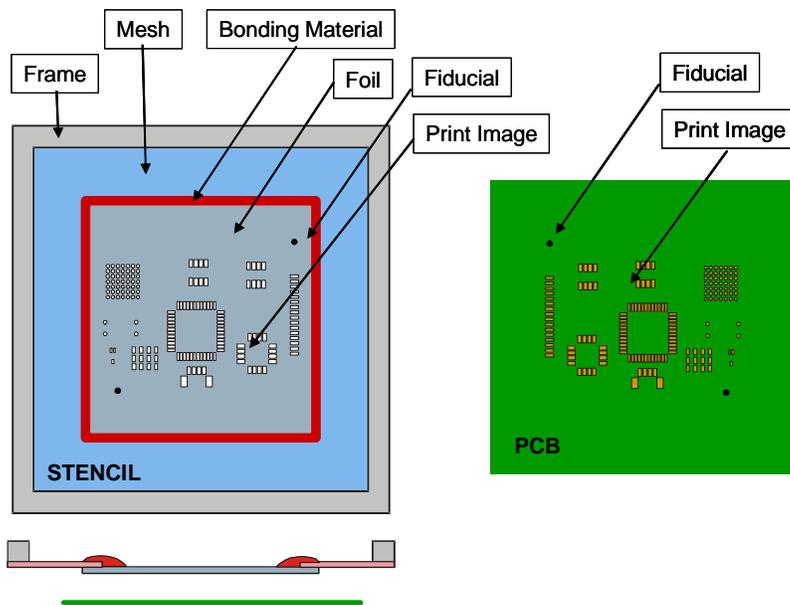


# Stencil Technology

## TERMS, TIPS & TERMINOLOGY

### GENERAL OVERVIEW



### GLOSSARY OF TERMS

- **Aperture:** An opening in the stencil that corresponds to a land area on the circuit board to be printed.
- **Aperture Design Rules:** A set or procedure of rules with recommendations for aperture reductions and modifications in order to optimize stencil design.
- **Aperture Reductions and Modifications:** Changes in size or shape of stencil apertures in relation to the corresponding landing pad. These changes are designed to improve paste deposition and yield.
- **Aperture Shape:** The outline of the opening in the stencil, as viewed from squeegee side. Common shapes include rectangles, squares, rounds, ovals and "home plate."
- **Aperture Size:** Refers to the width and length dimensions of the opening in the stencil.
- **Aspect Ratio:** The aspect ratio is the ratio of the aperture opening to the stencil thickness. For chemically etched stencils, this ratio should be greater than 1.5, for laser cut stencils it should be greater than 1.2 and for electroformed stencils which has the best solder paste release characteristics this ratio should be greater than 1.1. Anything less than these recommended ratios will cause the solder paste to stick into the apertures during release as the retaining force of the paste in the aperture will be stronger than the force pulling the paste out of the aperture. See formula.
- **Area Ratio:** The relation between the surface of the aperture and the inside surface of the aperture walls in the stencil. The major different with Aspect Ratio is that Area Ratio is more suitable shapes such as circles. Since solder paste has a certain adhesion force, it will stick to the walls of the aperture and to the pad. A ratio of 0.66 pad to wall is considered acceptable in our industry ( for example a 13.5 mil circle in a 5 mil thick stencil). See formula.
- **CAD/CAM:** Computer Aided Design and Computer Aided Manufacturing is the general terminology for file types used for computerized design and manufacturing.

The information contained herein is based on data considered accurate and is offered at no charge. No warranty is expressed or implied regarding the accuracy of this data. Liability is expressly disclaimed for any loss or injury arising out of the use of this information or the use of any materials designated.





- **Chem-Etch:** This is the least sophisticated manufacturing method of making stencils, a photosensitive process with a negative photo image is used onto the metal foil. A double sided chemical etching process is used to etch the material away in selected area's resulting in the apertures. The apertures are very smooth but have an hour-glass cross section.



