Stencil Design for Mixed-technology Placement & Reflow

By William E. Coleman, Denis Jean and Julie Bradbury

Stencil design requirements for printing solder paste around and in through-hole pads/openings is reviewed.

Reviewing stencil design requirements for mixed-technology boards is essential because full implementation allows through-hole components and surface mount devices (SMD) placement and the subsequent reflow of both simultaneously. This eliminates the need to wave or hand solder through-hole components.

Material type, pin type, lead length and standoff height of the through-hole components will be reviewed. Three stencil designs will be considered: single-thickness stencils with oversized stencil apertures for overprinting solder paste in the through-hole pad areas, step stencils with oversized stencil apertures for overprinting solder paste in the through-hole pad areas and thick stencils (0.015" to 0.025" thick) for printing solder paste in the through-hole pad areas. The thick stencil is the second stencil in the two-print stencil process.

The selection of stencil design depends on several factors. These factors include: total solder volume requirement to fill the plated through-hole around the lead to form a proper solder fillet; board thickness; lead diameter; plated through-hole size; position of the through-hole component in the board layout; component pitch; soldermask surface energy; solder paste activity level; and the metal solderability.

Background

Although the conversion from through-hole devices to SMDs has been dramatic, many printed circuit boards (PCB) are still populated with both types of technology. In the majority of electronic assemblies, there will be a few through-hole components on the board. In applications where robustness or power are required, the connector will continue to exist in a through-hole configuration. There is great interest in placing and reflowing the through-hole components along with the SMDs. This presents specific challenges to the solder paste printing process. The stencil must provide enough solder paste volume to fill the hole and provide a good solder joint. The through-hole component must be able to withstand the extra heat encountered during the reflow process. Special considerations must be given to the through-hole lead configuration and the overall PCB design. Recent publications by Gervascio[1] and Whitmore et al[2] have explored the pin-in-paste process for through-hole reflow.

Through-hole Component Selection

Component material. Components often fall into the "incompatible" category because they are designed for wavesoldering applications — where temperatures are typically 50° to 100°C lower at the component body than the reflow process.

The following is a list of acceptable and unacceptable materials for reflow process:

Acceptable materials: Diallyl Phthalate; Fluorinated Ethylene Propylene (FEP); Neoprene; Nylon 6/6; Perfluoroalkoxy (PFA) resin; Phenolic; Polyamide-imide; Polyarylsulfone; Polyester — Thermoset; Polyetherimide; Polyethylene Terephthalate; Polyimide; Polysulfone; Polytetrafluoroethylene (PFTE); Silicone; Polyphenylene Sulfide (PPS); Liquid Crystal Polymer (LCP); Polyetheretherketone.

Unacceptable materials: Acrylonitrile Butadiene Styrene (ABS); Acetal polymer; Acrylic; Cellulose Acetate Butyrate (CAB); Polybutylene Teraphthalate (PBT); Polybutylene; Polycarbonate; Polyethylene; Polyphenylene Oxide; Polypropylene; Polystyrene; Polyvinyl Chloride (PVC); Polyethylene Tedrephthalate (PET).

Pin Type: Straight vs. Locking

Through-hole components will frequently have high retention forces, which are designed to maintain the components in place during wave soldering. In intrusive reflow, these types of forces are not necessary. High-insertion forces will complicate the manual or automation insertion and create opportunities for defects during through-hole part insertion or placement. The insertion force of the through-hole components must be less than the placement equipment's Z-axis force; ideally the insertion force should be approaching zero. The insertion shock must be kept to a minimum to prevent dislodging of smaller SMDs. Components must be chosen with machine limitations (placement accuracy, placement force, vision capability and feeding mechanism) and ergonomic factors in mind. Through-hole components with a high-pin count should have locating features for manual placement to guide the operators to the proper registration in pad holes.

Lead Length

The lead length should not exceed 0.050" greater than the PCB thickness. When the lead is inserted, some of the solder paste is pushed out of the hole and remains on the lead. If the lead is too long, the solder paste will not flow back to the solder pad during the reflow process, reducing the final solder fillet volume.

PCB Design Issues

Hole size. In analyzing the design of a component hole size, several factors come into play:

- Placement accuracy tolerance (A)
- PCB hole location tolerance (B)
- PCB hole size tolerance (C)
- Part lead location tolerance (D)
- Lead diameter tolerance (E).

Experience shows that the worst case analysis is not always necessary. Assuming that the factors are normally and independently distributed, and the natural tolerance limits (±3 sigma) for each factor coincide with, or fall within, their respective specification limits, take the square root of the sum of the squares of the tolerances for the estimate of the total standard deviation of the insertion process. The equation for determining component hole size is as follows:

Finished hole diameter = Nominal lead diameter + $(A_2 + B_2 + C_2 + D_2 + E_2)_{1/2}$

The probability of successful insertion in the finished hole can be estimated from Z Tables, because in a normal distribution, the natural tolerance limits are:

However, it should be noted that greater clearance between the component lead and the finished hole size will require more solder paste volume to form a proper solder joint.

