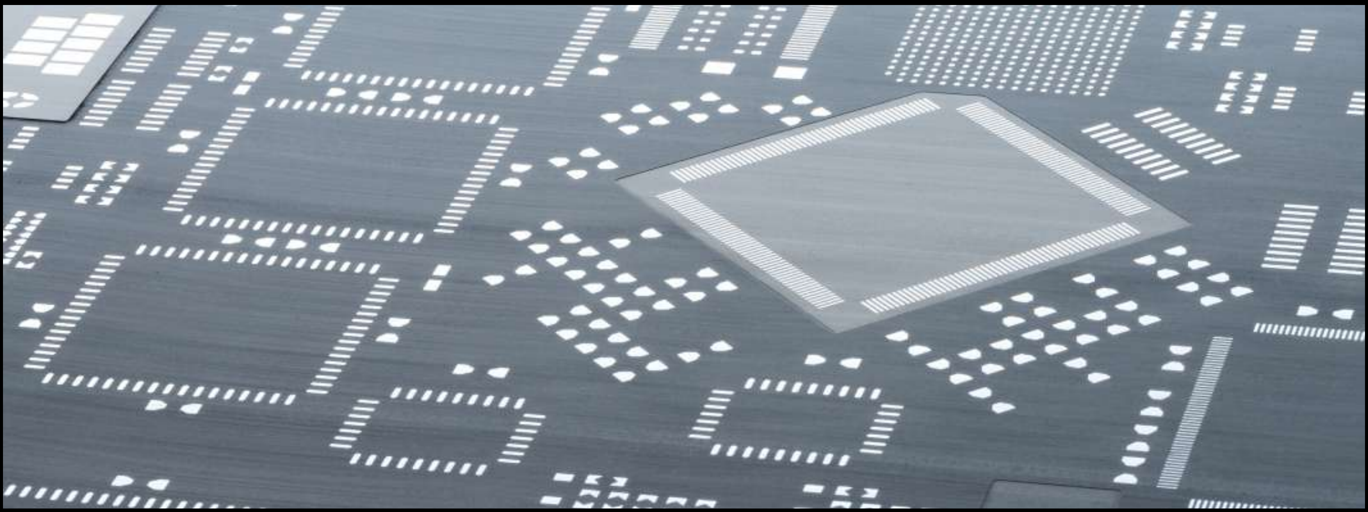
stencils handbook

a comprehensive guide

tecan



innovators in chemical etching since 1970

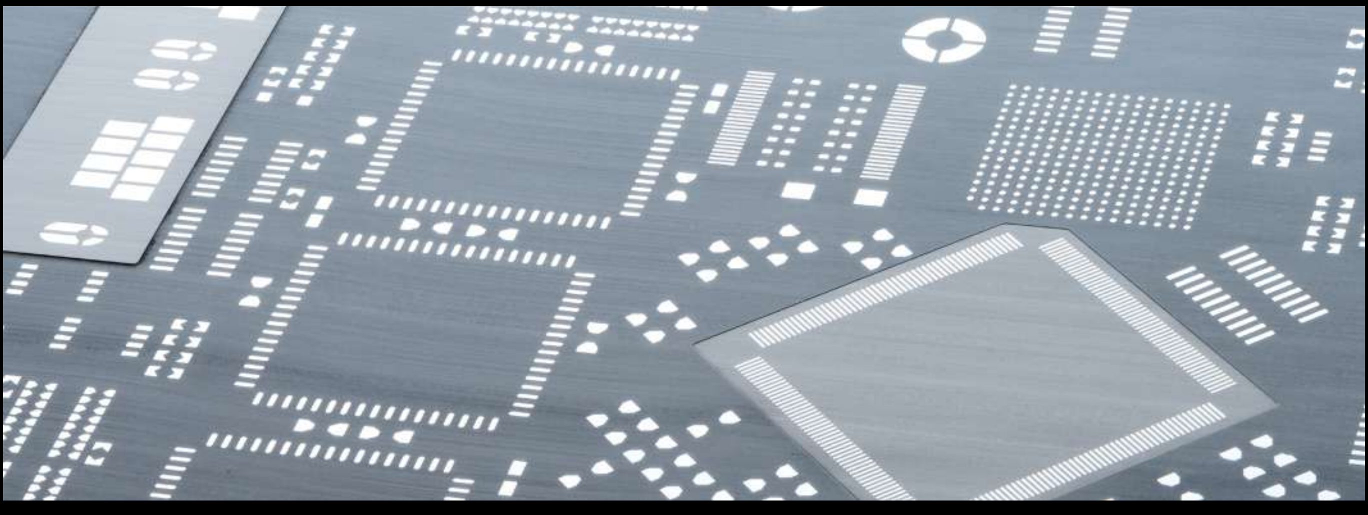
table of contents

In this handbook, we aim to provide a comprehensive guide to SMT stencils considerations, how to use them to optimum effect and their applications.

For further information and to buy online, go to www.stencils.co.uk.

1.0 an introduction to SMT and stencils

for optimum solder paste volume and accuracy



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.

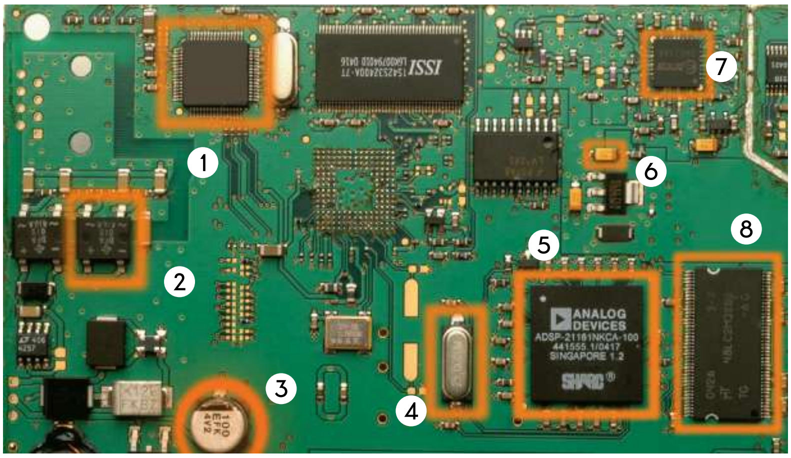


Figure 1.1: Assembly from a RF control application

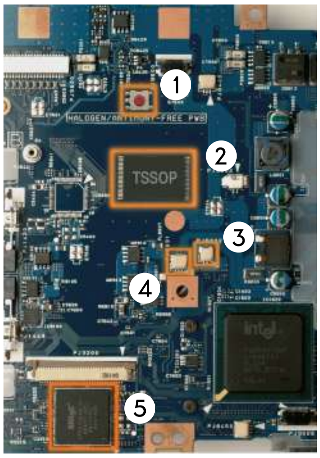


Figure 1.2: PC motherboard assembly

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

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.

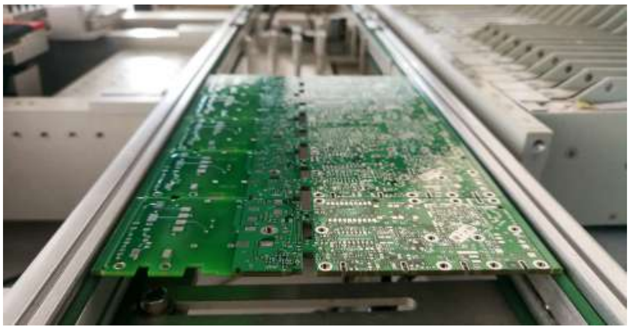
Finally, the boards are visually inspected for missing or misaligned components and solder bridging. If needed, they are sent to a rework station where a human operator corrects any errors. They are then sent to the testing stations to verify that they work correctly.

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.

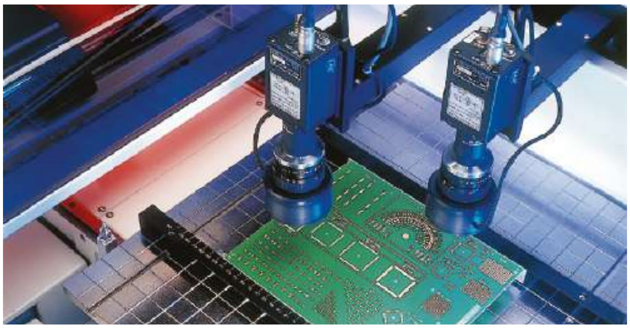
Step 2: printer set-up
The stencil is mounted, positioned; and the printer then calibrated in terms of:

- 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

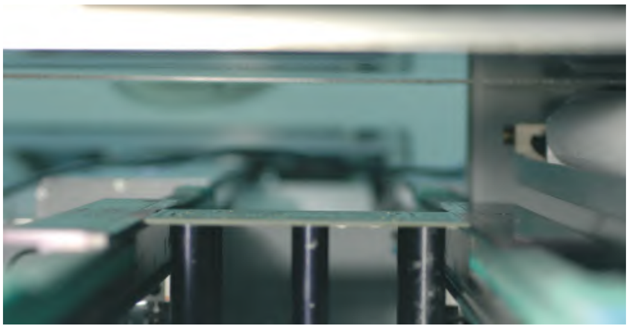


Step 4: image recognition
The alignment or matching of PCB features to stencil apertures with or without fiducials is one of the most critical pre-printing operations. The options include:

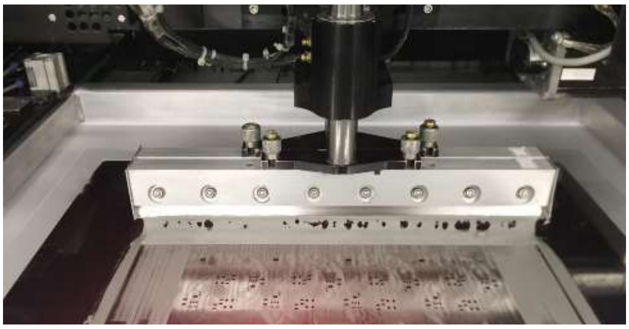
- Hand-eye
- Top down looking camera
- Camera/comparison (using stored images of before and after printing)
- In-line cameras as either:
  - Look-up/look-down, using a split field prism or alternative
  - Look-down/look-down

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

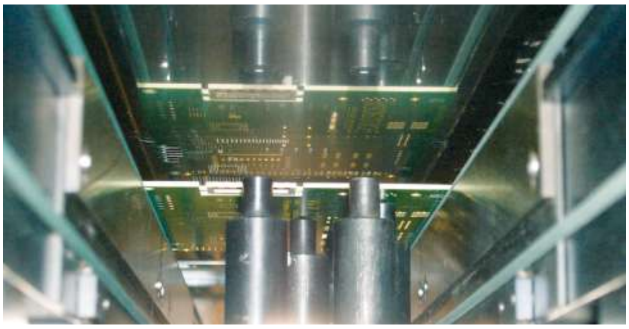
1.2.2. SMT flow diagram



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

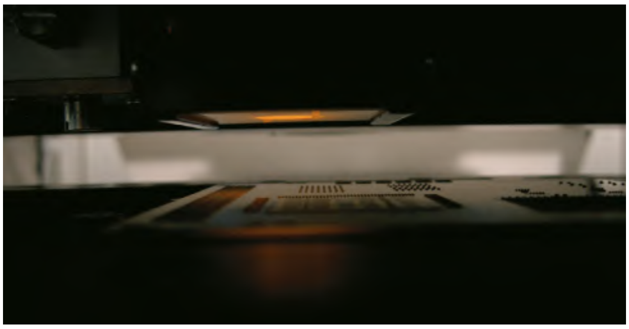


Step 6: print stroke
Travel of the squeegee blade, effectively rolling paste media to fill stencil apertures.



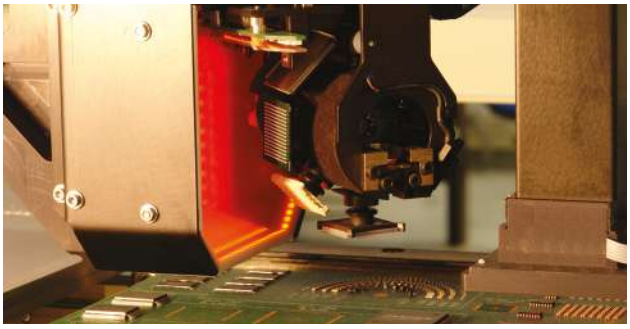
Step 7: separation
Effective separation of the PCB from the stencil for good paste release:

- 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:

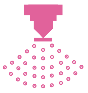
- Convection
- Infrared
- Vapour phase

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

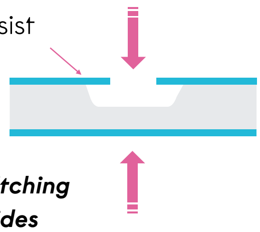


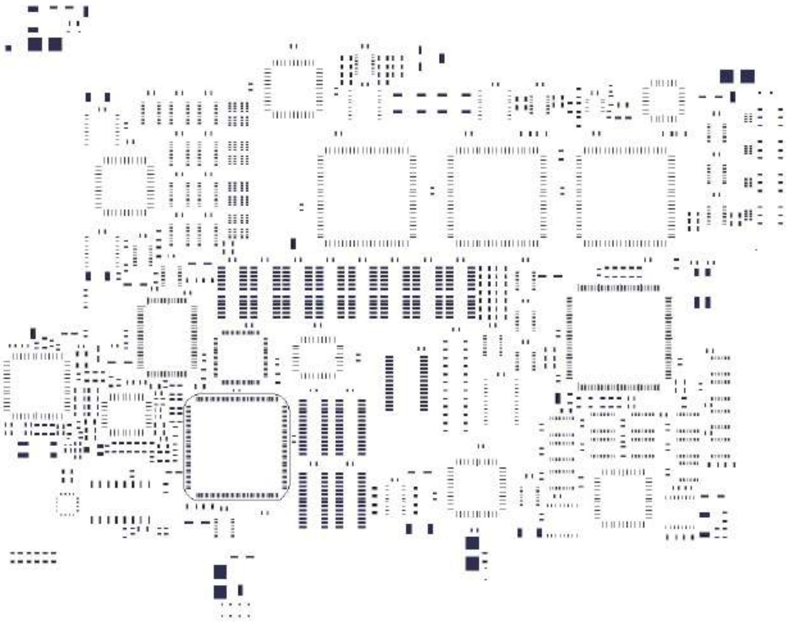
Figure 1.3: Etching from both sides

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.

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



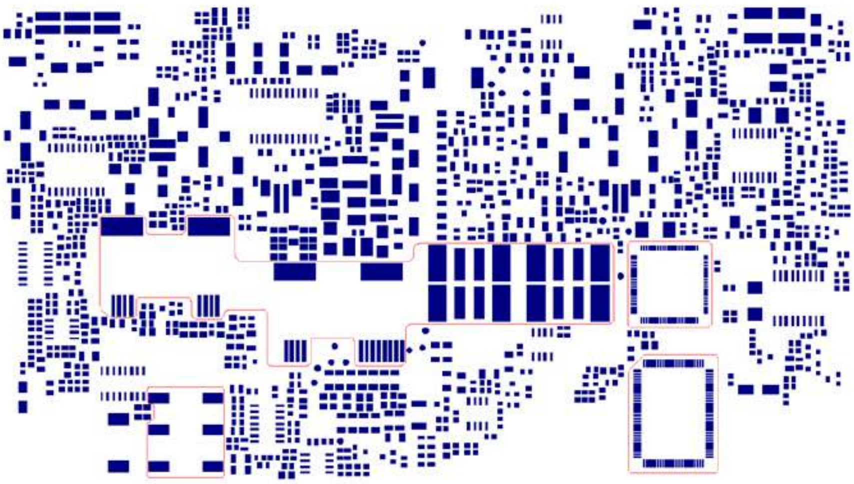
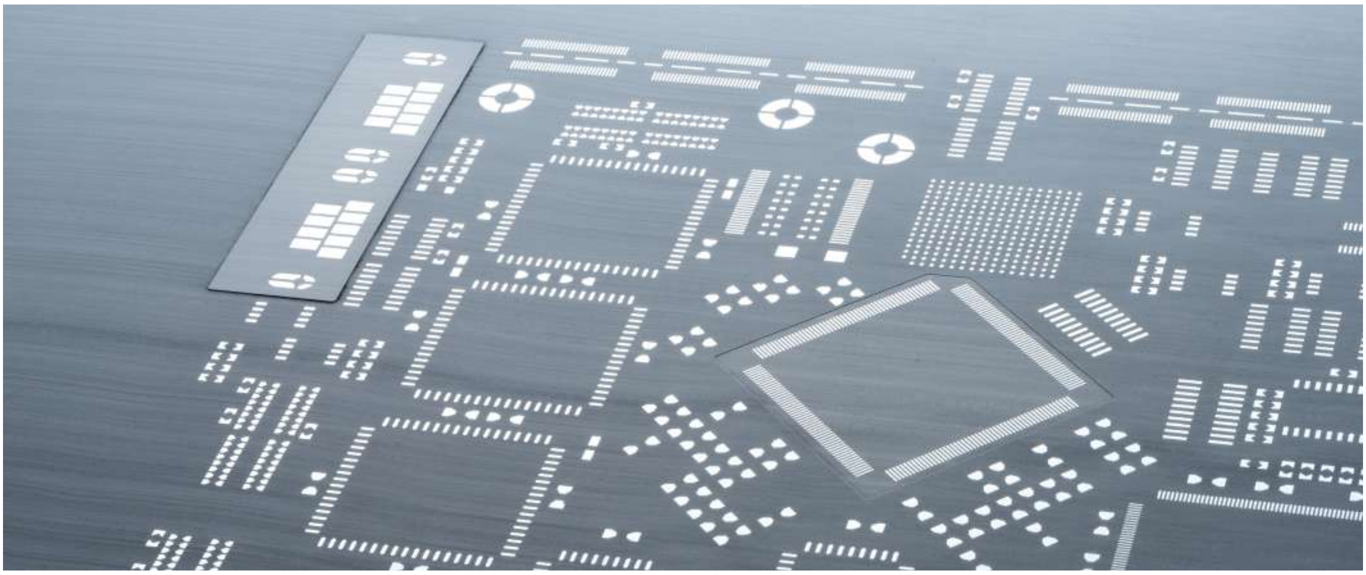


Figure 1.5 shows a stencil that was optimised with three thicknesses: 0.150mm general thickness, reduction to 0.120mm for the fine pitch and up to 0.200mm for the power components and connectors.

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

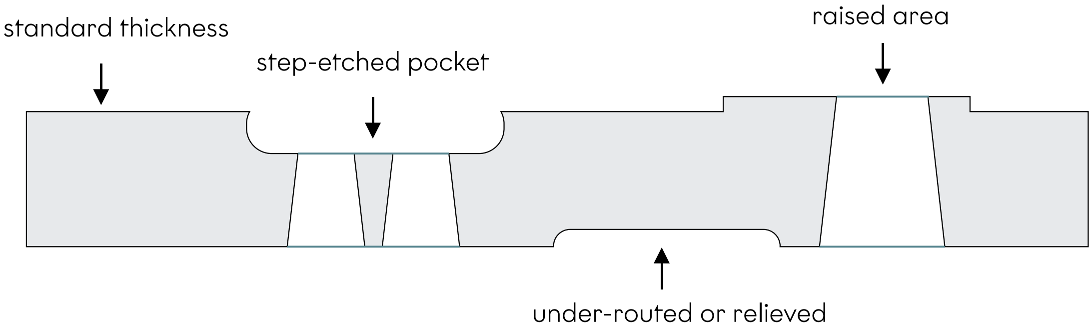
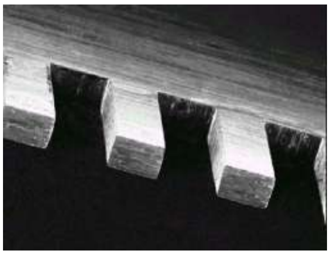


Figure 1.6: multi-level stencil

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

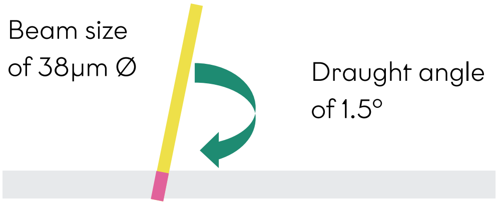


Figure 1.7: Laser cutting

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.

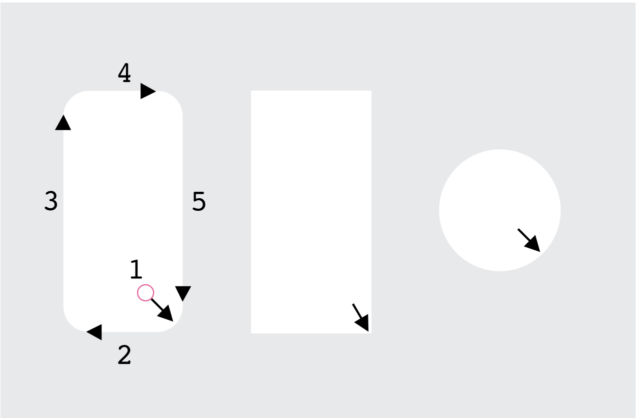


Figure 1.8: Laser-cut apertures

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

2.0 choosing your stencil

which format is right for your application?

2.1 Stencil formats

2.1.1 Remountable stencils

Many years ago, the tension offered by meshed stencils could not be fully replicated and as such image stability was a major concern when adopting remountable stencils for fine pitch printing. Today however, there are a number of four-sided tensioning frames in the market that are capable of ensuring optimum stencil tension and long stencil life.

Remountable stencils offer three major benefits:

- Space saving of up to 80%
- Cost saving of up to 65%
- No issues regarding selection of cleaning solutions to be compatible with adhesives

2.1.1.1 Genesis

Figure 2.1 Genesis mechanically tensioned stencil

Tecan's proprietary tensioning system, Genesis, enables foils to be quickly interchanged and held absolutely flat with equal and maintainable planar tension across the whole stencil.

This simple mechanical mounting system uses no adhesives - which may degrade over time, has no need for pneumatics - which are susceptible to rupture - and does not require a loading jig.

Description	Size	Tecan part number
Genesis 58 frame	584mm x 584mm x 25.4mm	FRG58
Genesis 73 frame	786mm x 786mm x 38.1mm	FRG73
Genesis frame adaptor	Convert a Gen 58 into a 73	FRGADAPT

2.1.1.2 OptiGuard

Figure 2.2 OptiGuard stencils for VectorGuard frames

A patented joint development between ASM and Tecan, OptiGuard is designed for use with standard VectorGuard™ frames.

Pin bars are used to mount the foil into an extruded aluminium profile, which is ideal when stepped or multi-level stencils are required and can be used with laser-cut, precision etched, or laser-formed stencils.

The foil is then mounted into the VectorGuard frame using air pressure.

Description	Size	Tecan part number
VectorGuard VG260 frame	584mm x 584mm x 30mm	FRV2323
VectorGuard VG265 frame	584mm x 736mm x 30mm	FRV2329
VectorGuard VG265 wide frame	736mm x 736mm x 30mm	FRV2929
VectorGuard VG248 frame	584mm x 584mm x 26.4mm	FRV248
VectorGuard frame adaptor	Convert a VG260 to VG265 or VG265 wide	FRVADAPT

2.1.1.3 TetraBond™

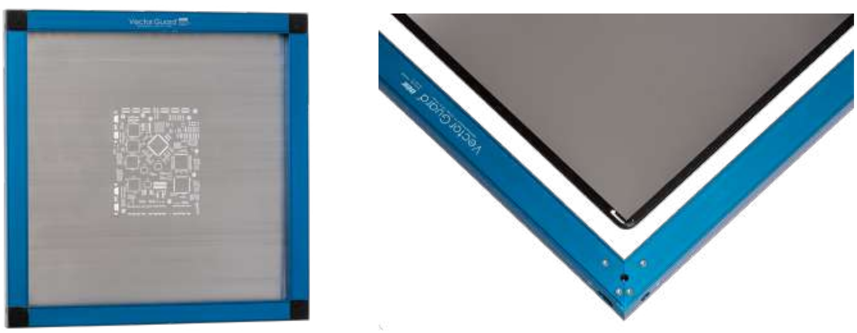
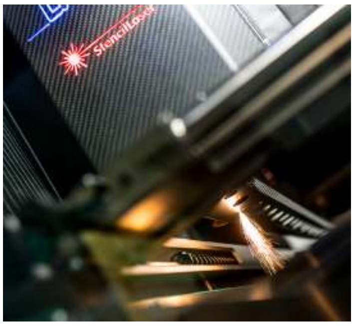


Figure 2.3 TetraBond stencil and VectorGuard frame

A simple system for safe mounting and demounting, designed to optimise rigidity in the stencil.
Foils are mounted into a thin aluminium extrusion using an advanced bonding system and are designed for use with Tetra™ and VectorGuard frames, whilst being backwards compatible with most other frames in the market.