- **Electroform:** The most sophisticated manufacturing technique of stencils, a photosensitive process with a positive image is used to make a disposable mandril. Electrolytic plating is used for this additive process where we actually grow the stencil around the apertures. The apertures are very smooth with a very fine resolution.



- **Fiducial:** A mark in the artwork that is etched in the stencil along with the apertures. It is used by the machine vision system to align the stencil to the PCB and to verify artwork orientation and location. Fiducials can be fully-etched or half-etched either on the top (squeegee side) or bottom (board side) of the stencil. Sometimes the fiducials are filled with black epoxy to provide contrast between the stencil and the fiducial for visual systems. "Global" fiducials are located outside the aperture footprint while "local" fiducials are placed within the image itself, usually in close proximity to an integrated circuit.



- **File Layer:** Describes the layer of the PCB that the Gerber information represents. The solder paste layer is a 1:1 reflection of the component lands on the board.
- **Frame Mount:** Stencils are usually secured to an aluminum frame with a tightly-stretched mesh border. Alternatively, a stencil can have mounting holes etched along the perimeter for temporary mounting on a universal sized frame, such as the ALPHA TETRA™ frame or competing frameless systems. The user requires typically one frame per printer and buys foils only with the stencil image made in the foil. This foil gets then tensioned into the re-usable frame.
- **Frame Size:** The size of the frame for mounting stencils is determined by the screen printer model. The smaller cast-aluminum frames are usually specified by the inside dimension (ID), e.g., 12 x 17 or 20 x 20 inches. Tubular aluminum frames are usually specified by the outside dimensions (OD), e.g., 29 x 29 inches.
- **Gasketing:** The degree to which a stencil aperture contacts and "seals" against a landing pad. Good gasketing decreases solder "squeeze balls" during printing, resulting in random solder balling and bridging.
- **Gerber Data:** Standardized PCB design language and operating commands which ultimately define the slope and location of apertures in a stencil plate.
- **Land:** The conductive area on a PCB to which components or separate circuits are attached.



- **Laser Cut:** The most popular manufacturing technique for current stencil technology, the CAD/CAM file is used directly to generate a CNC-file that drives the X-Y driven laser head. The laser will cut the perimeters of the apertures, resulting in tapered apertures. The wall structure is rougher than the other techniques due to the 'melting' effect of the laser beam.
- **ODB++:** is the most intelligent CAD/CAM data exchange format available today, capturing all CAD/EDA, assembly and PCB fabrication knowledge in one single, unified database. Originally developed by Valor Computerized Systems for use in its own PCB CAD/CAM systems, Genesis 2000, Enterprise 3000 and Trilogy 5000, ODB++ has already become widely accepted as the de facto industry standard, providing unprecedented power to PCB design, fabrication and



assembly, with the flexibility to expand as required. In parallel, ODB++ is providing the technological basis for a formal IPC standard

- **Step and Repeat:** Repeating a single Gerber file over two or more areas of the same stencil plate. PCB's are arranged on "break away" panels often referred to as 2 Up, 4 Up, etc. . . . Information on the center-to-center dimensions is essential to prevent misregistration of panel to stencil.
- **Step Down/Up:** Reduction of stencil thickness in specific areas of a stencil on either the top or bottom side(s). Step UP refers to an area higher/thicker than the majority of the stencil, Step DOWN refers to an area lower/thinner than the majority of the stencil. Used to control paste deposits on PCB's having both standard pitch and fine pitch components. Also, used for creating an underside cavity to accommodate surface circuitry, vias or other low profile surface components in order to optimize stencil gasketing.



- **Trapezoidal Opening:** The cross-sectional angle of the stencil aperture with the larger opening on the bottom side of the stencil and the smaller opening on the top side. This geometry improves solder paste deposition onto the PCB. The laser-cutting and electroforming processes provide trapezoidal openings, whereas, chemically-etched stencils must be etched from both sides of the stencil with a photoresist on the bottom side of the stencil which has slightly larger apertures than the top side photoresist ( hour glass)

## COMMONLY USED FORMULAS

