



## Critical Parameters for Stencil Performance

*The stencil plays an important role in the SMT printing process, and is a major factor in the overall SMT assembly process yield. A stencil should not be treated as a common commodity.*

The function of a stencil is to deliver a known and controlled volume of solder paste to device pads on the PCB. The printing process involves two steps: the aperture fill process, where solder paste fills the stencil aperture, and the paste transfer process, where solder paste is transferred from the stencil aperture to the PCB pads.

The fill process largely depends on solder paste, squeegee blade, solder paste roll, print speed, and aperture orientation with respect to print direction.

Paste transfer depends on the stencil technology and its wall smoothness, the stencil aperture design as related to the area ratio, solder paste, and the board's separation speed from the stencil. The paste transfer process can be viewed as a competing process, where the pad on the PCB below the stencil aperture is pulling the solder paste out of the aperture while the aperture sidewalls are holding the solder paste inside it. When thinking of the paste transfer process in this manner, it is easy to understand why area ratio and aperture sidewall smoothness claim a dramatic influence on paste transfer.

Critical parameters for stencil performance range from solder paste volume delivered to the PCB pad during printing to solder paste volume dispersion, stencil thickness, stencil aperture positional accuracy, stencil dimensional accuracy, missing or partially formed apertures, extra apertures, and stencil design check and verification before manufacture.

All stencils are not equal in solder paste transfer efficiency. The degree of inequality depends on the area ratio. The lower the area ratio, the larger the gap in solder paste transfer efficiency for different stencil technologies. Smooth aperture walls found in electroform stencils, as well as laser-cut stencils with electropolish and nickel-plate post-processing, provide higher paste transfer for lower area ratios, as found in 0.4-mm-pitch BGAs and 0201s.

Variation in solder paste volume also impacts good printing results. Stencils with smooth aperture walls provide lower paste volume dispersion, especially for small apertures found in high-density components with low area ratios.

Stencil thickness directly influences solder paste volume. The thickness should be certified by the stencil supplier, as it is difficult for the end user to check this when the stencil foil is mounted in a frame. When the stencil is still at the foil stage, however, it can be checked easily with a thickness gauge, which uses a small probe placed on the edge of an aperture or in the step pockets of a step stencil.

Stencil aperture position is an important attribute. The stencil apertures must align to the pads on the PCB receiving the solder paste. Misalignment paste-to-pad can cause bridges, opens, or tombstoning. Lead-free solder, which does not wet the pad as well as tin/lead solder does, is particularly sensitive to paste-to-pad mis-registration. It is important for the stencil supplier to verify aperture positional accuracy and clearly state this in a certificate of compliance (CoC). IPC-7525 establishes a guideline of 0.1  $\mu\text{m}/\text{inch}$  of active stencil image area as the maximum mismatch for aperture-to-Gerber-data registration.

The stencil supplier should verify aperture dimension in a CoC. Since stencils have tapered apertures, it is important to specify the opening on the squeegee or contact side of the stencil.

In any stencil manufacturing process - chem-etch, laser, electroform - it's possible to produce a defective stencil with a missing, partially formed, or extra aperture. Detecting this error with visual inspection never is 100% effective. A more reliable method to verify proper apertures is to automatically scan the stencil, create a database of the scanned information, and compare this database to the Gerber file's data field. Aperture presence/absence should be verified by the stencil supplier in a CoC.

Stencil aperture design is critical for printing performance as well as SMT assembly yields. If an aperture design violates the area-ratio guideline, poor paste transfer will occur. A useful tool to prevent bad designs is an area-ratio calculator, which provides the area ratio for a given aperture design. Apertures designed to prevent tombstoning of chip components and solder balls, QFN ground plane issues, and other special requirements are effective. Some stencil suppliers offer custom libraries for special aperture designs, which are used every time a stencil is ordered. This support service is designed to aid OEMs and EMS providers.

## Conclusion

When examining all the critical parameters that influence stencil printing performance, it is obvious that a stencil should not be treated as a common commodity. All stencils are not equal.

This becomes more evident with assembly using miniaturized SMT components. To achieve consistent printing performance, quality must be designed into the stencil, verified, and certified by the stencil supplier before shipping to the end user. The correct aperture design and stencil technology must be selected to provide a consistent, robust, trouble-free assembly process.