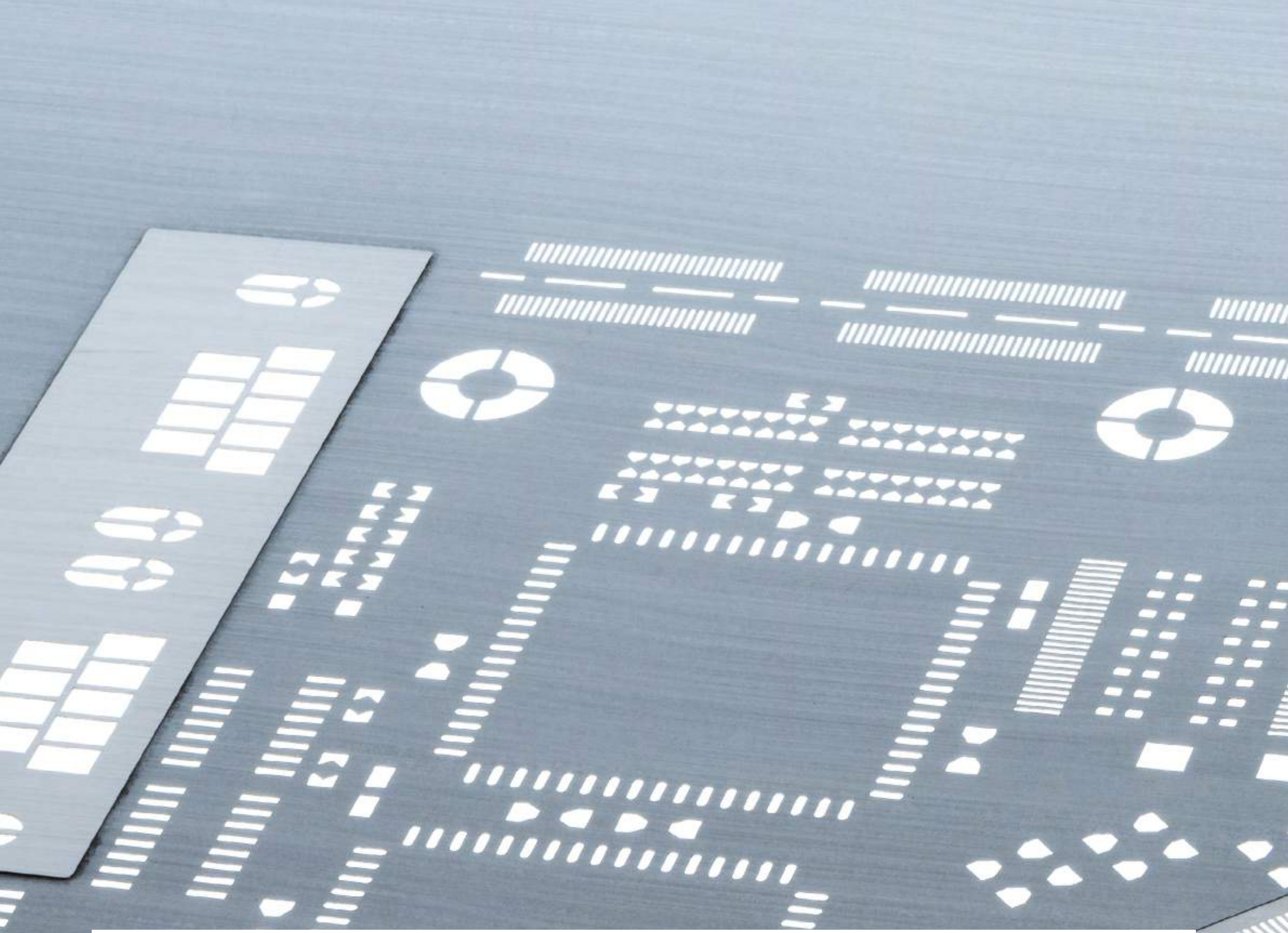


an introduction to SMT and stencils

for optimum solder paste volume and accuracy

tecan

innovators in chemical etching since 1970

1.1 Surface mount technology

Surface mount technology (SMT) is a method for constructing electronic circuits in which the components are mounted directly onto the surface of printed circuit boards (PCBs). Electronic devices made in this way are called surface-mount devices or SMDs.

In the electronics industry SMT has largely replaced the previous construction method of fitting components with wire leads into holes in the circuit board (also called through-hole technology).