Pad size. When considering a minimum pad size, the design must consider the finished hole size, PCB manufacturing tolerances and the minimum required annular ring. The minimum annular ring is the copper material measured from the edge of the hole to the outside diameter to the pad. This is a function of the minimum amount of layer-one copper needed to facilitate plating and mechanically anchor the plated through-hole. The following will give the minimum land diameter for any finished hole size:

Minimum pad diameter = Finished hole diameter + 2 x (minimum annular ring) + PCB manufacturing tolerances It should be noted that a minimum pad size reduces the solder paste volume required to form the top and bottom solder fillets.

Solder paste keep-out area. The through-hole components will require additional clearance that is free of components and unmasked vias for solder paste overprint. If there are unmasked vias that are too close, solder will be shared between the via and component pad, causing solder bridging. The solder paste keep-out area around the through-hole component should be clear of exposed metal (other than the pad), other holes and legend ink. The legend ink might impact the flow of solder paste during reflow and cause it to separate and form a solder bead.

Solder Paste Overprint Consideration

Successful solder paste overprint is a function of the solder paste rheology, soldermask and through-hole component standoff. Soldermask with high surface energy will permit a larger overprint than a soldermask with lower surface energy. In large overprint, a solder bead can be formed with a low-surface-energy soldermask. In addition, a more solderable land finish (e.g., HASL vs. OSP), lead finish and solder paste with higher wetting strength all assist in making the overprint reflow process more robust. Solder paste print should cover the entire land to maximize the wetting force exerted by the land on the overprinted areas.

The component standoff height required is a function of the solder paste volume extended beyond the component land. Because the shape with the least energy for molten solder on an unsolderable surface is a sphere, there is a tendency for the overprinted paste to form into a spherical shape as it moves inward toward the land during reflow. The component standoff must be greater than [2 x ((Solder volume printed beyond component land in any given direction x 3)/4π1/3]. The component housing material should not come in contact with the solder paste. Where the standoff height cannot meet this requirement and the overprint pattern cannot be rearranged, it may be necessary to decrease the size of the overprint and make up the paste deficit with a stepped stencil.

Electrical Test Consideration

This process will impact the electrical in-circuit test (ICT). When the intrusive reflow process is used, the through-hole component lead tip and bottom solder fillet is coated with flux residue and can cause an improper probe contact when the lead tip is being used as test site. An extra test pad will be required to eliminate the improper probe contact.
Another factor that might impact the ICT is the vacuum loss caused by the unplugged vias. For test purposes, the non-test vias can be masked with soldermask.

Stencil Designs

Solder volume. The objective of solder paste stencil printing for through-hole reflow is to provide enough solder volume after reflow to fill the hole and create acceptable solder fillets around the leads. The equation describing the required volume of solder paste is shown in Figure 1.

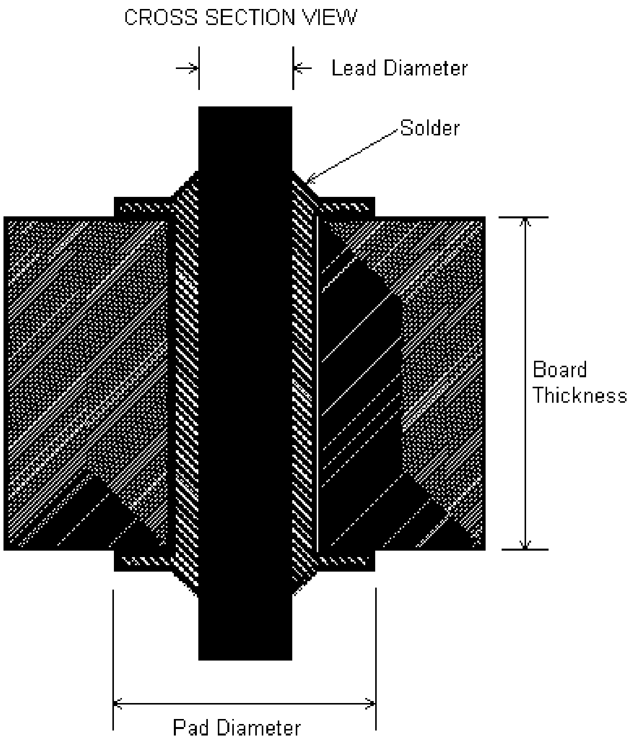

Figure 1

There are three stencil designs commonly used to deliver the through-hole solder paste: non- step stencil, step stencil and two-print stencil.

