



# STEP STENCILS: A Powerful Tool for Special Applications

by **William E. Coleman**  
PHOTO STENCIL

## IN SUMMARY

Mixed-technology applications of solder paste printing for through-hole/SMT, as well as solder paste/flux printing for flip-chip/SMT, require special step stencil designs. Several applications and the stencil design to achieve a solution are discussed in detail in this paper.

### Abstract

In the early days of SMT assembly, step stencils were used to reduce the stencil thickness for 25-mil-pitch leaded device apertures. However, as SMT requirements became more complex and, consequently, more demanding, so did the requirements for complex step stencils. Mixed-technology applications of solder paste printing for through-hole/SMT, as well as solder paste/

flux printing for flip-chip/SMT, require special step stencil designs. Thick metal stencils that have both relief etch pockets and reservoir-step pockets are very useful for glue and paste reservoir printing.

Electroform and laser-cut step-up stencils for ceramic BGAs and RF shields are a good solution to achieve additional solder paste height on the pads of these components. Special 3-D electroform stencils are an excellent solution for odd PCBs having some raised areas on the board (up to 0.090" high). The two-print stencil printing process is useful in dealing with the problem of small and large components coexisting on the same PCB. These applications and the stencil design to achieve a solution are discussed in detail in this paper.

### Introduction

Eight different applications for step stencils will be described. These include the following:

1. Step-up stencils for ceramic BGAs.
2. Step-up stencils for intrusive reflow of through-hole components.

“ *The two-print stencil printing process is useful in dealing with the problem of small and large components coexisting on the same PCB.* ”

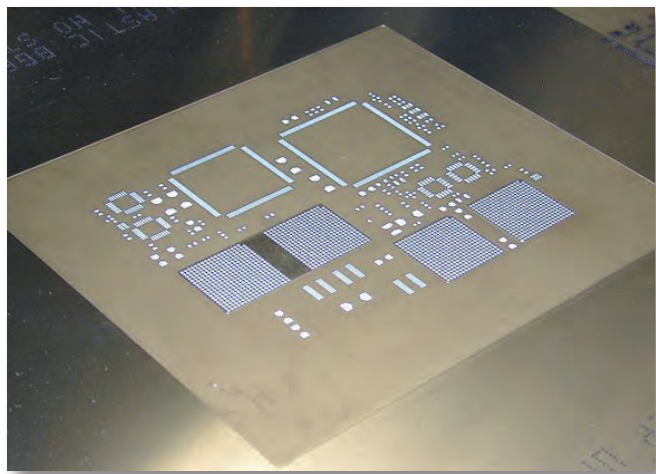
3. Step stencils with a relief etch pocket on the contact side (board side) for:
  - a. Raised vias
  - b. Bar codes
  - c. Board hold-down clips
  - d. Additive traces
4. Step stencils for two-print applications for mixed technology:
  - a. SMT/through-hole
  - b. Flip-chip/SMT
5. Thick stencils with etched reservoir pockets for printing glue for a variety of components with different stand-off heights.
6. Two-print stencil, with etched relief pockets, for printing glue for component attachment after SMT solder paste has been printed.
7. Three dimensional electroform stencils with a formed relief pocket for a high flex connector that connects two PCBs.
8. Two-print stencils for printing small devices like 0.3-mm-pitch uBGAs and 01005s at the same time as printing large devices like RF shields, SMT connectors, QFPs and other chip components.

## Step Stencil Applications and Solutions

### 1. Ceramic BGAs

Ceramic BGAs present a challenge to the SMT assembly process. Since the high melting temperature prevents the solder balls from melting at normal reflow temperatures, any slight coplanarity problem can result in an open contact to the CBGA balls. It is desirable to print higher solder bricks on the CBGA pads to prevent this problem. Normally, a solder paste brick height of 7 to 8 mils is desirable. On the other hand, SMT components like 0.5-mm-pitch QFPs, 0402 chip components and R-packs will not tolerate an 8-mil-thick stencil. Their aperture sizes are too small to achieve good paste release with a stencil this thick. Therefore, a step-up stencil is required for this application.

