DOYAN
SCREEN PRINTING

Overview

The printing process is the most critical part of PCB assembly, because one minor error can ruin large number of printed circuit boards. Such errors often occur when solder paste is printed on PCB pads. Therefore, solder paste stencil came into being, which serves two primary functions. The first is to ensure precise placement of a material, such as solder paste, flux or encapsulant, on a substrate. The second is to ensure the formation of properly sized and shaped deposits.

01 Laser Cutting Technology

Laser cutting stencils were introduced and used in mid-90's, just in time to meet the print requirements of 0.65-mm-pitch SMT devices. To meet this trend, Doyan has purchased LPKF SLG 6080 stencil laser system in 2015, which enables us to have technical ability to produce laser cutting stencils. Laser cutting is an extremely accurate and repeatable process. Therefore, it achieves improved aperture definition and superior dimensional tolerances for finer pitch apertures, down to 0.4mm (0.016″). On a 600mm×600mm area, it is possible to cut an aperture of ±9 micron and a positional tolerance of ±12.5 micron, and it's over 12,000 aperture/hour cutting speed allows us to deliver your finest stencils on time. The technical data of G6080 laser system is presented below:

Cutting range	600mm×800mm (23.6"×31.5")
Maximum frame size	740mm×950mm×40mm (29.1"×39.4"×1.6")
Maximum loose sheet size	650mm×850mm (25.6"×33.5")
Material thickness	Up to 1mm (0.04")
Laser repetition rate	Max. 45kHz
Axial precision	±2μm
Right-angle precision	4 angular seconds
Repeatability	±2μm
Power consumption	Max. 2800 VA over all
Dimension (W×H×D)	1575mm×1920mm×1820mm (61"×75"×64")
Weight Less than	2000 kg (4400 lbs)

02 Foil Thickness

Foil thickness and aperture size (width and length of the aperture or diameter of the aperture) determine the volume of solder paste released onto the printed circuit board. As the squeegee blade travels across the stencil during the printing process, solder paste fills stencil apertures and then is released onto circuit board during the separation between the board and stencil. The ideal result is that 100% of the paste that filled the aperture during printing process is released from the aperture walls and attaches to the circuit board, forming a complete solder brick. Therefore, the thickness and aperture ultimately determine solder paste volume, which is crucial to ensure a high-yield assembly process.

The choice of foil thickness is probably the most important decision in ordering a stencil. It affects everything from print release to solder joint reliability. Basic rule of thumb is to start with a .007 or .008 foil thickness and adjust thinner as dictated by the smallest aperture to be printed. Thicker foils will of course be ideal in giving more volume for robust solder fillets especially for larger passives and leaded SOIC's and connectors. But thicker foils cannot be used to reliably print smaller openings. Since the smallest openings must be printable, they are what ultimately determine the foil thickness you choose. Here is a quick and easy chart for determining foil thickness.

Components	Pads Pitch (mm)	Stencil Thickness (mm)
QFP, SOIC, SOP, TSOP, Connector	1.27	0.18/0.12
	0.8	0.15/0.18
	0.65	0.15
	0.5	0.10/0.12/0.15
	0.4	0.1
	0.3	0.08
BGA	1.5	0.15
	1.27	0.15
	1	0.13
	0.8	0.12
	0.65	0.1
	0.5	0.1
	0.4	0.08
603	*	0.12/0.15/0.18
402	*	0.10/0.12/0.15
201	*	0.1

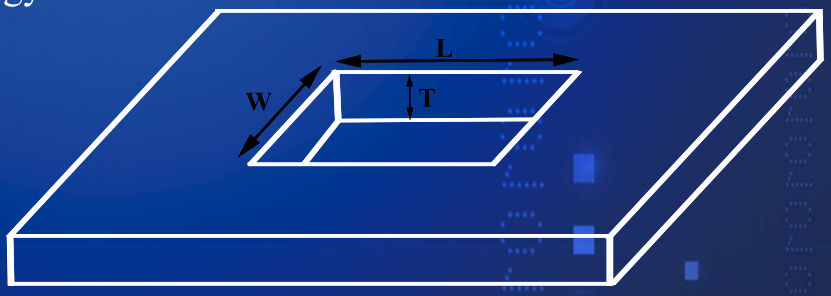
The above chart determines foil thickness based upon the types of parts on the assembly. It is based on our experience and is for recommendation only based on what we believe will yield the best results.

03 Aperture

Aperture on the other hand determine solder paste released from the inner aperture walls, which depends primarily on three major factors, including aspect/area ratio, aperture side wall geometry and aperture wall smoothness. The first factor is aperture design-related while the other two factors are stencil technology-related.

$$\text{Aspect Ratio}=\frac{\text{Width of Aperture}}{\text{Thickness of Stencil}}=\frac{W}{T}$$

$$\text{Area Ratio}=\frac{\text{Area of Aperture}}{\text{Area of Aperture Wall}}=\frac{L\times W}{2\times(L+W)\times T}$$



For small rectangular and square openings you will need an area ratio of 0.6, for circles a value of 0.66, and for longer rectangles an aspect ratio of 1.5. The formula changes depending upon the geometry of the stencil opening. Both aspect ratio and area ratio are illustrated above. When the stencil separates from the board, paste release encounters a competing process. Solder paste will either transfer to the land on the board or stick to the aperture side walls. When the area ratio is greater than 0.66 of the inside aperture wall area, a complete paste transfer should occur. (Noted: The 0.66 minimum area ratio discussion above refers to stainless steel stencils without nano-coatings.)

