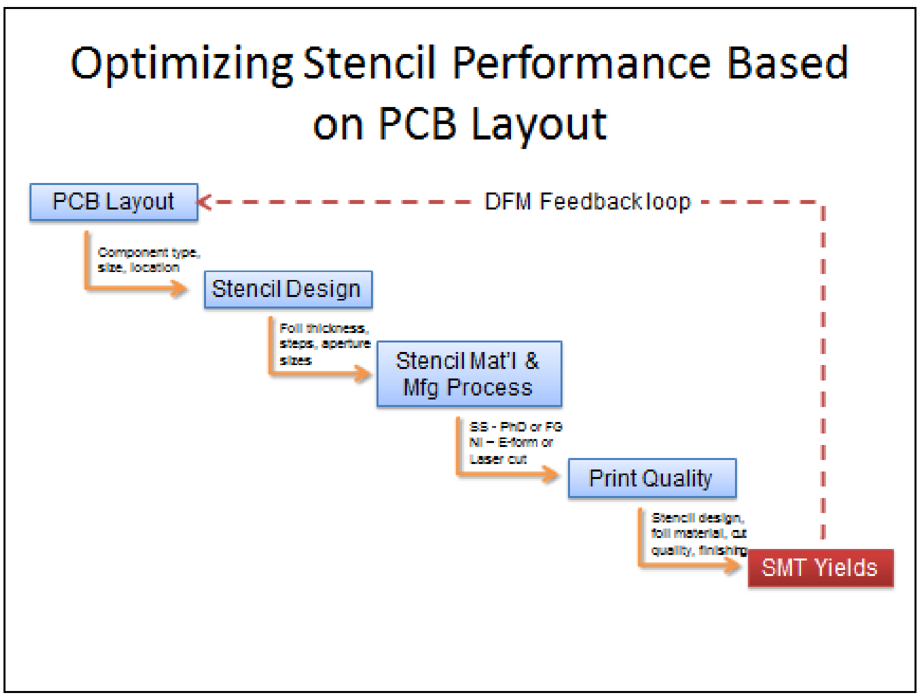


Figure 1. Overview of stencil optimization process[1]

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

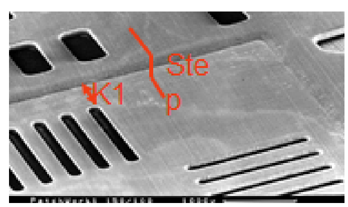
Steps can be created by:

- **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.

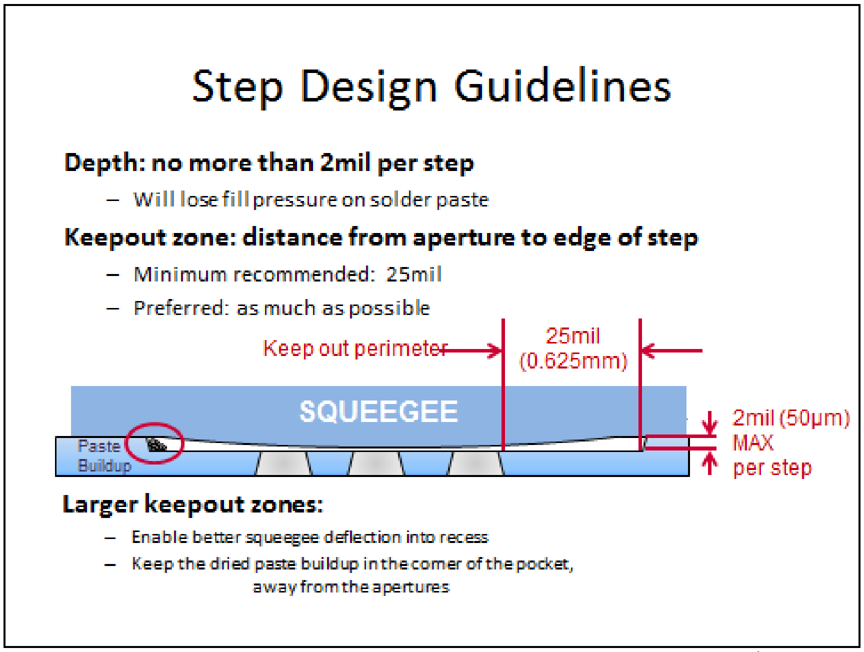


Figure 3. Alternate Stencil Step Down Design Guidelines[4]

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.

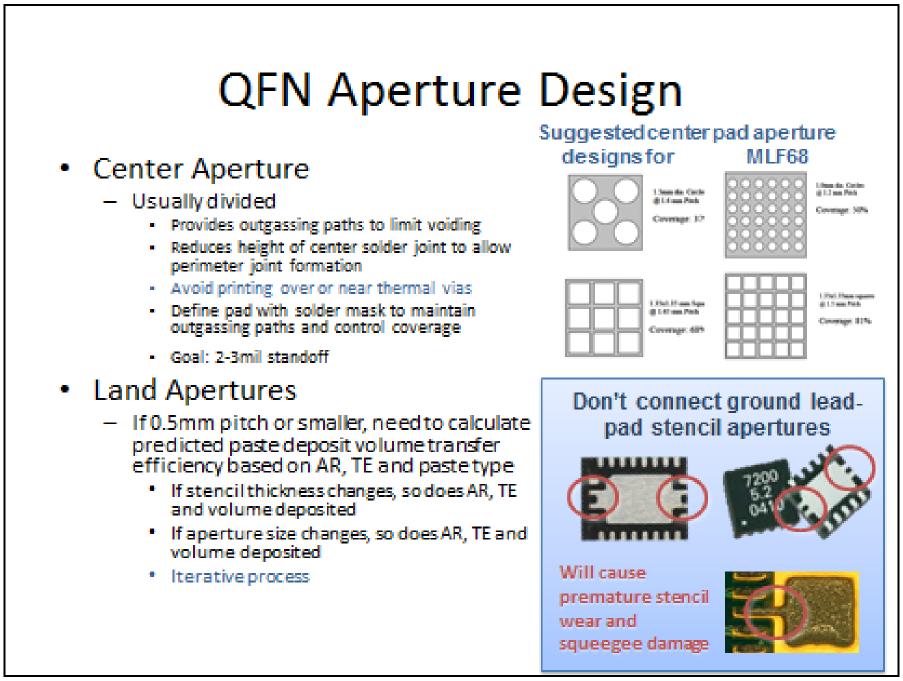


Figure 4. BTC aperture design considerations[4]

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]

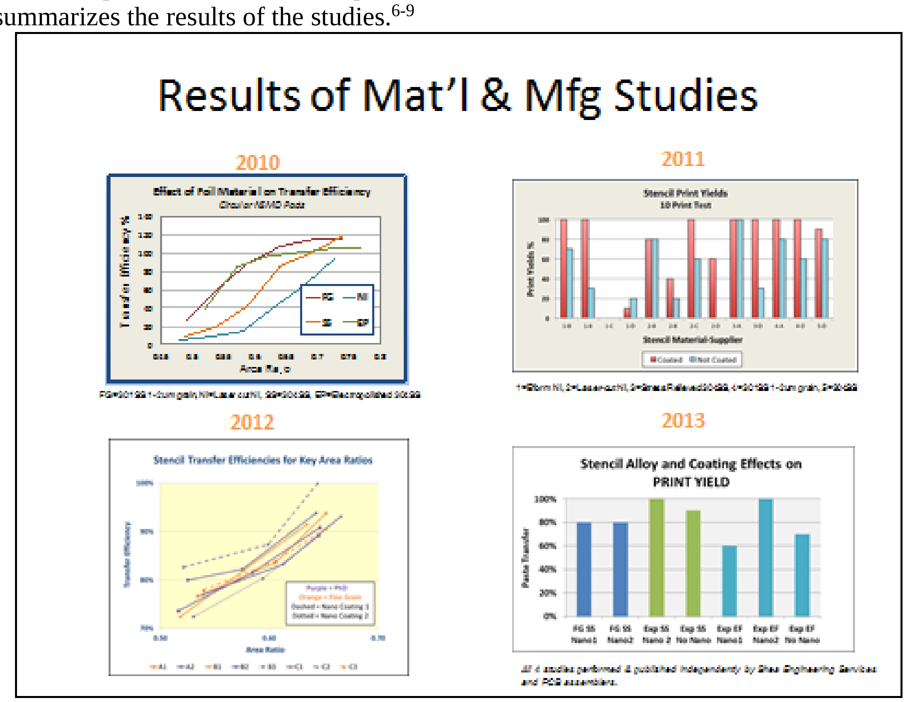


Figure 5. Results of print tests comparing foil materials

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).

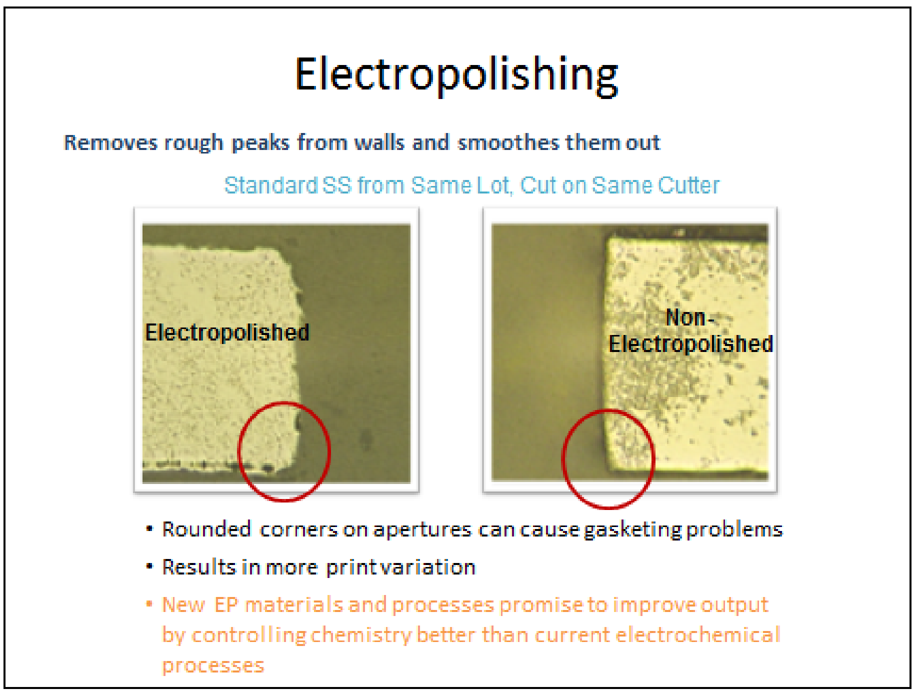


Figure 6. Cross sections of electropolished stainless steel apertures

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.