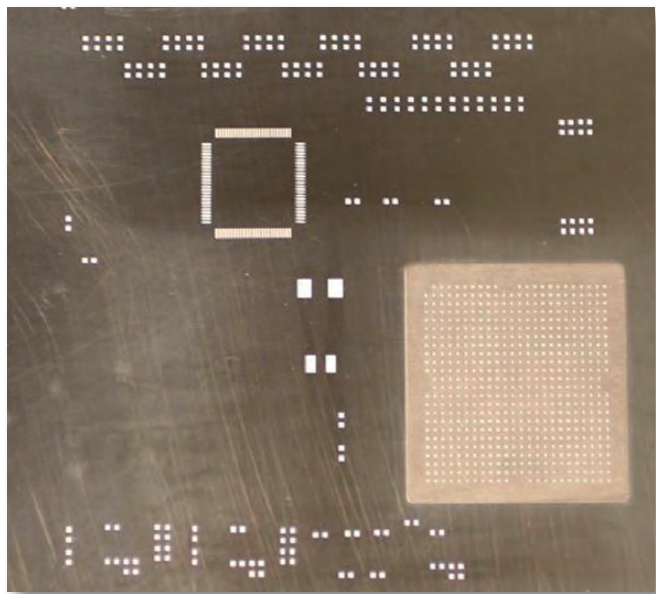
An example of this stencil is shown in Figure 1. This stencil starts out as an 8-mil-thick foil and the foil is etched back to 5 mils



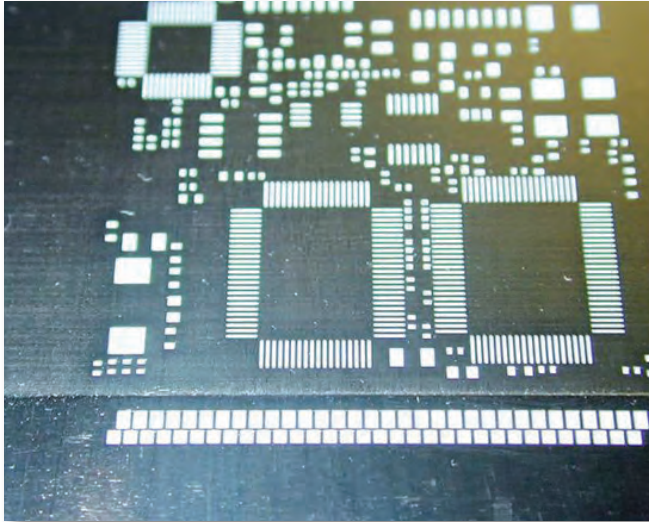
**Figure 1:** Step-up laser-cut stencil for CBGAs.

in all areas except in the CBGA area. The foil then has all the apertures, including the CBGA apertures, laser-cut into the foil. Electropolish and nickel plating are recommended for this stencil to achieve good paste release for the small apertures.

In cases where 0201 chip components and 0.5 mm pitch uBGAs are present, a stepped electroform stencil is recommended to achieve good paste release for these small apertures. A picture of a step-up electroform stencil is shown in Figure 2. The stencil is made by plating up to 5 mils in all areas and then continuing to plate up to 8 mils in the CBGA areas. Customers usually prefer the step on



**Figure 2:** Step-up electroform stencil for CBGA.



**Figure 3:** Step-up laser-cut stencil for through-hole connector.

the squeegee side of the stencil, and this is the case for the prior two examples. A general rule for spacing between the step ledge and first aperture in the lower thickness area (keep-out design) is 0.035 to 0.050" per 0.001" of step (IPC 7525 Stencil Design Guidelines). User feedback indicates that metal squeegee blades work fine with step-up stencils as long as these keep-out designs are followed.

## **2. Step-Up Stencil for Through-Hole Intrusive Reflow**

An increasing trend has emerged to reflow through-hole components, rather than using wave soldering, along with the SMT components [1]. To achieve this, solder paste is printed on, in and around the through-hole hole and pad (annular ring). Three stencil alternatives can achieve sufficient solder paste for this application:

1. Overprint the hole/annular ring with an oversized stencil aperture.
2. Step-up and overprint the hole/annular ring with an oversized stencil aperture.
3. Two-print stencil where the second print stencil is very thick and provides more solder paste for the requirement. (This is described in the "Step Stencil for Two-Print Operation for Mixed Technology" section below.)