2.1.1.4 Other formats



In addition to these most popular formats, there are a number of alternative remountable framing systems on the market. Tecan supports stencil production/ capability for most of these, including:

- Tecfoil
- Apex
- FTS
- LPKF
- ZelFlex

2.1.2 Meshed stencils

Figure 2.4 Stencil meshed in aluminum frame

Traditional meshed stencils, where the metal stencil element is bonded on to an aluminium frame, are still preferred by some operators.

Tecan only uses high grade epoxy adhesives and polyester fabrics in the meshed stencil manufacturing process as although meshed stencils are tensioned in four directions, the mesh material tension can degrade over time with use. Also, alternative adhesive bonds - between the frame and mesh and the mesh and stencil - may degrade with some cleaning solutions available.

Offset image positioning can often result in the start of the print stroke being close to the glue bond areas. When using semi-automatic or automatic printers with meshed stencils, don't be tempted to make the print stroke too long or damage to the glue bonds and mesh border could result.

Typical frame sizes (mm):

- 434 x 434 x 25.4mm
- 508 x 508 x 38.1mm
- 503 x 404 x 25.4mm
- 584.2 x 584.2 x 38.1mm
- 622.3 x 392 x 19.05mm
- 650 x 550 x 30mm
- 798 x 578 x 28/35mm
- 736.6 x 736.6 x 38.1mm
- Other sizes available on request

2.1.3 Extended stencils

Figure 2.5 Extended stencil meshed into aluminum frame

Tecan's extended precision screens are available in sizes of up to 1800mm x 900mm. Typically used for specialist applications such as OLED and LCD display technologies, these bespoke meshed stencils provide the same level of printing accuracy as standard SMT sizes.

2.2 Stencil thickness selection

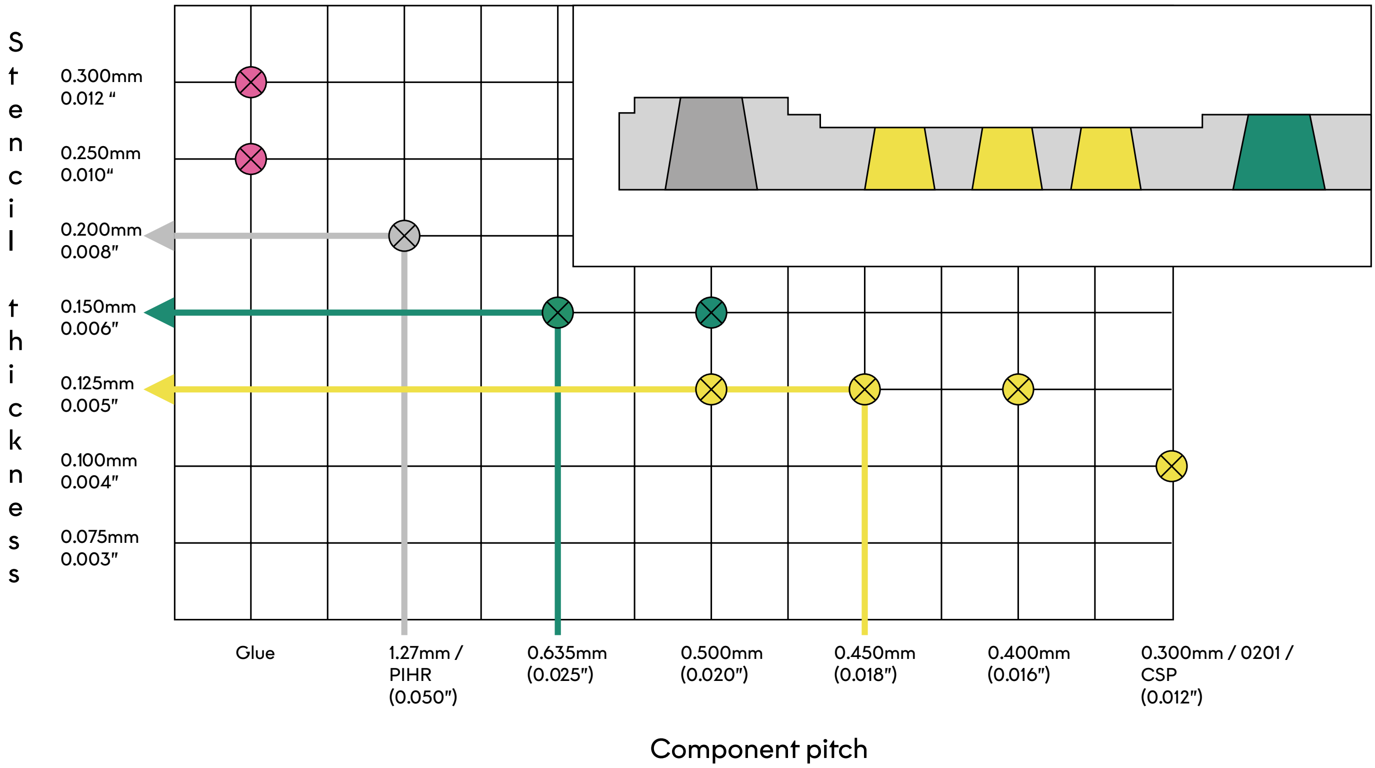


Figure 2.6 Component pitch vs stencil thickness

Stencil thickness ultimately determines the volume of solder fillet that is available for the component terminations. Any aperture design modifications, used to optimise the printing process, can be compromised by selecting an inappropriate thickness. Too thin and the fillets required may not be achieved. Too thick and paste retention may occur effectively starving the solder paste volume.

Figure 2.12 above highlights the different stencil thickness requirements for a range of surface mount components often found together on a typical assembly. In deciding which stencil thickness will give the best results it is always worth considering a multi-level stencil as this is the best way to provide components with conflicting paste volume requirements, the specific individual paste volumes they require.

The aspect ratios of materials currently available are listed below:

Stencil material	Simple aspect ratio
Stainless steel	690:1
Fine grain steel	615:1
Nickel	615:1

2.2.1. Multi-level

Multi-level stencils have become the preferred solution for many organisations due to their superior print quality and paste release properties. Other benefits include lower printed defects; less paste bridging, higher production yields and optimised quality.

Multi-level stencil features are created using photo chemical machining and subsequent laser-etching.

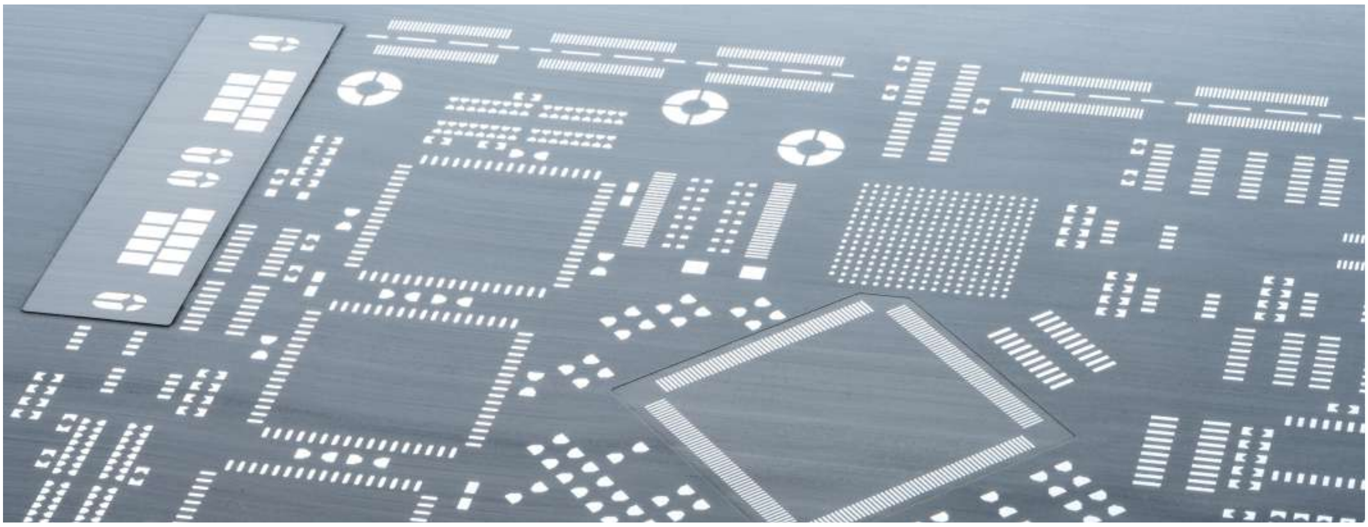


Figure 2.7 Multi-level stencil with raised and recessed areas

2.2.2. Stepped stencil

Stepped stencils are created by removing an isolated depth on the squeegee side of the stencil leaving the general thickness untouched.

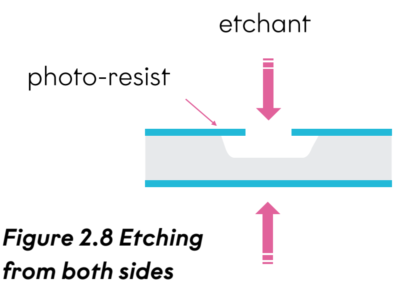


Figure 2.8 Etching from both sides

Stencils with raised areas on the squeegee surface are created by removing the majority of the top surface to leave raised islands. In these cases it is better to select nickel as the stencil material since it isn't affected by removal of the majority of one skin.

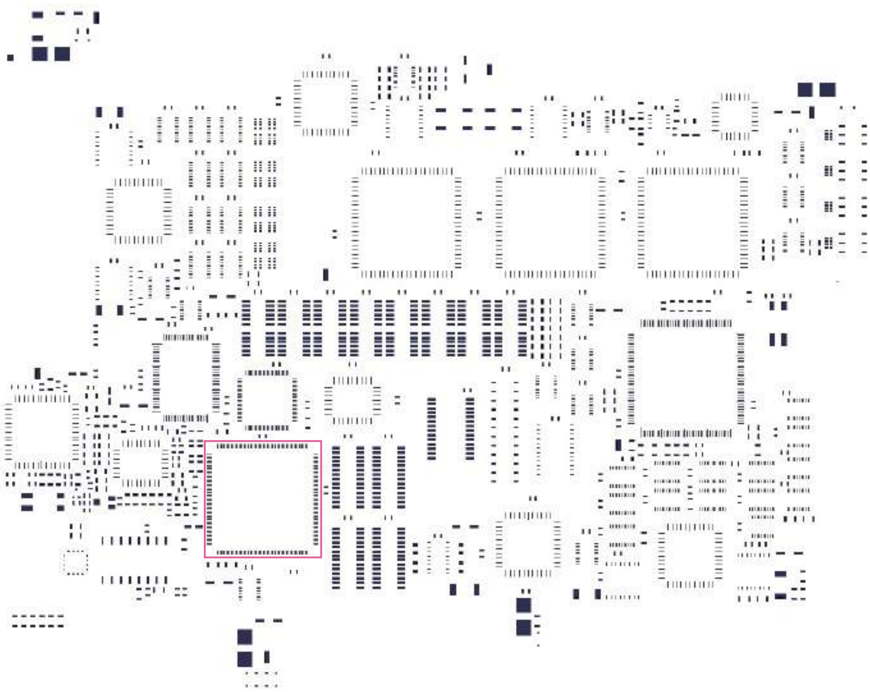


Figure 2.9: Stepped stencil containing a standard thickness of 0.150mm and a locally reduced area of 0.120mm tailored to the requirements of the fine pitch component

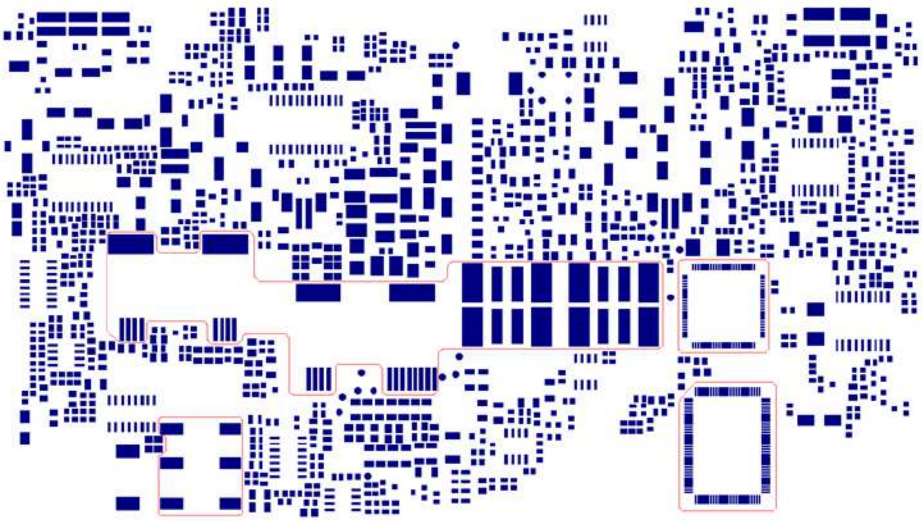


Figure 2.10: This stencil was optimised with three thicknesses: 0.150mm general thickness, reduction to 0.120mm for the fine pitch and up to 0.200mm for the power components and connectors.

Stencils with both raised portions and reduced thicknesses are the result of two separate processes.

2.2.3 Under-routed stencils

Localised pockets can be provided on the underside of the stencil to accommodate thicker solder pads, test points and badly registered solder resist. This enables optimum stencil gasketing.

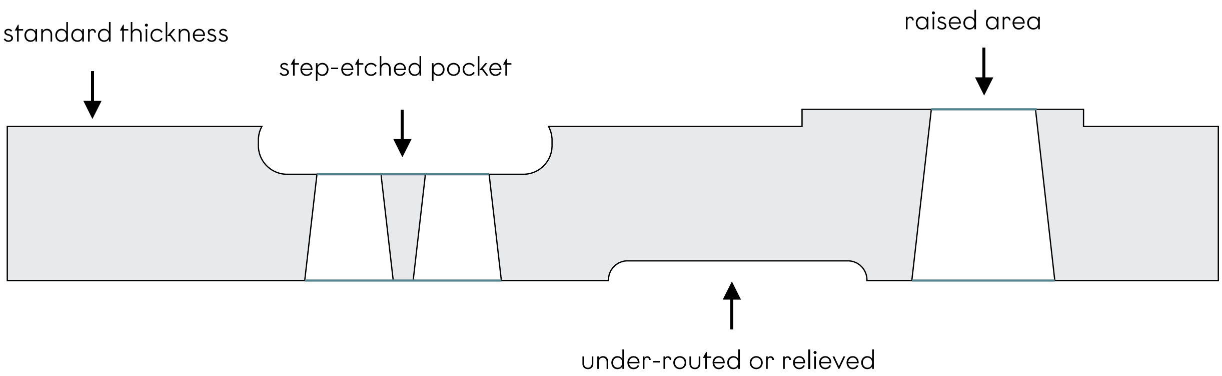


Figure 2.11: Multi-level stencil with under-routing

2.3 Aperture design

2.3.1 Aperture reduction

So why is it necessary to reduce stencil apertures from the CAD Gerber designs? It is recommended that stencil apertures are generally reduced from the copper feature sizes to be printed. Exceptions to this rule include BGAs, µBGAs, components with a known poor co-planarity and pin-in-hole reflow applications.

Let's consider that the Gerber is used as a datum from which both the PCB substrate and the stencil apertures are created.

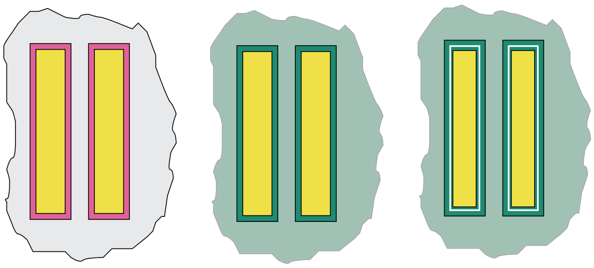


Figure 2.12: Designs can be created in CAD which are accurate to fractions of a micron. PCB fabricators amend the CAD design by adding etch factors to their photolithographic tools to realise the copper feature sizes required ± 50-70µm. Obviously thicker deposits of copper require increased tolerances.

PCB fabricators work to a manufacturing tolerance. Understanding the processes involved in producing the substrates can help to achieve successful printed results.

It is no good to design aperture footprints with IPC standards in mind if the PCB has been accepted with over-etched features.

Although today's stencils can be produced with tolerances of between 5-9µm, the loss of any stencil to PCB gasketing surface can lead to under-stencil contamination and ultimately bleeding, bridging shorts and rework.

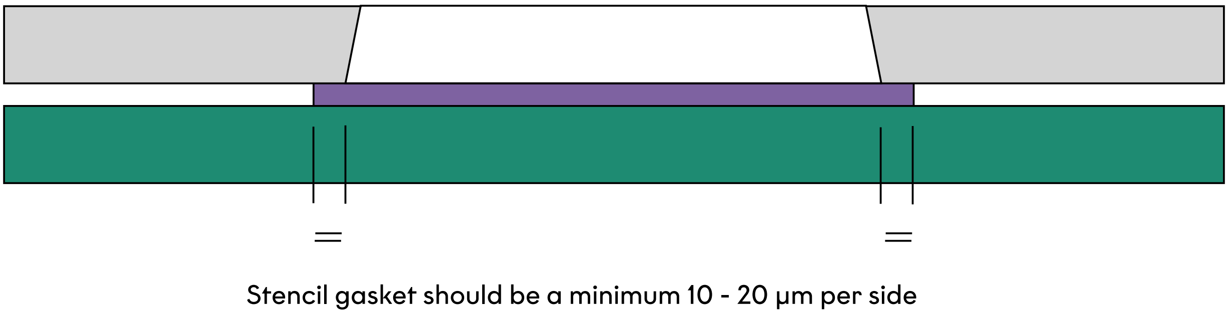


Figure 2.13: Stencil gasket

Printing accuracy must always be considered, even with the best printing machine available an acceptable tolerance on its accuracy is often ± 12 - 25µm so aperture reduction must accommodate this tolerance.

Reductions also allow for any paste slump: the "collapse" of the paste bricks from their printed formation due to environmental conditions and any paste spread associated with pick and placement pressures.

2.3.1.1 Aperture reduction example

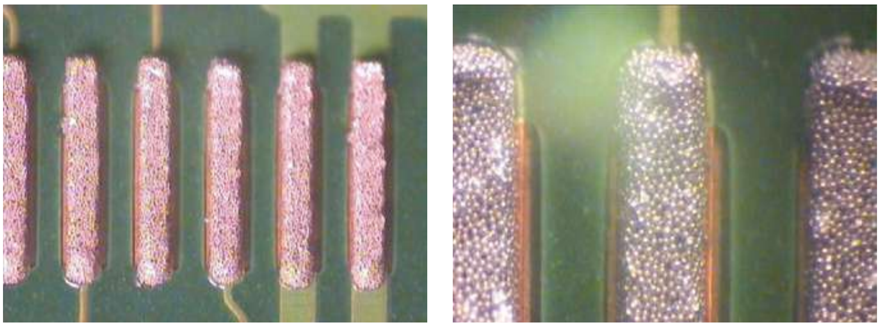


Figure 2.20
The paste has been printed accurately and the resultant brick deposits are uniform and consistent. Close up inspection reveals the evidence of a gasket surrounding the copper features.

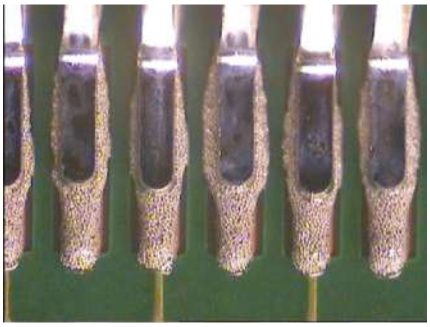


Figure 2.21
In this example the copper features were not over-etched but poor lead co-planarity caused several of the adjacent paste deposits to splay and almost merge after pick and placement.

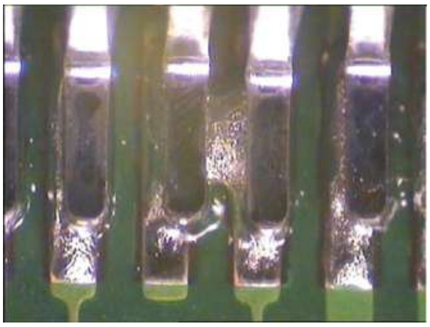


Figure 2.22
The results of reflow can be clearly seen in this example. A reduction in the width of the stencil apertures overcame the issues facing this company.

2.3.2 Aperture shapes

Simply reducing all the stencil apertures by a global dimension or percentage cannot offer the advantages that specific reductions or aperture shape modifications, tailored to individual components, can achieve to reduce end of line surface mount assembly defects.

For those engineers who recognise the problems but may not be aware of the solutions, Tecan can design the stencil aperture sizes and shapes to suit.

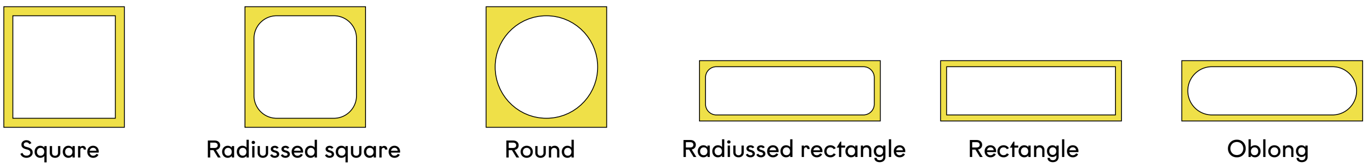


Figure 2.17: Stencil aperture shapes

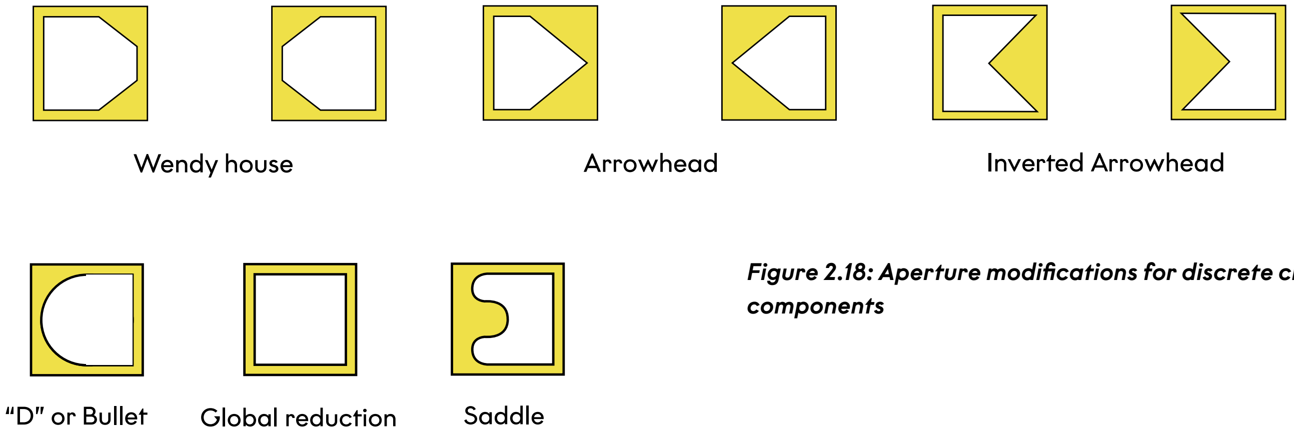


Figure 2.18: Aperture modifications for discrete chip components

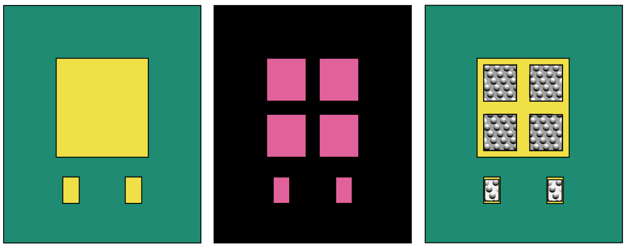


Figure 2.19: Modifications for D-PAK components

With the base pads of D-PAK components it is often necessary to reduce the outside dimensions of this pad and then window the resulting aperture to offer the solder paste volume required by the component without causing unnecessary component float or paste starvation.

2.3.3 Mid-chip solder ball elimination

Figure 2.20: Mid-chip solder ball

Solder beading or mid-chip solder balls are the result of excess solder paste beneath the non- wetting surfaces of discrete components. Upon reflow this paste is not able to retract and when the component is drawn towards the PCB it is squashed to emerge from beneath the component.