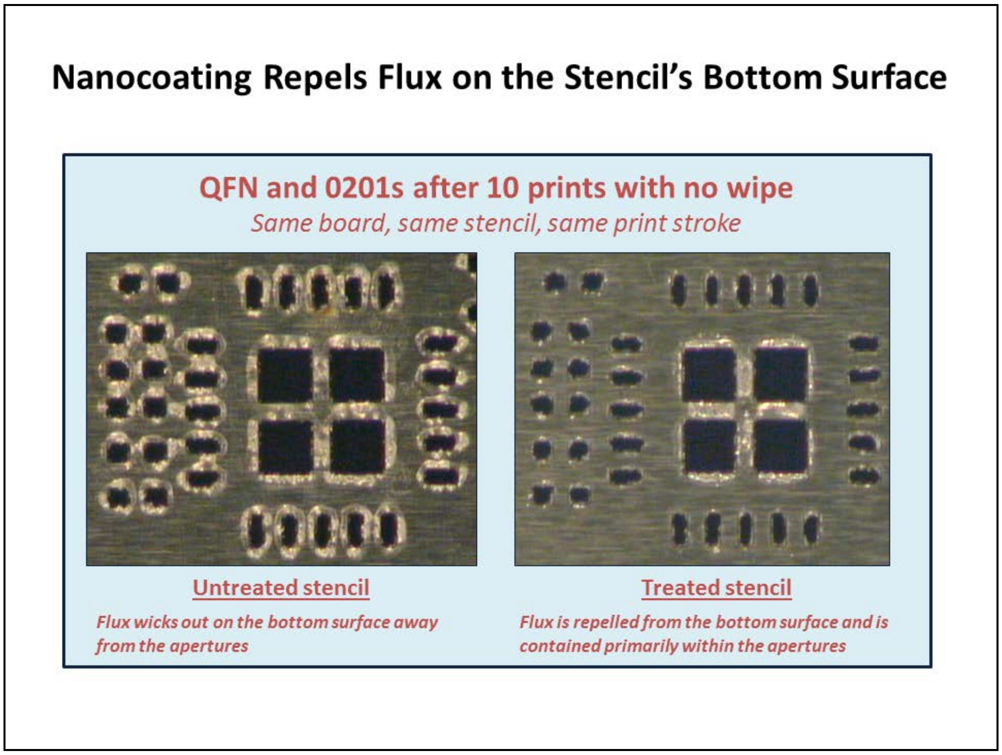


Figure 7. Flux spread on stencil bottom after 10 prints

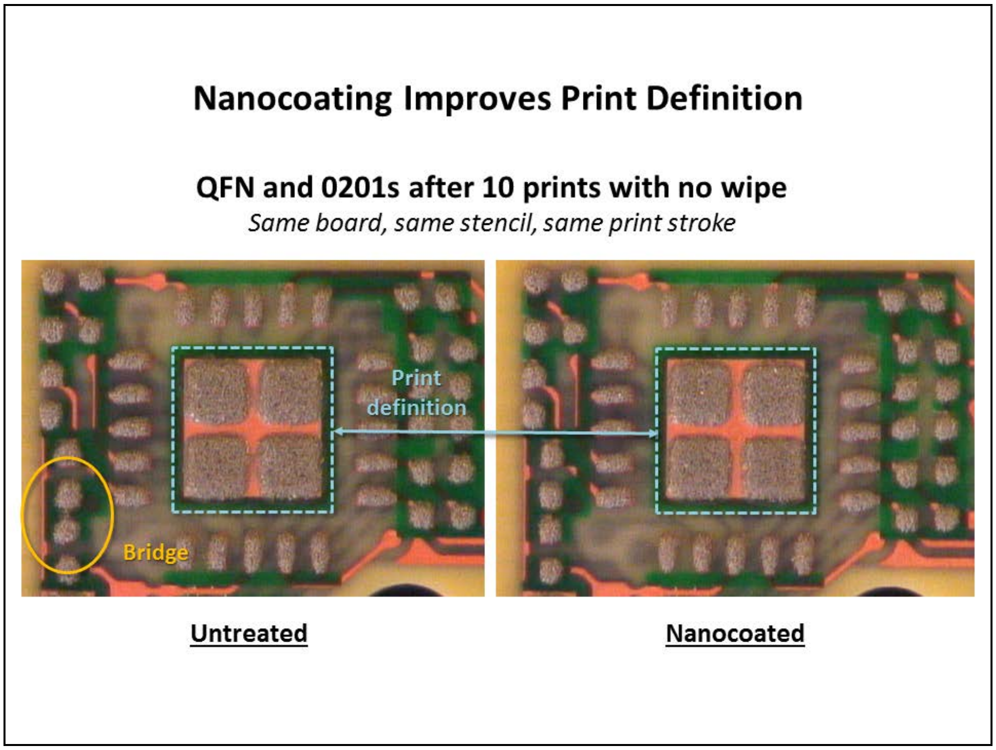


Figure 8. Print definition with and without nanocoating

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.

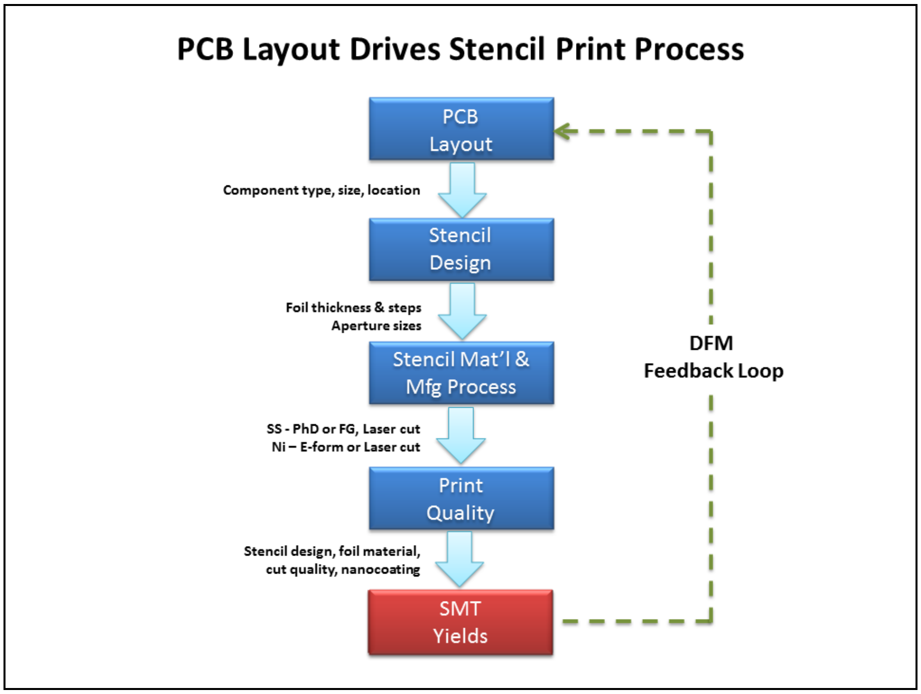


Figure 10. Influence of PCB layout on SMT printing process

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

Selecting Stencil Technologies to Optimize Print Performance

Chrys Shea

Shea Engineering Services

Communicating Expertise

Agenda

PCB Layout ← DFM Feedback loop

Component type, size, location →

Stencil Design

Foil thickness, steps, aperture sizes →

Stencil Mat'l & Mfg Process

SS - PhD or FG
Ni – E-form or Laser cut →

Print Quality

Stencil design, foil material, cut quality, finishing →

SMT Yields

Overview of available options

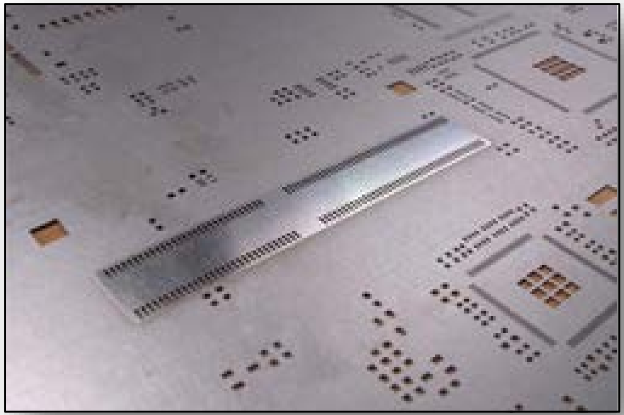
Tradeoffs Between Aperture Size and Foil Thickness

- Broad range of component sizes on PCB design
  - Big ones that requires higher volume solder paste deposits
    - Power components, PTH, SMT connectors
    - Rf shields
    - High I/O BGAs and LGAs
  - Small ones that requires high-precision, lower volume deposits
    - uBGAs, some QFNs, LGAs and BTCs
    - 0201s, 01005s
- Put extreme demands on stencil printing process
  - Larger deposits require thicker stencils
  - Smaller deposits require thinner stencils
  - Optimum print parameters change with feature size & density

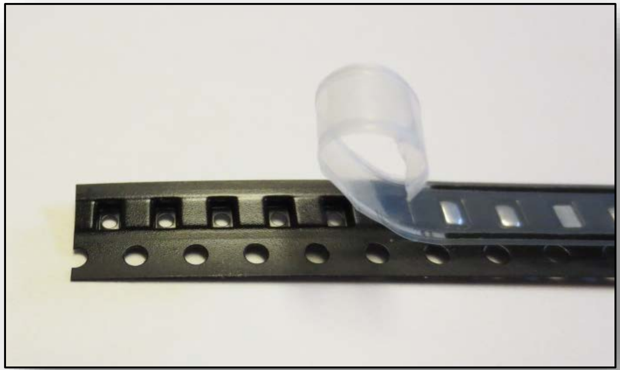
Managing the Mix

Traditional Approaches

- Stepped stencils
  - Different foil thicknesses accommodate different paste deposition requirements
  - Max step is 2mil (50um)
- Preforms
  - Add extra solder when printing can't achieve necessary volume
- Stencil design
  - Calculate volumes for Pin-in-Paste and other large solder joints
  - Calculate volumes for BGAs, QFNs and small solder joints
  - Determine tradeoffs in stencil thicknesses



