



## Step 4: Printing

**By William Coleman**

*A variety of designs are available for printed circuit board (PCB) applications requiring step stencils, polymer-application stencils and mixed-component technology stencils — surface mount, through-hole and flip chip. Additionally, various stencil technologies are available to meet the increased demand for higher-density assemblies.*

In the stencil printing process, solder paste, conductive polymer or adhesive is forced through the apertures and transferred onto a PCB, which is in contact with the stencil. A printing machine, in conjunction with a metal or rubber squeegee blade or a contained paste-transfer head, is used for this purpose. There are many variables that influence this process besides the printer: its setup, the PCB, the solder paste or conductive polymer, the squeegee, and the stencil itself. They are all interrelated and influence the final print-process yields.

### **Solder Paste Printing**

There are three main performance issues:

1. Control of aperture size, including length (L), width (W) and thickness (Figure 1).

$$PAR = \frac{\text{Area of aperture}}{\text{Area of aperture walls}} = \frac{L \times W}{2 \times (L + W) \times T}$$

*Figure 1. Stencil aperture cross-section. Control of an opening's size includes thickness as well as length and width.*

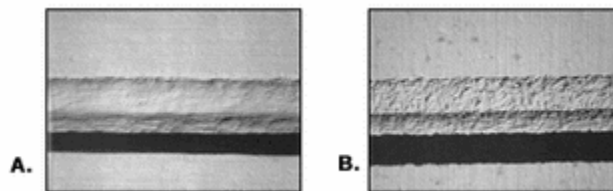
2. The stencil aperture's paste release characteristics. After the squeegee blade forces paste into the aperture and the PCB is separated from the stencil, it is desirable that all the material be released to the pads as complete bricks. However, in an alternate process, some of the paste is retained on the inside aperture walls and is referred to as percent paste release.

As the apertures become smaller to match the smaller component pads and higher component-lead densities, it is more difficult for paste to be released completely. The print-area ratio (PAR)<sup>1</sup> is used as a guideline to quantify the paste-release capability of an aperture design, which also is shown in Figure 1. A PAR above 0.66 typically is acceptable, although recent tests have shown that some types of electroformed stencils provide good paste release down to a PAR of 0.50.<sup>2</sup> The condition of a stencil aperture's walls also plays a major role in percent paste release. Smooth aperture walls with a slight trapezoidal taper away from the squeegee side increase the efficiency of paste release.

3. Positional accuracy of the aperture locations (compared to pad sites) must be well controlled to avoid misregistration of the paste bricks. PCB stretch or shrink also is a factor in solder brick-to-pad misregistration.

## Stencil Technology

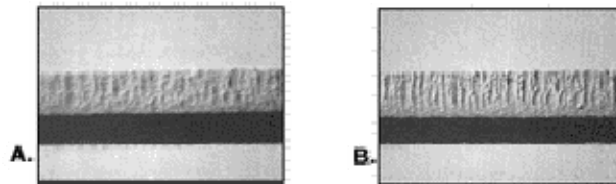
**Chemically Etched Stencils.** This stencil type has been manufactured since the 1970s. In this process, a metal foil is laminated on both sides with photoresist on which a latent image is exposed via a pin-registered phototool. After developing the resist, the apertures are etched from both sides using a dual-sided, spray etching process. Figure 2 shows a typical chemically etched aperture along with the so-called knife-edge effect. Large apertures have different etch rates than small openings because it is more difficult for the etchant to reach the inside.