04 Aperture Size & Pad Size

When squeegee moves across stencil and fill solder paste into apertures. After stencil separates from circuit board, solder paste is then released on it. To guarantee successful transfer from stencil to PCB, The adhesion to the pad must be greater than the adhesion to the stencil to ensure a good transfer. Therefore aperture openings should be slightly smaller then the landing pad size. This is primarily for:

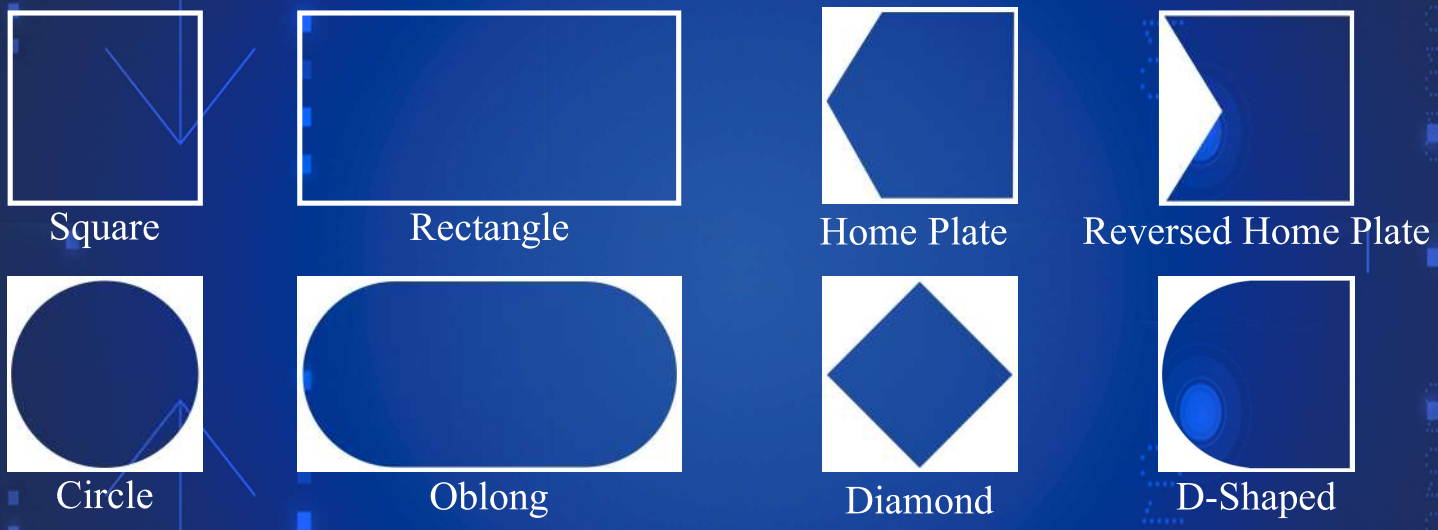
- Improved gasketing between the landing pad and the underside of the stencil.
- Prevent bridging on fine pitch component.

Here is a quick and easy chart for industry standard aperture size and pad size.

Component Pitch	Aperture Width	Pad Width
50 mil	25 mil	25 mil
40 mil	20 mil	20 mil
31 mil	16 mil	17 mil
25 mil	12 mil	15 mil
20 mil	10 mil	12 mil
16 mil	8 mil	10 mil
12 mil	6 mil	8 mil

05 Aperture Shape

Different aperture shapes have been found to offer the benefits of less paste utilization, consistent paste release and reduced or eliminated solder balling. Shapes to consider include:



06 Aspect Ratio

This relationship between foil thickness and aperture width is also known as “aspect”. Aspect is the difference in forces that either pull paste from aperture to pad or cause paste to be held within aperture. These forces can be quantified and represented as a measurement called Aspect Ratio. In simple terms, to achieve successful solder paste transfer, the paste surface tension on pad must be stronger than the surface tension on the aperture wall. If stencil is laser cut, the ratio of foil thickness to minimum aperture width is 1:1.2. For example, A laser-cut stencil with an 8 mil aperture width should have a maximum foil thickness of 6.6 mil (8 mil/1.2=6.6 mil).

07 Framed Stencil

Framed stencils are the strongest form of laser-cut stencils available in the market today, which are designed for high volume screen-printing.

Key Features and Advantages:

- Unique Process for Smooth Aperture Walls.
- Very Clean Laser-Cut Apertures.
- Excellent Print Performance.
- Excellent for High-Volume Stencil Printing.
- Unique Process Creates Permanent Non-removable Non-fading Fiducial