An SMT component is usually smaller than its leaded counterpart because it has either no leads or smaller leads. It can have short pins, flat contacts, a matrix of balls (BGAs), terminations on the body of the component (passives), or short leads in a gull-wing formation (QFPs).

1.1.1 Typical SMT components

The following close-ups of printed circuit boards include a PC mother board and an assembly from an RF control application, both showing typical surface mount technology components.

The large square component (Item 5) in Figure 1.1 is a ball grid array (BGA) device - having its terminations on the underside facilitates increased lead-outs at larger pitches than is possible with QFPs. The copper pads relating to a BGA can be seen in the middle of this PCB. BGA devices are becoming ever smaller as can be seen in applications such as mobile phones. MLF devices (Item 7) contain no outwardly visible leads or terminations and care has to be taken to ensure the paste volume matches the requirement exactly. QFP devices (Item 1 being 0.5mm pitch) have been produced down to 0.3mm pitch but are widely used only to 0.4mm pitch with BGA technology providing more robust assembly solutions for finer pitch.

Figure 1.2 contains both 0.5mm pitch TSSOP (Item 2) and BGA devices (Item 5) together with a surface mount switch (Item 1) and inter-board connectors (Items 3 and 4). Often the inclusion of connectors can create problems when their co-polarity is less than ideal. Solder paste requirements to overcome these problems often need to be greater.

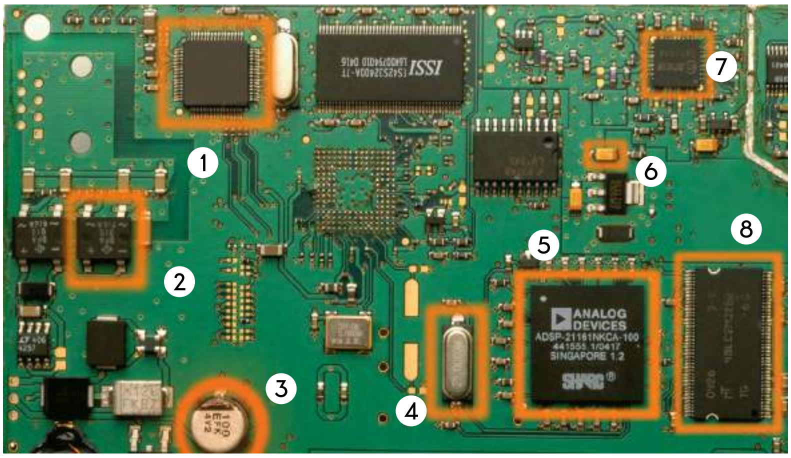


Figure 1.1: Assembly from a RF control application

1. QFP
2. Bridge rectifier
3. Wet film electrolytic capacitor
4. Crystal oscillator
5. BGA
6. Tantalum capacitor
7. MLF
8. TSOP

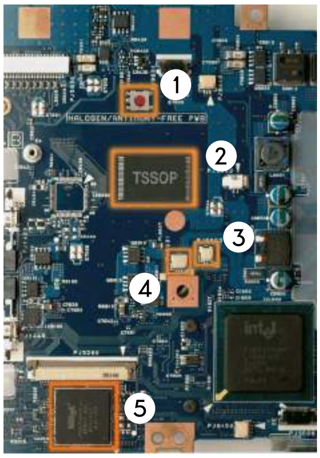


Figure 1.2: PC motherboard assembly

1. Switch
2. TSSOP
3. Surface mount header
4. Surface mount header
5. BGA

1.1.2 Advantages of SMT

The main advantages of SMT over through-hole are:

- Smaller, lighter components.
- Fewer holes need to be drilled through abrasive boards.
- Simpler, automated assembly.
- Small errors in component placement are corrected automatically (the surface tension of the molten solder pulls the component into alignment with the solder pads).
- Components can be fitted to both sides of the circuit board.
- Lower lead resistance and inductance (leading to better performance for high frequency circuits).
- Better mechanical performance under shake and vibration conditions.