*Figure 2. A chemically etched aperture wall without electropolish (a) is compared to one with electropolish (b). Post-processing technologies of electropolishing and plating will assist solder paste release.*

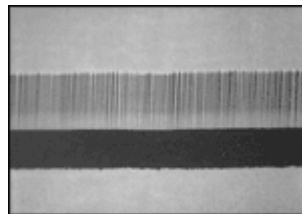
Generally, it is good practice to keep the PAR above 0.90 for chemically etched apertures. A practical example of a 0.90 PAR is an aperture 0.013 (W) x 0.060" (L) in a 0.006" thick stencil for a 0.025" pitch quad flat pack (QFP). Post-processing technologies of electropolishing and nickel plating do a good job of making the aperture walls smoother as seen in Figure 2. Trapezoidal chemically etched apertures also are possible by adjusting the phototool to have a smaller opening on the stencil's squeegee side. Because electropolishing removes material from the aperture wall and nickel plating adds material, special adjustments must be performed to control the final size. Additionally, small apertures have different electropolishing and nickel-plating rates compared to those of the larger openings, making aperture size-control difficult.

**Laser Cut.** In this process, a stencil foil is held taut while a laser beam, having a spot size of 0.001 to 0.002", traces and cuts the aperture. Because laser travel is computer numerically controlled (CNC), aperture-size tolerance and positional-accuracy tolerances are very tight; typically  $\pm 0.0003$  and  $\pm 0.0005$ ", respectively. The opening for a laser-cut aperture typically is 0.0005 to 0.001" larger on the entry side of the foil compared to the exit side, depending on laser setup and type. This provides a natural trapezoidal effect. As with chemically etched apertures, the post-processing technologies of electropolishing or nickel plating make their walls smoother, as seen in Figure 3. Also, as in the chemically etched case, small laser cuts have different electropolishing and nickel-plating rates compared to those of larger apertures, making size control difficult.



*Figure 3. Examples of laser-cut aperture walls, comparing a "rough" cut (a) with one electropolished and nickel plated (b).*

**Electroformed (E-FAB/AMTX)\*.** While the previous two stencil technologies are subtractive processes — the material is removed from the foil to form the apertures — the E-FAB process is an additive procedure. Photoresist layers are laminated onto a copper mandrel and a latent image is exposed through a phototool. After developing, photoresist pillars, which are taller than the desired stencil thickness, remain on the mandrel at any point an aperture is to be formed. The mandrel then is placed in a plating tank in which nickel is built around the photoresist pillar until the desired stencil thickness is achieved. Next, the nickel foil is peeled off the mandrel, leaving the stencil free and clear.



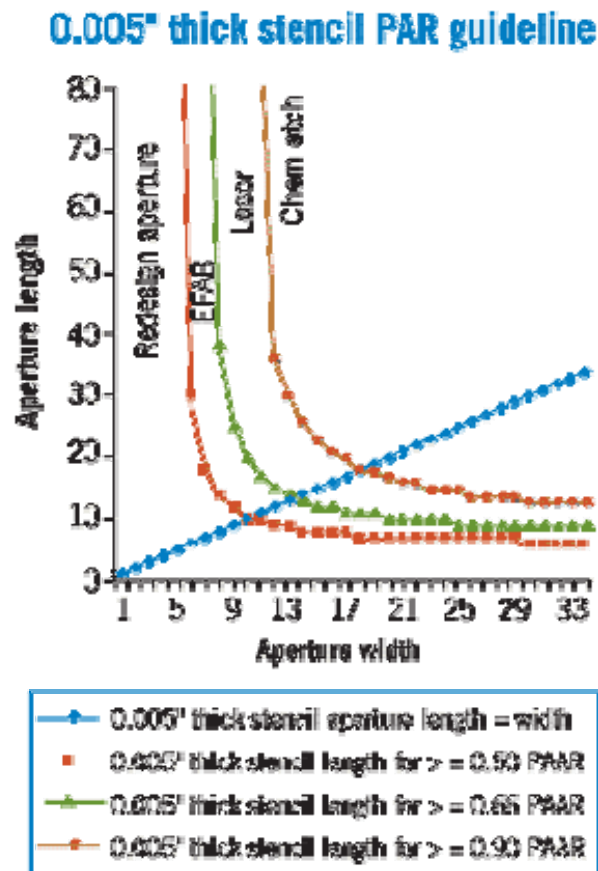
*Figure 4. E-FAB aperture side walls exhibit a smoothness and a "natural" trapezoidal character for full solder paste release without extra*

[processing.](#)

The E-FAB side walls are very smooth, as seen in Figure 4; no additional processing such as electropolishing or nickel plating is required and tight aperture tolerances of  $\pm 0.025''$  are the result. With proper processing, three additional benefits are inherent in the E-FAB stencil: The stencil strength is greater than that of a full-hard stainless-steel stencil; the apertures are trapezoidal with the contact side about  $0.0005''$  larger than those on the squeegee side; and a "lip" on the edge of the aperture assures that it is always the highest point on the stencil surface so that the stencil will "gasket" well to the PCB pads.