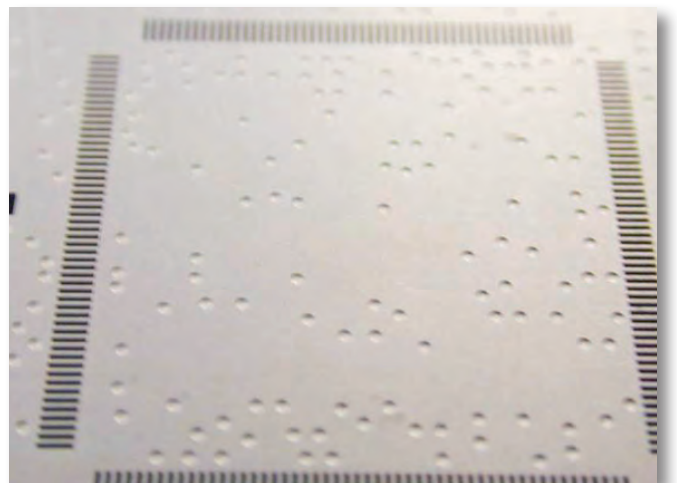
Figure 3 shows a step-up stencil with the step on the squeegee side for a through-hole edge connector. The stencil is 10 mils thick in the through-hole area and 5 mils thick elsewhere. The squeegee stroke is parallel to the step-down ledge so the entire length of the blade drops down to 5 mils during the squeegee stroke.

## **3. Relief Step Stencil with Relief Etch Pockets on Contact Side of Stencil**

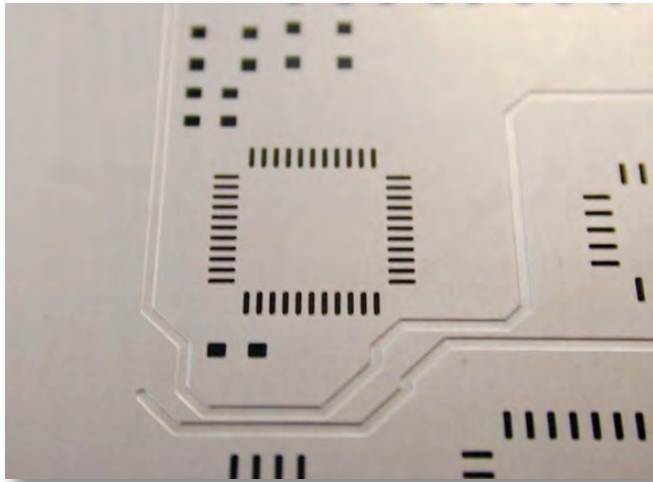
**1. Raised via pads on the PCB.** If a board has raised via pads, they will prevent the stencil from gasketing to the PCB. To achieve good stencil/board contact a relief pocket is etched on the contact side of the stencil wherever there is a raised via. Typically, the relief pocket depth is half of the stencil thickness, which is usually enough to clear the raised via. A picture of this stencil is seen in Figure 4.

**2. Bar code on the PCB.** Many PCBs have bar code identifiers attached to the board surface. If it gets too close to board pads it can prevent the stencil from gasketing to these pads during printing. A simple solution is to use a stencil that has a relief pocket etched in the area of the bar code. This allows the stencil to sit flat on the PCB during printing.

**3. Board hold-down clips.** Some stencil printers have edge clips which hold the PCB down during the print operation. If there are component pads close to the edge of the board, the stencil may not be able to gasket to



**Figure 4:** Raised via pad relief step stencil.



**Figure 5:** Additive trace relief pocket stencil.

the board on the edge. Again, a stencil solution is to provide a relief etch pocket in the area of the hold-down clips.

**4. PCB with additive traces.** Additive traces are added to the surface of the PCB to correct a design problem by altering the lead wiring. Unfortunately, this trace adds height to the board surface. A stencil solution is to provide a relief pocket around the additive trace. An example of this stencil is shown in Figure 5.