1.1.3 Package sizes

Surface-mount components are usually much smaller than their leaded counterparts, and are designed to be handled by machines rather than by human hand. The electronics industry has defined a collection of standard package shapes and sizes. These include:

Rectangular passive components (mostly resistors and capacitors):		
01005	0.01″ × 0.005″ (0.3mm × 0.15mm)	two terminals
0201	0.02″ × 0.01″ (0.6mm × 0.3mm)	two terminals
0402	0.04″ × 0.02″ (1.0mm × 0.5mm)	two terminals
0603	0.06″ × 0.03″ (1.5mm × 0.8mm)	two terminals
0805	0.08″ × 0.05″ (2.0mm × 1.3mm),	two terminals
1206	0.12″ × 0.06″ (3.0mm × 1.5mm),	two terminals
1812	0.18″ × 0.12″ (4.6mm × 3.0mm),	two terminals
2010	0.20″ x 0.10″ (5.0mm x 2.5mm),	two terminals

Tantalum capacitors:	
Size A (EIA 3216-18):	3.2mm × 1.6mm × 1.6mm
Size B (EIA 3528-21):	3.5mm × 2.8mm × 1.9mm
Size C (EIA 6032-28):	6.0mm × 3.2mm × 2.2mm
Size D (EIA 7343-31):	7.3mm × 4.3mm × 2.4mm
Size E (EIA 7343-43):	7.3mm × 4.3mm × 4.1mm

- **SOIC** - Small outline integrated circuit, dual-in-line, 8 or more pins, Gull-wing leads, pin spacing 1.27mm.
- **PLCC** - plastic leaded chip carrier, square, J-lead, pin spacing 1.27mm.
- **TSOP** - thin small-outline package, thinner than SOIC with smaller pin spacing of 0.5mm.
- **SSOP** - shrink small-outline package, pin spacing of 0.635mm.
- **TSSOP** - thin shrink small-outline package.
- **QSOP** - quarter-size small-outline package, with pin spacing of 0.635mm.
- **VSOP** - even smaller than QSOP; 0.4, 0.5 mm or 0.65mm pin spacing.
- **DPAK** - discrete packaging. Developed by Motorola to house higher powered devices.
- **SOT** - small-outline transistor, with three terminals.
  - **SOT-23** - 3mm × 1.75mm × 1.3mm body – three terminals for a transistor, or up to eight terminals for an integrated circuit.
  - **SOT-223** - 6.7mm × 3.7mm × 1.8mm body – four terminals, one of which is a large heat-transfer pad.
- **LQFP** - Low-profile Quad Flat Package, 1.4mm high, varying sized and pins on all four sides.
- **PQFP** - plastic quad flat-pack, a square with pins on all four sides, 44 or more pins.
- **CQFP** - ceramic quad flat-pack, similar to PQFP.
- **TQFP** - thin quad flat pack, a thinner version of PQFP.

- **QFN** - quad flat pack, no-leads, smaller footprint than leaded equivalent.
- **PQFN** - power quad flat-pack, no-leads, with exposed die-pad[s] for heat sinking.
- **BGA** - ball grid array, with a square or rectangular array of solder balls on one surface, ball spacing typically 1.27mm.
- **CGA** - column grid array, circuit package in which the input and output points are high temperature solder cylinders or columns arranged in a grid pattern.
- **CCGA** - ceramic column grid array, circuit package in which the input and output points are high temperature solder cylinders or columns arranged in a grid pattern. The body of the component is ceramic.
- **BGA** - micro-BGA, with ball spacing less than 1mm.
- **COB** - chip-on-board; a bare silicon chip that is usually an integrated circuit, is supplied without a package (usually a lead frame over-moulded with epoxy) and is attached, often with epoxy, directly to a circuit board. The chip is then wire bonded and protected from mechanical damage and contamination by an epoxy "glob-top".
- **COF** - chip-on-flex; a variation of COB, where a chip is mounted directly to a flexible circuit.
- **COG** - chip-on-glass; a variation of COB, where a chip is mounted directly to a piece of glass - typically an LCD display.
- **MLP** – Lead-frame package with a 0.5mm contact pitch.
- **MQFP** - Metric Quad Flat Pack, a QFP package with metric pin distribution.
- **CSP** – Chip Scale Package. A 4 pin round grid array. Can be classified into the following groups:
    - Customised lead frame-based CSP (LFCSP)
    - Flexible substrate-based CSP
    - Flip-chip CSP (FCCSP)
    - Rigid substrate-based CSP
    - Wafer-level redistribution CSP (WL-CSP)

There are often subtle variations in package details from manufacturer to manufacturer, and even though standard designations are used, designers need to confirm dimensions when laying out printed circuit boards.