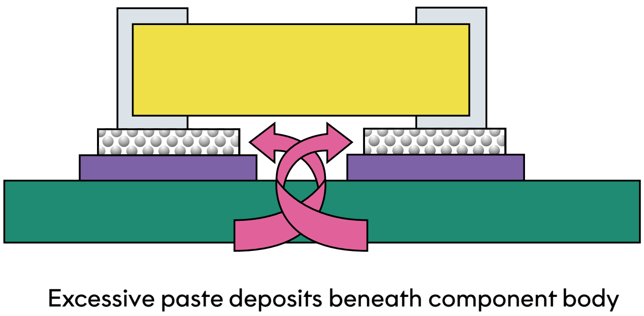


Figure 2.21: Excessive paste deposits beneath component body

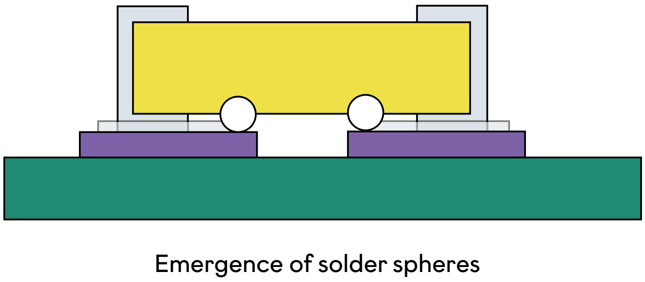


Figure 2.22: Emergence of solder spheres

3.0 nanocoating and SMT stencils

for quality results and a cleaner process

3.0 Nanocoatings

With a thickness of no more than 1-100 nanometers, nanocoatings are ultra-thin layers or chemical structures that are applied to surfaces by a variety of methods and applied to a wide range of substrates and chemically bond with non-porous surfaces. To put this in perspective, consider the thickness of paint used in the automotive industry of which is typically 125 microns or 125,000 nanometers.

While they can be one molecule thick, multiple molecular layers can be built up to deliver a particular chemical or physical property to a surface such as water-resistance (hydrophobic) and oil-resistance (oleophobic).

3.1 Nanocoating SMT stencils

The underside of solder paste stencils can be nanocoated at a typical thickness of ~5 nanometers to provide a non-stick surface which:

- reduces the number of cleaning cycles required during the paste printing process
- improves paste transfer efficiency for fine pitch apertures

Nanocoating SMT stencils delivers immediate and measurably improved results from the first print and can also be applied on the production line to previously used stencils.

3.2 When to use nanocoatings

There are a number of factors to take into account when considering whether to nanocoat a stencil:

1. What is the smallest aperture and pitch on the stencil and what is its thickness?
In the case of a large board with large appertures, the benefit of nanocoating is insignificant. On the other hand, stencil performance can be considerably improved in the case of a dense image with fine pitches and small apertures.

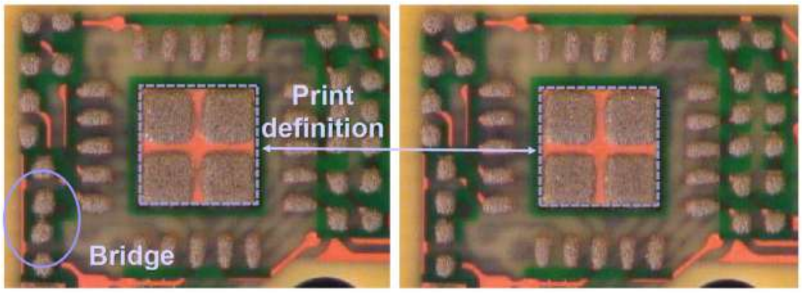
2. How many prints are you using the stencil for?
If you're simply printing a prototype, nanocoating is not necessary but where volume runs are concerned, nanocoating significantly reduces the frequency of cleaning cycles.

3. What paste are you using?
In the case of SMT stencils, the denser the paste you are using, the greater the benefit of nanocoating.

3.3 Nanocoating benefits

3.3.1 Better quality printing

Because the flux is repelled from the aperture walls by nanocoating, there is a reduction in bridging, resulting in better results. Figure 3.1 below shows the print definition improvements that can be achieved with a nanocoated stencil, highlighting QFN and 0201 devices after ten prints with no wipe using the same board, same stencil design and same print stroke.



Results from stencil with no nanocoating

Results from nanocoated stencil

Figure 3.1: Print definition improvements with nanocoated stencil. Source: "Fine Tuning the Stencil, Manufacturing Process and Other Stencil Printing Experiments", Shea C. and Whittier R., Proceedings of SMTA International.

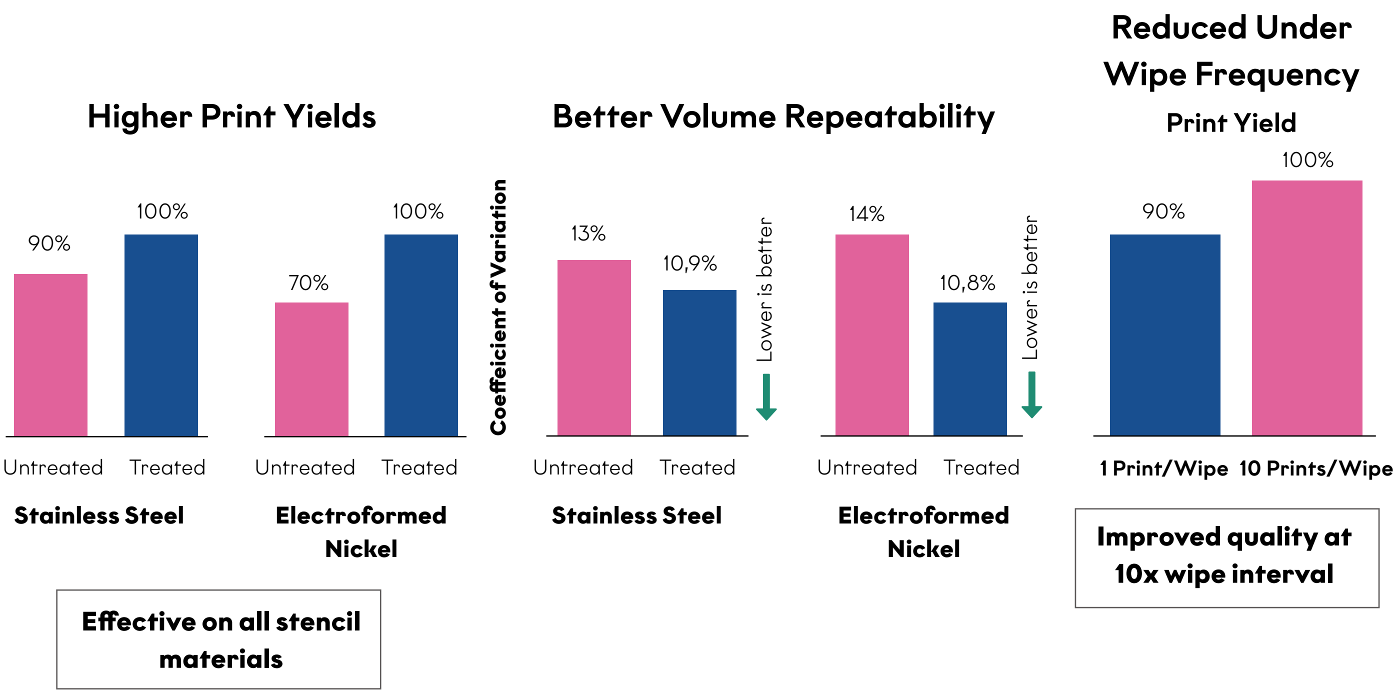
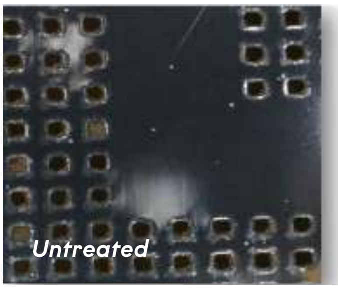


Figure 3.2: Results of 10-print test in large DOE. Source: "Fine Tuning the Stencil, Manufacturing Process and Other Stencil Printing Experiments", Shea C. and Whittier R., Proceedings of SMTA International.

3.3.2 Improved productivity, reduced costs

Flux-resistant nanocoating applied to the underside of the stencil and stencil aperture walls can boost productivity and reduce costs for volume runs:

- less underwiping is required
- less downtime for paper changes
- there is less damage to stencil mountings, particularly for meshed stencils, from exposure to aggressive cleaning solvents.
- lower paper and solvent consumption



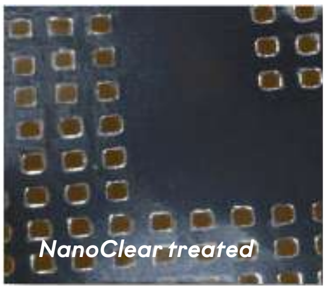


Figure 3.3: Comparison of untreated and NanoClear treated stencil.
Source: https://www.aculon.com/nanoclear-stencil-wipes/

3.4 SMT nanocoatings available from Tecan

A choice of nanocoatings are available for all Tecan stencils both of which create a robust, abrasion resistant surface that stands up to repeated cleaning.

- **MicroShield** is a two-part coating that is applied by Tecan prior to stencil dispatch
- **NanoClear®** coating from Aculon is supplied in a pouch containing both the primer and nanocoating to be applied on the production line.

3.4.1 MicroShield

This on-contact coating has a unique chemistry. Upon contact, it forms a self-assembling monolayer that is highly hydrophobic and oleophobic. It is applied to Tecan's solder paste stencils after they are cut and prior to dispatch. MicroShield demonstrates both the printing and cleaning benefits of nanocoating.

MicroShield does not "cure" like a traditional polymer coating but instantly transforms the surface on-contact. Performance typically improves with time.

Physical Properties	Values
Appearance	Clear
Specific Gravity @ 23 °C	0.80 g/cm^3
Viscosity @ 23 °C	2.1 cP
Nonvolatile content	1 %
Static contact angle, water	103 Degrees
Static contact angle, n-hexadecane	69 Degrees
Abrasion resistance, ASTM D2486, Isopropyl Alcohol	>2000
Abrasion resistance, ASTM D2486, IPA Based Flux	>2000
Pencil hardness	N/A

Figure 3.4: MicroShield test results

3.4.2 NanoClear®

NanoClear is a SAMP Coating (Self-Assembling Monolayer Phosphonate) supplied in a two-part pouch and can be applied by SMT operators to a new stencil on the production line or to an existing stencil to improve performance.

Aculon NanoClear repels flux which improves print quality, increases efficiency, lowers total costs and enhances printing with small apertures.

Application of NanoClear® is very simple. With a NanoClear® dual wipe, a water source, and just five minutes of time you can have an SMT stencil that has been properly treated with a robust and high performing coating.

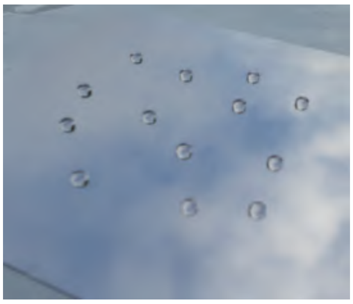
Figure 3.5 NanoClear SMT Stencil Nanocoating"

3.4.2.1 Applying NanoClear

Step one (unprimed stencil)
Take the clean stencil and hold under running water for a few seconds, if the water does not wet out evenly onto the stencil (see figure 3.6), then the stencil still has surface contaminants and needs to be primed with Aculon Primer (Part A).

Figure 3.6: Uneven wetting

Step two (primed stencil):
After using Aculon Part A rinse the stencil under running distilled or deionised water for at least 60 seconds. Please note a primed stencil will have an unbroken film of water that should remain on the surface without beading up for 30 seconds. If the stencil looks like this, then completely dry with a cleanroom wipe and proceed to the application of Part B.



Step three (treated stencil):
Upon opening, immediately apply Aculon Nanoclear Part B by wiping it on the stencil for one minute. Wipe off excess coating with a cleanroom wipe. You can test a stencil's performance by evaluating if water droplets bead tightly on the surface (as shown in figure 3.7) and shed from the stencil easily when tilted.

Figure 3.7: Tight beading on stencil surface

3.5 How long does a nanocoating last?

Aculon states that users typically report **25K print cycles**, although durability depends on many factors, including:

- Abrasiveness of wiper paper/fabric
- Wipe frequency
- Solvent or dry wipe
- pH of under wipe and off-line cleaning solvents
- Solder paste chemistry

In order to maximise durability, Aculon suggests:

- Use soft, non-abrasive understencil wiper paper such as Eco Roll SCER360 or Hyperclean PP4200
- Use a solvent wipe rather than a dry wipe – engineered solvents are best for lead-free no-clean pastes
- Use pH neutral cleaners
- Reduce understencil wipe frequency

4.0 SMT printing parameters

understanding the print cycle and the factors that can affect board assembly

4.0 Printing parameters

Success with SMT printing depends on many factors, the most significant being:

- Printer set-up
- PCB flatness and solderable finish
- Solder paste, its condition, solder sphere size
- Squeegee blade or paste delivery system
- Environmental conditions
- Operator's experience
- Stencil design and optimisation

Figure 4.1 Printer set-up

4.1 The printing cycle

A printer must be able to carry out several operations to complete the print cycle, which comprise:

1. Transporting the substrate into position and contacting the stencil
2. Alignment of stencil to board
3. Application of solder paste - enabling paste roll and aperture filling
4. Controlled separation of the substrate from the stencil
5. Inspection of solder paste deposits (optional)
6. Under screen cleaning to remove stray solder balls and contamination (after predetermined number of cycles)

The printing cycle time is an important factor in determining the line capacity or throughput. Typically, the printing cycle is completed between 9 seconds - for mass manufactured items such as mobile phones - up to approximately 25 seconds for small batch runs involving fine pitch. Today, the more significant factors are pick and placement and reflow times. Often separate offline operations are used for inspection to avoid delays to the total cycle time.

The sequence of printing operations is as follows:

1. PCB enters printer and travels to board stop.
2. PCB is clamped.
3. Printer camera moves to fiducial 1 and locates its position
4. Printer camera moves to fiducial 2 and locates its position
5. Camera moves to rest position away from printing operation
6. Stencil is aligned
7. PCB support/platen moves up and PCB contacts underside of stencil
8. Squeegee operation commences
9. Board support moves down
10. Board is unclamped and exits machine on the conveyor

This description of a printer integrated into a production line also applies to standalone printers in respect of board fixturing, alignment of substrate to stencil, board separation and cleaning.

There are many variations in the way mounting, alignment and cleaning are implemented, and standalone or shuttle printers will also handle boards manually rather than automatically.

Printers can vary from the simplest hinged frame and platen arrangement, where the printing is achieved using a manual squeegee blade, through semi-automatic printers offering control of the squeegee pressure and print stroke, to fully automatic printers that include all the features required of a flexible inline machine. In essence, the finer the component pitch being printed combined with the quantity being produced and the quality standard offered defines the type and attributes of the printer required.

Left: Figure 4.2 Typical semi-automatic printer
Right: Figure 4.3 Typical automatic printer

4.2 Board mounting

The PCB or substrate is usually transported into and out of the printer by conveyor belts. The PCB arrives in the working area and is stopped in the desired position using either mechanical end stops or an optical sensor. It is essential that the PCB is clamped rigidly in position to prevent lateral movement, whilst also being supported to resist the downward forces during the print stroke, which would otherwise lead to warping and solder bleed under the stencil.

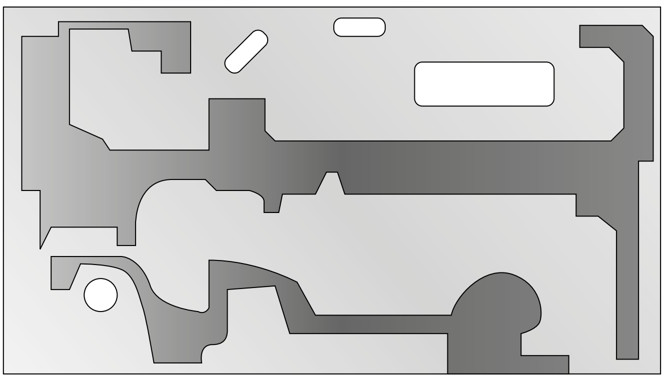
Underboard support can take several forms, including:

- A dedicated tooling plate with dowel pins to align the board
- A matrix of manual or programmable universal mounting pins

The exact solution chosen will depend on the application, but providing adequate support during second-side printing can be problematic, especially when the first-side assembly is densely packed with components, or the board is thin or flexible. In such applications a dedicated tooling plate can be employed which is machined to accommodate the components. This often provides improved support which is better than a bed-of-nails fixture, especially around the extents of the substrate.

Figure 4.4 Pin board support

A 3-D profiled aluminium nest plate can be created complete with supporting pillars and routed areas to accommodate the first-side components whilst offering a flat substrate to the underside of the stencil.



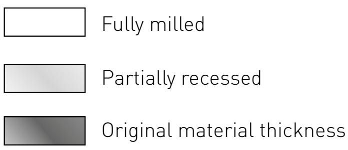


Figure 4.5 Nest for second-side printing

4.3 Board clamping

Clamping the PCB to the conveyor rails can be achieved mechanically, using edge clamps which are thin enough not to impede squeegee travel (and which are correspondingly sharp!). It must be remembered that using clamps on PCBs where the surface mount features are too close to the edge of the board can and does lead to bleeding, bridging and or insufficient paste deposits.

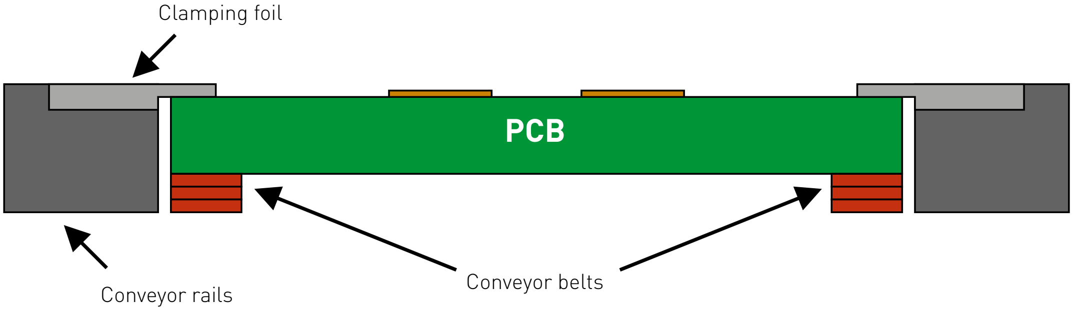


Figure 4.6 Clamping foils

Some PCB designs do not have salvage or snap off regions whilst also containing fine pitch components very close to the edge of the PCB itself. Although with the older shuttle printers this is not likely to be a problem, modern inline printers utilise clamp foils to retain the PCB in registration throughout the print cycle. Even with these thin foils it is not possible to ensure the stencil gaskets to the PCB - thereby creating paste release problems, paste retention or stencil clogging and also paste bleeding bridging and short circuits.

Where possible ensure the PCB design has a snap-off or salvage area whenever fine pitch components are located in close proximity to the finished PCB edge. The red shaded area shown in figure 4.7 represents approximately 2-4mm where the stencil cannot achieve good contact with the substrate.

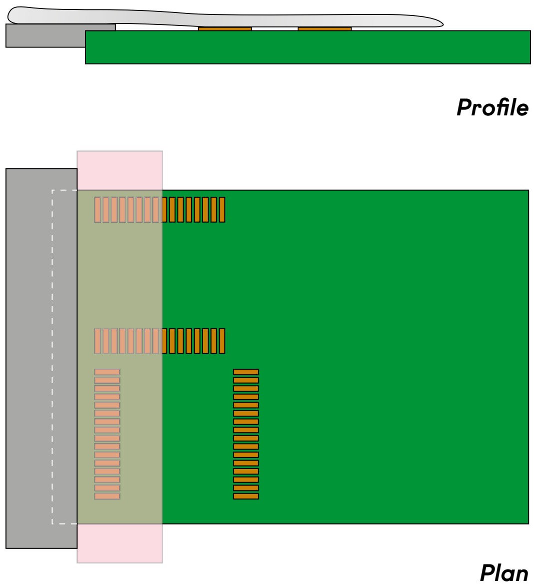


Figure 4.7 Poor flatness of the stencil at the edge of the PCB

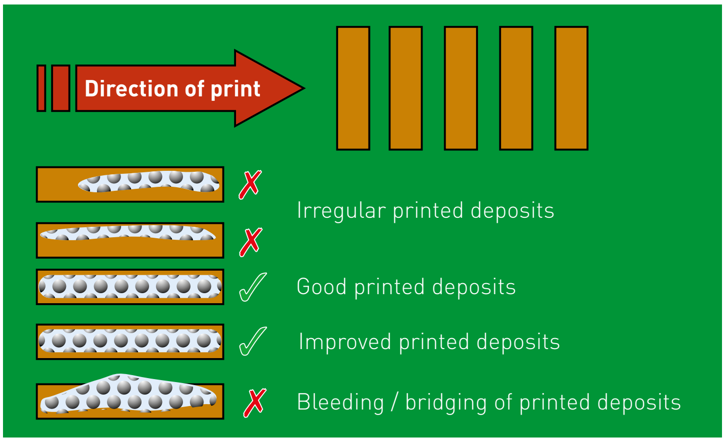
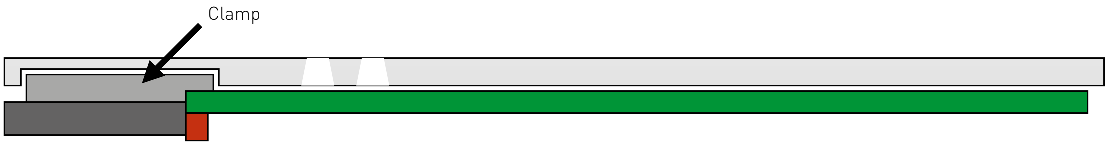


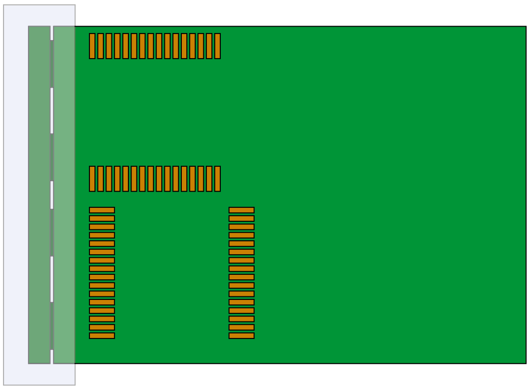
Figure 4.8 Typical results when the clamp foil thickness affects the stencil's ability to seal or gasket to the PCB features.

To ensure the stencil is able to seal to the solder pads at the edge of a PCB, when using in-line printers the underside of the stencil can be relieved to accommodate the clamping foils.

The part etched nests on the underside of the stencil enable fault free printing.



Profile



Plan

Figure 4.9 Nesting of clamping foils

4.3.1. Vaacum tooling plates

Other holding options include the use of vacuum tooling plates - where the substrate is held against the tooling plate with vacuum assistance. This method can be used to assist in overcoming flatness problems associated with warped boards (although this task is largely carried out by the downward pressure of stencil and squeegee blade).

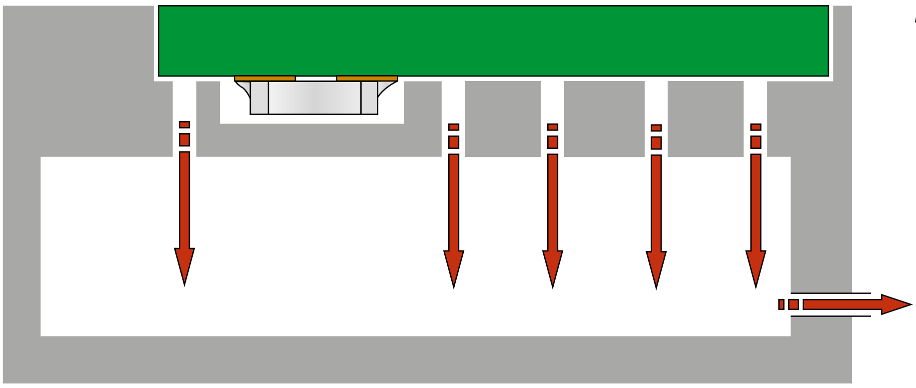


Figure 4.10 Vacuum tooling plates

4.3.2. Pin bar arrangements

Semi-automatic printers often have simple pin bar fixtures to retain the PCB throughout the transport and printing modes. The inherent accuracy of these printers is less than their equivalent automatic counterparts since drilled holes in PCBs used with the pins as shown in figure 4.11 often tend to be ± 50 - 100µm of the nominated dimension required.