Minimum distance between apertures. The web distance between apertures should be maximized to eliminate solder bridging or solder starvation caused by solder paste slumping. A thin web between aperture in stencils of 0.006" or less will distort and elongate during the printing process. Some of the following factors will affect the web dimension:

- Aperture size
- Stencil thickness
- Board flexure
- Squeegee material
- Printer setup.

Overprint Without Step

This is a single-thickness stencil with oversized apertures for the through-hole components to meet the solder paste volume requirement to form a solder joint. A cross-section of this stencil type is shown in Figure 2.

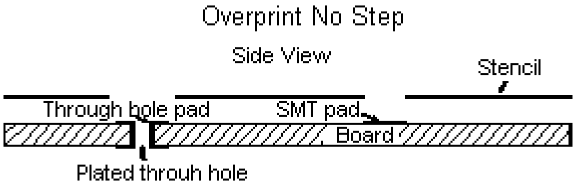

Figure 2

An example of when this stencil could be used is a two-row connector on 0.10" pitch with 0.045" diameter through-holes and 0.035" lead diameter with a 0.048" thick PCB and no other components or vias within 0.150" of the through-hole openings. An overprint stencil aperture of 0.085" wide and 0.170" long with a stencil thickness of 0.006" can deposit a sufficient solder paste volume to form a solder joint with solder fillets on both sides of the PCB.

Overprint with Step

A step overprint stencil can be used when a single-thickness overprint stencil will not provide the proper solder paste volume to form an acceptable solder joint. An application for the step overprint stencil is for a through-hole component with multiple lead rows (three or more) or a densely populated board with a minimum keep-out area between the SMT components and the through-hole component. An example of this stencil type is shown in Figure 3.

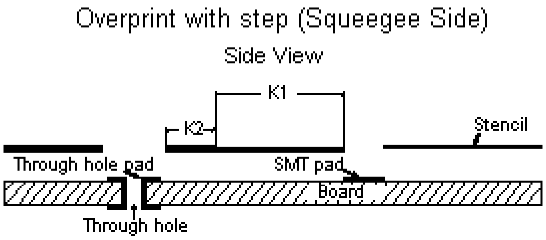

Figure 3

K1 and K2 are keep-out distances. K2 is the distance between the through-hole aperture and the step edge. As a design guide rule, K2 can be as low as 0.025". K1 is the distance from the step edge to the nearest aperture in the step-down area. As a design guide rule, K1 should not be less than 0.035" for every 0.001" of step-down thickness. For example, a 0.008" with a step down to 0.006" would require a K1 keep-out distance of 0.070". It is also possible to put the step on the contact side of the stencil instead of the squeegee side (Figure 4

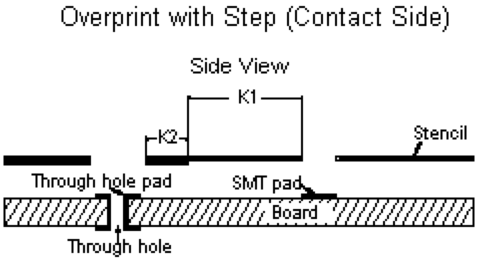

Figure 4

This type of step is sometimes more convenient when using metal squeegee blades. Contained solder paste heads do not have print blades, but they do have wiper blades. These wiper blades will catch on any step on the squeegee side of a stencil. Therefore, contact-side steps are required for this type of print head. The same keep-out rules apply for contact- or squeegee-side steps.