1.2 The SMT process

1.2.1 SMT assembly

Where components are to be placed, the printed circuit board has flat, usually tin, silver, gold plated, tin-lead or copper pads without holes, known as solder pads. Solder paste, a mixture of flux and tiny solder particles, is first applied to all the solder pads using a squeegee and thin metal stencil. If components are to be mounted on the second side, the assembly can either pass through the entire process (as described in 1.2.2) again or the second side components can be fixed in place using surface mount adhesive which can be printed through a stencil or dispensed as small liquid adhesive dots at the locations of all second-side components, with wave soldering creating the soldered terminations.

The boards then proceed to the pick-and-place machines, where they are placed on a conveyor belt. Small SMDs are usually delivered to the production line on paper or plastic tapes wound on reels. Integrated circuits are typically delivered stacked in static-free plastic tubes or trays. NC pick-and-place machines remove the parts from the reels or tubes and place them on the PCB. Second-side components are placed first, and the adhesive dots are quickly cured with application of low heat or ultra-violet radiation. The boards are flipped over and first-side components are placed by additional NC machines.

The boards are then conveyed into the reflow soldering oven. They first enter a pre-heat zone, where the temperature of the board and all the components is gradually, uniformly raised. This helps minimise thermal stresses when the assemblies cool down after soldering. The boards then enter a zone where the temperature is high enough to melt the solder particles in the solder paste, bonding the component leads to the pads on the circuit board. The surface tension of the molten solder helps keep the components in place, and if the solder pad geometries are correctly designed, surface tension automatically aligns the components on their pads.

There are a number of techniques for reflowing solder. One is to use infra-red lamps; this is called infrared reflow. Another is to use a hot gas, known as convection reflow. Also available are special fluorocarbon liquids with high boiling points employed in a method called vapour phase reflow. Nitrogen gas can be used in convection reflow ovens to prevent re-oxidation of pads and terminations. Each method has its advantages and disadvantages. With infrared reflow, the board designer must lay the board out so that short components don't fall into the shadows of tall components. Component location is less restricted if the designer knows that vapour phase reflow or convection soldering will be used in production.

Following reflow soldering, certain irregular or heat-sensitive components may be installed and soldered by hand, or in large scale automation, by focused infrared beam (FIB) equipment.

After soldering, the boards are washed to remove flux residue and any stray solder balls that could short out closely spaced component leads. Rosin flux is removed with fluorocarbon solvents, high flash point hydrocarbon solvents, or limonene, derived from orange peels. Water soluble fluxes are removed with de-ionised water and detergent, followed by an air blast to quickly remove residual water. When aesthetics are unimportant and the flux doesn't cause shorting or corrosion, flux residues are sometimes left on the boards, saving the cost of cleaning and eliminating the waste disposal.

Finally, the boards are visually inspected for missing or misaligned components and solder bridging. If needed, they are sent to a rework station where a human operator corrects any errors. They are then sent to the testing stations to verify that they work correctly.

1.2.2. SMT flow diagram

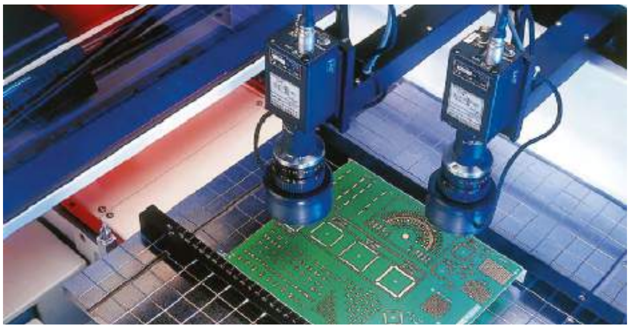
Step 1: stencil creation
The majority of stencils are created by modifying the original design of the PCB in CAD using Gerber files or their derivatives to optimise the printing process. Stencils are then typically cut on laser-stencil machines.

Step 2: printer set-up
The stencil is mounted, positioned; and the printer then calibrated in terms of:

- Adjustment of print speed and squeegee pressure (to suit the PCB design and paste used)
- Setting printing table height (achieving contact between the PCB and the stencil)
- Ensuring adequate under-board support
- Defining the limits of print stroke or travel
- Fiducial recognition
- Paste application

Step 3: carriage
The PCB along the assembly line can be:

- Manual
- Shuttle
- Or conveyor

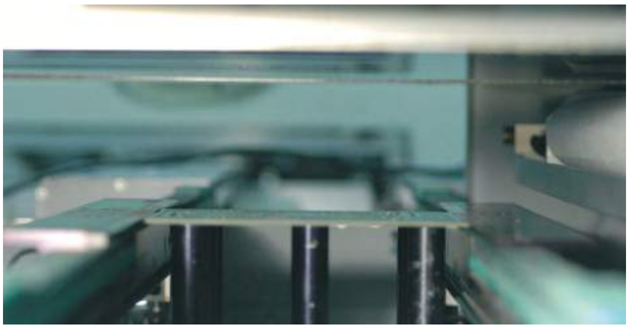


Step 4: image recognition
The alignment or matching of PCB features to stencil apertures with or without fiducials is one of the most critical pre-printing operations. The options include:

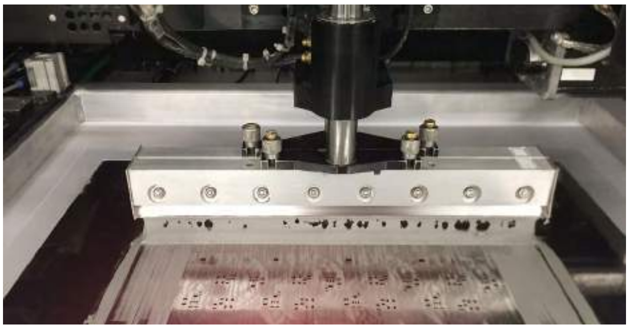
- Hand-eye
- Top down looking camera
- Camera/comparison (using stored images of before and after printing)
- In-line cameras as either:
  - Look-up/look-down, using a split field prism or alternative
  - Look-down/look-down

Poor alignment is probably the biggest cause of surface mount defects on finer pitched components.

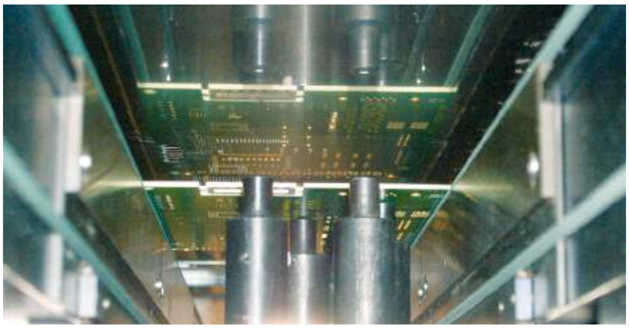
1.2.2. SMT flow diagram



Step 5: lifting into contact
The PCB is raised into good contact with the underside of stencil.



Step 6: print stroke
Travel of the squeegee blade, effectively rolling paste media to fill stencil apertures.

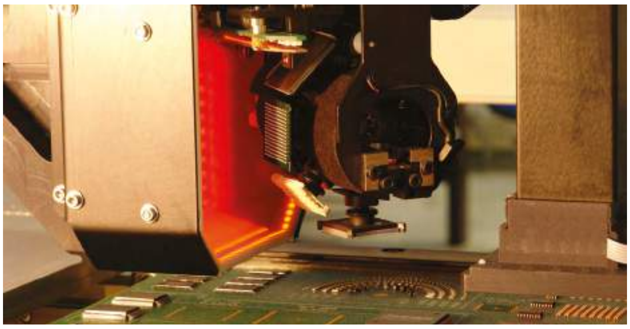


Step 7: separation
Effective separation of the PCB from the stencil for good paste release:

- Straight lift separation is recommended for fine pitch requirements
- Table separation speed depends on the solder paste selected (usually consisting of a slower initial speed, to facilitate paste separation from the stencil)
- Cantilever or clam shell separation will affect the quality of the fine pitch printed deposits. paste media to fill stencil apertures

Step 8: inspection
Checking the printed PCB for correct paste alignment:

- Optical
- 2D/3D auto microscopes



Step 9: pick and place
Placing the components onto the correct pads on the PCB.

Step 10: reflow
Melting the solder to ensure electrical and mechanical connection of the device to the PCB:

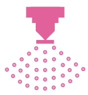
- Convection
- Infrared
- Vapour phase

1.3 Stencils

Surface mount stencils are more than just sheet of metal, with apertures replicating the PCB layout, used to deposit solder paste. They are the single most important tool whose design and optimisation influences the success of every surface mount assembly line.

Achieving success with lead-free printing, either fine-pitch or pin-in-hole reflow, is very similar to using traditional solder alloys. The stencils used have multi-level aspects that contain distinct thicknesses on the same stencil to deliver the individual paste volumes required by the diversity of components on the PCB.