### Print Performance

A recent study comparing the print performance of laser-cut stencils with that of the E-FAB stencil demonstrated that the latter provides good paste release for PARs as low as 0.50.<sup>2</sup> This is significant when considering high-density packages such as microBGAs and  $0.016''$  pitch QFPs as well as 0201 chips. From this study, a Stencil Technology Selection Guideline has been proposed.



*Figure 5. With a 0.005" thick stencil, the PAR is given at the intersection of aperture width and length.*

Figure 5 shows PARs, held as a constant, vs. aperture width on one axis and aperture length on the other for a 0.005" thick stencil. The intersection gives the PAR. For areas below the 0.50 PAR curve, the stencil needs to be redesigned, making the stencil thinner or the aperture larger.

E-FAB is recommended for PARs greater than 0.50; laser-cut stencils are acceptable for PARs greater than 0.66. Chemically etched stencils generally are acceptable if the PAR is above 0.90. However, it should be noted that all laser stencils are not equivalent in performance.

Electropolishing and nickel plating play important roles in stencil print performance. Care should be exercised in selecting a laser-cut stencil so that good wall quality is assured. The table shows a stencil technology selection guideline for various surface mount device (SMD) types.

### Design and Applications

**Aperture Design.** The stencil aperture should be smaller than its intended pad on the PCB. The typical reduction is 0.001" per side. (IPC 7525 shows stencil aperture recommendations for most common SMDs.) This reduction is necessary to assure that the stencil is flush to the pads during the paste-transfer process.

**Chip Design (to eliminate solder balls).** If the stencil aperture is 1:1 with the pad when using no-clean solder paste, free solder balls may result. This is caused by excess solder paste under the chip. The "home-plate" aperture design with the tip pointing toward the chip's center is effective in eliminating this defect.

**Step Stencils.** There are several applications in which step stencils are useful. Steps typically are chemically etched into the stencil foil before the apertures either are chemically etched or laser-cut. Ceramic BGAs (CBGA) generally feature solder bumps that do not melt during the reflow process and, thus, require higher paste heights to avoid opens. A step-up stencil, which is 0.008" thick in the CBGA area and 0.005" thick in the remaining area, is used to avoid the potential open problem. Any protrusion above the PCB pad height will result in poor pad-to-stencil gasketing and may result in paste smearing under the stencil and on the board, risking an electrical short. A relief pocket-step stencil in the area of the protrusions will solve this problem. Examples of protrusions on the board include raised vias, board-identification labels, raised heat sinks and additive traces.

TABLE		
Stencil Technology Selection Guideline		
Component	PAR*	Stencil Recommendation
BGA 20	0.5 to 0.7	E-FAB
BGA 30	0.6 to 0.8	E-FAB, L-EP/NP
BGA 50	1.6 to 1.6	All four stencils
0.006" QFP**	0.52 to 0.56	E-FAB
0.008" QFP**	0.67 to 0.74	E-FAB, L-EP/NP
0.010" QFP**	0.80 to 0.91	E-FAB, L-EP/NP, L-EP
0.012" QFP**	0.92 to 1.07	All four stencils
0402 chips	0.80 to 1.00	E-FAB, L-EP/NP, L-EP
0201 chips	0.60 to 0.75	E-FAB
Flip Chip	0.50 to 0.65	E-FAB

\*For 0.005" thick stencil \*\*Aperture width

Step stencils also are required in a two-print process. In this application, a deep-relief step pocket is required on the board side of the stencil at any point where either solder paste or flux was printed during the first operation. Examples of applications requiring relief pockets for two-print

stencils include intrusive reflow<sup>3</sup> for reflowing SMDs together with through-hole devices; flip chip assembly where flux is printed first and paste second (as both devices are reflowed simultaneously); and applying both paste and adhesives for chips, which requires a pocket for solder-paste relief clearance during the adhesive-printing operation.