Stepped stencils



Solder preforms in tape and reel

Basic Metrics in Stencil Printing

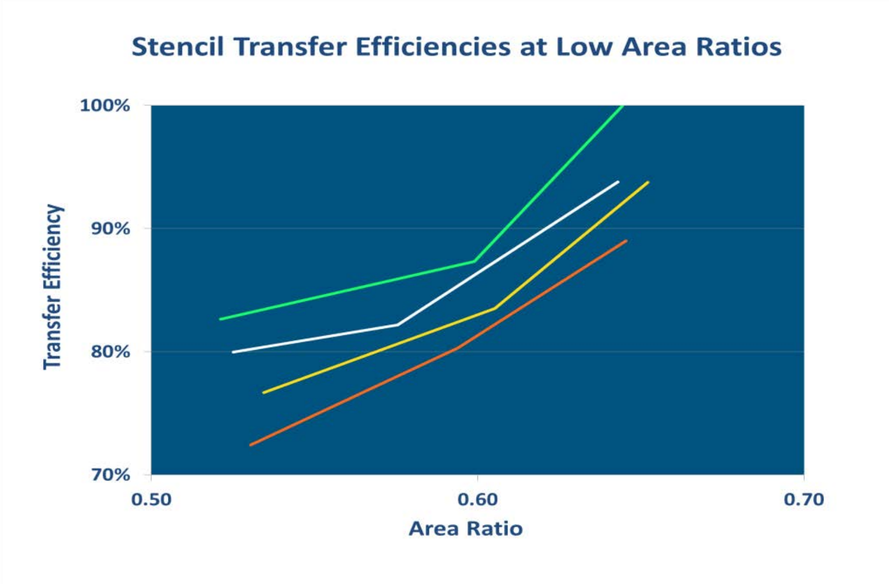
Area Ratio, AR

$$AR = \frac{\text{Area of circuit side opening}}{\text{Area of aperture walls}}$$

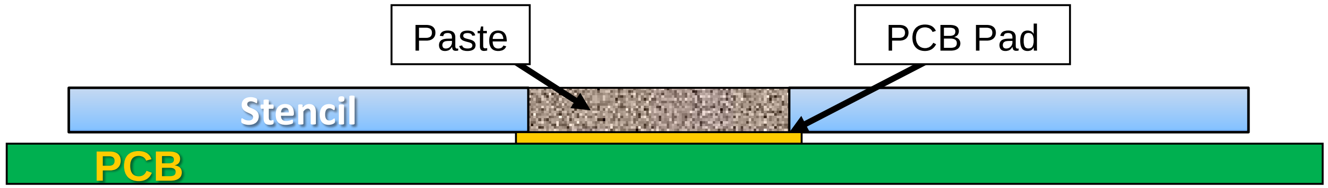
Transfer Efficiency, TE

$$\%\ TE = \left(\frac{\text{Volume of paste deposited}}{\text{Volume of stencil aperture}}\right) \times 100$$

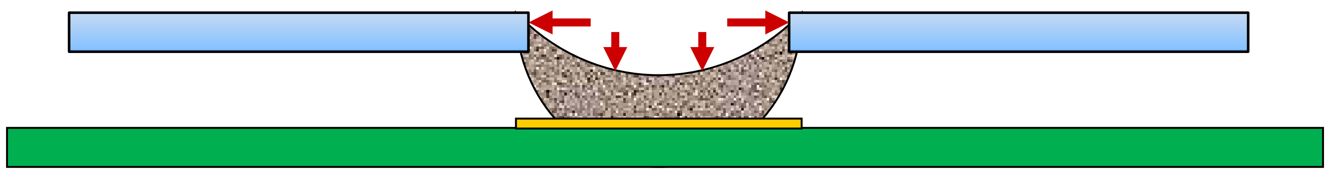
- A stencil aperture's **Area Ratio** helps predict the volume of paste deposited on the PCB
- The aperture volume is multiplied by the **Transfer Efficiency** to predict the paste deposit's volume
- ***Changing aperture size or foil thickness changes AR***
- ***Changing paste, stencil or print parameters can change TE***



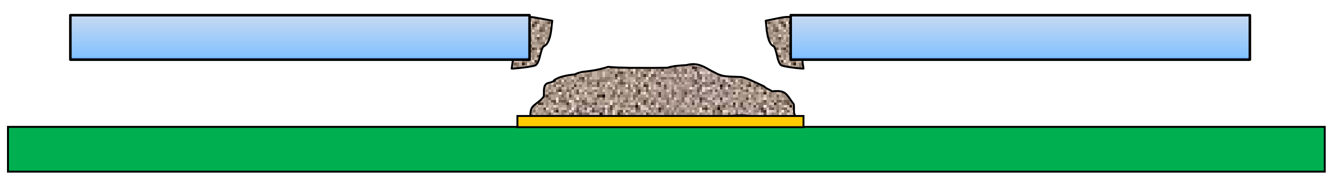
Solder Paste Release from Stencil



After the aperture is filled, the solder paste sets up and sticks to both the stencil walls and the pads.



At separation, the forces holding the deposit to the pad must overcome the forces holding the deposit to the stencil walls



Depending on area ratio, a portion of the paste will release to the PWB, while some will stay in the aperture. Some paste may also stick to the bottom of the stencil due to stringing, bad gasketing or pump out

The smaller the AR, the lower the TE

Stepping

Stepping is critical in many processes, especially when stencil design calculations are being performed based on aperture volumes and area ratios

- Steps are chemically etched prior to laser cutting
- Step Types:
  - **Step Up:** Thickens stencil locally
  - **Step Down:** Thins stencil locally
  - **Top or Bottom side steps,** or both
  - **"Stepless" steps:** Smooth the transition (used w/encl print heads)
  - **Angled steps:** Reduce squeegee damage (also w/encl print heads)
  - **Cavity relief:** For labels or other PCB topographical features
- **Precision steps** are often required for high-density assemblies

Precision Stepping

- From top or bottom
- May have very tight keepout zone
- Needs well defined walls
- May have irregular shape
- Low tolerance on thickness variation
- Fine Grain (FG) stainless steel is best choice

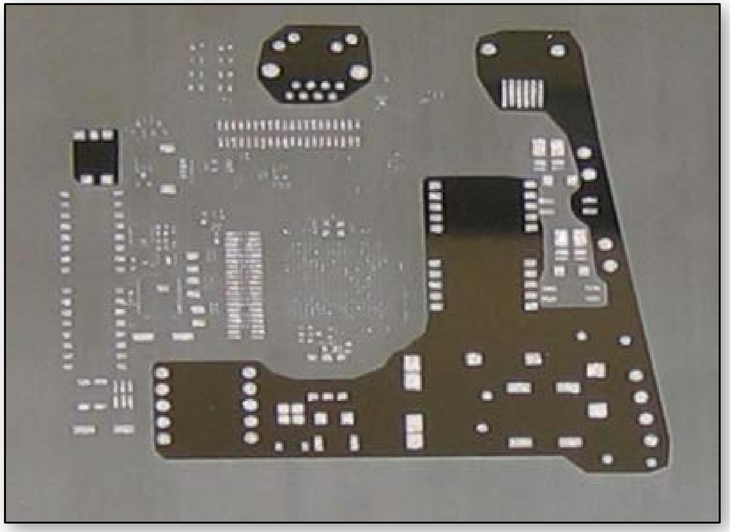


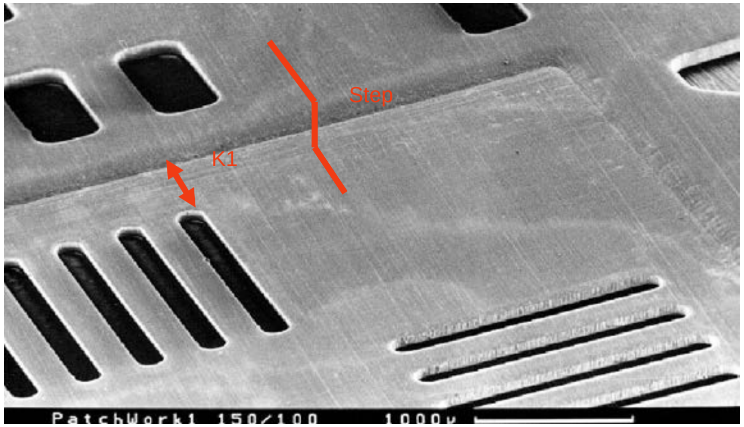
Image Source: HP Etch

Stencil Design

Keys to a successful print process

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

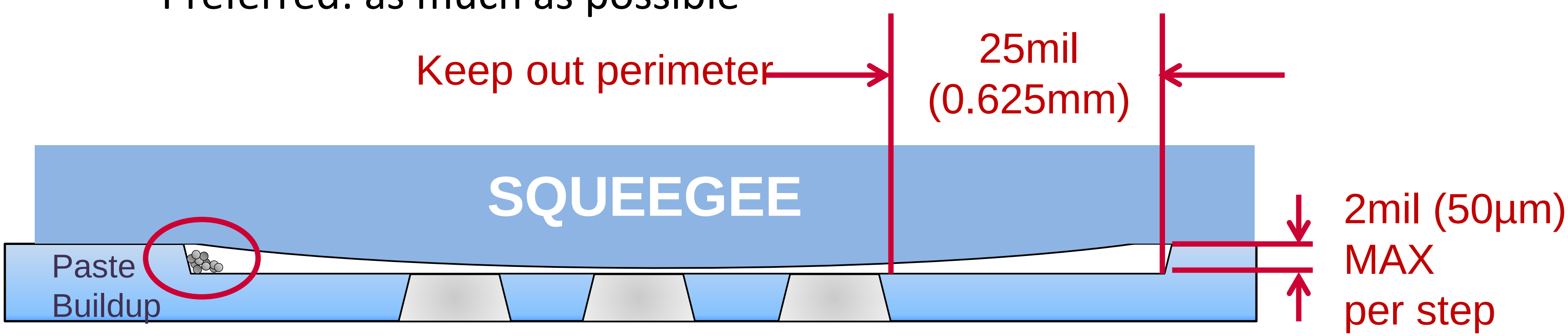
Step Design Guidelines

Depth: no more than 2mil per step

- Will lose fill pressure on solder paste

Keepout zone: distance from aperture to edge of step

- Minimum recommended: 25mil
- Preferred: as much as possible



Larger keepout zones:

- Enable better squeegee deflection into recess
- Keep the dried paste buildup in the corner of the pocket, away from the apertures

PTH Stencil Design Calculations

- Through Hole/PiP
  - Solder volume needed
    = Hole vol – pin vol + solder fillets (assumption)
  - Solder paste deposited
    = Aperture volume (overprint) + solder volume pushed into hole (assumption)
    - Aperture volume changes with changes in foil thickness
  - Preform volume (if used)
    =LxWxH, also available from on-line chart
  - Solid solder volume
    =~50% of paste volume + 100% of preform volume
- Fine features/uBGA/0201
  - Deposit volume
    = Aperture volume * TE for the aperture's AR and paste type
    - AR and TE change with changes in foil thickness

Solder preforms placed in solder paste add volume to PTH and other large solder joints

Image Source: Alpha

BTC/QFN Stencil Design

- This is a very economical, reliable and popular package
  - Some chipsets are only available in this package type
  - Some assemblers have up to 15 years' experience with package; some have 0.
  - Thermal/ground pad causes issues:
    - Too much paste on center pad prevents perimeter joint formation
    - Not enough paste on center pad limits thermal transfer
    - Themal vias in pad rob paste from bond, causing voids and creating printing problems on second side
    - Flux in solder paste causes voids
  - Voiding in pad may affect thermal and electrical performance

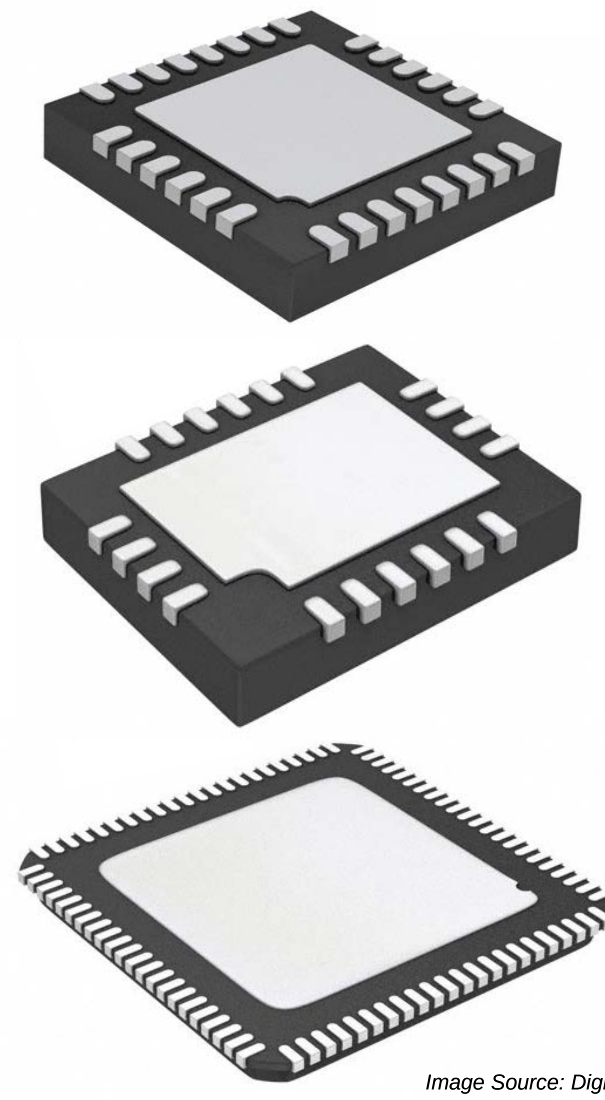
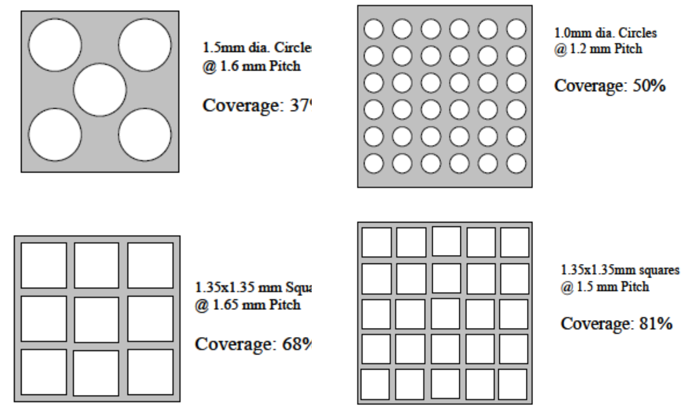


Image Source: Digikey

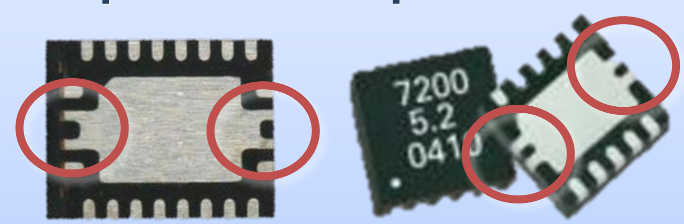
QFN Aperture Design

- Center Aperture
  - Usually divided
    - Provides outgassing paths to limit voiding
    - Reduces height of center solder joint to allow perimeter joint formation
    - Avoid printing over or near thermal vias
    - Define pad with solder mask to maintain outgassing paths and control coverage
    - Goal: 2-3mil standoff
- Land Apertures
  - If 0.5mm pitch or smaller, need to calculate predicted paste deposit volume transfer efficiency based on AR, TE and paste type
    - If stencil thickness changes, so does AR, TE and volume deposited
    - If aperture size changes, so does AR, TE and volume deposited
    - Iterative process

Suggested center pad aperture designs for MLF68



Don't connect ground lead-pad stencil apertures



Will cause premature stencil wear and squeegee damage

Automating Stencil Design

Reference: Alpha Stencils

Unprotected | Lead Free | Tin Lead

Customer Copy | Update | Reset

Units: Imperial

Grams per board: 1.080

Area Ratio & Transfer Efficiency Analysis

Dcode	Shape	X Value mils	Y Value mils	Thickness mils (5)	Aperture Count	Area Ratio	Transfer Efficiency	Aperture Volume mils³	Paste Volume mils³	TOTAL Aperture Volume mils³	AR Filter (0.66)	Original Thickness mils	Original Paste Volume mils³	Percent of Change	Paste Volume Loss/Gain mils³	Suggested Preform Type	Preform Select
10	Round	10	10	5	0	0.5	24%	393	94	0	Warning	5	94	0%	0		
46	Custom	11.2	11.2	5	1031	0.56	35%	627	219	225,[illegible]89	Warning	5	219	0%	0		
111	Rectangle	19.6	9	5	12	0.62	49%	882	432	5,[illegible]84	Warning	5.0	432	0%	0		
112	Rectangle	9	19.6	5	12	0.62	49%	882	432	5,[illegible]84	Warning	5.0	432	0%	0		
120	Rectangle	11.4	14.25	5	536	0.63	52%	812	422	226,[illegible]92	Warning	5.0	422	0%	0		
121	Rectangle	14.25	11.4	5	484	0.63	52%	812	422	204,2[illegible]8	Warning	5.0	422	0%	0		
45	Custom	13.3	13.3	5	868	0.67	64%	884	566	491,288	Ok!	5	566	0%	0		
50	Custom	10.24	19.68	5	4	0.67	64%	1,008	645	2,580	Ok!	5	645	0%	0		
114	Rectangle	9.8	21.6	5	7	0.67	64%	1,058	677	4,739	Ok!	5.0	677	0%	0		
103	Oblong	22.32	9.31	5	48	0.68	68%	1,039	707	33,936	Ok!	5.0	707	0%	0		
104	Oblong	9.31	22.32	5	44	0.68	68%	1,039	707	31,108	Ok!	5.0	707	0%	0		
58	Custom	33.7	8.7	5	8	0.69	71%	1,466	1,041	8,328	Ok!	5	1,041	0%	0		
48	Custom	31.5	9.06	5	5	0.7	75%	1,427	1,070	5,350	Ok!	5	1,070	0%	0		
49	Custom	31.5	9.06	5	5	0.7	75%	1,427	1,070	5,350	Ok!	5	1,070	0%	0		
115	Rectangle	9.8	25.6	5	1	0.71	71%	1,254	890	890	Ok!	5.0	890	0%	0		
88	Oblong	25.65	10.45	5	36	0.77	77%	1,340	1,032	37,152	Ok!	5.0	1,032	0%	0		
89	Oblong	10.45	25.65	5	36	0.77	77%	1,340	1,032	37,152	Ok!	5.0	1,032	0%	0		
118	Rectangle	39.4	9.8	5	1	0.78	78%	1,931	1,506	1,506	Ok!	5.0	1,506	0%	0		
61	Custom	23.62	11.82	5	4	0.79	79%	1,396	1,103	4,412	Ok!	5	1,103	0%	0		
62	Custom	23.62	11.82	5	4	0.79	79%	1,396	1,103	4,412	Ok!	5	1,103	0%	0		
63	Custom	11.82	23.62	5	8	0.79	79%	1,396	1,103	8,824	Ok!	5	1,103	0%	0		
64	Custom	11.82	23.62	5	8	0.79	79%	1,396	1,103	8,824	Ok!	5.0	1,103	0%	0		
117	Oblong	37.4	9.8	5	5	0.80	80%	1,833	1,466	7,330	Ok!	5.0	1,466	0%	0		
101	Rectangle	11.21	29.83	5	12	0.81	81%	1,672	1,354	16,248	Ok!	5.0	1,354	0%	0		
102	Rectangle	29.83	11.21	5	12	0.81	81%	1,672	1,354	16,248	Ok!	5.0	1,354	0%	0		
109	Rectangle	19.7	14	5	4	0.82	82%	1,379	1,131	4,524	Ok!	5.0	1,131	0%	0		
110	Rectangle	14	19.7	5	4	0.82	82%	1,379	1,131	4,524	Ok!	5.0	1,131	0%	0		
75	Oblong	20.52	13.01	5	32	0.83	83%	1,335	1,108	35,456	Ok!	5.0	1,108	0%	0		
76	Oblong	13.01	20.52	5	32	0.83	83%	1,335	1,108	35,456	Ok!	5.0	1,108	0%	0		
113	Rectangle	9.8	53.1	5	48	0.83	83%	2,602	2,160	103,680	Ok!	5.0	2,160	0%	0		
80	Oblong	31.82	11.21	5	96	0.86	86%	1,784	1,534	147,264	Ok!	5.0	1,534	0%	0		
81	Oblong	11.21	31.82	5	96	0.86	86%	1,784	1,534	147,264	Ok!	5.0	1,534	0%	0		
107	Rectangle	9.8	68.9	5	432	0.86	86%	3,376	2,903	1,254,096	Ok!	5.0	2,903	0%	0		
116	Oblong	57.1	9.8	5	10	0.86	86%	2,798	2,406	24,060	Ok!	5.0	2,406	0%	0		
98	Oblong	11.21	33.63	5	20	0.87	87%	1,885	1,640	32,800	Ok!	5.0	1,640	0%	0		
99	Oblong	33.63	11.21	5	20	0.87	87%	1,885	1,640	32,800	Ok!	5.0	1,640	0%	0		
91	Rectangle	13.01	31.73	5	16	0.92	92%	2,064	1,899	30,384	Ok!	5.0	1,899	0%	0		
72	Rectangle	18.7	18.6	5	3	0.93	93%	1,739	1,617	4,851	Ok!	5.0	1,617	0%	0		
73	Rectangle	18.6	18.7	5	1	0.93	93%	1,739	1,617	1,617	Ok!	5.0	1,617	0%	0		
30	Square	18.7	18.7	5	3	0.94	94%	1,748	1,643	4,929	Ok!	5	1,643	0%	0		
31	Square	18.7	18.7	5	1	0.94	94%	1,748	1,643	1,643	Ok!	5	1,643	0%	0		
47	Custom	18.7	18.7	5	64	0.94	94%	1,748	1,643	105,152	Ok!	5	1,643	0%	0		
[illegible]	Oblong	11.21	56.05	5	50	0.96	96%	3,142	3,016	150,800	Ok!	5.0	3,016	0%	0		
95	Oblong	56.05	11.21	5	50	0.96	96%	3,142	3,016	150,800	Ok!	5.0	3,016	0%	0		