Tecan combines its knowledge of the latest surface mount industry requirements with its extensive manufacturing experience to offer stencil solutions based on three technologies as follows:



Precision etching is used to reduce the stencil thickness locally, creating recesses or leaving raised areas, ready for the apertures to be subsequently created using laser technology.

Laser-cut stencils deliver improved aperture definition and superior dimensional tolerances for finer pitch apertures. In this process each aperture is created consecutively, with larger aperture count stencils requiring more time.

Laser-formed stencils are used to deliver stencil solutions for finest pitch components on very quick turnarounds. The stencil material is electroformed nickel and the apertures are subsequently created using laser technology. During the laser cutting process the laser beam normally liberates trace elements to the cut surface of the aperture walls. Since the material used here is more than 99% pure hard nickel the resultant apertures are naturally polished and offer optimum paste release.

1.3.1 Precision etching technology

- Accurate, cost-effective solution primarily used to reduce the stencil thickness locally to create recesses or raised areas.

Photo Chemical Machining (PCM), or 'photo etching', is a subtractive process that selectively removes metal by chemical action. PCM is an extremely precise and cost-effective process that facilitates the production of a wide range of burr- and stress-free parts and tools in virtually any metal.

Virtually all metals are suitable for Tecan's Photo Chemical Machining (PCM) process. The following metals are most widely used for producing stencils:

- Stainless Steel 302, 304 & 430
- Fine grain steel
- Semi-fine grain steel
- Nickel

1.3.1.1 Multi-level stencils

Multi-level stencil features are created using the Photo Chemical Machining technique. Stepped stencils are created by removing an isolated depth on the squeegee side of the stencil leaving the general thickness untouched.

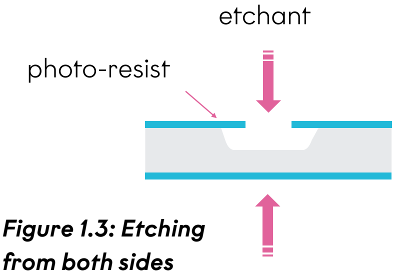


Figure 1.3: Etching from both sides

Stencils with raised areas on the squeegee surface are created by removing the majority of the top surface to leave raised islands.

Figure 1.4 shows a stepped stencil containing a standard thickness of 0.150mm and a locally reduced area of 0.120mm tailored to the requirements of the fine pitch component.