#### 4. Step Stencil for Two-Print Operation for Mixed Technology

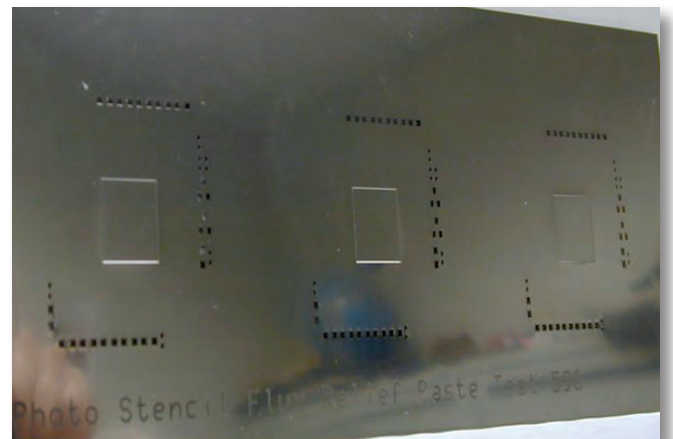
**1. SMT/through-hole mixed technologies.** When overprint (oversized apertures for the through-hole) with a normal step-up stencil does not provide enough solder paste volume for proper intrusive reflow a thicker stencil must be used. However, the normal SMT components will not tolerate a thick stencil (15 to 20 mils thick) and a step down (20 mils down to 5 mils) is impractical. The two-print stencil operation is a viable solution. Consider a PCB that has a fully-populated pin grid array (PGA). Overprint is limited because of the fully-populated geometry. The only alternative to providing more solder paste is to make the stencil thicker. In this case, a stencil 15 mils thick was required. In the two-print operation all the surface mount solder paste is printed with a normal SMT stencil of 5 or 6 mils thick. The second print stencil is 15 mils thick with

relief etch pockets etched on the contact side of the stencil any place that SMT solder paste was printed during the first print. As a rule of thumb, the relief pocket depth should be 4 mils deeper than the SMT first print stencil [2].

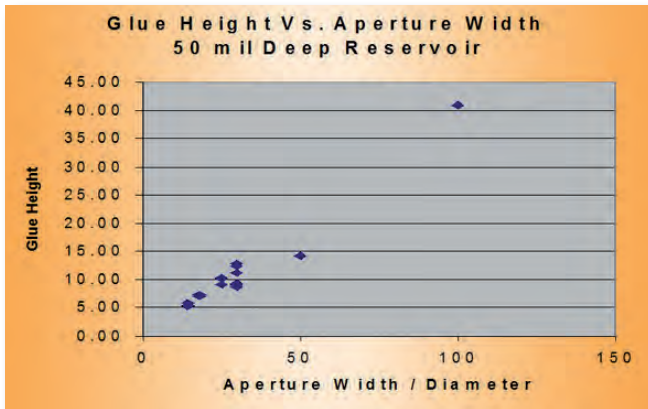
**2. Flip-chip/SMT mixed technologies.** There are applications when it is desirable to print flux or solder paste for a flip-chip component and solder paste for normal SMT devices. Both are then placed and run through the reflow cycle. Normally, the stencil thickness for the flip-chip printing is 1 to 2 mils thick; much too thin for normal SMT printing. Two-print stencils are ideal for this application. A thin (1- to 2-mil-thick) electroform is used to print either flux or solder paste on the flip-chip pad sites on the PCB. Then a SMT stencil (5 mils thick) is used to print solder paste on all SMT pads. This stencil has relief pockets formed on the contact side anywhere flip-chip flux or paste was previously printed. Figure 6 shows a 3-D AMTX electroform stencil that is 5 mils thick with 3-mil relief pockets formed on the contact side which is ideal for this application. This stencil has a formed step-up relief pocket. The height of the stencil in the flip-chip area is 8 mils (5 mil stencil thickness and 3 mil relief pocket).

#### 5. Thick Stencil with Deep-Etched Reservoirs for Printing Glue for Component Attachment

Advantages exist to printing glue rather than dispensing it. Set-up time is reduced and



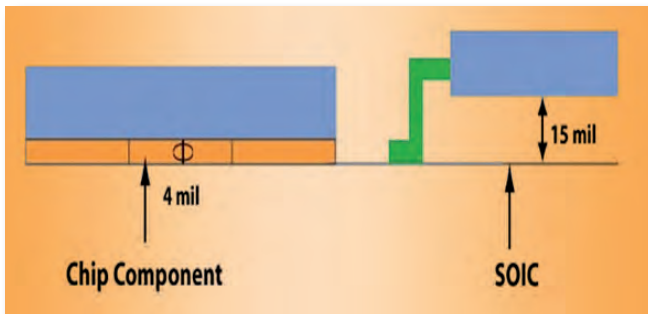
**Figure 6:** 3-D electroform stencil for paste printing with flux relief.



**Figure 7:** Glue height versus aperture size.

process time is shortened since glue bricks are deposited in parallel rather than one dot at a time. Glue does not print the same way solder paste does and it is acceptable, as well as useful, to leave glue remaining inside the stencil aperture. As a benefit, this allows different heights of glue to be printed using the same stencil thickness. Large apertures will release all the glue while small apertures will release only a portion of the glue. Glue height versus aperture size for a 50-mil-deep reservoir pocket is shown in Figure 7. This is very useful when the PCB contains components with different stand-off heights.