Figure 4.11 Typical pin bar arrangements on semi-automatic printers

4.4 Image positioning

Examples of image positioning are shown below:

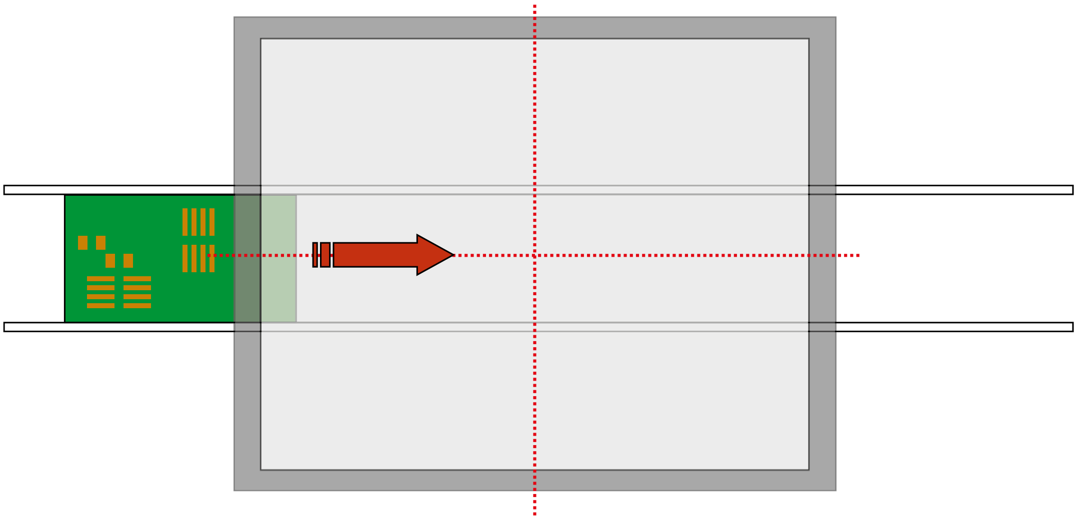


Figure 4.12 Offset image for use on DEK 265 (29″ x29″) machines

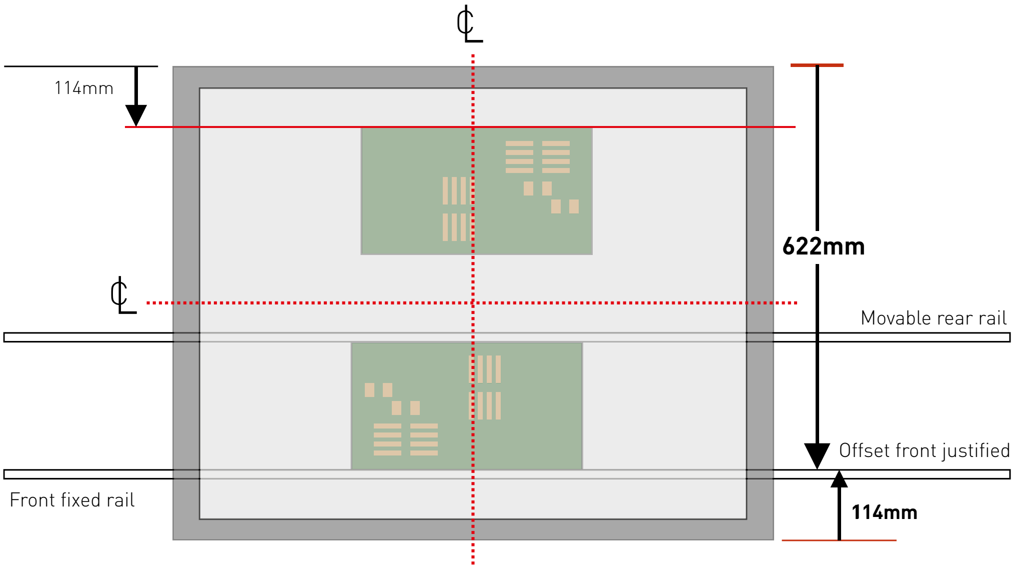


Figure 4.13 Dual image offset details for DEK 265 (29″ x29″) machines

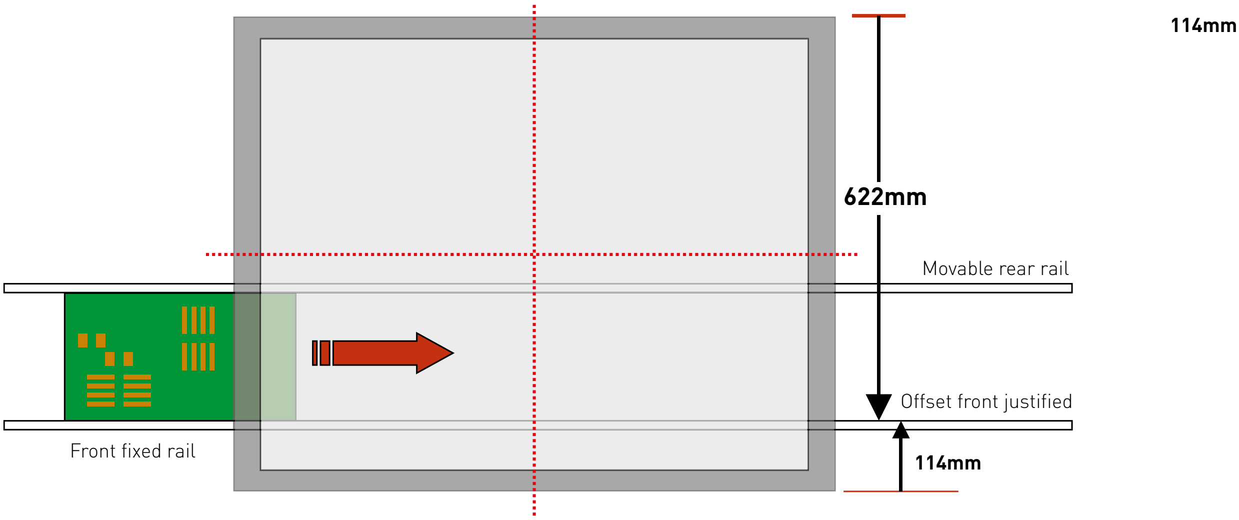


Figure 4.14 Centred image positioning

4.4.1. Roll on / Roll off distance

It is important to provide the solder paste with every opportunity to roll; as such a distance at the start and end of the printing stroke, known as the roll on / roll off distance should be accommodated.

The RO/RO distance should be between 30-40mm. Its purpose is to provide sufficient momentum to the paste to promote good paste roll, which enables aperture filling.

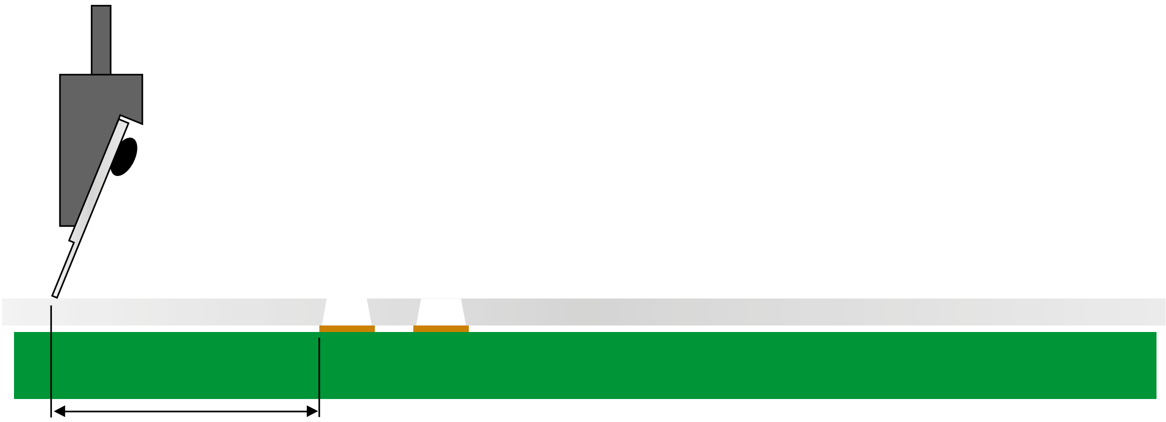


Figure 4.15 RO/RO distance

Squeegee separation also must be considered. This is a fixed dimension and on most modern printing machines is between 40 and 50 mm.

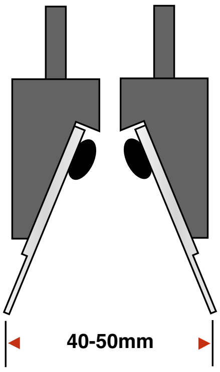


Figure 4.16 Squeegee separation

4.5 Alignment

Alignment accuracy is critical to achieving success with the printing process. Positioning the board accurately and providing alignment repeatability to align the copper features of the PCB pattern with the apertures in the stencil is the minimum required in this operation. This requires three adjustments (X, Y, Ø): since the range of angular adjustment is small, alignment is often implemented using X1, X2, and Y adjusters as shown in figure 4.17.

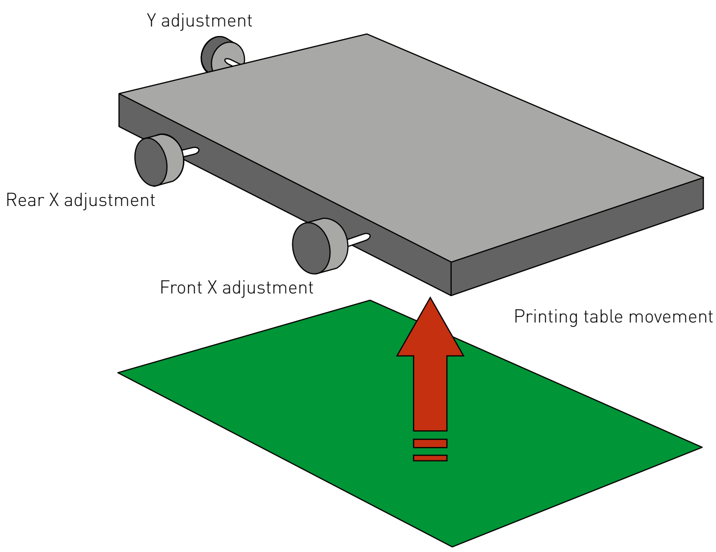


Figure 4.17 Stencil alignment mechanism

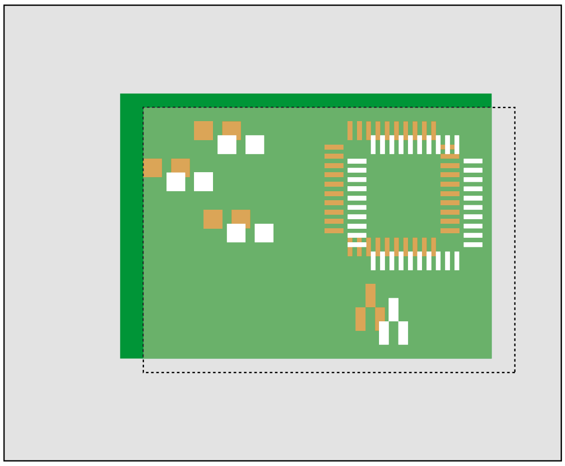


Figure 4.18 Adjustment to X and Y axis controls

The alignment of PCB pads to stencil apertures requires adjustments to X and Y axis controls as shown in figure 4.18.

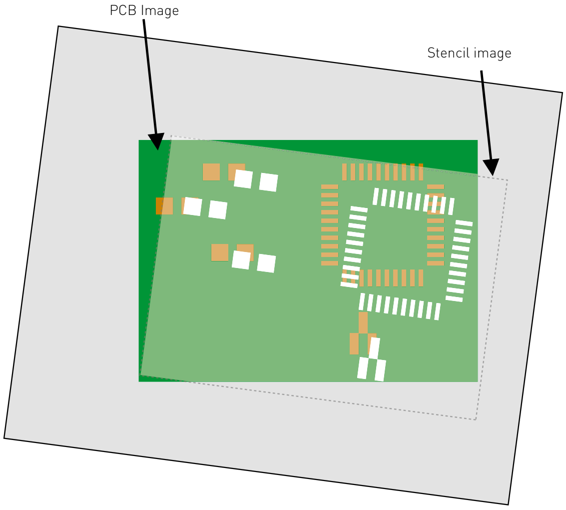


Figure 4.19 Final adjustments

Final adjustments may involve an angular or theta adjustment where the front and rear X axis controls are used.

4.5.1 Vision alignment systems

Manual adjustment has now largely been replaced by vision systems using a CCD camera to image fiducial marks on the board. A number of different shapes of fiducial have been used in the past, although the industry tends now to use either a solid filled circle or diamond between 1mm and 3mm in diameter. It is essential to ensure the fiducials are unique so that the recognition factor involved in locating them with a vision system presents minimal confusion or problems.

When repeatability is required, assisted-vision systems are a necessary part of any printer. There are several systems that can be used to achieve good alignment:

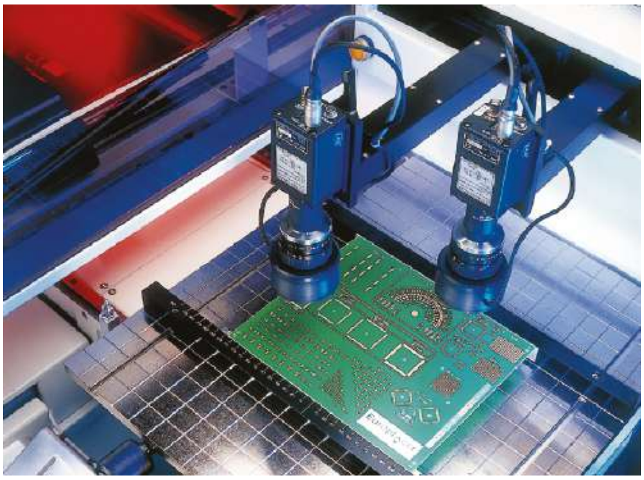


Figure 4.20 Down-looking camera fitted to a semi-automatic printer

- Basic look-down camera systems used on semi-automatic printers that store the image of the features on a PCB and subsequently compare the printed deposit positions with the original copper pad
- Automatic fiducial recognition using either look-up / look-down camera technology - where the fiducials on the PCB and their counterparts on the stencil are compared and "best fit" alignment is achieved, using split field prisms
- A look-down / look-down camera that also offers subsequent printed deposit inspection

In many systems, the camera is inserted between the stencil and the PCB, looking down onto the PCB as well as up onto the underside of the stencil to view the fiducials simultaneously. The camera is free to rove beneath the stencil as it registers the fiducials and then it returns to its rest position away from the vertical motion of the printer rails.

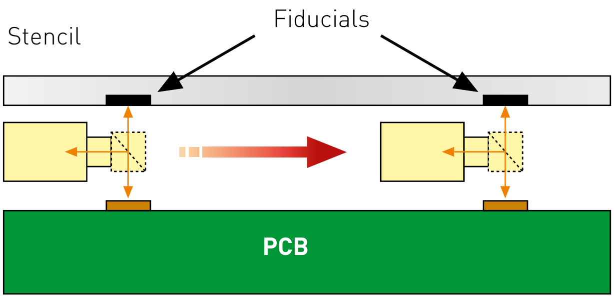


Figure 4.21 Vision alignment system

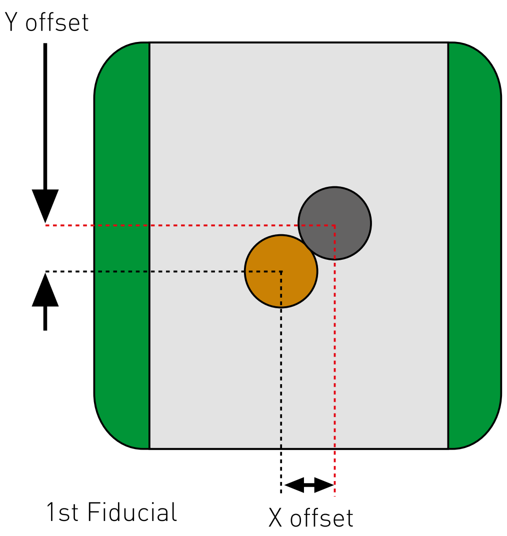


Figure 4.22 First fiducial

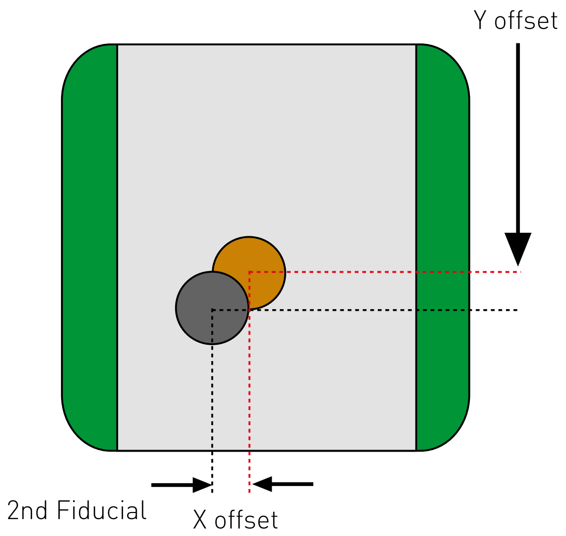


Figure 4.23 Second fiducial

Some printer manufacturers adopt different methods of employing cameras to recognise the fiducials including precision look-down / look-down cameras that store an image of the PCB fiducial or pad to compare with the corresponding stencil feature.

Fiducial identity can be referenced to either a lighter background, as in the case of the stencil or a darker background as on a PCB or substrate. Printer camera recognition systems are used to calculate the fiducial's position, by scoring against target/accept scores. The ideal situation would result in a perfect match.

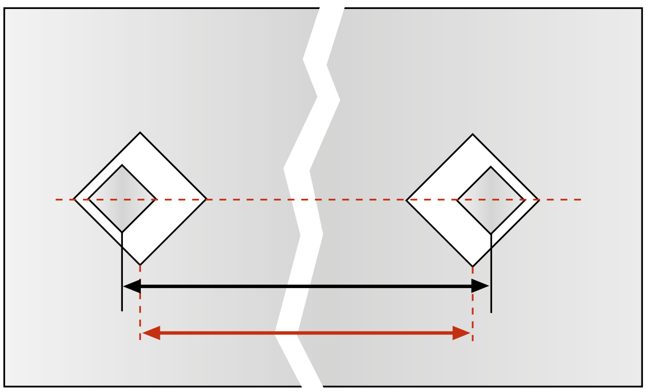
Image alignment accuracy depends on:

- Optical and incident lighting system employed in the printer
- Resolution, in terms of pixels into which the PCB image is converted by the camera
- Algorithm used to determine the necessary location information from the board image

4.5.2 Alignment compromises

Environmental variability - such as differences in temperature and humidity during production - can lead to small differences in dimensions, particularly on large boards, so there is no absolute guarantee that the stencil will exactly match the board even though these have been generated from the same CAD artwork.

Because of minor variability, the best an alignment system can do is to provide perfect alignment between the stencil and the corresponding feature on the PCB at one point, and to minimise the errors at all others. Vision alignment systems look at two or more fiducials, and then correctly align the fiducials to the line joining them averaging the error along this line as shown in figure 4.24.



The one point of perfect match is somewhere between the two fiducials. Often this point is midway between them, but some boards may have non-central areas which are particularly difficult to print. In such cases, it is possible to 'weight' the fiducial correction, so that the optimum alignment occurs at the desired board location. In all cases, however, exact alignment in X, Y and theta, is only achieved at one place, and the inaccuracy increases with distance from this point.

Figure 4.24 Averaging errors in fiducial positions

4.6 Environmental conditions

If possible, temperature should be kept constant within close limits: 21°C ±2°C is a typical specification. This is primarily in order to avoid any effect on the process resulting from changes in the viscosity of the solder paste, but also helps keep alignment consistent by maintaining constant stencil dimensions.

4.7 Squeegee Blades

Squeegee blades are effectively the paste delivery mechanism and to ensure the solder paste fills the finest of stencil apertures several factors need to be considered. These are covered in more detail in chapter 5 in this series.

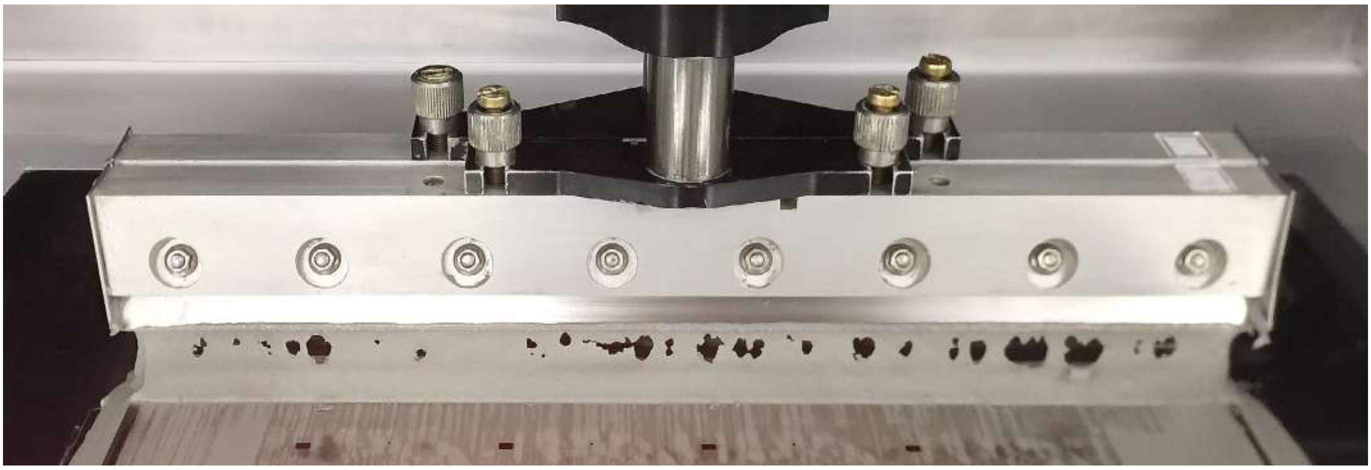


Figure 4.25 Squeegee blades

4.8 Speed

Printing speed needs to be considered carefully. The first consideration should always be the finest aperture present on the stencil, parallel to the direction of print. Printing with an excessive speed may cause incomplete or inadequate aperture filling resulting in inconsistent printed deposits.

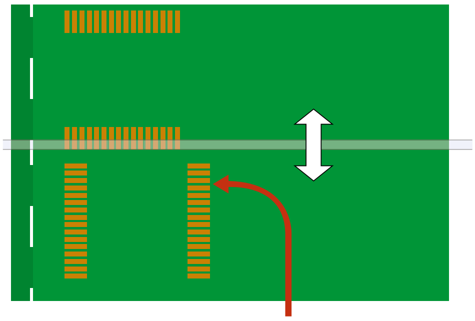


Figure 4.26 Speed should always allow the finest apertures parallel to the printing direction to be filled.

Always consult the paste manufacturer's recommendations with regard to printing speeds required for their products. Equally poor results can be obtained by printing high speed paste at lower speeds - where the rheological properties of the paste do not reach the optimum conditions - as printing lower speed pastes above the recommended speeds.

4.9 Pressure

Squeegee pressure depends mostly on length of blade being used. Blade lengths should be the equivalent of the board width +25mm at each end. If the squeegee blade employed is longer than this dimension it is often not possible to achieve good printed results without increasing the pressure significantly. As a good rule of thumb 0.5-1.0kg / 50mm should satisfy most solder paste printing requirements. Beware of using excess pressure as inconsistent printed results and under-stencil contamination usually occur.

4.10 Stencil cleaning

Stencil cleaning may not be the most glamorous part of the printing process but is nonetheless essential. Cleaning is required at the end of printing when the stencil is removed from the printer, to remove any accumulations of solder paste from the aperture walls and the top and bottom stencil surfaces before the deposits harden. It is also necessary as an in-process activity to ensure printed deposit consistency. The frequency of under-stencil cleaning will depend on the finest pitch component present as much as the quality of the PCBs being printed, together with the accuracy of the printer.

Stencil cleaning frequency can be reduced by using nanocoated stencils. This is covered in more detail in chapter 3 in this series.

The underside of the stencil must enable gasketing to the copper features below or it will gradually acquire solder paste through:

- Stencil bleeding
- Misalignment
- Poor paste release

The rate at which this happens will depend on the print parameters selected as well as the stencil technology, substrate condition, paste rheology and the prevailing environmental conditions.