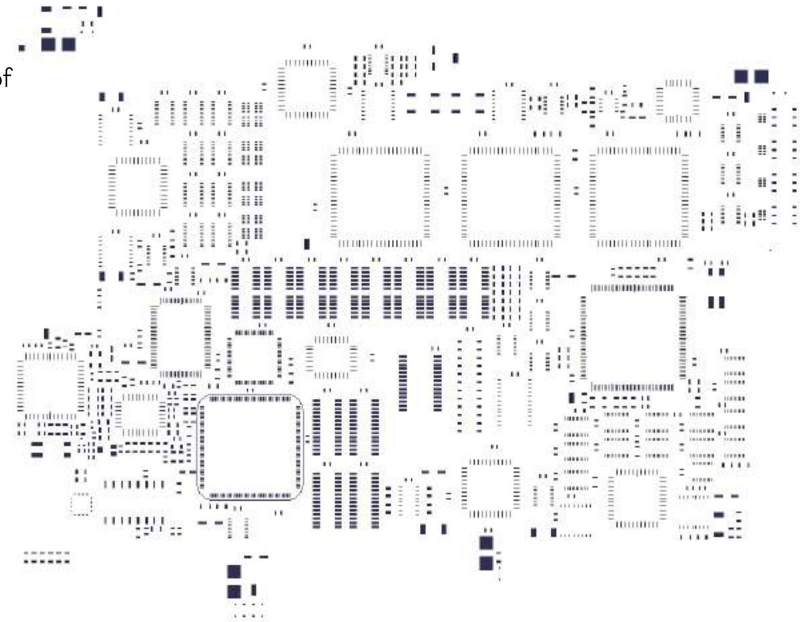


Figure 1.4: Stepped stencil

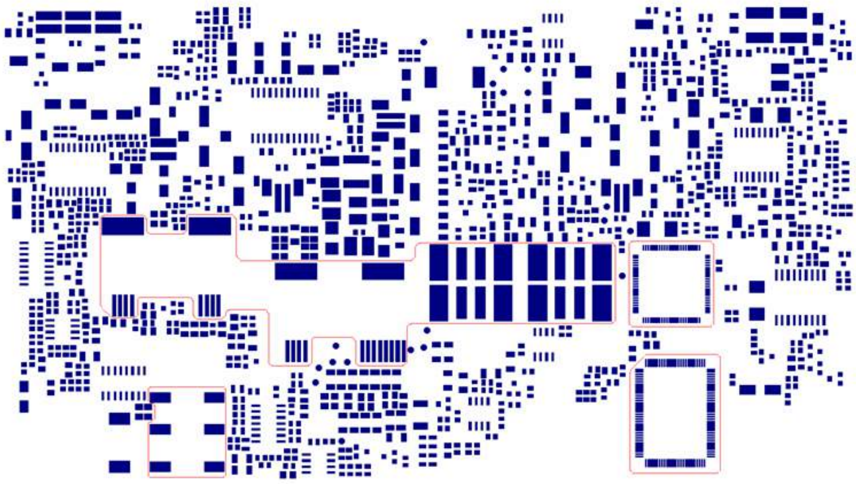
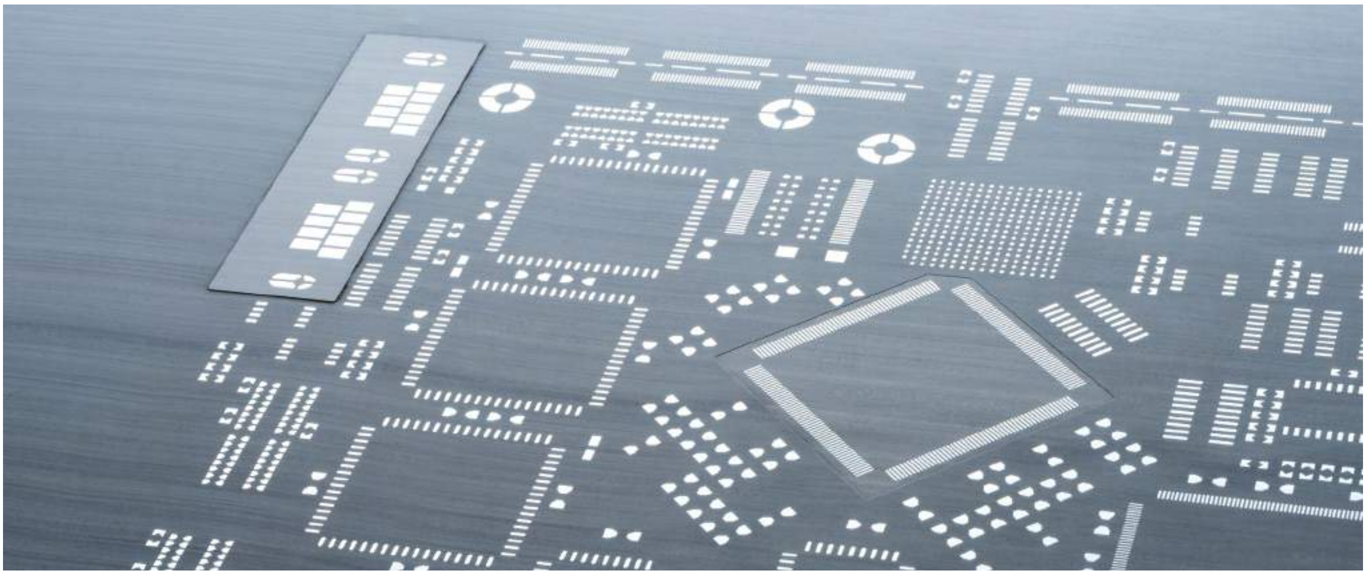


Figure 1.5 shows a stencil that was optimised with three thicknesses: 0.150mm general thickness, reduction to 0.120mm for the fine pitch and up to 0.200mm for the power components and connectors.