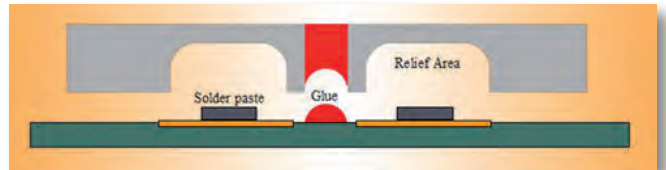
An example of an application is a PCB with chip components with a stand-off of 4 mils and an SOIC with a stand-off of 15 mils. This is shown as a schematic in Figure 8. By using a 15-mil-thick stencil the aperture sizes can be adjusted to provide 6 mils of glue height for the chip components and 15 mils high for the SOIC component.



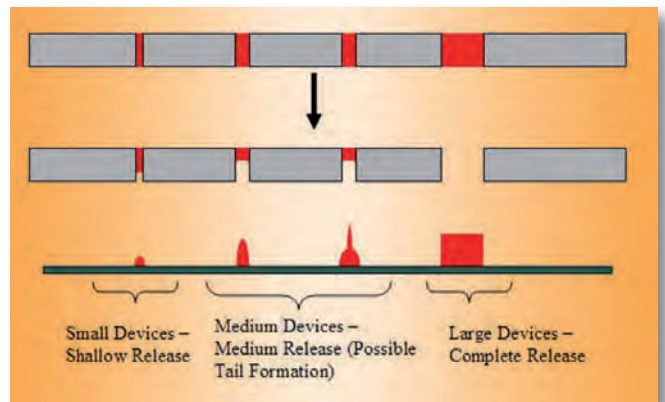
**Figure 8:** Chip component and SOIC with different stand-offs.

## 6. Two-Print Stencil for Printing Glue After Paste

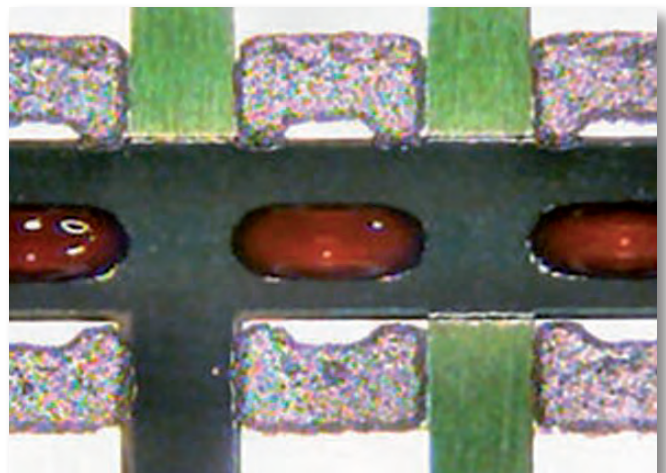
It was reported that printing glue compared to dispensing glue resulted in lower defect rates for large and complex server type PCBs [3]. The process used is to print solder paste first for the back side components then print glue, with a 20-mil-thick stencil with 15-mil-deep relief pockets for the back side component solder bricks. The next three figures are courtesy of Mike Kochanowski: Figure 9 shows a schematic



**Figure 9:** Glue two-print stencil 20 mil thick with 15 mil relief pockets.



**Figure 10:** Different glue heights for different aperture sizes.

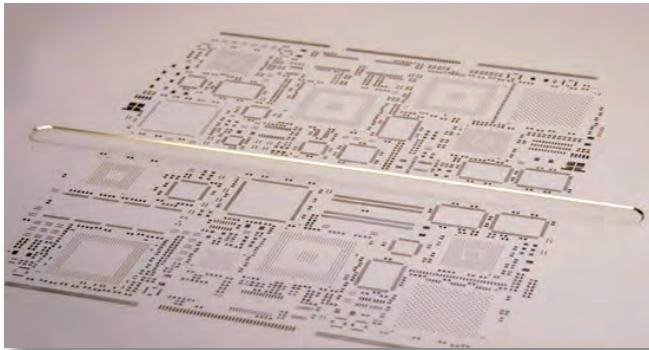


**Figure 11:** Glue and paste bricks for 0805 device.

cross section of the glue two-print stencil; Figure 10 illustrates the fact that some glue remains in the glue aperture depending on the size of the glue aperture; and glue and paste for a 0805 chip component is shown in Figure 11.