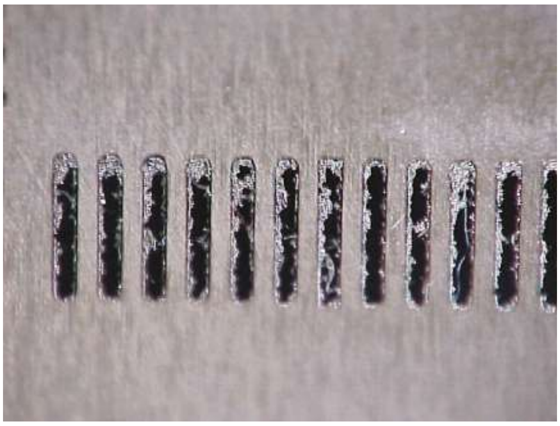


Figure 4.27 Paste clogging

Solder paste can accumulate on the underside of the stencil because small amounts of paste bleed resulting from an imperfect seal between the stencil and board. This phenomenon is made worse by misalignment, using squeegee pressures outside the process window or by poor release of paste from the stencil.

Periodic cleaning of the underside of the stencil is particularly important for fine-pitch applications, because even a small degree of contamination of the substrate by solder balls or flux from the solder paste will degrade the printed results through smearing. If the contamination is not removed, the resulting print smearing increases the incidence of bleeding, bridging and solder shorts or solder balling.

Cleaning can be carried out by hand or completed automatically. Programmable in-process stencil cleaning can be built into modern automated stencil printers, while separate automatic spray-cleaning tanks may be used for stencils after printing.

Automatic stencil cleaners are designed to enable unassisted cleaning of the underside of the stencil at user-programmable intervals (typically after a predetermined number of print cycles). The frequency of under-stencil cleaning will depend on whether the stencil is nanocoated, the alignment of the stencil to the PCB and also the environmental conditions.

Typical systems for cleaning the stencil underside use a lint-free wipe, running between supply and take-up rolls, so that an unused area of the paper is pressed against the underside of the stencil. Contaminants and paste removed from the stencil are trapped on the material roll.

Figure 4.28 Under-stencil cleaning mechanism

On some machines, this operation can be run dry or wet, using a cleaning fluid that is sprinkled onto the absorbent paper. Machines can also be programmed for different combinations of wet and dry wiping, using wet cleaning to loosen any dried solder paste residues.

The cleaning process can also be assisted by vacuum, which can help remove solder paste from stencil openings and improve the clearance of partially clogged apertures. The vacuum system operates in conjunction with twin blades on the under-stencil wiper to draw contaminants into the absorbent paper below the stencil surface.

4.11 Paste conditioning

Paste needs to be stored correctly and allowed to reach operating temperature before the container is exposed to (potentially moist) air. Preparation for use should also include a degree of 'paste conditioning' to ensure that:

- The paste is homogenous – during extended storage, solder spheres and flux may separate. Ideally, paste should be rolled slowly and continuously, which is especially difficult if the paste has been stored refrigerated to extend shelf life and is not brought to ambient temperature in a controlled manner
- Initial folding or stirring of the solder paste does not introduce or entrap air
- Once the solder paste has achieved a stable rheological state it is essential that it rolls or flows to fill the apertures, and then reverts to provide solid solder brick deposits before separation of the substrate from the stencil.

One technique available on the printing machine itself is the knead function where the paste is moved backwards and forwards over a portion of the stencil containing no apertures until suitable paste rolling action is achieved.

4.12 Print quality

This usually involves a visual assessment of selected paste deposits to offer a quick and helpful guide to whether the process is under control. Good results can be obtained with relatively low magnification (×4 to ×10) using a magnifier or projection microscope, as these allow the whole area to be scanned relatively quickly. So, what should you be looking for?

A working definition of acceptable print quality is one that has good definition and registration without any defects such as slumping, scavenging, bridging and peaking. These defects are covered in more detail in chapter 6 in this series.

4.12.1 Paste measurement

Measurement of the paste deposit is crucial to quality control, there being two significant aspects:

- Has the whole of the intended pattern been printed successfully?
- Have you achieved the volume of paste deposits required for the component population? (For which paste height and area are useful measurements).

The methods of evaluating a printed substrate vary between fully automated inspection, both for coverage and paste height, and occasional operator visual checks. Today, with the increasing use of smaller components, often with terminations below their bodies, there is a trend towards implementing automatic checks at the end of the print cycle, using either the printer itself or a separate machine.

Until recently inspection of the paste deposits using the printer camera systems impacted or limited the required printing cycle times. But advances in technology - employing the existing sophistication and speed of the optical arrangements, combined with software - make it possible to complete a post print inspection in under a minute.

Optical inspection for coverage has in the past relied on there being a visual difference between a pasted and bare pad: this is very easy when printing onto a nickel-gold finish, but significantly more difficult when printing onto solder surfaces.

A 'z-check' for paste height can be carried out easily with a light-section microscope or laser equivalent. As shown in figure 4.29, the height of the print, and some information on the topography of the surface, can be gained using oblique illumination through a slit, and viewing from above.

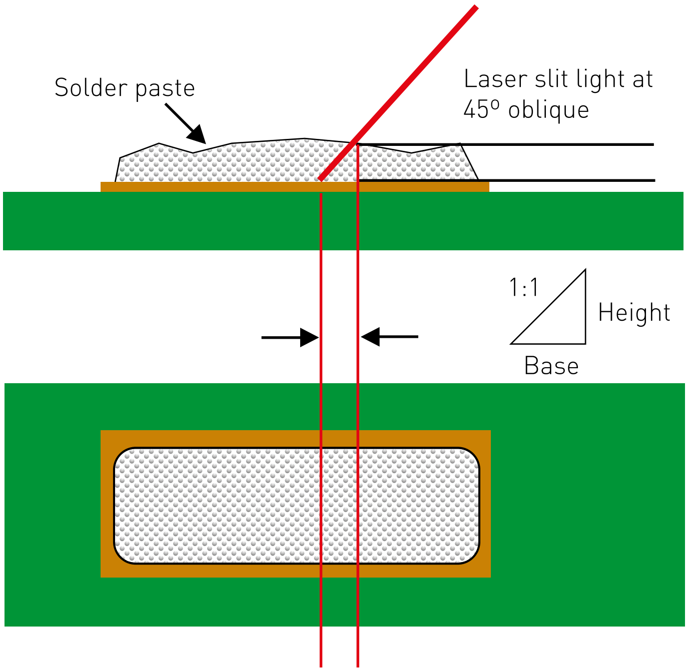


Figure 4.29 Method of operation of a light-section microscope

5.0 squeegee blades

applying solder paste for optimum results

5.0 Squeegee Blades

There are two main methods of applying solder paste to a circuit board using an SMT stencil printer: squeegee blade printing and enclosed head printing. This section focuses on the most common method – squeegee blade printing.

With squeegee blade printing, three print parameters can typically be controlled, squeegee speed, blade angle and downward squeegee pressure. Excessive pressure can result in damaged stencils, coining and breaking of webbing between fine pitch apertures. Too little pressure can result in skips if the stencil is not wiped clean.

To achieve the best results, the blades must possess flexibility to consistently deliver the expected paste volumes.

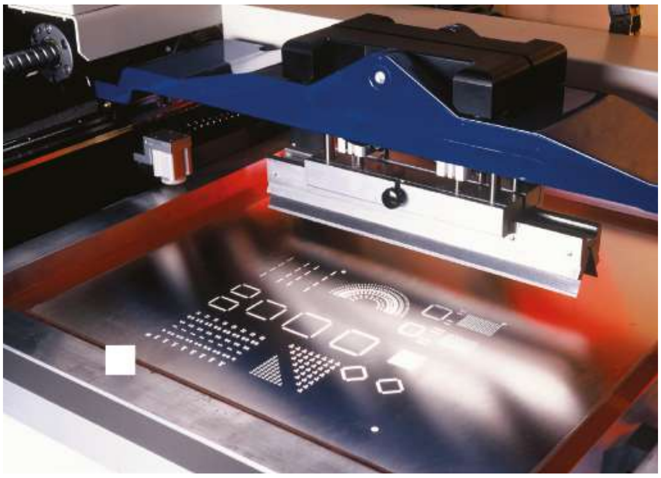


Figure 5.1 Squeegee blade printing

5.1 Squeegee blade types

There are two squeegee material types: rubber/polyurethane squeegees and metal squeegees.

Metal squeegees blades are most commonly used and operate at lower pressure than rubber variants. As such, they do not scoop paste from apertures, and because they are metallic, they do not wear as easily as rubber types - and hence do not need to be sharpened. The popularity of metal squeegees has grown with the prevalence of finer pitch components.

Metal squeegee blades are typically made from stainless steel in either a single thickness (0.150mm) configuration or can be manufactured with an etched mobile/recessed edge where the base material is 0.300mm thick and the edge is tapered down to 0.150mm.

Figure 5.2 Metal squeegee blades

5.1.1 Single-thickness metal squeegee blades

Single-thickness squeegee blades are laser-cut from specially selected 0.150mm annealed stainless steel and are ideal for applications that contain a mix of components. They offer a wide pressure window, excellent consistency and minimal wear.

Figure 5.3 Single thickness metal squeegee blades

5.1.2 Mobile edge metal squeegee blades

Mobile edge metal squeegee blades are etched from 0.300mm stainless steel with the edge feathered down to 0.150mm thickness.

Blade flexibility and its reaction to under-stencil interruptions such as proud vias, poor silk-screening or solder resist imperfections can influence the success of the printing process.

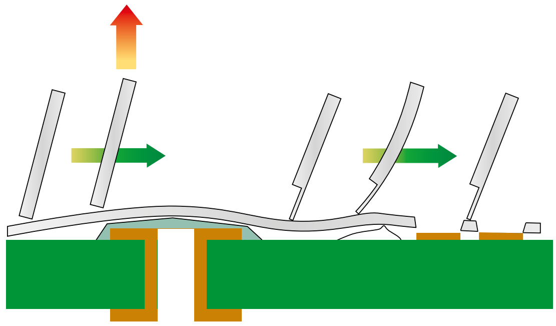


Figure 5.4 Uneven stencil surfaces can result from proud features on the substrate

Figure 5.5 Mobile edge metal squeegee blades

Mobile edge blades adapt better than single thickness blades. In instances where a raised PCB feature is adjacent to fine pitch apertures it can create interruptions to the stencil flatness that cause the squeegee blade to "jump" or skip over an aperture leading to either retention of paste in the aperture, leading to an insufficient or possible bleeding and bridging.

Mobile edge blades possess extra flexibility at the interface with the stencil surface and can adjust their profile, local to the under-stencil deformation, without sacrificing the print quality to achieve cleaner more consistent printed deposits.

5.2 Squeegee printing parameters

Printing with squeegee blades involves angle, speed and pressure.

5.2.1 Blade angle

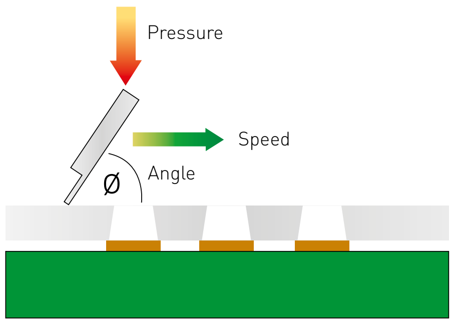


Figure 5.6 Squeegee printing parameters

Generally, the squeegee blade should be angled at 60° to the stencil surface. Sufficient downwards force should be exerted as the squeegee blade traverses the stencil, on the printing stroke, to ensure the paste rolls and fills the finest stencil apertures.

Decreasing the blade angle to 45° will provide extra downwards force to encourage increased aperture filling. Care should be taken to ensure the squeegee blade pressure is not excessive at this angle as it is possible to break the gasket seal between the stencil and the PCB resulting in bleeding, bridging and shorts.

5.2.2 Speed

The speed at which the squeegee moves across the stencil plays a large part in the effectiveness of the solder paste application by determining how much time is available for the solder paste to 'roll' into the stencil apertures and on to the PCB pads. Typically operators use a speed of 25 millimetres per second, but this can be adjusted depending on the size of the apertures and the type of solder paste being used.

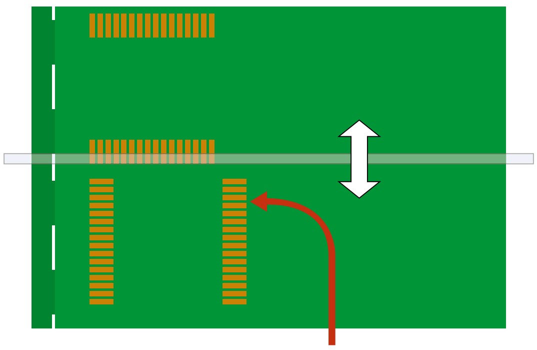


Figure 5.7 Speed should always allow the finest apertures parallel to the printing direction to be filled

5.2.3 Pressure

Squeegee pressure depends mostly on the length of blade being used. Blade lengths should be the equivalent of the board width + 25mm at each end. If the squeegee blade employed is longer than this dimension it is often not possible to achieve good printed results without increasing the pressure significantly. As a rule of thumb 0.5-1.0 kgs / 50mm should satisfy most solder paste printing requirements. Beware of using excess pressure as inconsistent printed results and under-stencil contamination usually occur.

5.3 Squeegee condition and maintenance

Squeegees need to be carefully stored and maintained as any damage can lead to poor printing results. They should be checked before use and thoroughly cleaned after use, ideally using an automated cleaning system so that any solder paste residue is removed. If any damage is noticed to squeegees they should be replaced to ensure a reliable and repeatable process.

Squeegee life and quality should be discussed with all your process engineers and operations and should be established as a standard.

With stainless steel metal squeegee blades, extra care must be taken in handling. All squeegee blades should be inspected before installation and be replaced if they contain any edge flaws such as nicks or dents.

Operators should flag and stop any printer / squeegee setup which is not wiping properly. This is an indication of a squeegee or printer setup problem. The swipe of the squeegee should produce a smooth shiny stencil surface with no puddles or streaks of solder paste left behind.

5.4 Squeegee blades available from Tecan

5.4.1 Standard ranges

Tecan offers a range of off-the-shelf squeegee blades that are fully compatible with most leading SMT printer types. These can be purchased directly from www.stencils.co.uk

5.4.1.1 DEK/ASM compatible single thickness blades

Figure 5.8 DEK/ASM compatible single thickness blade

Part No	OEM part number	Description
LSQD170	129924	Single thickness DEK compatible 60° squeegee blade 170mm
LSQD200	129925	Single thickness DEK compatible 60° squeegee blade 200mm
LSQD250	133584	Single thickness DEK compatible 60° squeegee blade 250mm
LSQD300	133585	Single thickness DEK compatible 60° squeegee blade 300mm
LSQD350	129926	Single thickness DEK compatible 60° squeegee blade 350mm
LSQD400	133586	Single thickness DEK compatible 60° squeegee blade 400mm
LSQD440	129927	Single thickness DEK compatible 60° squeegee blade 440mm
LSQD483	133587	Single thickness DEK compatible 60° squeegee blade 483mm
LSQD510	133588	Single thickness DEK compatible 60° squeegee blade 510mm
LSQD535	129928	Single thickness DEK compatible 60° squeegee blade 535mm

5.4.1.2 DEK/ASM compatible mobile edge blades

Figure 5.9 DEK/ASM compatible etched mobile edge blade

Part No	OEM part number	Description
ESQD170	129924	Mobile edge DEK compatible 60° squeegee blade 170mm
ESQD200	129925	Mobile edge DEK compatible 60° squeegee blade 200mm
ESQD250	133584	Mobile edge DEK compatible 60° squeegee blade 250mm
ESQD300	133585	Mobile edge DEK compatible 60° squeegee blade 300mm
ESQD350	129926	Mobile edge DEK compatible 60° squeegee blade 350mm
ESQD400	133586	Mobile edge DEK compatible 60° squeegee blade 400mm
ESQD440	129927	Mobile edge DEK compatible 60° squeegee blade 440mm
ESQD483	133587	Mobile edge DEK compatible 60° squeegee blade 483mm
ESQD510	133588	Mobile edge DEK compatible 60° squeegee blade 510mm
ESQD535	129928	Mobile edge DEK compatible 60° squeegee blade 535mm

5.4.1.3 Speedprint compatible mobile edge blades

Figure 5.10 Speedprint compatible mobile edge blade

Part No	Description
ESQS170	Speedprint compatible 60° squeegee blade 170mm
ESQS220	Speedprint compatible 60° squeegee blade 220mm
ESQS270	Speedprint compatible 60° squeegee blade 270mm
ESQD300	Mobile edge DEK compatible 60° squeegee blade 300mm
ESQS320	Speedprint compatible 60° squeegee blade 320mm
ESQS370	Speedprint compatible 60° squeegee blade 370mm
ESQS420	Speedprint compatible 60° squeegee blade 420mm
ESQS450	Speedprint compatible 60° squeegee blade 450mm
ESQS490	Speedprint compatible 60° squeegee blade 490mm
ESQS550	Speedprint compatible 60° squeegee blade 550mm

5.4.1.4 Reprint compatible mobile edge blades

Part No	Description
ESQR150	Reprint compatible 60° squeegee blade 150mm
ESQR200	Reprint compatible 60° squeegee blade 200mm
ESQR220	Reprint compatible 60° squeegee blade 220mm
ESQR300	Reprint compatible 60° squeegee blade 300mm
ESQR360	Reprint compatible 60° squeegee blade 360mm
ESQR420	Reprint compatible 60° squeegee blade 420mm
ESQR450	Reprint compatible 60° squeegee blade 450mm
ESQR460	Reprint compatible 60° squeegee blade 460mm
ESQR520	Reprint compatible 60° squeegee blade 520mm

5.4.2 Bespoke squeegees

Bespoke mobile edge etched squeegee blades for all other printers and applications are available on request. For a price and lead-time please provide the dimensions as indicated in figure 5.11 below.

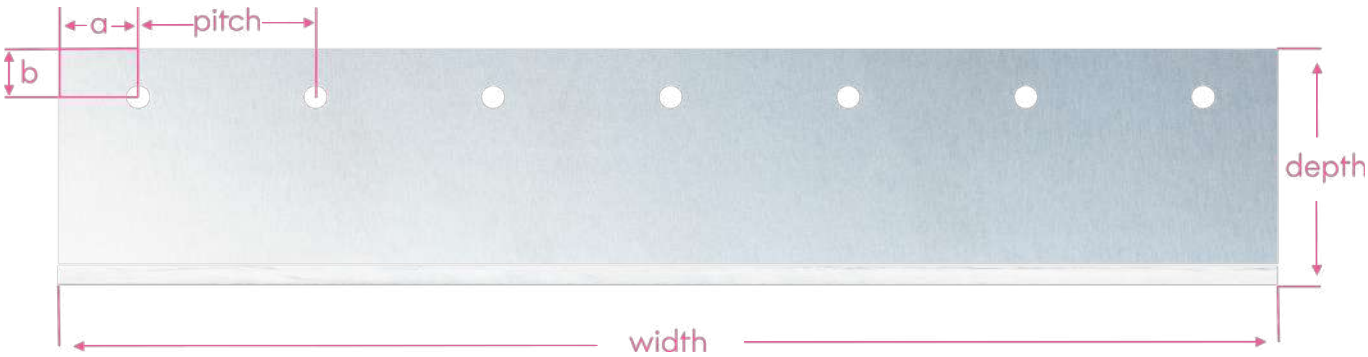


Figure 5.11 Bespoke squeegee blade

All brands, logos and trademarks are the property of their respective holders. Any reference to these trademarks is purely for the purpose of identifying the SMT manufacturers' compatible components. Tecan is neither affiliated with, authorised by, distributors for, or related in any way to the descriptive SMT brand manufacturers.

6.0 reflow

overcoming faults, solder paste printing effects and screening can requirements

6.1 Reflow and profiles

During reflow soldering the printed solder paste deposits melt and form solder joints to interconnect the component terminations to the substrate pads. There are a considerable number of solder paste manufacturers each with a range of solder pastes with subtle nuances in the formulations that require small adjustments to the reflow profile to achieve the results required.

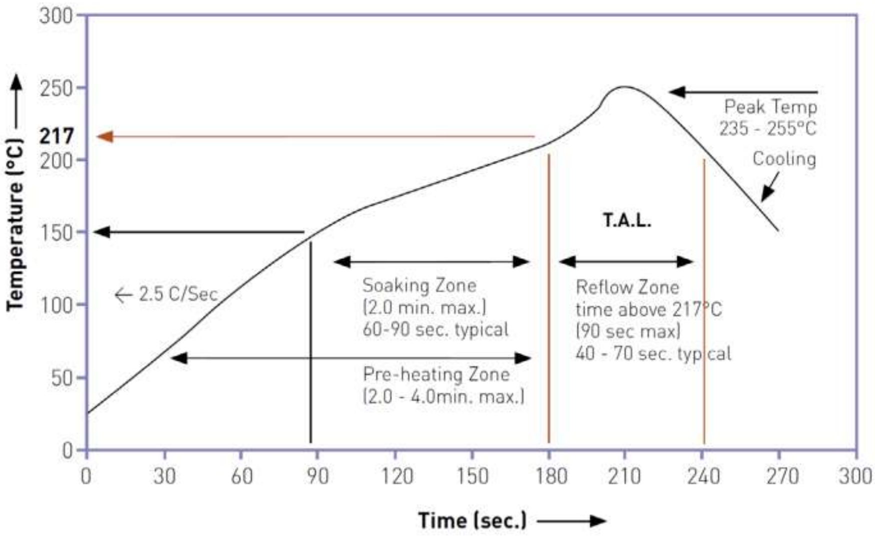


Figure 6.1 "Typical lead-free reflow profile"

Alloys: Sn96.5/Ag 3.0/Cu 0.5 and Sn 96.5/Ag 3.5

When surface mount began, the heat sources used were infrared. Infrared ovens have improved in the last thirty years to achieve good results but today many people use forced air convection ovens to reflow the solder paste. Vapour phase technology was also used to good effect in the development of reflow soldering and is popular with lead-free process requirements. The boiling point of the chemical used is a constant and every part of the PCB assembly benefits from the same temperature.

Achieving a reflow profile that is suited to both the solder paste and the components used is critical to the success of the surface mount process. Guidance is always provided by the solder paste manufacturer and should be used as exactly that. The PCB you are processing may require small variations from the manufacturer's suggested reflow profile to enable achievement of acceptable quality levels.

In order to adjust a profile you need to understand what's happening on the PCB. As such, a profiler should be used with thermocouples mounted to monitor the temperature at various critical points throughout the process to ensure the temperatures achieved enable reflow of the solder paste.

Reflow profiles can be broken down into four distinct areas:

- **Pre-heating:** This raises the temperature of the PCB and components without the risk of thermal shock to either.
- **Soaking:** Allows the stabilisation of the temperature across the variation of different components.
- **Reflow:** This occurs once the temperature is elevated above that of liquidus. Typically 40-60 seconds is spent above this temperature (time above liquidus) to ensure all the solder fillets are fully reflowed.
- **Cooling:** Necessary to ensure the prevention of excessive intermetallic layers, which could cause embrittlement of the resultant solder fillets.

6.2 Reflow related faults

6.2.1. Solder balling

Sometimes also known as spattering, this is mostly caused by the explosive evaporation of the solvents within the solder paste. As part of the reflow process, when the solder deposit's temperature is raised, pressure builds up within the deposit until the solvents are able to migrate through this flux membrane. Since these solvents are contained within the printed deposits it is important that the reflow profile ramp rate is not excessive or the soak zone temperature too high so that the release of these solvents is more gradual.

A similar but far more spectacular fault can be caused by the solder paste absorbing moisture which upon reflow causes mini-steam explosions with the accompanying spattered effects.

6.2.1.1 Isolated solder balls in close proximity to a solder pad.

This generally occurs when either the stencil to substrate alignment wasn't good during printing or the stencil aperture couldn't gasket to the copper feature beneath. Common reasons include over-etched PCBs, incorrect aperture reduction or printer registration errors.

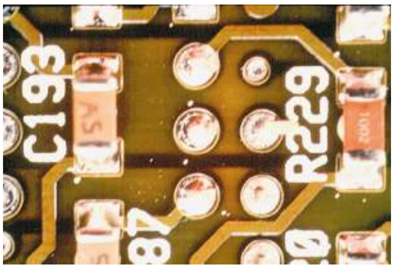


Figure 6.2 Isolated solder balls

6.2.1.2 Dewetting and solder-balling

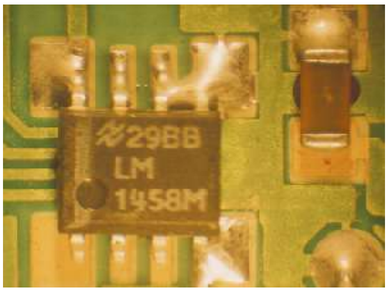


Figure 6.3 Dewetting and solder balling

Poor wetting and solder balling on the same pad is most likely caused by creating an excessive soak phase in the reflow profile. Firstly, the activated flux will clean the solder but extending the soak time can cause reoxidation resulting in poor wetting and a large solder ball. This can often become more of a problem with the solder pastes used for finer pitch requirements as the smaller solder spheres have a larger surface area on which oxides can form or re-form. Lower residue pastes can also offer a smaller operating window so it is prudent to consult the technical data sheet for the paste selected.