'0201
'0202
'3015
'0402H
'0402
'0402B
'0603H
'0603

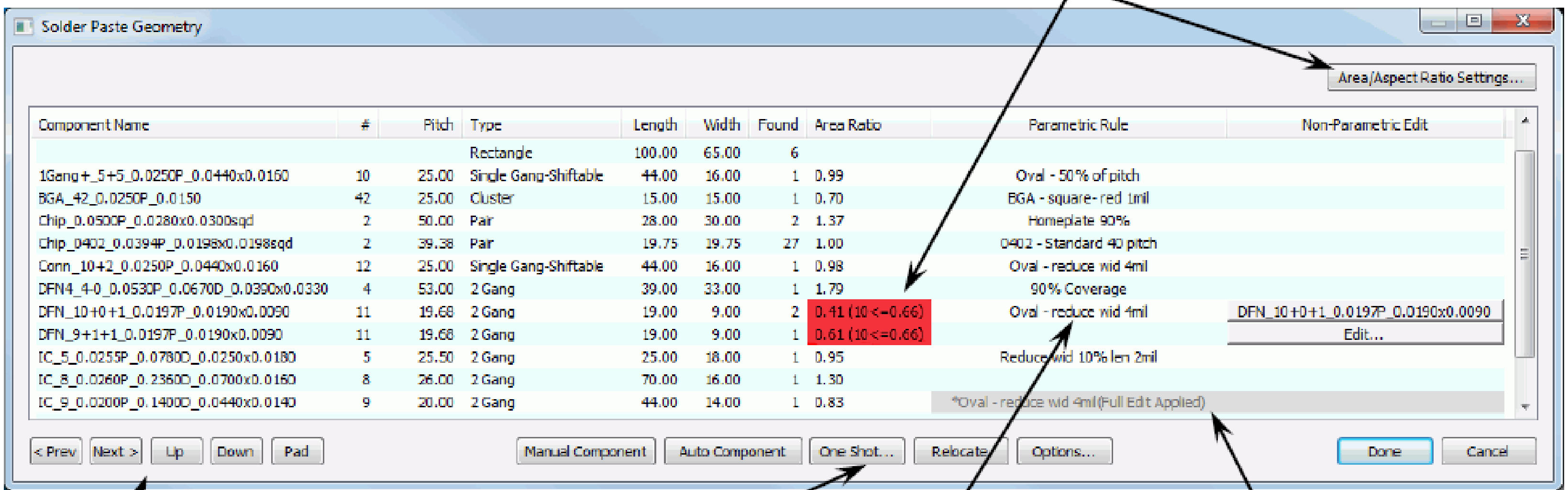
Update Preform Volume

ARTE

Automating Stencil Design

Area/Aspect Ratio Calculated to Specification
Click on the button to adjust settings or turn ratio calculations on/off.

Results for the smallest stencil aperture are shown for each component.
Stencil apertures which don't meet the target values are flagged in red.



Component Name	#	Pitch	Type	Length	Width	Found	Area Ratio	Parametric Rule	Non-Parametric Edit
			Rectangle	100.00	65.00	6			
1Gang+_5+5_0.0250P_0.0440x0.0160	10	25.00	Single Gang-Shiftable	44.00	16.00	1	0.99	Oval - 50% of pitch	
BGA_42_0.0250P_0.0150	42	25.00	Cluster	15.00	15.00	1	0.70	BGA - square- red 1mil	
Chip_0.0500P_0.0280x0.0300sqd	2	50.00	Pair	28.00	30.00	2	1.37	Homeplate 90%	
Chip_0402_0.0394P_0.0198x0.0198sqd	2	39.38	Pair	19.75	19.75	27	1.00	0402 - Standard 40 pitch	
Conn_10+2_0.0250P_0.0440x0.0160	12	25.00	Single Gang-Shiftable	44.00	16.00	1	0.98	Oval - reduce wid 4mil	
DFN4_4-0_0.0530P_0.0670D_0.0390x0.0330	4	53.00	2 Gang	39.00	33.00	1	1.79	90% Coverage	
DFN_10+0+1_0.0197P_0.0190x0.0090	11	19.68	2 Gang	19.00	9.00	2	0.41 (10<=0.66)	Oval - reduce wid 4mil	DFN_10+0+1_0.0197P_0.0190x0.0090
DFN_9+1+1_0.0197P_0.0190x0.0090	11	19.68	2 Gang	19.00	9.00	1	0.61 (10<=0.66)		Edit...
IC_5_0.0255P_0.0780D_0.0250x0.0180	5	25.50	2 Gang	25.00	18.00	1	0.95	Reduce wid 10% len 2mil	
IC_8_0.0260P_0.2360D_0.0700x0.0160	8	26.00	2 Gang	70.00	16.00	1	1.30		
IC_9_0.0200P_0.1400D_0.0440x0.0140	9	20.00	2 Gang	44.00	14.00	1	0.83	*Oval - reduce wid 4mil(Full Edit Applied)	

Want to Review?

Need to handle ambiguous data?

Want to change an Edit?
Edits are made according to the edit rule you selected, but

Want to edit special cases?

Software is from Infinite Graphics; interfaces with CAD systems to recognize footprints.
Controls aperture library and calculates AR, but not TE or preform volumes
Good for aperture design control

Stencil Foil Materials and Manufacturing Processes

Influence on print process quality

Alloys/Foil Materials & Mfg. Processes

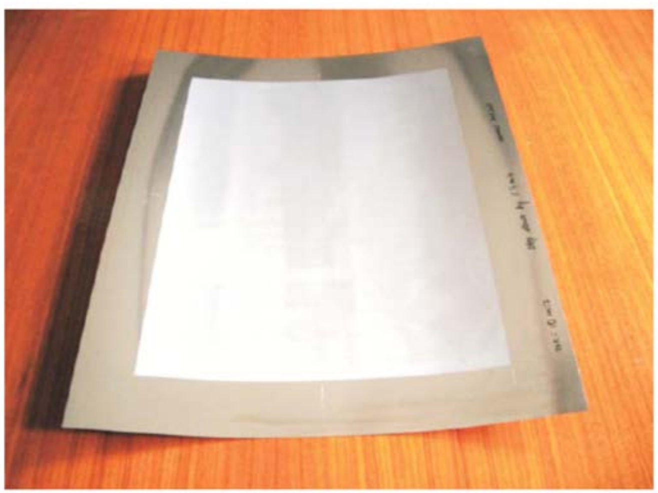
Recent evolution

- Stress relieved stainless steel (7 yrs)
- Fine grain stainless steel (5 yrs)
- New electroforming processes (always a new one!)
- New nickel plating processes (3-4 yrs)
- Laser-cut Ni (not new at all)
- Fiber lasers in cutting machines (3-5 yrs)

Images courtesy of Datum Alloys

Stress-Relieved Stainless Steel

- Cold rolling work hardens SS and imparts residual stresses
- Annealing removes them
- Stress-relieved materials deform less during cutting and enable tighter aperture density



Standard 304 SS used for stencils

Stress-relieved 304 SS used for stencils

Images courtesy of Datum Alloys

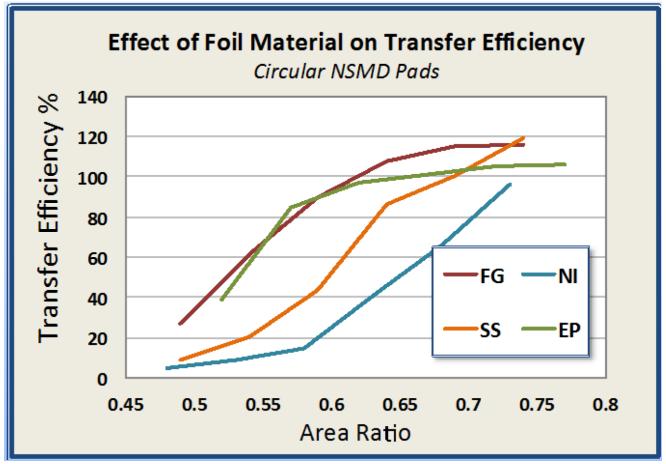
Studies on Foil Materials & Mfg Processes

- **2010**
  - FG outperforms std SS, electropolished SS, Laser-cut Ni
- **2011**
  - FG outperforms stress-relieved SS, E-form, Laser-Ni
  - Nanocoating* improves quality
- **2012**
  - SS outperforms E-form and Ni-plated SS
  - Nanocoating* improves release
- **2013**
  - New nanocoating* better than previous nanocoating
  - FG still better than E-form, Experimental SS shows promise
  - Reducing under wipes with nanocoating improves quality