**7. 3-D AMTX Electroform Stencil for Special PCBs Connected with High-Profile Flex Connector**

Two PCBs are connected with a flexible connector, which rises up 90 mils above the board surface. The challenge was to be able to print solder paste on the SMT pads with the flexible connector obstructing a normal stencil from making contact with the board. The solution to this problem is a 3-D Electroform stencil, which is 5 mils thick, but has a 95-mil-high relief pocket formed in the stencil. Figure 12 shows a picture of the stencil. Figure 13 shows the squeegee side of the stencil along with the special E-Blade that has a notch formed in the blade for clearance of the relief pocket.



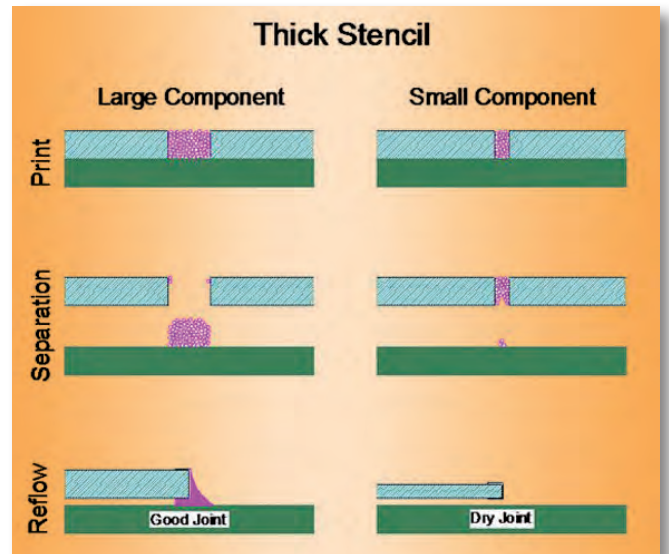
**Figure 12:** 3-D electroform stencil with 90-mil-high relief pocket.



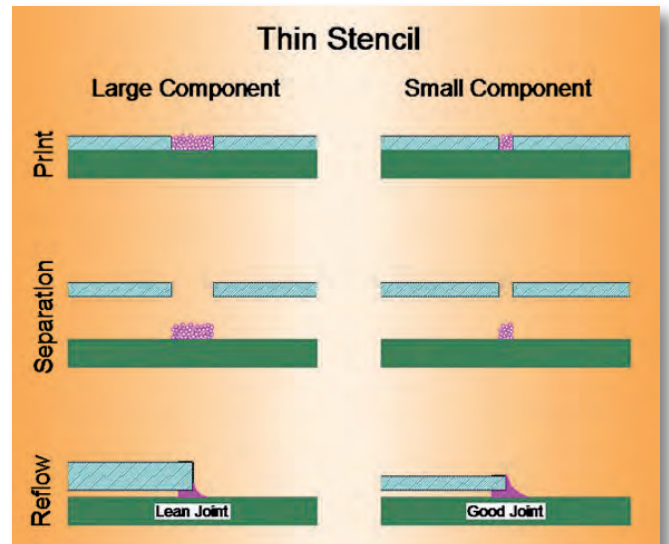
**Figure 13:** 3-D electroform stencil with notched e-blade.

**8. Two-Print Stencils When Very Small and Large Devices are Present on PCB**

A problem exists when printing small devices like 0.3-mm-pitch uBGAs and 01005s at the same time as printing large devices such as RF shields, SMT connectors, QFPs and other chip components [4]. This problem is illustrated in Figures 14 and 15. If a thick stencil is used, good paste release is realized for the large apertures resulting in good solder fillets. However, small apertures have incomplete paste transfer because of low area ratios, resulting in dry solder joints for the small components.



**Figure 14:** Print sequence for thick stencil.



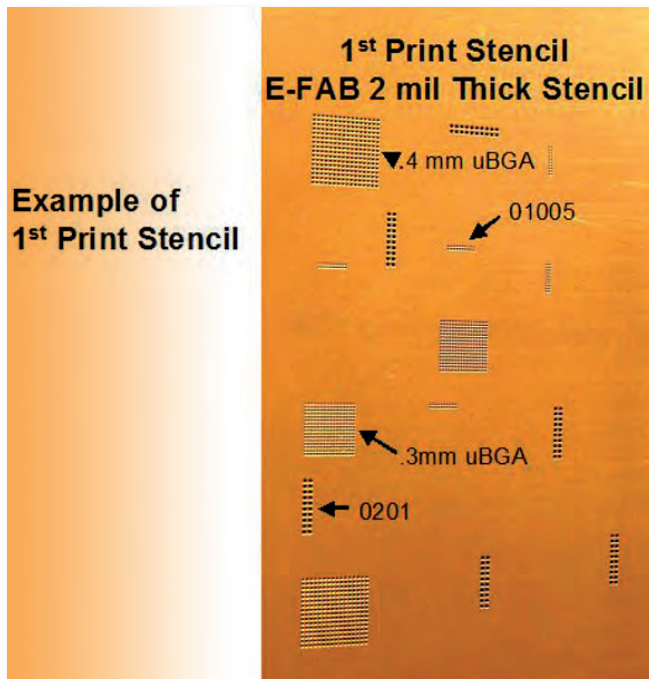
**Figure 15:** Print sequence for thin stencil.