6.2.1.3 Mid-chip solder balls or solder beading

This is usually evident as an emerging ball or balls beside discrete chip components and is usually the result of inappropriate reduction to the stencil aperture. Excessive paste deposits are unable to retract to the component terminations or solder pad before the chip component is attracted to the board upon reflow. Any solder not able to flow to the component terminations becomes trapped and is compressed to form solder balls. Aperture modifications to greatly reduce or eliminate this problem are found in chapter 2.

Figure 6.4 Mid-chip solder balls

6.2.2. Wicking

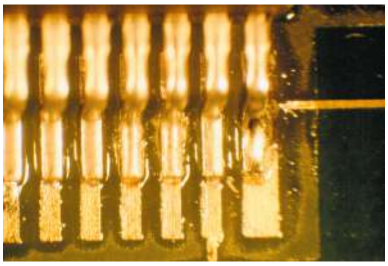


Figure 6.5 Wicking

Wicking occurs when a component lead or termination attracts or absorbs the meted solder and can often result in an open circuit. Typical reasons include too large a temperature difference between the pad to be soldered and the component leads. If the lead is hotter than the adjacent copper pad then the solder will flow onto the lead away from the pad. One obvious area to explore is prolonging the soak time to ensure there is minimal difference between the temperatures of the components and the substrate.

6.2.3 Wetting

Poor wetting can be the result of increased or excessive oxidation. This can be limited by a reduction in the heat impact on the solder paste selected. Reductions can be achieved by shortening the overall heating time or lowering the temperature rise (ΔT) in the pre-heat and soak zone.

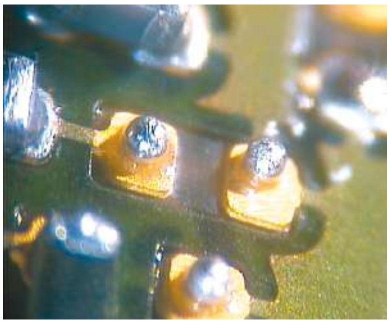


Figure 6.6 Poor wetting

Dewetting is caused by overheating during the reflow process. To overcome this problem, peak temperature can be lowered or the dwell time reduced.

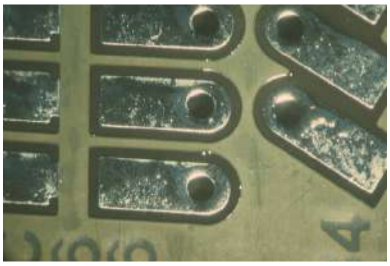


Figure 6.7 Dewetting

6.2.4 Tombstoning/component lifting

Tombstoning on chip components can be caused by unequal wetting forces on the two terminations. Surface tension at the end that reflows first can attract the component and make it rise - often causing it to stand on its end.

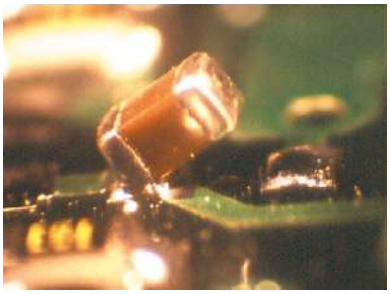


Figure 6.8 Tombstoning

Unequal wetting can be caused by incorrect solder pad design, poor solderability of either the component terminations or solder pad, different volumes of solder paste available at each termination or unequal temperature at the two pads. A reduction in temperature difference is required to solve this problem. The temperature rise (ΔT) used in the pre-heat phase must be reduced and the soak time extended accordingly.

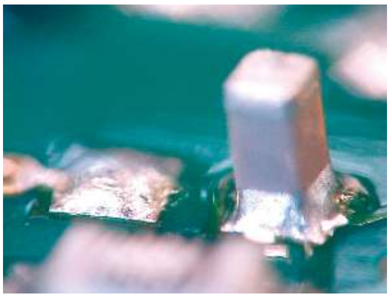


Figure 6.9 Tombstoning

There are a number of reasons for the appearance of tombstoning with discrete components including:

- Different pad surface areas, from one side of the component to the other
- Variations in the thermal demand of the solder pads attributed to tracks and internal layers
- Different volumes of solder paste applied to the two pads
- Solder paste printed beneath the body of the component
- Poor solderability of one or both of the solderable component terminations
- Solder resist thickness. Excessive thickness may cause the component to rock
- Inadequate pick and placement pressure causing poor initial adhesion to the paste deposits

With discrete components getting ever smaller the surface area of the solder pads influences the surface tension experienced by the components. Tombstoning can be more prevalent with smaller component sizes.

6.2.4.1 Incorrect pad design

Solder pads should be designed to accommodate the component terminations approximately central to the pad centroids to ensure the forces acting on the two ends of the component are balanced. When the component is out of position, relative to the centroids, the forces at each end will be different causing the possibility of lifting the component. Once movement has started the momentum continues until the component has no solder termination at one end.

6.2.4.2 Variations in thermal demand of solder pads.

When designing the layout of the PCB it is important to achieve, as near as possible, equal solder pad and track interconnections at each end of the component. This is because during the reflow process the component terminations do not reflow at precisely the same moment. Differences in the surface area of copper connected to the component terminations, including connecting tracks, can affect the wetting speed and lead to an imbalance of forces on one side of the component compared to the other - increasing the incidence of component lifting.

6.2.4.3 Different volumes of solder paste

There may be occasions, even when the copper pads are equal in size, that the solder paste volumes might be different. Poor registration of the stencil to the PCB can cause just such a problem. Smaller paste deposits may reflow before the larger deposits, causing wetting to the termination faster than the other side of the component. This logical process is also evident with variations in small paste deposits used for μBGAs where the smaller paste deposits reflow before the larger ones.

6.2.4.4 Paste under the component body

Slumping of the solder paste deposits, caused either by squashing during placement or cold slump due to environmental conditions, are both undesirable as this can cause component lift during reflow. Paste should always be contained within the boundary of the copper pad limits.

6.2.4.5 Solderability

Solderability issues can exist when the plating finish itself is suspect or subsequent environmental reactions, for example from incorrect storage, have degraded the finish to cause large variations in the wetting attraction forces evident on the two pads. Where the solderability of the terminations is worse on one end of the component, the forces acting on the component would pull the component to the better wetting pad before reflow had been effected at the other pad. It may not cause a tombstone but it is important to remember that even a small lift can cause an inadequate solder fillet or an open circuit.

6.2.4.6 Solder resist thickness

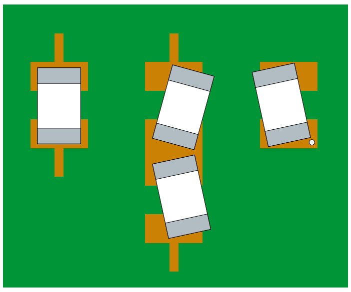
Thicker solder resist and resist variability between adjacent solder pads can cause the component to seesaw rather than sit correctly. This effect can also be seen when interconnecting copper tracking exists between the two pads.

6.2.4.7 The effect of pick and placement pressure

This can cause the component to simply rest on the surface of the paste deposit, as opposed to being held by the tackiness of the flux. In such cases, when the solderability of the component is in question, wetting of the solder may be limited. Care has to be taken with pick and placement as excessive force can lead to both increases in mid-chip solder balls and shorting.

6.2.5. Component skew

Skewing is also caused by an unequal wetting at the two terminations on a discrete component. The surface tension, at the end to reflow first, often acts to realign the component out of its original placed or desired position. With chip components, the solution is the same as for tombstoning. Pad design can also affect the surface tension at one end of a chip component. Placing two chips on shared pads or having differing width connection tracks or via holes within the pad can influence the realignment of the components concerned.



Left: Figure 6.10 Misplacement

Right: Figure 6.11 Component skew

Some integrated circuits used today, such as QFNs, have comparatively large grounding planes on their underside between the terminations. If solder paste is applied to the corresponding central copper pad without reductions, the volume of solder available causes the component to skew out of position.

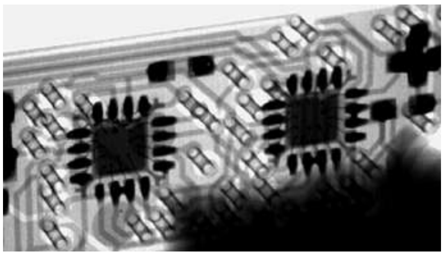


Figure 6.12 The shorts visible in the x-ray picture above could have been avoided by optimising the apertures as shown in the figures 6.13 – 6.15.

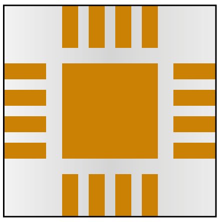


Figure 6.13 QFN package

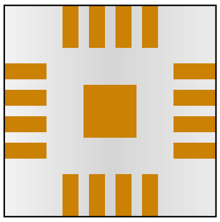


Figure 6.14 Corresponding aperture design required to prevent skew

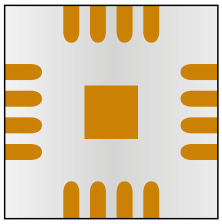


Figure 6.15 Corresponding aperture design required to prevent skew and bridging

6.2.6 Cold joints

Cold joints will be caused by low peak temperatures not fully reflowing the solder mass. To achieve the required solder fillet appearance, it is essential that minimum peak temperatures are achieved and also the time above liquidus (TAL) is sufficient.

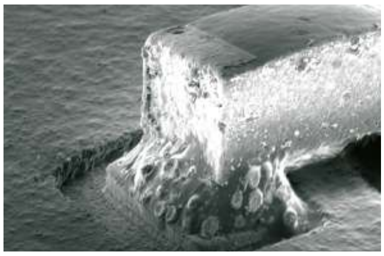
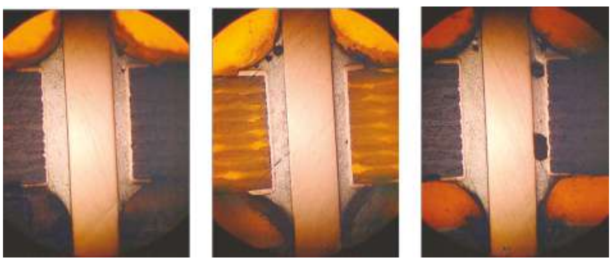


Figure 6.16 Cold joints

6.2.7 Voiding

Voiding or non-metallic pockets within a solder joint are largely caused by outgassing of solvent materials. Pockets of flux can also become trapped within a solder joint as shown in the cross sections in figure 6.17.



Flux entrapment is clearly visible on the right-hand end view, whereas evidence of surface voiding can still be seen on the middle view. Minimising this problem relies on reducing the soldering time or the pre-heat ramp-up rate.

Figure 6.17 Cross sections showing the solder fillets resulting from PIHR.

6.2.8 Intermetallic growths

Excessive intermetallic growths are caused by exposure to heat over time, where resultant absorption of metallic elements form layers within the solder joint and also at the interface with the surface mount pads and component terminations.

Thicker intermetallic layers make a solder joint harden and become brittle. Care should always be taken to ensure excessive peak temperatures and TAL are avoided.

The appearance of large grain sizes in the solder joint is indicative of a slower cooling rate. This can be avoided by ensuring the cooling rate is set between 3 - 4° C / second.

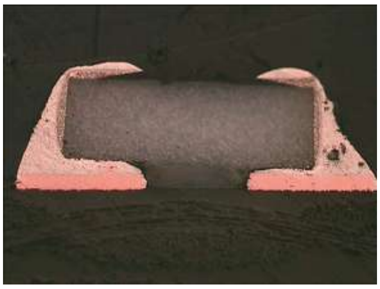


Figure 6.18 Formation of intermetallic layers in solder joint

6.2.9 Cracks

Figure 6.19 Component cracks

Component cracks develop due to thermal stress inside of the component. Care should be taken when heating or cooling to ensure both the PCB and the component population do not undergo rapid temperature changes.

Solder joint cracks are usually the result of mechanical stress at the component terminations. This can be caused by substrate flex, which in some cases is exacerbated by the use of inappropriate solder pad designs which creates too much solder on component terminations.

Figure 6.20 Cracked solder filet

6.2.10 Fillet lifting

Pad lifting, fillet lifting and fillet tearing are effects that result in part from the differences in thermal coefficients of expansion (TCE) between the substrate material and the copper barrels and tracks on the PCB.

Figure 6.21 Fillet lifting

There is a relatively large expansion of the laminate material in the z axis throughout contact with the molten solder in the wave or selective soldering processes. This expansion causes conical deformation of the copper pads.

When solder joints start to solidify, the board material cools down and returns to its planar shape. This movement can create stress on the surface of the solder joint, which at this stage has not gained its full strength. Such stress may cause pad lifting or – if adhesion between the copper pad and substrate is at that point stronger than the solder – will cause cracks in the solder fillet surface, known fillet tearing.

Figure 6.22 Fillet tearing

6.3 Solder paste printing effects

Many individual company workmanship standards exist alongside those published by the SMART group and other bodies, but generally they all refer to qualifying the printing process in terms of:

- Registration of the printed deposits to the copper pad pattern
- The effective solder paste coverage of the pads
- The appearance of the printed deposits and their definition and consistency
- Paste thickness achieved
- Appearance of obvious defects such as slumping, bridging and spikes

6.3.1 Good print

Paste is printed and contained within the boundary of the copper pads.

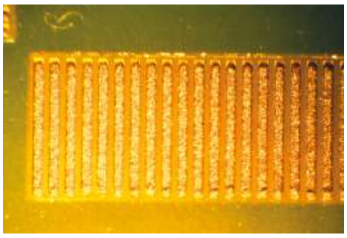
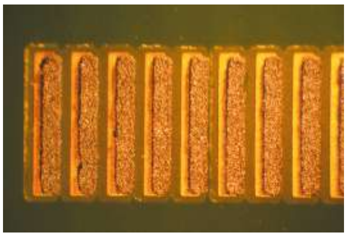


Figure 6.23 Good print

6.3.2 Misaligned print



Poor registration of the stencil image to the corresponding PCB features.

Figure 6.24 Misaligned print

6.3.3 Bridged

The result of incorrect gasketing between the stencil aperture and the copper pad or feature on the PCB. Never assume that the PCB or substrate features match the original data sizes exactly. Dependent on the thickness of the copper on the PCB, it is possible to lose up to 100µm from the target size required.

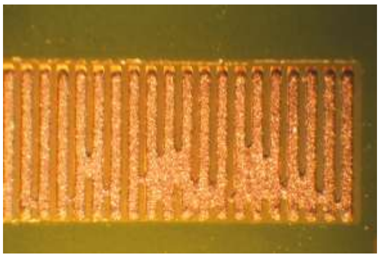


Figure 6.25 Bridging

Bridging is often a problem with finer pitched devices and can be caused by a number of factors including:

- Inadequate gasketing through incorrect stencil aperture reductions, over-etched PCBs, or inaccurate stencil to substrate alignment

- Component coplanarity errors or excessive pick and placement pressure

- Paste slump caused by too rapid a temperature gradient in the pre-heat zone

- Dirt and particulate contamination, for example a hair or fibre filament across two adjacent pads, causes an attraction and flow path between them - often resulting in shorts

When printing fine pitch requirements, the stencil must gasket onto the copper pad in order to confine the printed deposit to the pad itself.

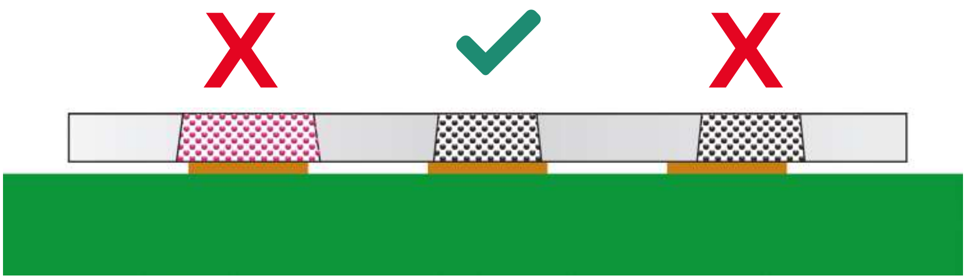


Figure 6.26 Over-etched copper pads cause a loss of gasket that results in flux and solder balls – causing bleeding, bridges and shorts.

6.3.4. Slumping

Paste slump can be caused by using solder paste outside of its usual environmental/temperature/humidity range. The same effect can occur if using out-of-date solder paste.

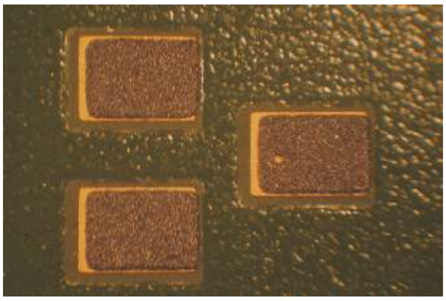


Figure 6.27 Good print

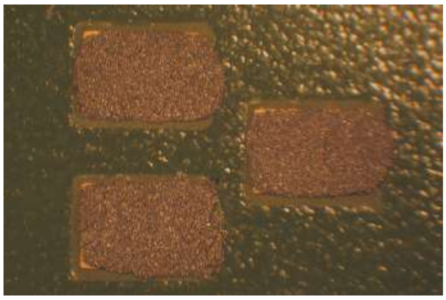


Figure 6.28 Slumped print

6.3.5. Scavenging

Loss of the solder paste volume, for larger stencil apertures, such as base pads of D-PAK devices. The answer here is to brace the aperture to make several smaller apertures within the original pad border.

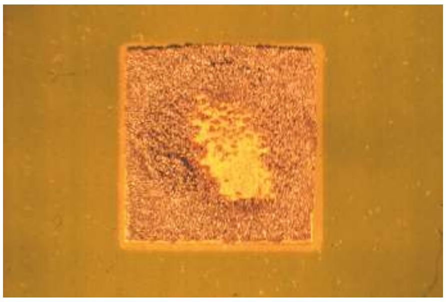


Figure 6.29 Scavenged print

6.3.6 Ripped print

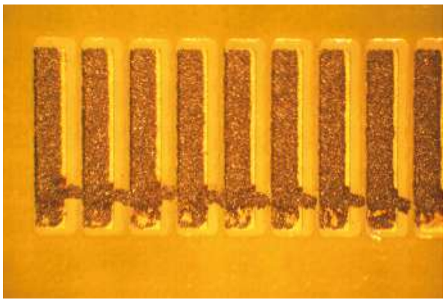


Figure 6.30 Ripped print

Ripped print can be caused by movement at the end of the print stroke or the use of cantilever, not straight, lift separation of the substrate from the stencil post-printing. If using a simple cantilever separation make use of a two-axis or table up/down control to separate by a minimum of 1.5 x stencil thickness.

6.3.7 Insufficient

Too little solder paste printed - usually the result of incorrect aperture sizes or excessive printing speeds.

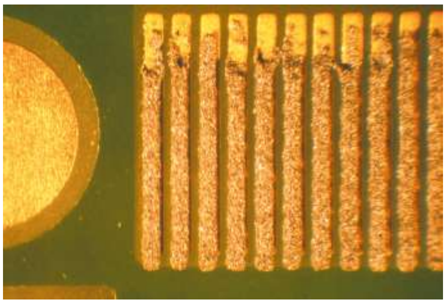


Figure 6.31 Insufficient

6.4 Screening can requirements

Coplanarity issues are not limited to semiconductors; RFI shielding cans play an increasing role in the quality of finished assemblies today. Often 'pressed' (or stressed) cans will not sit flat on the PCB itself. Several methods are currently used to overcome such problems, including preheating to anneal the cans and placing weights on the cans during reflow, but this can affect the reflow characteristics of the assembly and is also likely to lead to problems with joint integrity during cooling and use.

Some screening cans do not exhibit severe coplanarity problems as they are manufactured using photo chemical machining which does not induce stress into the material. Once plated and assembled the seating faces on all four sides largely fulfill the flatness tolerances required.

The use of selective thickness printed solder deposits can also help to overcome any coplanarity problems. Printing increased height paste deposits for the screening can, on the same printing pass, eliminates problems associated with this non-conformance and provides stronger solder fillets which increase mechanical security and prevent problems associated with voided fillets.

Often paste deposits are printed in isolated blocks. The solder does not, however, reflow sufficiently and creates voids and blowholes in the sealing fillet, which may require reworking. One answer is to ensure the paste deposits do not have these large interrupts that cause the effect in the first place. It is possible to produce integral deposits, which assist in reflowing without voids. Replacement of the solid metal tags with suitable mesh aperture patterns that join the isolated deposit apertures, permits the deposition of sufficient solder material and flux in the paste. This encourages the necessary conditions to provide surface tension, ensuring solder flows over all areas giving an even fillet all around the can.

Since both the PCB and screening can initially possess good solderable finishes, the addition of printed solder paste, combined with the heat applied during reflow often encourages the migration of solder up the can walls away from the PCB. The result is sometimes less than pleasing in terms of appearance and may not provide effective shielding or the mechanical strength required in the solder fillet.

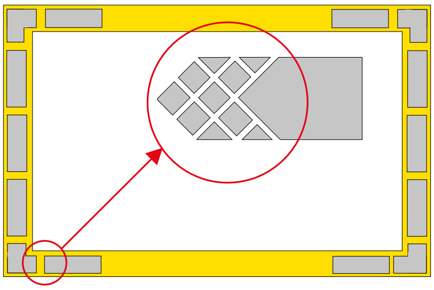


Figure 6.32 Paste deposits for screening cans

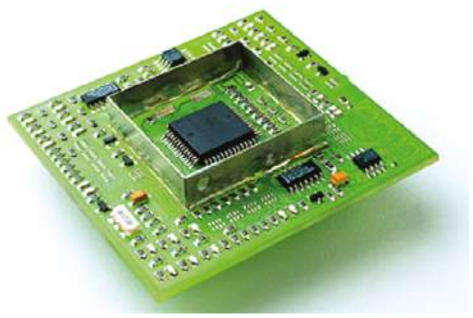


Figure 6.33 Conventional assembly method showing paste migration.

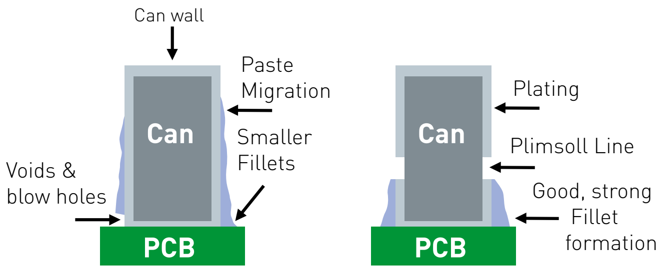


Figure 6.34 Paste migration

A unique solution to this problem of solder fillet migration is the Reflow Plimsoll Line (RPL). The interruption in the plating finish on the can wall allows the solder deposit only minimal migration and ensures fillet strength and volume are maximised. It also avoids the problems associated with the formation of blowholes, gaps and the untidy appearance of the can wall finish.

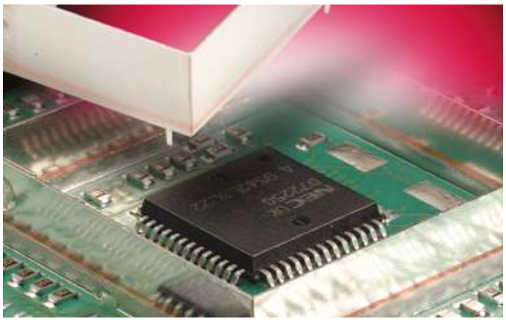


Figure 6.35 Reflow Plimsoll Line (RPL)

6.4.1 Stencil design considerations for screening can applications

A traditional approach, including webs between adjacent apertures as shown in figure 6.36, often results in isolated paste deposits that do not always deliver integral solder fillets and may need rework.