* Three different nanocoatings were used in three different tests

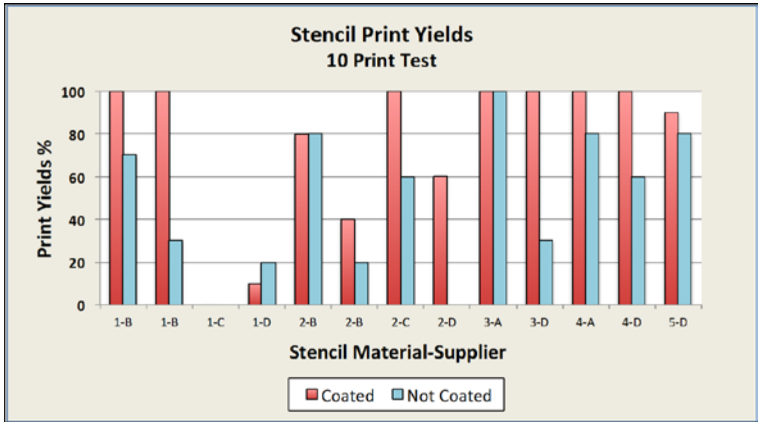
Results of Mat'l & Mfg Studies

2010



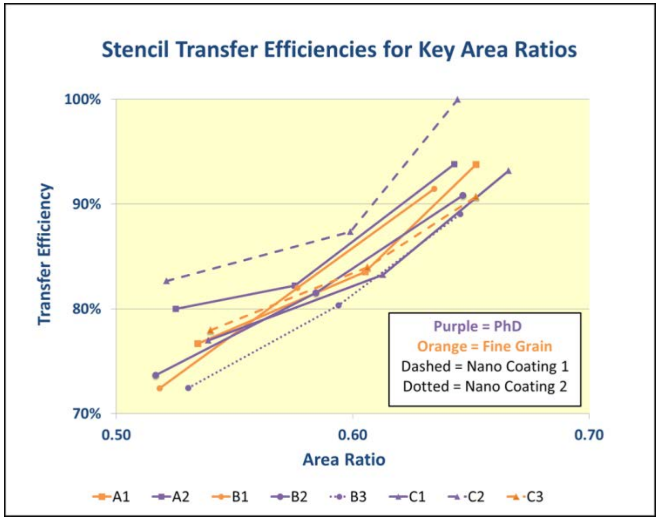
FG=301SS 1-2um grain, Ni=Laser cut Ni, SS=304SS, EP=Electropolished 304SS

2011

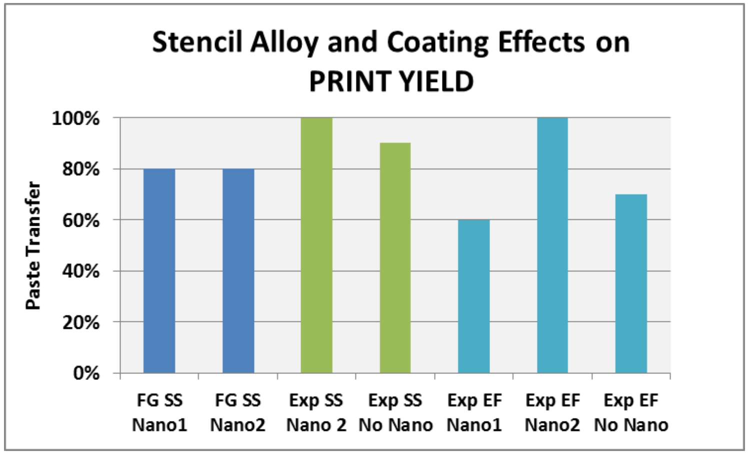


1=Eform Ni, 2=Laser-cut Ni, 3=Stress Relieved 304SS, 4=301SS 1-2um grain, 5=304SS

2012



2013



All 4 studies performed & published independently by Shea Engineering Services and PCB assemblers.

SS Foil Alloy Selection

When does FG benefit the printing process?

	Stress-Relieved 304SS	Fine Grain 301SS
Miniaturized or high-density assembly		√
Area ratios <0.66		√
General SMT, lead pitches≥ 0.5mm, leadless pitches≥ 1.0mm	√	
Stepped stencil for µBGA, CSP, QFN, BTC		√
Uniform foil thickness ≥150µm	√	
Powder size Type: 4,5,6		√
Powder size: Type 3	√	√

Based on empirical information

Electroforming

- Stencils are grown in electroplating tank
- Nickel is very hard material
  - Good for high pressure or high volume processes
- Can exhibit dimensional problems
  - Plating processes are notoriously hard to control
- Entire stencil grown with apertures
  - Very smooth walls
- Nickel "blank" grown then cut on laser cutter
  - Improves dimensional accuracy
  - Modern lasers can cut very clean walls
  - Can do ½ mil thicknesses: 3.5, 4.5, 5.5, etc.

Video on electroforming (Reference: Alpha)
: https://www.youtube.com/watch?v=UtemNdRMGdo

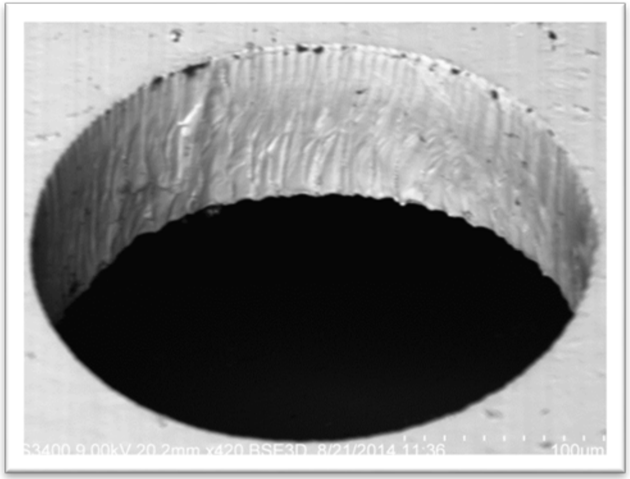
Laser Cutting

- Produces most accurate stencils
- Stencil quality depends on machine quality, age, calibration, maintenance
- Newer cutters have very fine laser beams, on-board optical inspection to make sure the hole is fully cut, on-board aperture measurement for SPC, and remote control for cals, tune-ups or troubleshooting
- Ask your stencil vendor about his cutter

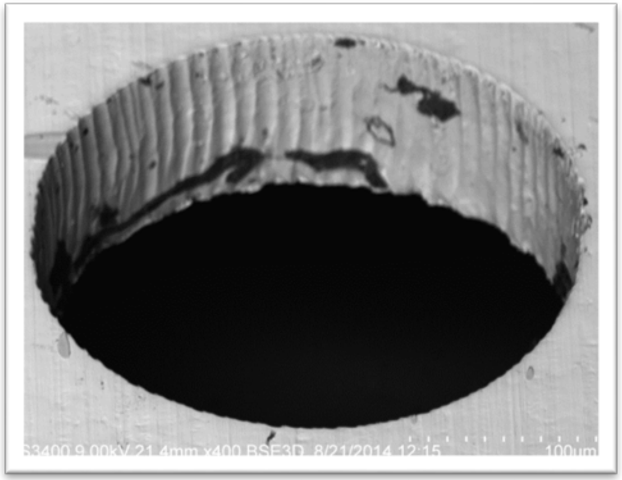
Image source: www.tannlin.com

Cut Quality is Critical

- The finer the feature, the more important cut quality
- Rougher walls do not release paste as well as smoother walls
- Burrs can impair solder paste fill and release flow
- Slag on contact side can create gasketing problems

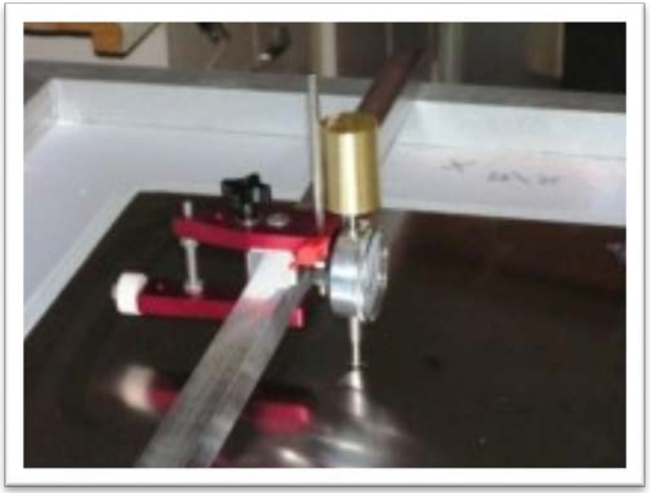


Best print performance in recent study



Worst print performance in recent study

High Tension Foils and Frames



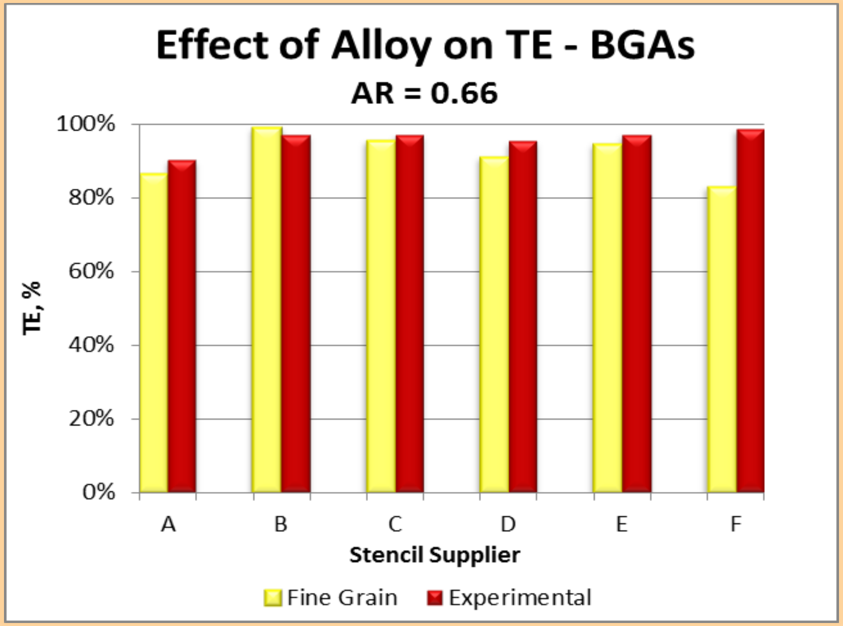
- Typical tension – 35-40 N/cm
- “High” tension – 50+ N/cm
- Promise to have less “snap back” and cleaner release
- Need a more rigid frame to carry higher tension without warping
- May need a harder steel to carry higher tension on thin webs