Component and typical Aperture Size	Stencil Thickness →					
	2 mil 50u	2.5 mil 62u	3 mil 75u	3.5 mil 87u	4 mil 100u	5 mil 125u
<b>01005</b>						
6 mil (150u)	0.75	0.60	0.50	0.43	0.38	0.30
7 mil (175u)	0.88	0.70	0.58	0.50	0.44	0.35
<b>.4mm CSP</b>						
6 mil (150u)	0.75	0.60	0.50	0.43	0.38	0.30
7 mil (175u)	0.88	0.70	0.58	0.50	0.44	0.35
8 mil (200u)	1.00	0.80	0.67	0.57	0.50	0.40

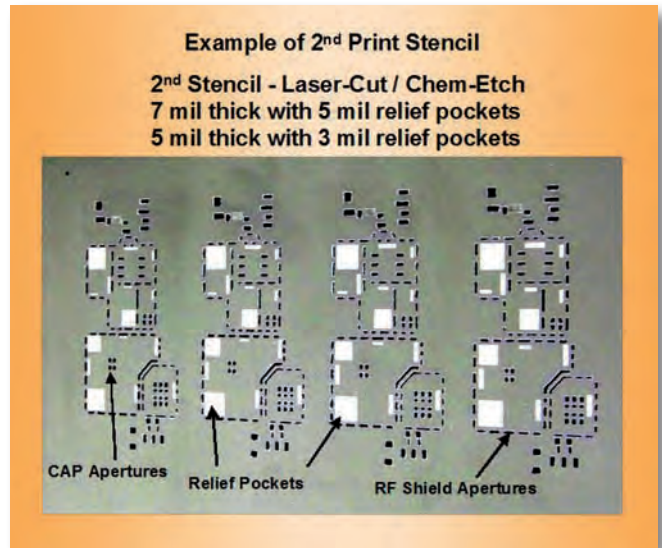
Green = OK Orange = Warning Red = Stop

**Figure 16:** Area ratio matrix.

On the other hand, a thin stencil provides good paste release for both large and small components, but too little paste for the large components results in lean solder joints. The area ratio matrix, shown in Figure 16, illustrates acceptable stencil thickness and aperture sizes for 01005 and 0.4 mm uBGA devices. One possible solution to this problem is a two-print stencil process, where the small component's solder paste bricks are printed first with a thin stencil.



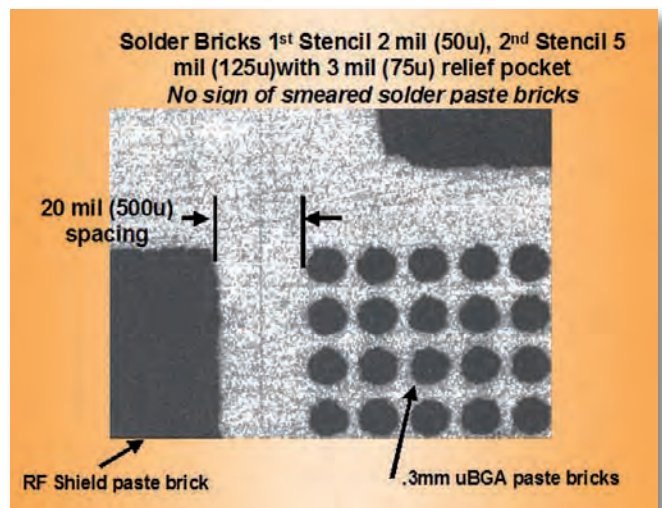
**Figure 17:** First print electroform stencil.