Figure 1.5: Stencil optimised with three thicknesses



Typical multi-level steps

	Steps of up to:
Using standard thickness of 0.100mm	+/- 0.025mm
Using standard thickness of 0.125mm	-0.050 and + 0.075mm
Using standard thickness of 0.150mm	-0.075 and + 0.100mm
Using standard thickness of 0.175mm	-0.075 and + 0.075mm
Using standard thickness of 0.200mm	-0.100 and + 0.050mm

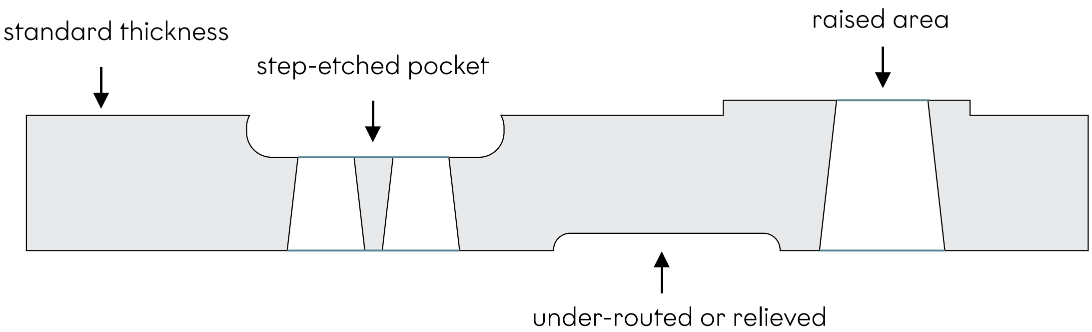
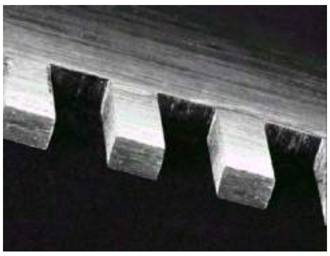


Figure 1.6: multi-level stencil

1.3.2 Laser manufacturing technology

- Improved dimensional accuracy for finer pitch requirements.
- Used for component pitches down to 0.3mm.

Laser cutting machine



SEM of laser cut apertures

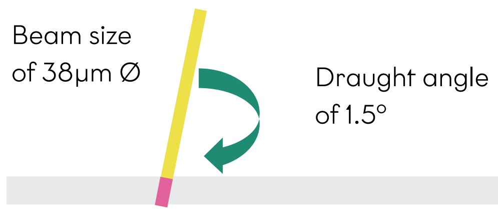


Figure 1.7: Laser cutting

Approximately 1.5° - 2° taper, from squeegee side to release side, provides the necessary trapezoidal aperture geometry which improves paste release to achieve consistent printed deposits for finer pitched components.

Using laser technology the stencil apertures are created consecutively, as such - stencils containing larger aperture counts take longer to complete.

Stencil apertures are cut with a fine beam from inside the aperture (1) towards its boundary (2) and then tracing around (3 - 5) until the beam passes the point where it first met the boundary. The resultant metal shape is then deposited into the vacuum tray beneath. Aperture wall definition and smoothness affect paste release; this is why the speed of cut is fundamental to achieving the results intended.

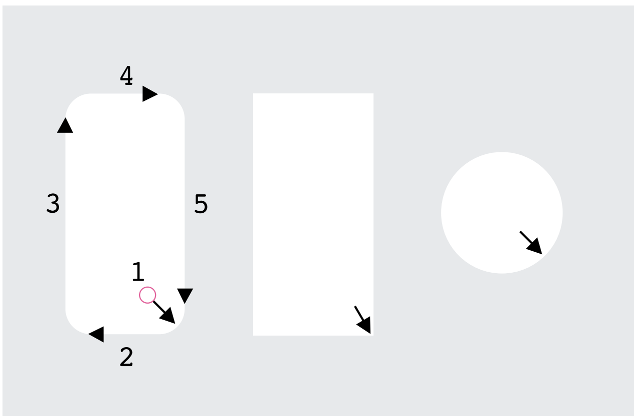


Figure 1.8: Laser-cut apertures

1.3.3 Laser-formed stencils and fine grain steel

- Stencil material is electroformed from nickel and the apertures are lasered.
- Fine grain steel can also be used as base material to improve aperture wall smoothness.

Laser-formed stencils are a hybrid technology offering the precision of laser cut apertures and the enhanced paste release properties of nickel. Improved paste roll activation and multi-level technology are combined with exceptionally quick turnarounds to deliver optimised print deposit consistency.

Fine grain steel can also be used as a base material for stencil thicknesses up to 0.250mm.

When laser cutting stainless steel materials, trace elements are liberated to the aperture walls and as such, care has to be taken to ensure the relative wall roughness doesn't impede the transfer of solder paste.

With laser-formed or fine grain steel stencils the base material results in significant improvements to the aperture wall smoothness to offer enhanced printed deposits.

Laser cutting machine