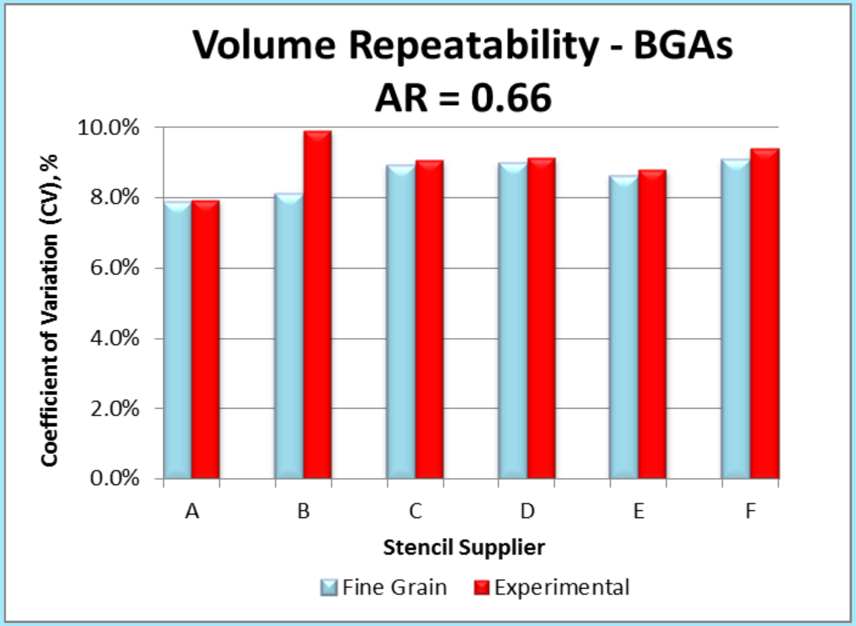
Image source: www.smtstencilsupply.com

New “Tension” SS Alloy

New alloy developed for high tension applications was tested (at std tensions) from 6 different stencil cutters.

- Performance was compared to Fine Grain (current best) as benchmark
- TE’s slightly better in most cases
- Print variation comparable



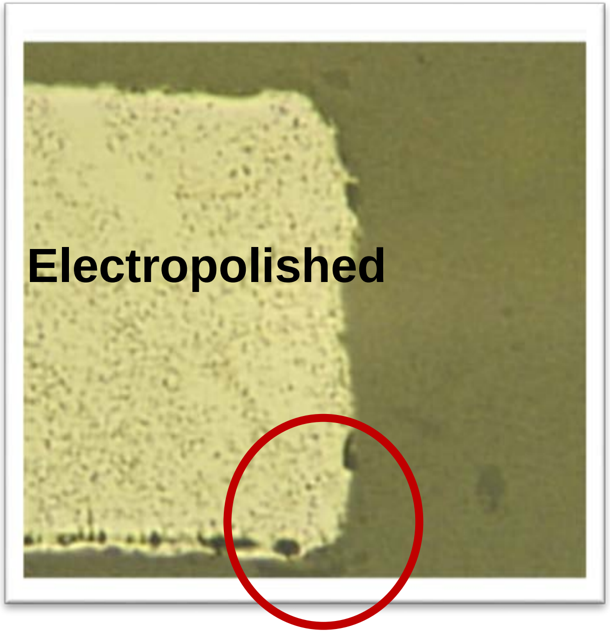


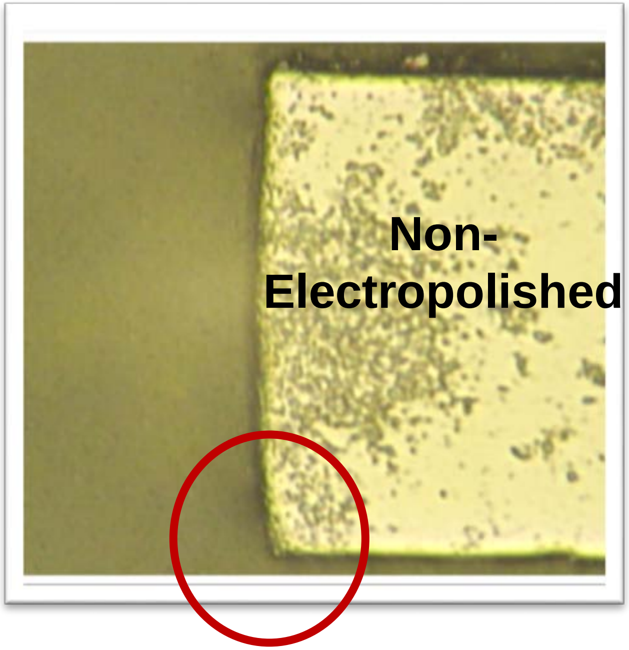
Source: “The Effects of Stencil Alloy and Cut Quality on Solder Paste Print Performance,” C. Shea and R. Whittier, Proceedings of SMTA International 2014.

Electropolishing

Removes rough peaks from walls and smoothens them out

Standard SS from Same Lot, Cut on Same Cutter





- Rounded corners on apertures can cause gasketing problems
- Results in more print variation
- New EP materials and processes promise to improve output by controlling chemistry better than current electrochemical processes .

Stencil Nanocoatings

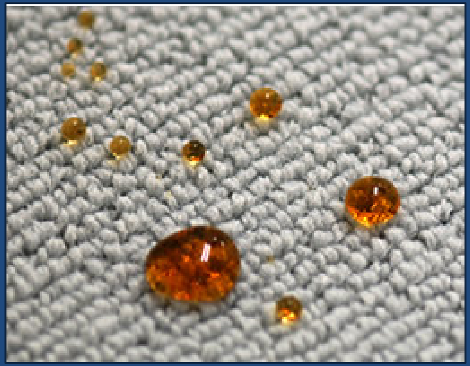
Treatments that improve print quality

Nanocoatings are a Repellency Treatment

- Water repellent: ***hydrophobic***
- Oil repellent: ***oleophobic***

Examples of Common Water and Oil Repellency Treatments

On fabric



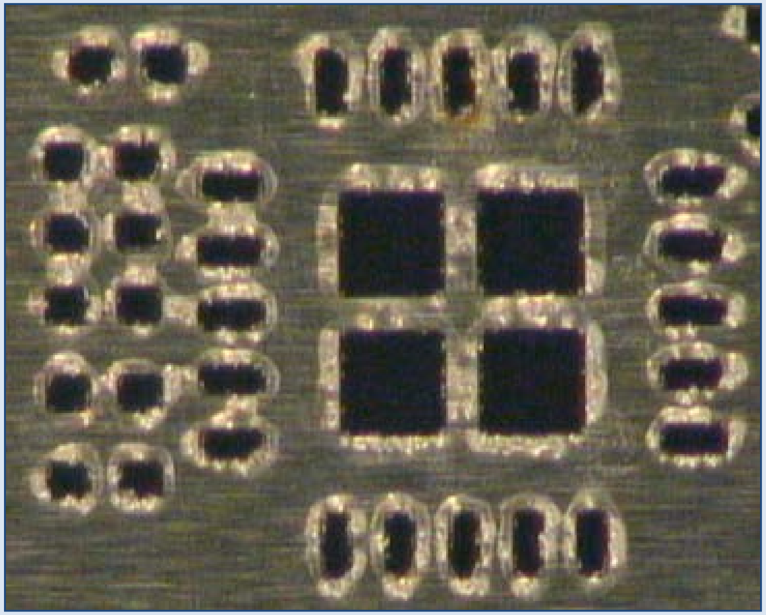
On carpet

On paper food containers

Image source: Daikin Industries (UNIDYNE web page)

