Stencil Controls

By William E. Coleman, Ph.D., Photo Stencil

It is generally stated that over 70% of defects in surface mount assembly are related to the printing process. There are many factors involved with print, including the printer, squeegee blades, solder paste, printer set-up, and the stencil. All of these factors have a major influence, but I will confine this discussion to an analysis of stencil controls.

The stencil is a simple tool — typically a sheet of metal or plastic with holes in it for applying solder paste, flux, or glue to a substrate. The base sheet is stretched tightly into a frame with a mesh border of a tensioning mechanism built into it. The holes, or apertures, are formed by one of three methods: chem-etching, laser-cutting, or electroforming. The first two processes are subtractive in that the erosion of material from the stencil foil, either by chemical etching or laser cutting, creates the aperture. The third process, electroforming, is an additive process whereby the stencil foil is created by electroforming metal, in this case nickel, around small photoresist pillars that represent the apertures in the foil. Photoresist is applied to a temporary mandrel, imaged, and developed to form the pillars on the mandrel. When the material is plated up to the desired stencil thickness, the foil is separated from the mandrel, creating the electroformed stencil.

Controllable Elements of the Stencil
Following are key stencil control issues. A good reference document is IPC 7525A "Stencil Design Guidelines."

Aperture size. The volume of solder paste transferred onto the printed circuit board (PCB) is determined primarily by the size of the stencil aperture and stencil thickness. The aperture size is normally held to ±0.3 mils (0.008 mm) from the nominal value for laser-cut and electroform stencils. Stencil thickness is typically held to ±0.2 mils (0.005mm) for laser-cut stencils and ±10% of thickness for electroform stencils. Given these nominal values for aperture size and stencil thickness, the actual amount of paste transferred to the PCB is determined by five additional factors: the area ratio of the stencil aperture design; the aperture side-wall geometry; aperture wall smoothness; stencil/board separation speed; and the solder paste's particle size. The generally accepted area ratio for paste release is >0.66. However, for very smooth aperture walls, normally achieved in electroform or laser-cut stencils with electropolish and nickel-plate coatings, area ratios as low as 0.5 may be acceptable. Lower stencil/board separation speeds usually result in better paste transfer, giving the paste more opportunity to pull out of the stencil aperture. A good guideline for solder paste particle size is the average particle size should be <1/5 the smallest aperture size.

Aperture position. When solder paste is printed onto the PCB, positioning on the pad is crucial. Mis-registration can cause tombstoning for discrete components and shorts or opens for leaded and BGA devices. The IPC 7525 guideline suggests the maximum mis-registration between true position and actual aperture position be less than 0.1 mil (0.00254 mm) per 1" (25.4 mm) of aperture pattern. Stencil aperture registration to PCB pad is important for lead-free solder pastes since pad wetting is normally less than that for tin/lead solder pastes.

Step stencil design There are situations where it is desirable to have a stencil with more than one thickness: a step stencil. One application requiring a step stencil is a PCB having a CBGA and fine pitch QFPs and/or microBGAs. Additional paste height is required for the CBGA due to potential coplanarity issues. Another step stencil requirement is a PCB having intrusive reflow apertures for through-hole devices along with standard SMT devices. There are two keep-out designs to consider. K1 is the distance between an aperture, in the thinner part of the stencil, to the wall where the stencil becomes thicker. IPC 7525 suggests K1 should be a minimum of 35 mils (0.9 mm) for every 1 mil (0.025 mm) of step-down thickness. K2 is the distance between the step-down wall and an aperture in the thicker part of the stencil. The recommended design for K2 keep-out is a minimum of 25 mils (0.65 mm).

Chip component aperture design. Proper aperture design for chip components can prevent problems such as micro solder balls and tombstoning. "Homeplate" and "bow-tie" apertures are popular designs to eliminate micro solder balls. Proper design of the spacing between apertures is effective for tombstoning prevention.

QFN aperture design. Most fine-pitch QFNs have pad lengths that are not much more than the pad width. Pay close attention to the stencil thickness, since these small aperture sizes can have unacceptably small area ratios if the stencil is too thick. Printing the ground plane 1:1 tends to make the component float during reflow. A 50% area aperture reduction compared to pad area is recommended to eliminate this problem. This reduction is typically achieved by a windowpane aperture design.

Conclusion

Stencil controls are important for successful SMT assembly printing. Proper aperture design and stencil thickness selection will ensure acceptable area ratios. Proper selection of stencil technology will provide smooth aperture walls for clean paste transfer. Finally, component-specific aperture design will eliminate problems associated with particular package types.

William E. Coleman, Ph.D., is an *SMT* Editorial Advisory Board member and vice president of technology at Photo Stencil. He is Chairman of IPC's Stencil Design Guidelines Subcommittee 5-21e, and a member of SMTA and IEEE. Coleman has presented Technical Stencil Design reviews to SMTA chapters globally. He may be contacted at (719) 535-8528; bcoleman@photostencil.com.