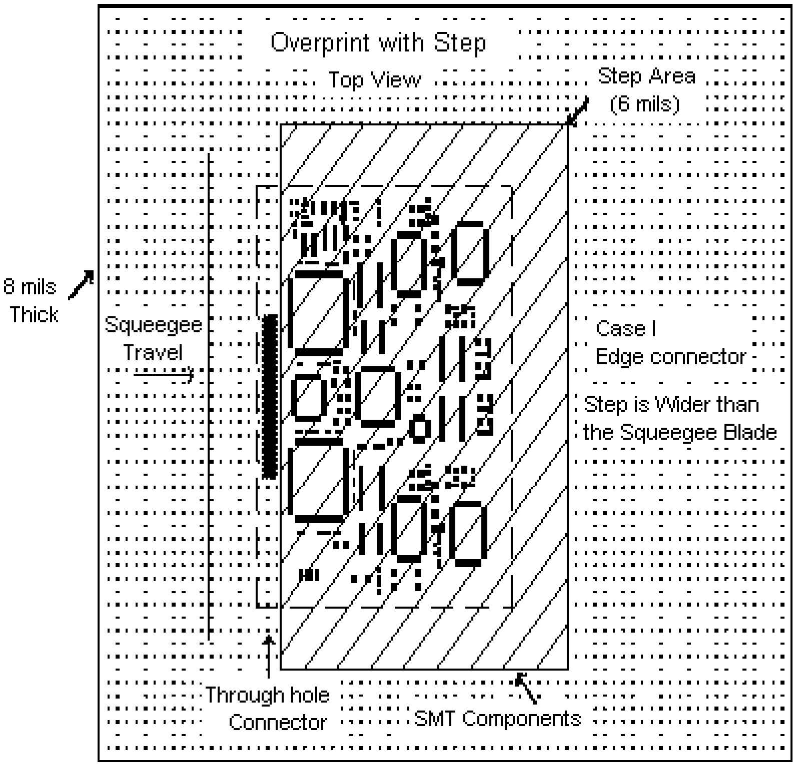

Figure 5

Step layout will depend on PCB layout. Figure 5 shows a PCB with an edge connector. The PCB outline is indicated by the dotted line. In this case, the step should be wider than the squeegee blade length as shown. The squeegee blade will not be held up on the ends as it passes into the step pocket for a squeegee side step. The metal foil will easily deflect downward and make good gasketing contact to the PCB for the contact side step.

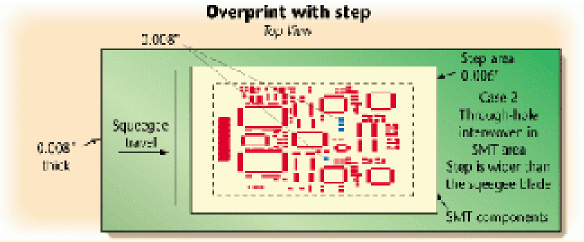

Figure 6. If through-hole components are interwoven with SMDs, the stencil can be stepped down and then back up for the through-hole components.

Figure 6. If through-hole components are interwoven with SMDs, the stencil can be stepped down and then back up for the through-hole components.

Figure 6 shows through-hole components interwoven among SMT components. In this case, the stencil is stepped down to 0.006" in a very large area that includes the board area. Again, the step is larger than the squeegee blade length as shown. The stencil is stepped up to 0.008" in the through-hole components area. The step-up can be on the squeegee side or the contact side.

Two-print Stencil

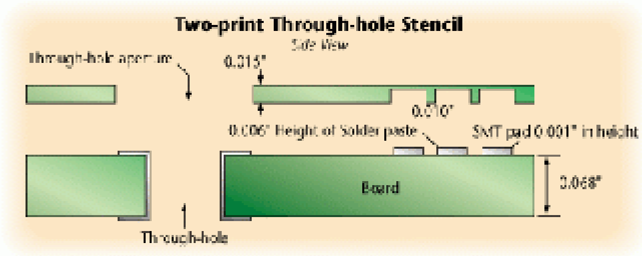

Figure 7. A cross-section of the two-print through-hole stencil.

Figure 7. A cross-section of the two-print through-hole stencil.

Some through-hole devices have small leads with large holes or dense spacing with thick boards. In either case, insufficient solder paste volume is available using the first two stencil designs. The two-print stencil can deliver large amounts of solder paste volume into the plated through-holes. In this design, a normal SMT stencil (0.006" thick) is used to print the SMD solder bricks. While the SMD paste is still tacky, a thick stencil is used to print the through-hole solder paste. Normally, this requires a second stencil printer setup in-line to perform this printing. This stencil can be as thick as required — 0.016" to 0.030" is typical. When stencil thickness requirements exceed 0.020", laser-cut electropolished apertures provide better paste release and overall print performance because of the excellent wall geometry. The contact side of this stencil is relief etched at least 0.010" deep in any area where SMD bricks have been previously printed. A cross section of the two-print through-hole stencil is shown in Figure 7.

Conclusion

Many SMT assembly process engineers have been successful in converting through-hole assembly from a wavesoldering process to a reflow process. This allows both types of components to be placed on or into the PCB and soldered in place with a single reflow process. There are a number of design issues to consider when converting to this process — as well as a myriad of tradeoffs. However, careful selection of the through-hole component material type, pin design, lead length, PCB and stencil design will ensure success.

REFERENCES

1. T. Gervascio, "Developing the Paste-in-hole Process," Proceedings of Surface Mount International 1994, p. 333.
2. D. Manessia, M. Whitmore, J.H. Adriance and G.R. Westby, "Evaluation Study of Proflow System for Stencil Printing of Thick Boards (0.125") in the Alternative Assembly and Reflow Technology (AART) or Pin-in-paste Process," Proceedings of the Technical Program, NEPCON West '99, p. 416.

WILLIAM E. COLEMAN may be contacted at Photo Stencil, 4725 Centennial Blvd., Colorado Springs, CO 80919; (719) 535-8528; Fax: (719) 535-8557; E-mail: bcoleman@photostencil.com DENIS JEAN may be contacted at 3Com, Mount Prospect, IL. JULIE BRADBURY may be contacted at Qualcomm, 5775 Morehouse Drive, San Diego, CA 92121; (858) 587-1121; Fax: (858) 658-2100.