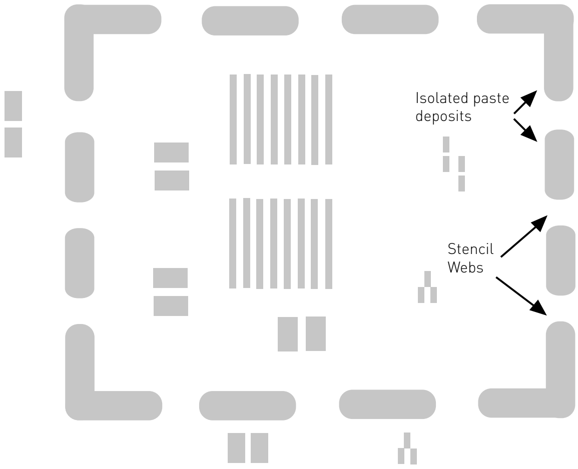


Figure 6.36 Stencil layout with webs between apertures

The enhanced stencil design, shown in figure 6.37, includes metal mesh and ensures paste deposits on the whole of the can wall - the presence of flux and paste encourages the reflow process.

Multi-level stencils can also ensure appropriate paste deposits as illustrated in figures 6.38 and 6.39.

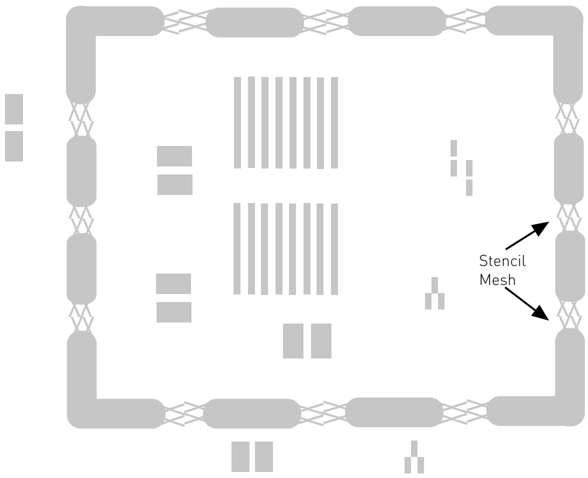


Figure 6.37 Enhanced stencil design

Figure 6.38 Conventional single-thickness paste deposits

Figure 6.39 Selective multi-height paste deposits can be achieved using multi-level stencils

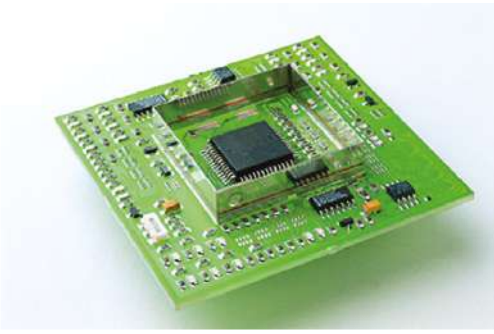


Figure 6.40 Screening can assembled using reflow technique, in combination with multi-level stencil, replacement of stencil webs and Reflow Plimsoll Line.

7.0 rework

a range of solutions designed to ensure consistent reworking

7.1 Rework solutions

Electronic assemblies today often contain diverse component populations; some can be reworked using conventional skills such as hand soldering.

A different approach is, however, required for many surface mount components, including:

- PLCCs
- QFPs
- BGAs
- µBGAs
- SM connectors

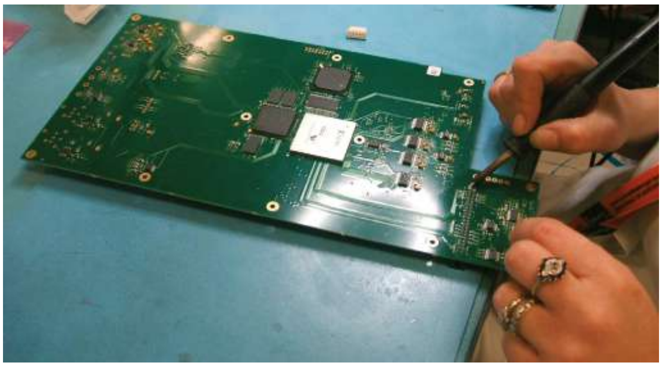


Figure 7.1 Hand soldering

Rework stencils mimic the footprint and aperture sizes, corresponding to both the device requirements and the original stencil used in production. In combination with a rework station, alignment is simple, resulting in a controlled 'right first time' process.

Offering optimum reliability and consistent printed results, rework stencils are both strong and rigid to ensure positive gasketting with the PCB surface. The compact designs include minimal stencil projection beyond the image to be printed, enabling use in all situations, especially high-density component areas.
Accurately folded edges and run-on / run-off areas ensure effective paste roll and containment without causing interference to other components on the populated PCB.

In addition to specialised direct component printing or PCB printing rework stencils, other rework solutions include dip transfer plates, BGA re-balling stencils and reflow reflector shields.

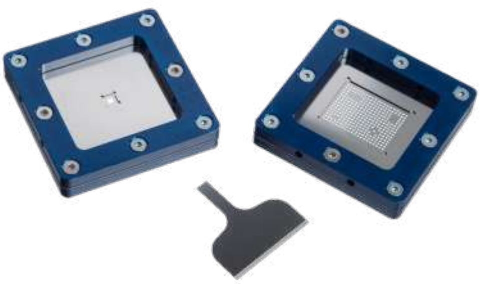


Figure 7.2 Direct printing stencils

Figure 7.3 PCB printing rework stencils

Dip transfer plates enable the accurate application of precise flux volumes to the lower quartile of the solder balls of Ball Grid Array (BGA) components, thereby eliminating excessive flux contamination on the underside of the reflowed component.

BGA re-balling stencils offer advantages in two areas. Firstly, to ensure the precise registration of new solder spheres onto pre-printed flux deposits, and secondly, in the form of the thicker stencils necessary for printing solder paste onto the BGA itself to create new solder sphere terminations.

For depositing paste onto the pads of the BGA device itself, where the printed device will be picked and placed onto the printed circuit board (PCB) and subsequently reflowed, direct component printing stencils are used.

During reflow, reflow reflector shields provide critical protection for heat sensitive components on densely populated PCBs, where an adjacent device is to be reworked / reflowed. The reflow shields are custom-made to fit over leaded components such as connectors, crystal oscillators, switches, inductors and capacitors, during reflow. In the reflow oven the shield reflects heat energy, thereby preventing damage to the component body, while simultaneously allowing conducted heat from the bottom of the PCB to reflow the solder paste effectively, ensuring a good joint every time.

Figure 7.4 Reflow reflector shields

7.2 Fine pitch suitability

Stencil technology	Dimensional tolerance	Paste release	Ease of forming
Precision etched 	± 9µm	4	✓✓
Laser cut / laser-formed 	± 5µm	4.5	✓

7.3 Rework stencil design considerations

Although rework stencils are used to print solder paste for single component footprints their design and suitability are examined in the same way as a stencil used to print the solder paste for an entire component population in terms of:

- Aperture size / shape
- Stencil thickness
- Aspect ratio / area ratio
- Paste release / wall roughness
- Solder paste rheology / particle size

7.4 Rework option 1 – Printing paste on to the PCB

1. Remove faulty component using hot air reflow
2. Clean copper pads to level/flatten footprint. Once cleaned it should be remembered that the solder remaining will invariably be more than was present when the PCB was assembled and as such a thinner stencil may give better results
3. Align stencil and print paste. It is recommended that a holding jig is used so that once alignment is achieved, the operator only has to concentrate on printing the paste. Solder paste must roll across the stencil to fill the apertures and during printing it is essential that the stencil is not disturbed
4. Once printing has been completed the stencil should be lifted vertically from the PCB to leave the paste deposits on the solder pads
5. Place new component
6. Localised reflow
7. Inspect

Poor solderable pad finish flatness will prevent stencil gasketting, causing bleeding and bridging which can result in further rework. When preparing the solder pads before printing care should be taken to achieve the flattest possible finish.

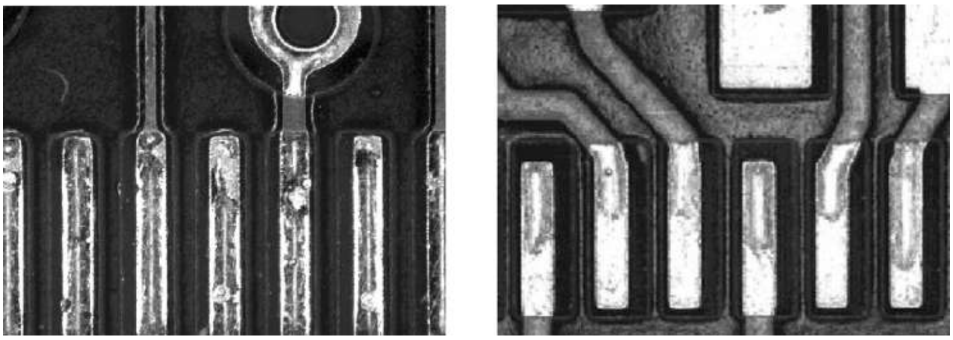
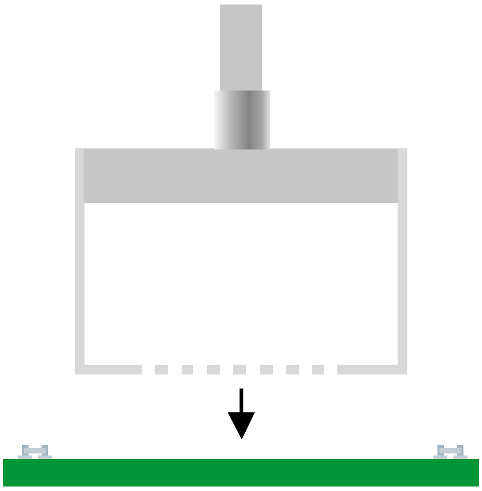
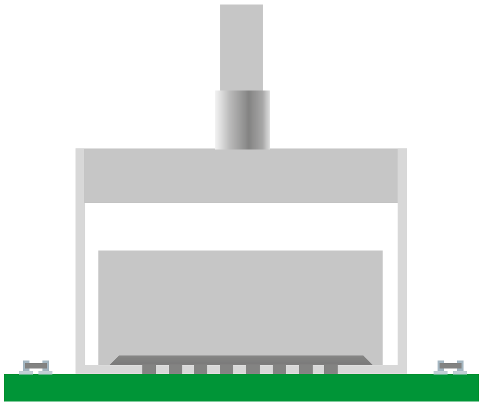


Figure 7.5 PCB pad finish

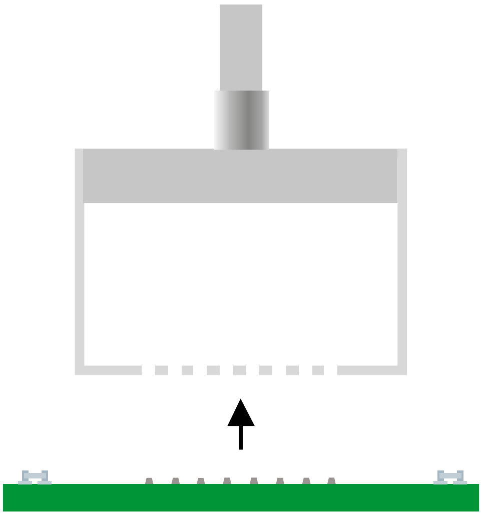
7.4.1 Mini rework stencil printing



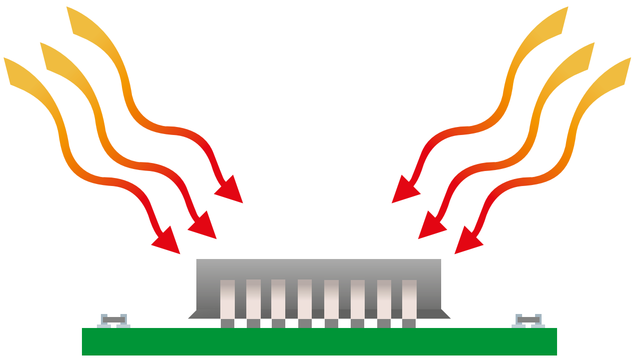
Step 1: Accurately align stencil above the solder pads.



Step 2: Ensure solder paste rolls to fill the apertures.



Step 3: Ensure straight lift separation of the stencils from the PCB.



Step 4: Localised reflow of solder paste to form solder fillets.

Figure 7.6 Mini rework stencil printing

7.4.2 Rework stencil keep out zone

Keep-out zones are the distances between the terminations of the device being reworked and the nearest adjacent component. Once a PCB has been fully assembled the packing density will affect the clearance surrounding the rework stencil. Sometimes it can be more effective to remove any discrete components that interfere with achieving a minimum clearance of 2.50mm.

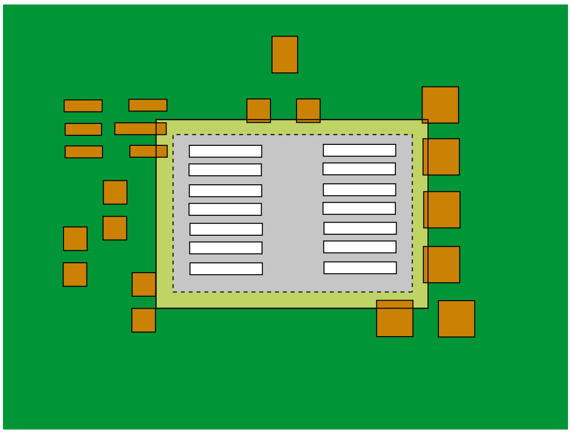


Figure 7.7 Rework stencil keep-out zone

Creation of a “ski-ramp” at the end of the rework stencil can both accommodate adjacent components and allow uninterrupted printing with any excess paste remaining on the ramp.

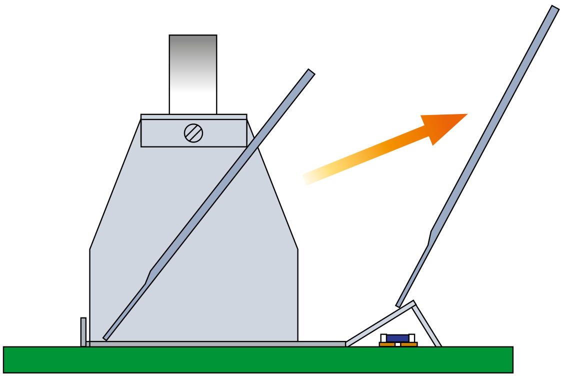


Figure 7.8 Ski-ramp

It is essential that rework stencils are provided with straight lift to the PCB surface and also after printing. Any angular variation from 90° will create pressure on one edge of the stencil, interrupting its correct seating, causing gasketting problems.

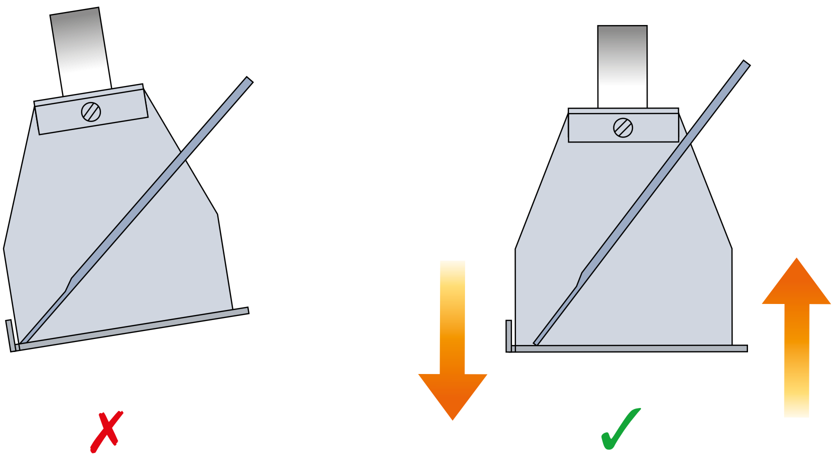


Figure 7.9 Rework stencil angle

7.4.3 Rework stencil design options

Hollow stalks, vacuum nozzles, screwed threads and simple handheld interfaces can all be provided. With any stencil format, straight lift of the stencil after printing and providing sufficient head height to permit a printing angle of approx. 60 degrees are essential.

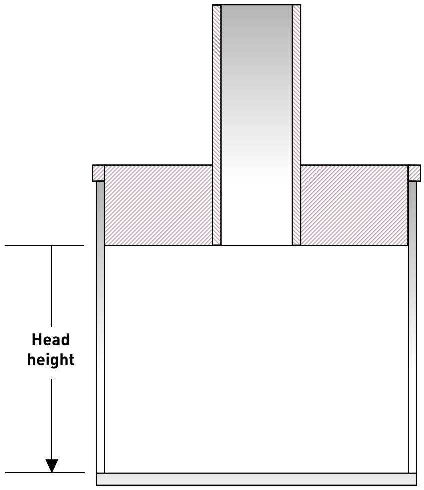


Figure 7.10 Stencil head height

7.5 Rework option 2 – Dip transferring sticky flux (gel flux)

1. Remove faulty component
2. Clean copper pads
3. Apply gel flux to component
4. Place new component
5. Localised reflow
6. Inspect

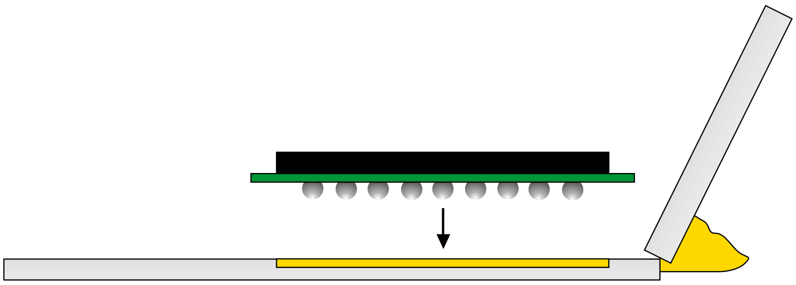


Figure 7.11 Dip transferring sticky flux

Sticky flux can be used as an alternative to solder paste for plastic BGAs. The application of excessive volumes of flux leads to problems when the assembly requires visual inspection or cleaning as the hardened flux is difficult to see through or remove.

Dip transfer plates ensure that only the lower quartile of each ball is coated to provide sufficient flux for reflow without the risks associated with excess volumes. The plates are profiled using photo etching to provide the precise recess depths required.

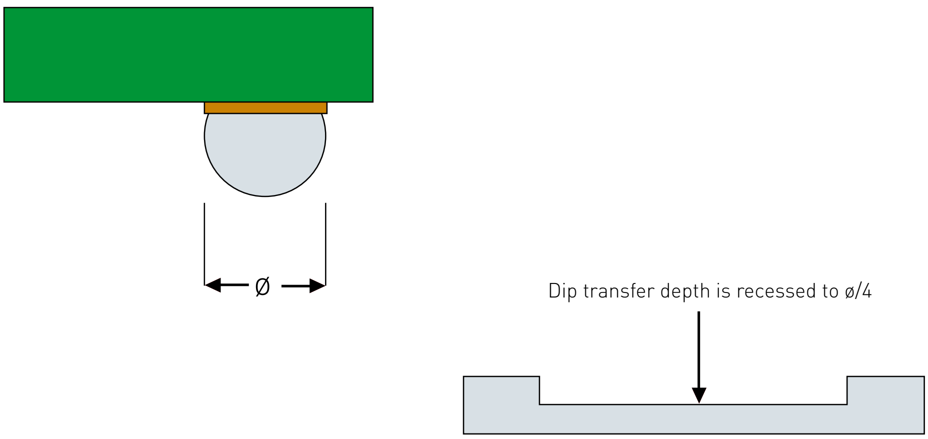


Figure 7.12 Dip transfer depth

7.6 Rework option 3 - Direct printing onto device terminations

1. Remove faulty component
2. Clean copper pads
3. Place component in stencil nest
4. Hold component in place in nested mini rework stencil using finger
5. Turn over nested rework stencil
6. Put paste on stencil
7. Use mini squeegee blade to apply paste to component
8. Turn over stencil
9. Place nested mini rework stencil in rework machine clamp (if applicable)
10. Localised reflow
11. Inspect

Component should be nested to avoid any stress before stencil is aligned.

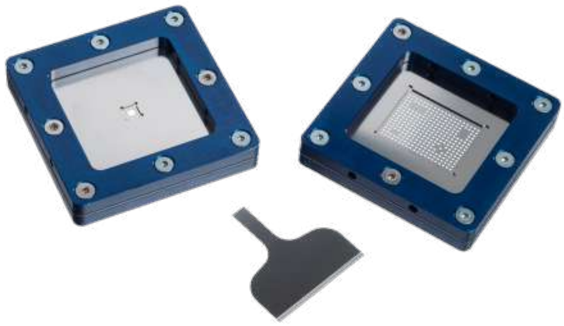


Figure 7.13 Direct printing mini rework stencil and mini squeegee blade

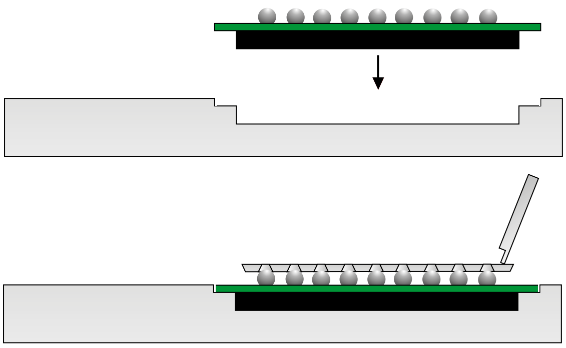


Figure 7.14 Component nesting

Stencil thickness and aperture diameter need to be considered carefully to ensure the solder paste is released in consistent volumes on top of the BGA solder balls.

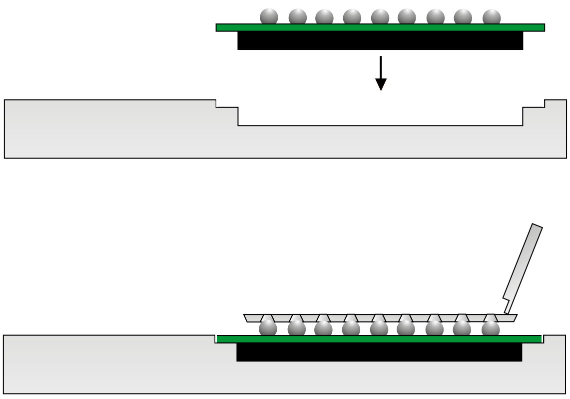


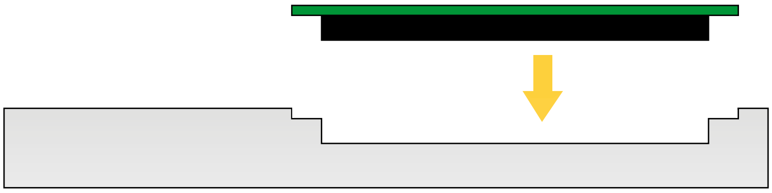
Figure 7.15 Direct printing

Figure 7.16 Results from a 0.150mm direct printing rework stencil

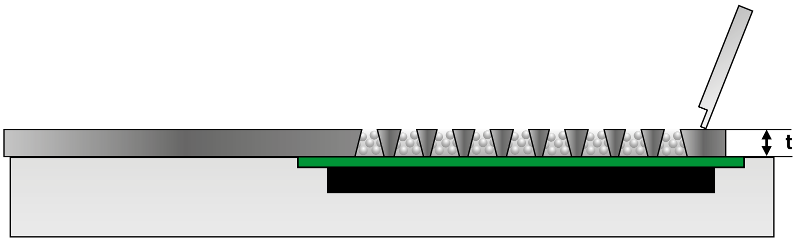
7.7 Rework option 4 - BGA re-balling stencil

It is possible to re-ball BGAs using thicker stencils to provide the necessary ball diameters the device was created with. It must be remembered however that solder paste although 88-93% metal by weight, is only 50% by volume. As such, typical stencil thicknesses required are 0.5 – 0.65mm depending on the sphere diameter.

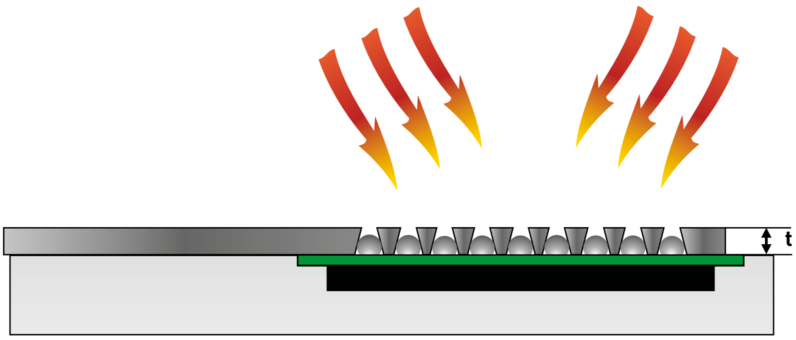
The Component should be nested to avoid any stress before the stencil is aligned.



Step 1: The BGA must be nested carefully before the stencil is aligned.



Step 2: The exact thickness of the stencil (t) depends on the sphere diameter required.



Step 3: Subsequent reflow with the stencil in place creates the spheres required.