**Adhesive Printing.** There are two options for applying adhesives for chips as well as leaded devices before wave soldering: dispensing and printing. Stencil printing of adhesives is a parallel process wherein all the bricks are printed in a single stroke. A robust process, this offers excellent material height and volume control.<sup>4</sup> A laser-cut metal stencil, 0.006 to 0.008" thick, is used if only chips are present. When both chips and leaded devices are present, two stencil technologies are available for printing adhesives: plastic stencils (pump printing)<sup>5</sup> and thick metal stencils (adhesive reservoir printing).<sup>6</sup> Both technologies take advantage of the fact that material height may be controlled by aperture size. In this manner, heights of 0.006" can be achieved for chips (with 0.004" stand-offs) and heights of 0.030" for leaded devices (with stand-offs of 0.025"). The reservoir-type stencil for this application is 0.050" thick with a 0.040" thick adhesive reservoir positioned above the apertures. The adhesive apertures are laser-cut in a 0.010" thick foil with 0.020" wide apertures for the chips and 0.050" wide apertures for the leaded devices. In some applications, clinched leads for through-hole devices protrude above the PCB. For these, a deep-relief pocket is designed into the stencil to provide clearance.

### Summary

Stencil technology and aperture design play important roles in print performance. It is critical to pay close attention to the PAR of all apertures in a stencil. When the PAR is below 0.75 and approaches 0.66, it is necessary to select a laser-cut stencil with smooth side walls, preferably one with electropolishing or electropolishing with nickel plating. When the PAR is below 0.66 (but above 0.50), electroformed (E-FAB) stencils are recommended. Devices such as 0.016 pitch QFPs, 0.020 pitch microBGAs and 0201 chips normally fall into this PAR range. When the PAR is below 0.50, however, it is prudent to redesign the aperture/stencil.

There are numerous stencil designs available for various printing applications, including no-clean paste-aperture modifications to eliminate solder balls for chips; step stencil designs for CBGAs and through-hole components; relief pockets on the board side of the stencil for many applications having protrusions that rise above the PCB pads; and stencils for printing adhesives for chips and leaded devices with or without clinched through-hole leads.

- E-FAB is a patented process of Xerox Corp. and licensed to Photo Stencil.

### REFERENCES

1 W. Coleman "Stencil Design for Advanced Packages," SMT Magazine, June 1996.

2 W. Coleman, "Stencil Print Performance Studies," 2001 Conference Proceedings, SMTA International, pp. 94-101.

3 W. Coleman, D. Jean, J. Bradbury, "Stencil Design for Mixed-technology Through-hole/SMT Placement and Reflow," SMT Magazine, July 2000.

4 R Lathrop, "Optimizing the Print Performance of SMT Adhesives," Proceedings of the Technical Conference, APEX 2002, pp. S27-2-1 to S27-2-9.

5 A. Hobby, J. Hingtgen, "How to Print Thirty Million Glue Dots Per Hour (And Then Print Three Dimensional Solder Paste)," Proceedings of the Technical Conference, APEX 2002, pp. S27-4-1 to S27-4-7.

6 W. Coleman, A. Wadhwa, "Metal Stencils for Adhesive Printing," Proceedings of the Technical Conference, APEX 2002, pp. S27-1-1 to S27-1-6.

**William Coleman** may be contacted at Photo Stencil, 4725 Centennial Blvd., Colorado Springs CO 80919; (719) 535-8528; E-mail: [bcoleman@photostencil.com](mailto:bcoleman@photostencil.com).