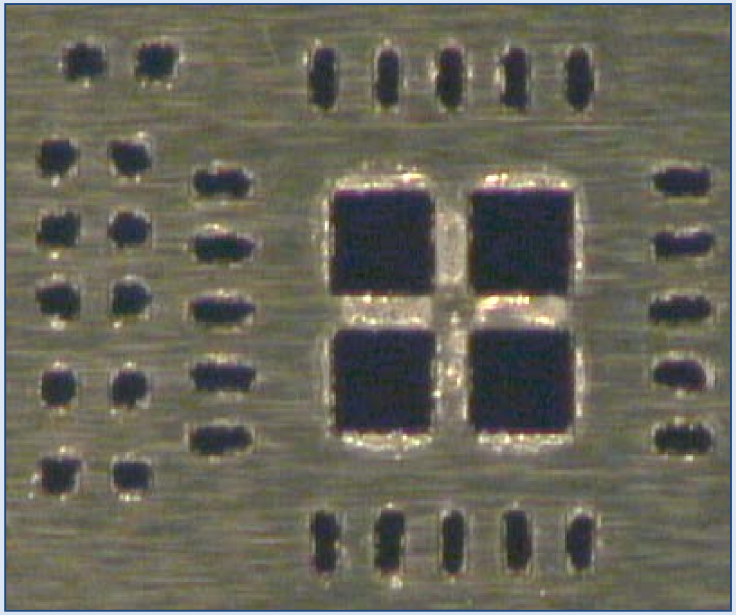
Flux Repellent: *Fluxophobic*

Example of Fluxophobic Stencil Treatment



Untreated stencil

Flux wicks out on the bottom surface away from the apertures

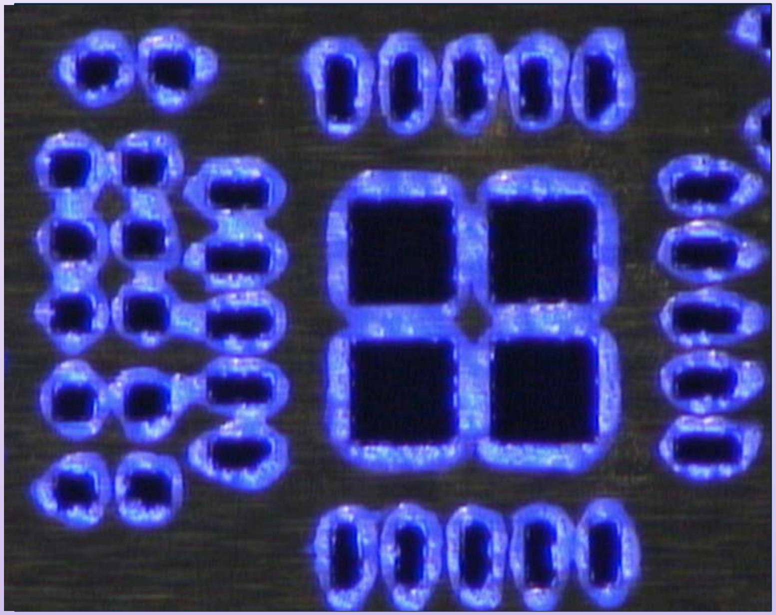


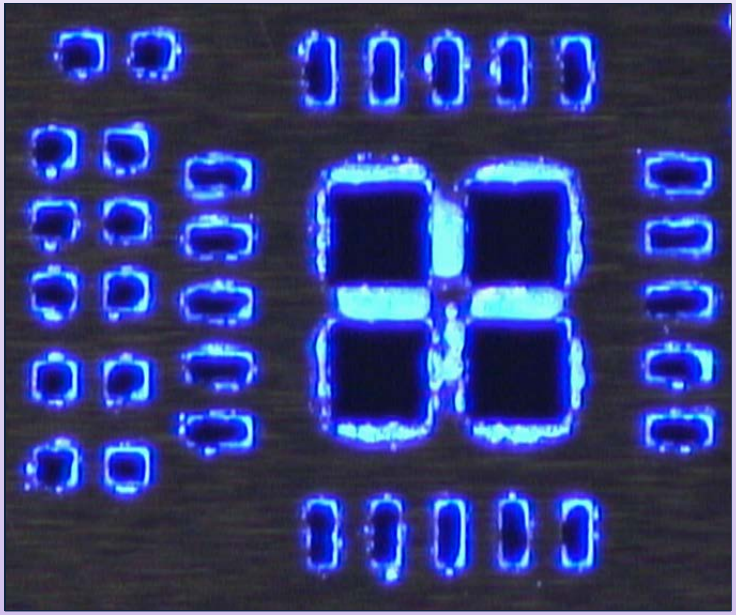
Treated stencil

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

Flux Repellency on Stencils

Flux Treated with UV Tracer Dye





Untreated stencil

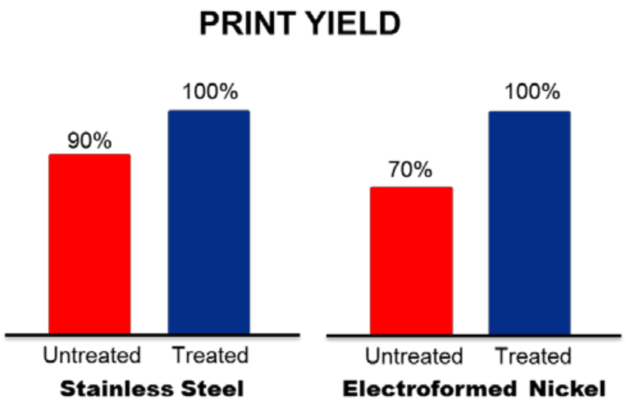
Flux wicks out on the bottom surface away from the apertures

Treated stencil

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

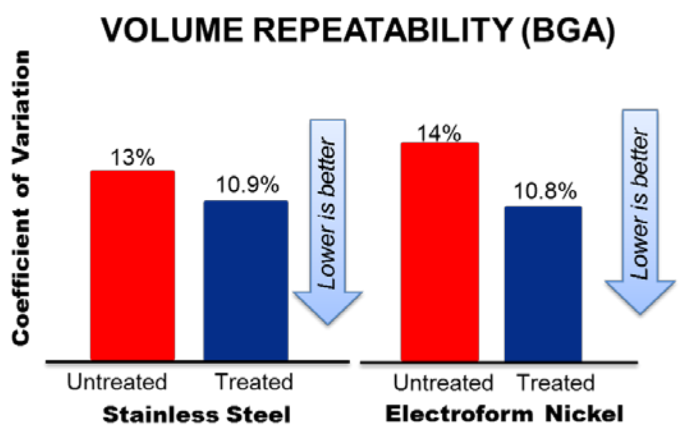
Nano Coating Print Quality Improvements

Higher Print Yields

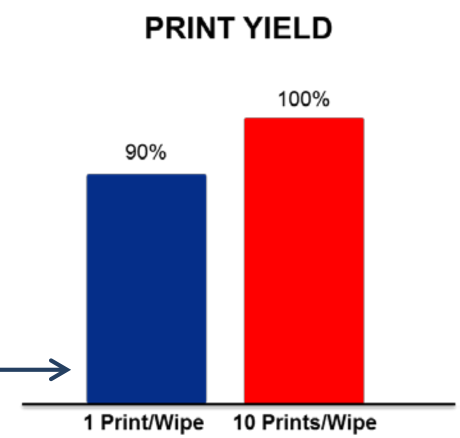


Effective on all stencil materials

Better Volume Repeatability



Reduced Under Wipe Frequency



Data from 10-print tests in large DOE
Coating is Aculon NanoClear

Improved quality at 10X wipe interval

Data Source: *Shea, C. and Whittier, R., "Fine Tuning The Stencil, Manufacturing Process and Other Stencil Printing Experiments" SMTAI 2013*

Nanocoatings Comparison

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

Summary

Stencil Technology Selection

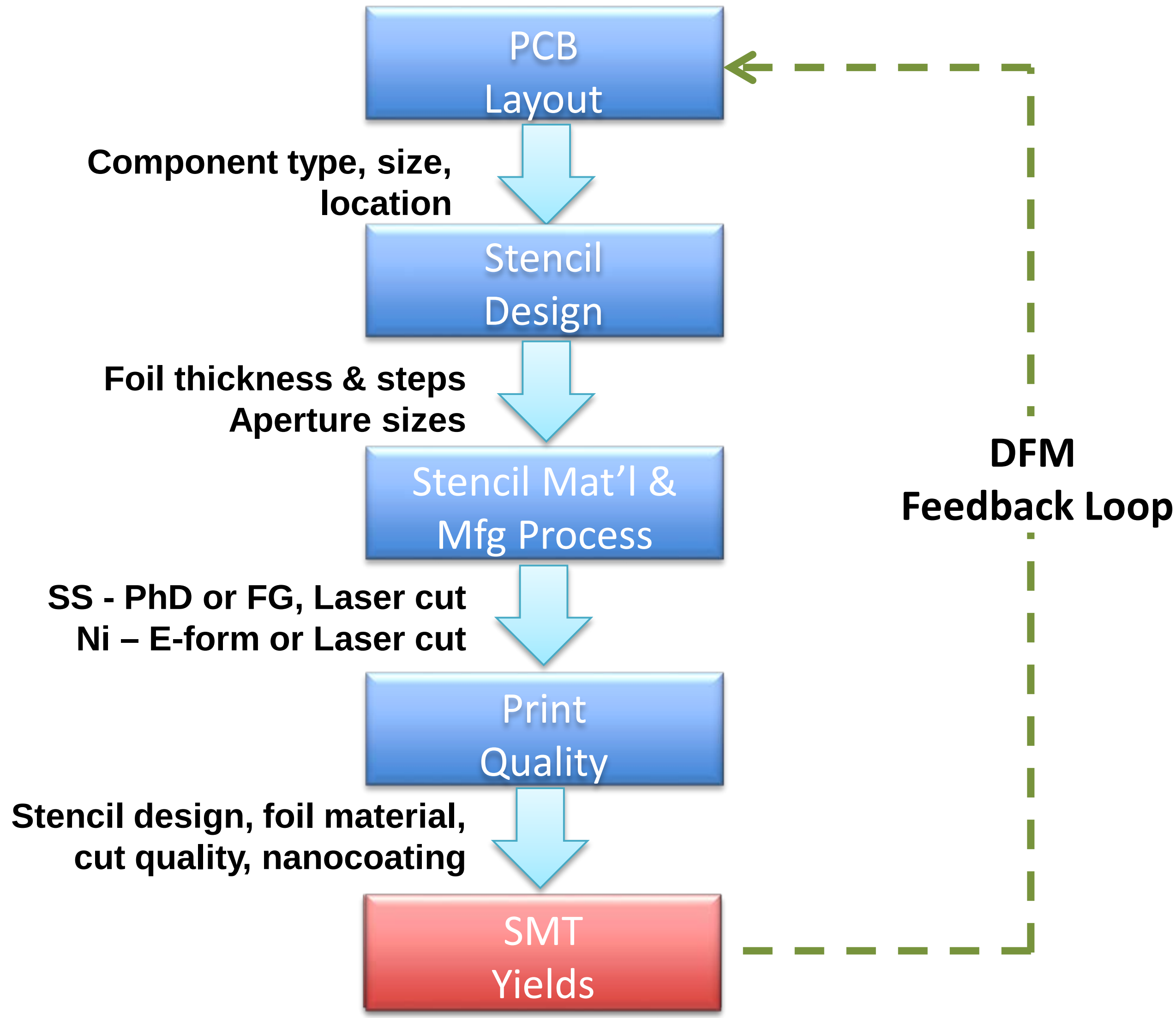
Summary (1)

- PCB layout heavily influences stencil design
  - Power components and shields require heavy paste deposits
  - QFNs and other small packages require small, precise paste deposits
  - Many tradeoffs with foil thickness, aperture size, steps, overprints, preforms, etc
- Design analysis software speeds and error-proofs calculations
  - Calculates Area Ratio & Transfer Efficiency
  - Predicts deposit volumes
  - Selects best size preforms

Summary (2)

- Laser cutting technology is better than ever
  - Machines must be tuned for good cut quality
- Alloy
  - 4 years in a row, FG has beaten every other candidate in print performance
  - Smaller grain size, smoother walls, better release, more consistent stepping
  - New SS shows excellent performance and lots of promise
- Nanocoating
  - Lowers the stencil's surface energy so it repels solder paste flux instead of attracting it
  - Improves print yields, print definition and volume repeatability

PCB Layout Drives Stencil Print Process



Acknowledgements

- Ray Whittier, Vicor
- Zina Lewis & Ben Scott, Datum Alloys
- Carol Wood, Grant Burkhalter, Stentech
- Paul Keop, ALPHA
- Mike Bixenman, Kyzen
- Edward Hughes & Eric Hanson, Aculon

Questions?

Contact:

Chrys Shea

Shea Engineering Services

609 239-2524

chrys@sheaengineering.com