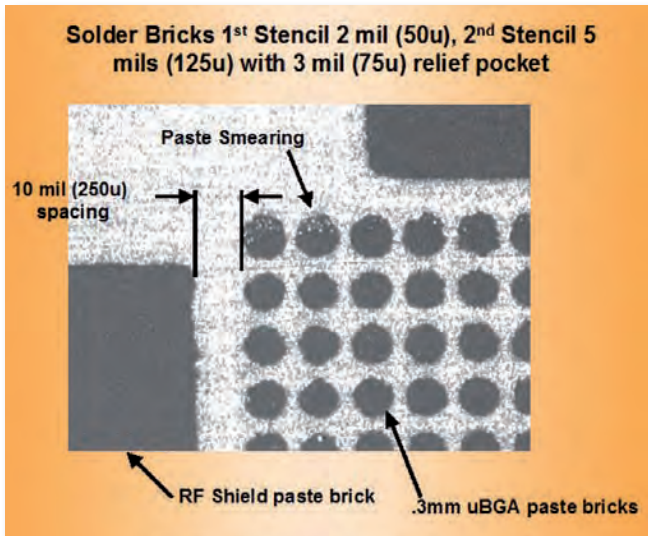
**Figure 18:** Second print chem-etch relief/laser-cut apertures.

Next, a second in-line screen printer prints solder paste for the large components. This stencil has relief pockets etched on the board side of the stencil anywhere solder paste was printed with the first stencil. This process is illustrated for a cell phone application in Figures 17 and 18. Figure 17 shows a 2-mil-thick electroformed stencil for printing 0.3 and 0.4 mm uBGAs and 01005 and 0201 chip components. Figure 18 shows the second print stencil with relief pockets for the first print stencil solder paste bricks.

The design question that needs to be



**Figure 19:** Solder bricks showing no smearing.



**Figure 20:** Solder bricks showing smearing.

answered is: What is the smallest spacing between apertures in the first and second print? Figure 19 shows that 20-mil spacing between apertures with a 3-mil-deep relief pocket works very well with no smearing of the solder paste bricks, in this case the 0.3mm uBGA bricks. Figure 20 shows smearing when the spacing is 10 mils. This suggests that somewhere around 15 mils between apertures is a workable spacing for this process.

**Conclusion**

Step stencil technology offers unique solutions for various printing applications. Step-up stencils offer printing solutions for CBGA and through-hole paste volume/height requirements. Step-up electroform stencils not only offer solutions for higher solder volume/height for the above applications, but also provide excellent paste transfer for 0.5 mm uBGAs and 0201 devices. Two-print stencils, including a thick second print stencil with deep relief step pockets, offer solutions for both glue-attach printing and intrusive reflow printing. Normal SMT stencils 5 to 6 mils thick with relief step pockets on the contact side of the stencil offer solutions when there are raised areas on the PCB that would prevent a normal stencil from gasketing to the board.

3-D AMTX electroform stencils provide an excellent solution for two-print stencil

operations when printing flux/paste for flip-chip and paste for SMT in a mixed technology application flip-chip/SMT. 3-D AMTX electroform stencils with formed-relief pockets are excellent for printing on boards when high protrusions exist on the board surface. Two-print stencils are also a viable solution for printing very small components along with large components on PCBs as in cell phone applications. **SMT**

**References**

1. William Coleman, Denis Jean and Julie Bradbury, "Stencil Design for Mixed Technology Through-Hole/SMT Placement and Reflow," *SMT Magazine*, July, 2000.
2. William Coleman and Wadhwa Ashok, "Metal Stencils for Adhesive Printing," Proceedings of APEX Conference, 2002.
3. Michel Kochanowski, "Combined Printing Solder Paste and Printed Adhesives Process for Devices Going through Wave Solder," Proceedings of the SMTAI Conference, September, 2005.
4. William Coleman and Chris Anglin, "Stencil Options for Printing Paste for .3mm CSPs and 01005 Components," Proceedings of APEX Conference, 2010.



Dr. William E. Coleman earned his Ph.D. in Physics from West Virginia University. His early career was spent with NCR developing memory and visual display devices. Dr. Coleman has spent the past 23 years at

Photo Stencil as Vice President of Technology working closely with customers to understand their SMT printing requirements. At Photo Stencil he has developed several innovative solutions for these requirements. Dr. Coleman has published over 20 papers in this field and is presently Co-Chair of IPC 5-21e committee, which produced IPC 7525 "Stencil Design Guidelines." He is on the Editorial Advisory Board for *SMT Magazine* and the Advisory Board for West Virginia University.