Figure 7.17 BGA re-balling stencil printing

7.8 Reflow reflector shields

The primary use of this protection is to protect susceptible plastic components or connector bodies from the affects of heat during the Pin in Hole Reflow process. Being thin and reflective they can be designed to reduce the temperature within the shield by up to 40°C relative to the outside temperature.

Simple 5-sided shields for prototype assembly can be augmented using spot welding to provide shields that will endure many reflow operations.

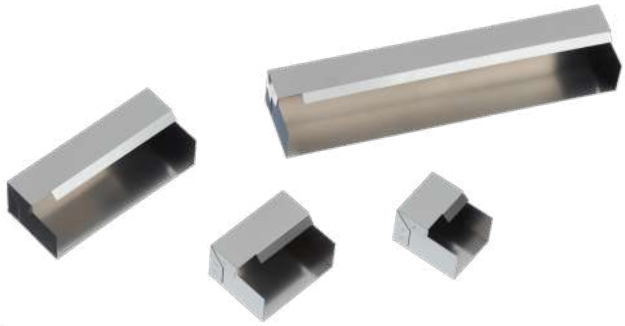


Figure 7.18 Reflow reflector shields

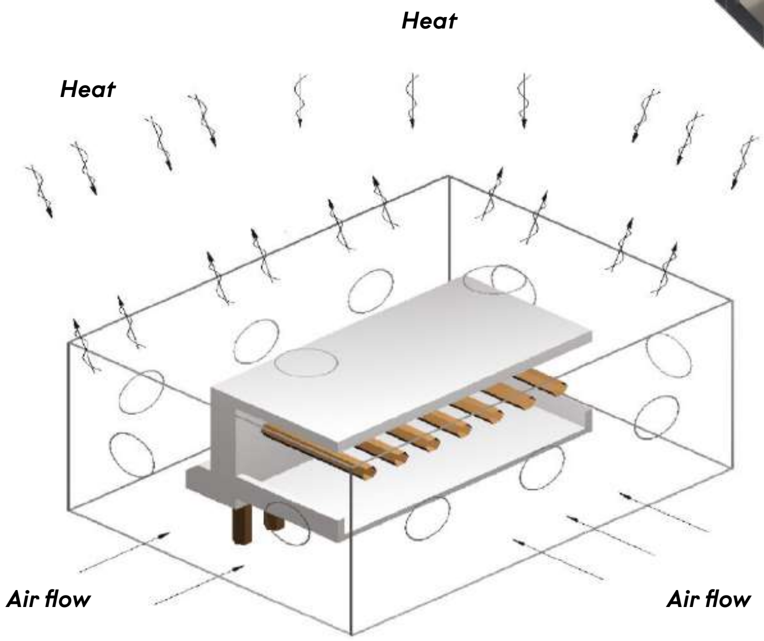


Figure 7.19 Schematic diagram showing a reflow reflector shield in operation

7.8.1 Reflow reflector shield results

Reflow shields can also be used to good effect, protecting surrounding devices from the effects associated with subsequent reworking of an adjacent component. Simply locating a reflow reflector shield over the surrounding susceptible components protects them from any adverse effects resulting from the reflow temperatures involved.

Reflow reflector shield solves the heat susceptibility problem

Some components go into meltdown if not protected

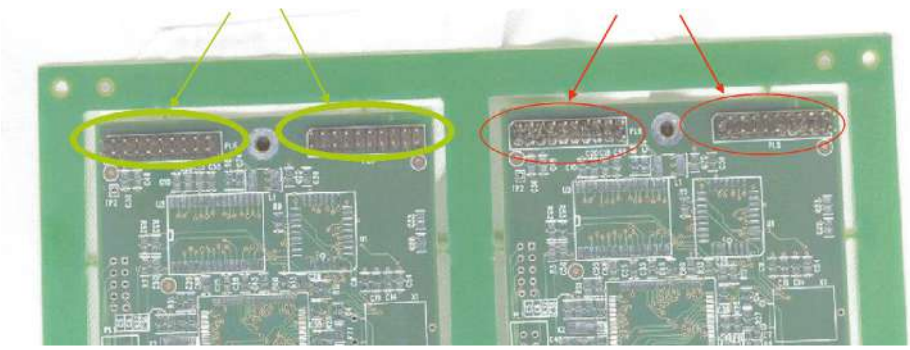


Figure 7.20 PIHR assembly using conventional non-reflow capable headers

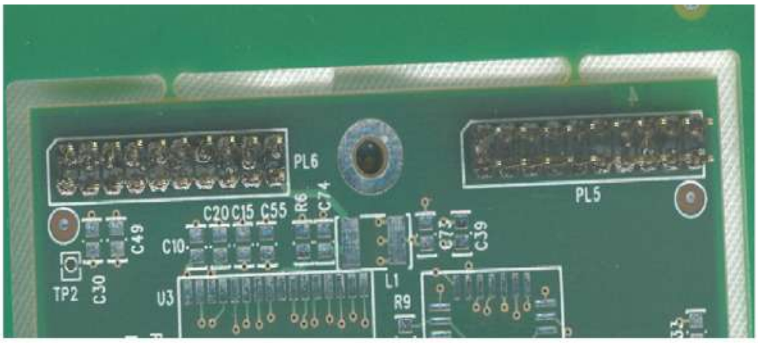


Figure 7.21 Evidence of severe heat distortion after reflowing without a reflow reflector shield

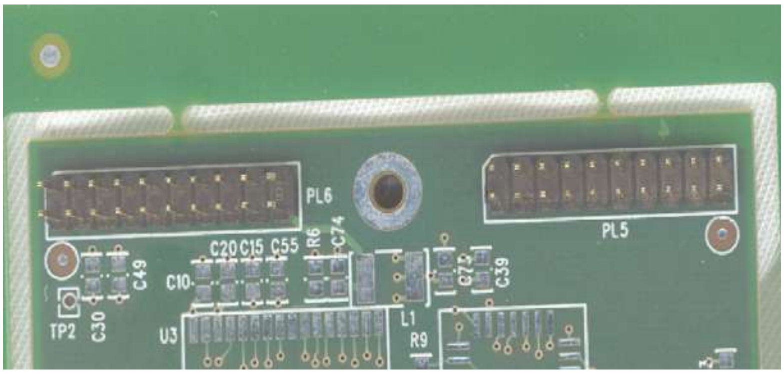
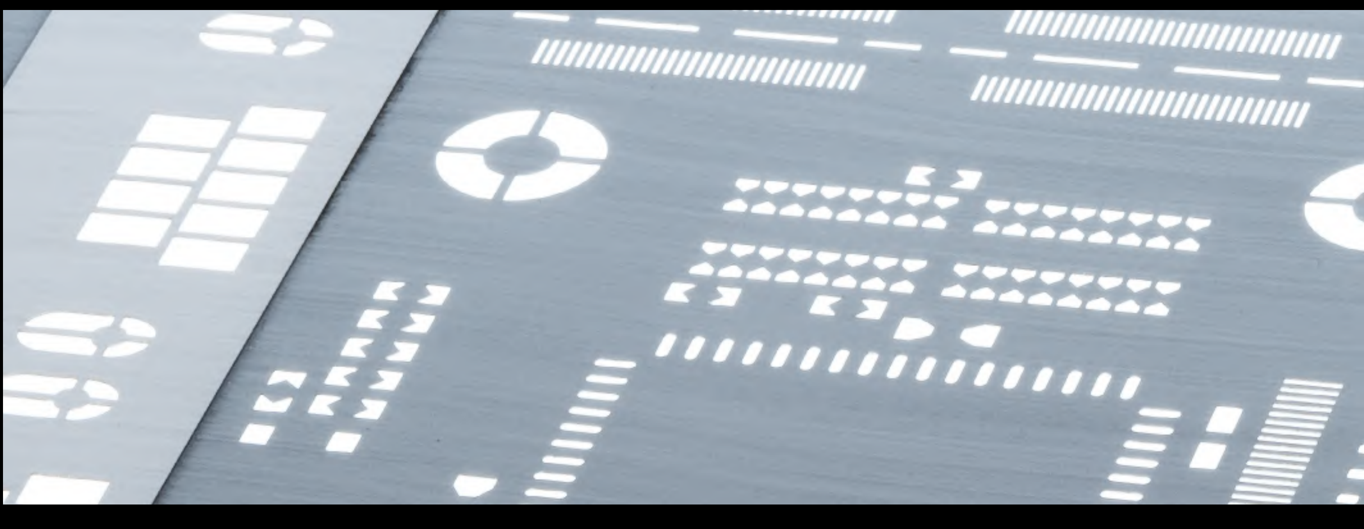


Figure 7.22 Use of a reflow reflector shield has preserved the mechanical integrity and thermal stability of the headers

8.0 other stencil applications

considerations for glue and pin-in-hole reflow



8.1 Glue

Epoxy glue is used as a surface mount adhesive (SMA) to attach and maintain surface mount devices (SMD) to the PCB surface throughout placement and wave soldering. As such, thicker stencils which enable increased adhesive deposits are required.

Since the development of single component epoxy technology for use as a SMA, the methods of application themselves have evolved, driven by the requirements for increased assembly speed and improved process repeatability.

Adhesive stencil printing utilises existing production technology and techniques to optimise the efficiency required from a surface mount line, particularly relevant when densely populated assemblies are processed. Total print times of less than 15 seconds are achievable.

8.1.1 Wave soldering

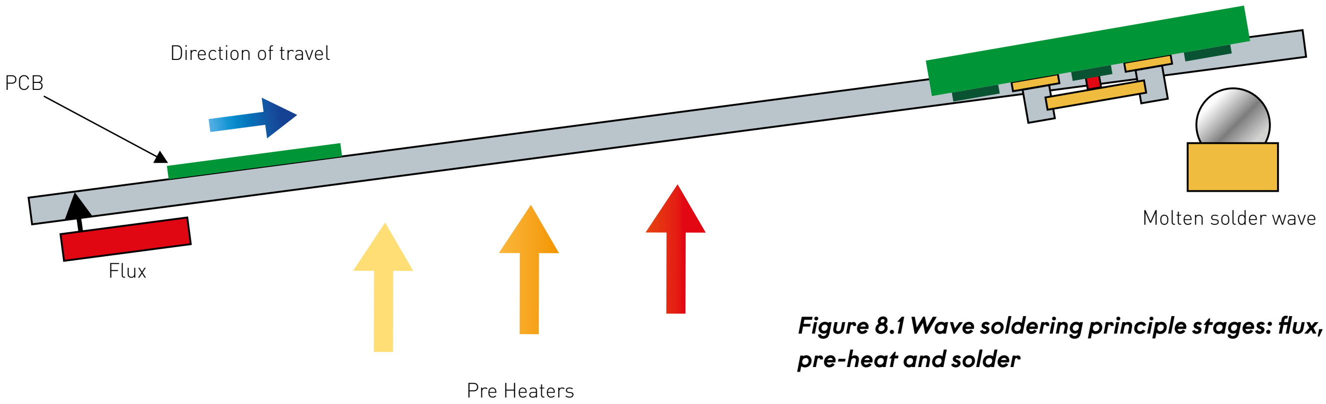


Figure 8.1 Wave soldering principle stages: flux, pre-heat and solder

Figure 8.2 Wave soldering machine

Figure 8.3 Conveyorised board transport

Figure 8.4 PCB entering solder wave

8.1.2 Glue stencils

The stencil technology differs from that used to print solder paste, where the intention is to transfer the entire contents of the aperture onto the PCB. When glue printing, however, good use is made of the adhesive's ability to remain as a partial retention in the apertures. This is based on the simple relationship of the surface area of the deposit base (PCB contact area) relative to the aperture wall surface area.

8.1.3 Glue printing principles

For components of a similar stand-off a stencil of 0.008" should be used. Where a variety of components exist a stencil of 0.010" should be used. The resultant glue dot heights are a direct result of the aperture diameter relative to the stencil thickness.

Where diameter = stencil thickness the dot heights are approx. 1/3 of the stencil thickness used. Using apertures of twice the stencil thickness delivers dot heights of between one and two stencil thicknesses. Increasing the aperture further delivers dots of approx. the stencil thickness.

Glue deposits can exist as single dots, double dots or simple slots. In each case it is important that the aperture dimensions do not exceed 1/3 of the distance between the inner most edges of the discrete component pads. The reason is that when a component is placed onto the adhesive dot it will flatten and spread out and the contamination of the copper pads on the board with adhesive should be avoided at all cost.

Photo etched stencils can provide an aperture profile on the top of the stencil that assists with increased glue dot heights. It is also possible using laser stencils to manufacture stencils with aperture sizes the same as the stencil thickness, 0.200mm in 0.200mm (0.008" in 0.008"), this can be useful when the pads on the PCB are too close together.

8.1.4 Multi height deposits

Where apertures are small, for example 0.012"/0.3mm, the adhesion between the glue and stencil effectively retains some of the deposit and the resultant dots have a small or low Glue Dot Height (GDH).

Stencil apertures of 0.8mm (0.032") ensure a larger percentage of the adhesive is transferred onto the PCB. When the stencil and board are separated the stencil drags the adhesive and the resultant dots will be higher.

For apertures of between 0.060" to 0.080" (1.5mm to 2mm) most of the adhesive is transferred onto the PCB and the GDH will be similar to the thickness of the stencil.

Each adhesive has individually distinct properties and characteristics which may require some process variations. Most manufacturers offer guidelines and basic design rules to ensure compatibility with screen printing.

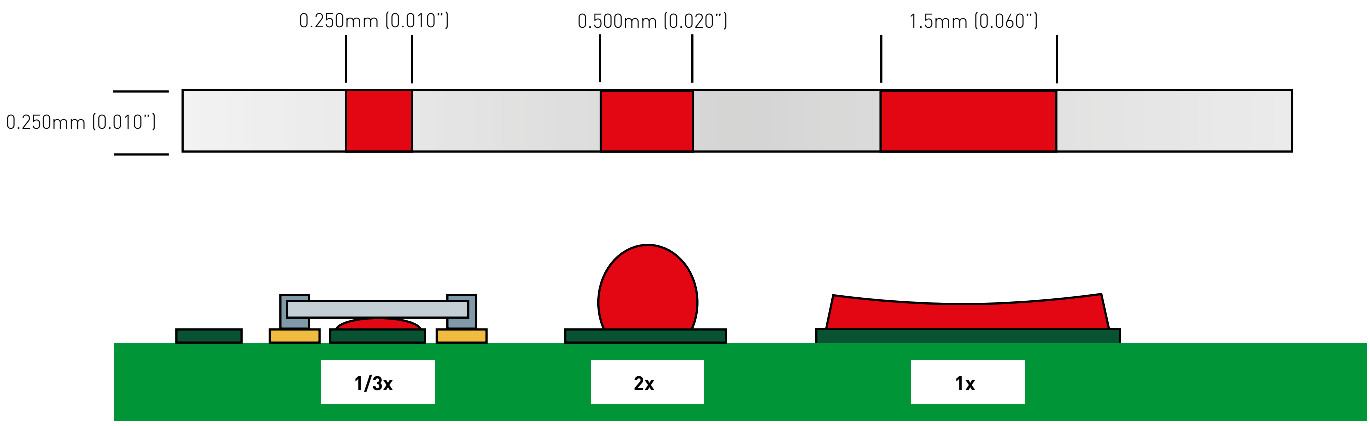


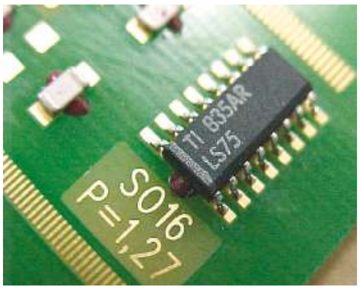
Figure 8.5 Deposit height relative to glue dot diameter and stencil thickness

8.1.5 Glue stencil design

There are three variables to consider when designing the most effective pattern for SMD attachment:

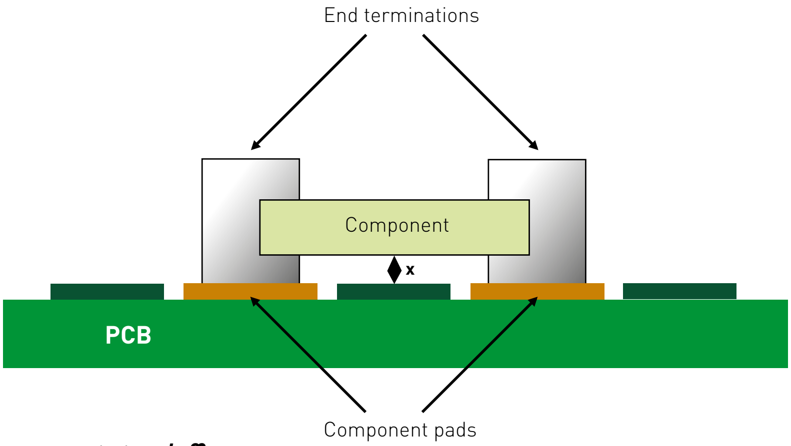
- Component standoff
- Stencil thickness
- Pad design

8.1.5.1 Component Standoff



This refers to the distance between the PCB and the underside of the component, shown as x in figure 8.7 below.

Figure 8.6 Component standoff



A standoff of 0.002" (0.05mm) will cause the deposit to spread 100% more than if it was 0.005" (0.125mm), see figure as shown in figure 8.8.

Figure 8.7 Component standoff

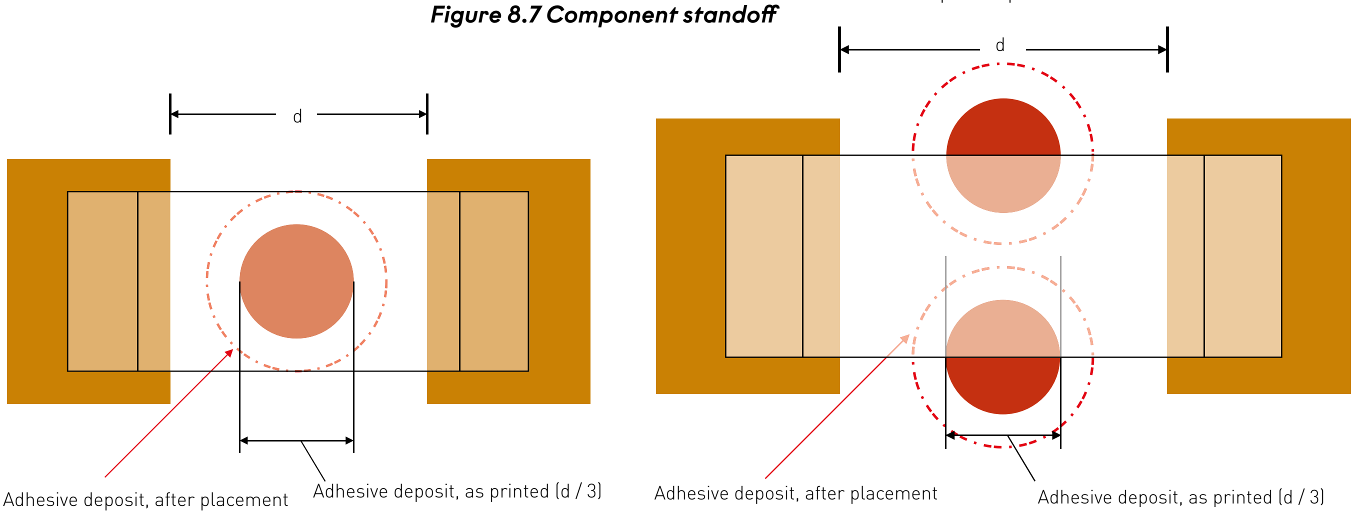


Figure 8.8 Effect of placement

The difference could be enough to allow the adhesive to spread onto the solder pads, causing joint contamination and preventing effective soldering of the component. To avoid this problem, double dot printing is recommended as shown in figure 8.9.

8.1.5.2 Stencil thickness

This is dependent upon the type of SMDs being used but is usually between 0.008" - 0.012" (0.2-0.3mm). 0.008" (0.2mm) is more appropriate for components with similar standoffs and 0.010"-0.012" (0.25- 0.3mm) is used where the mix of components require deposits of different heights (multi-height deposits).

8.1.5.3 Pad Design

This is usually dependent upon the clearance between the deposit and the component pads after placement. Three alternatives are available as shown in figure 8.10

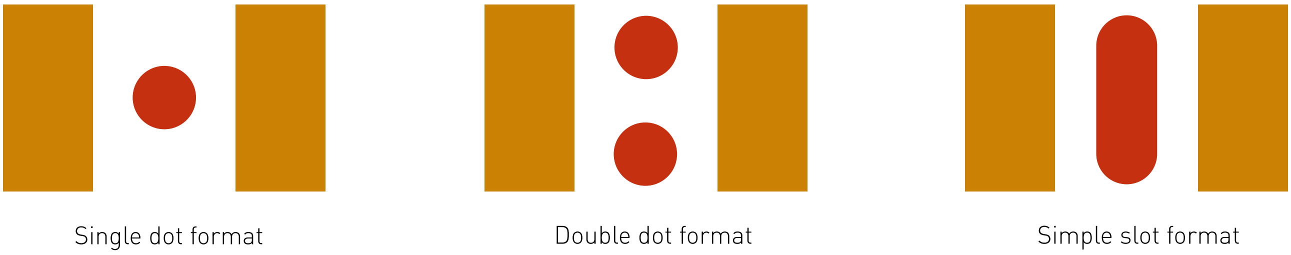


Figure 8.10 Dot formats

8.1.6 GDD and GDH

Glue dot diameter (GDD) and glue dot height (GDH) of the deposits depend on:

- Diameter of stencil apertures
- Stencil thickness selected
- Viscosity/rheology of adhesive (manufacturer defined)
- Surface roughness of the stencil (friction between adhesive and aperture walls)
- PCB surface condition. An uncontaminated dry surface is necessary to ensure optimum adhesive printing

The following table shows approximate GDDs required when the adhesive has been printed with a metal squeegee blade through a 0.010" (0.25mm) Tecan glue stencil.

Component Size	GDD in mm	Component size	GDD in mm
0402	0.4	Mini-melf	1.1
0603	0.5	1812	1.4
0805	0.6	SO8	3 x 1.5
1206	0.8	SO14	3 x 1.7
SOT 23	0.9		

Equally good results have been achieved using multi-level stencils with 0.008" (0.200mm) general thickness rising to 0.014" (0.350mm) in selected areas.

Tecan can create a glue stencil from the copper pad and silk screen data layers in combination with the component designation file.

Some components have larger stand-offs and as such higher printed glue dots are required. It is crucial to identify the components that need higher glue dots since too little glue will only result in loss of the components to the solder wave.

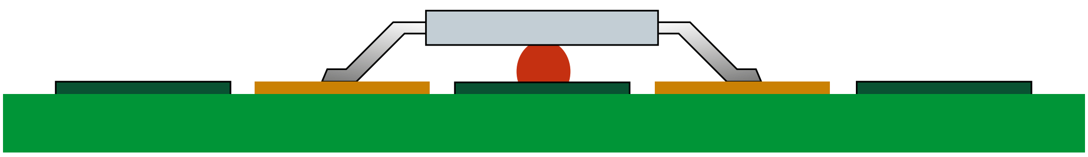


Figure 8.11 Aperture modifications to increase GDH

Aperture design and stencil thickness can be used to provide extra glue dot height.

The compass design enables the glue deposit to achieve a good solid base with the glue in the four points contributing to glue dot height in the centre when the substrate separates from the stencil.

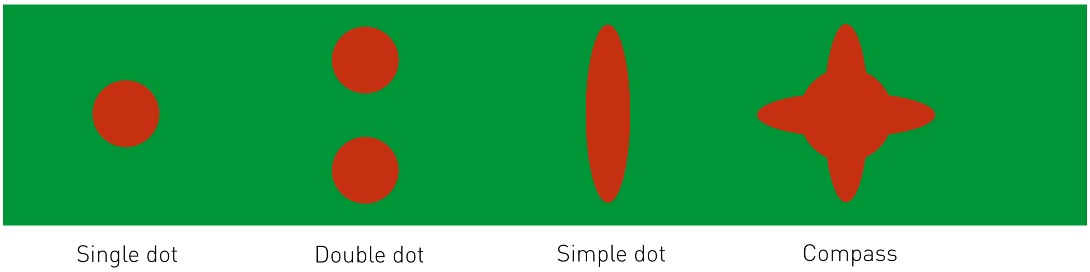


Figure 8.12 Aperture design

When higher glue dots are required to overcome increased component standoffs, use of the compass design may need to be combined with both a normal print and a subsequent flood. Flooding is carried out without pressure and can leave a residue on the stencil that collapses into the larger apertures to provide an extruded dot height of between three and five times the stencil thickness selected.

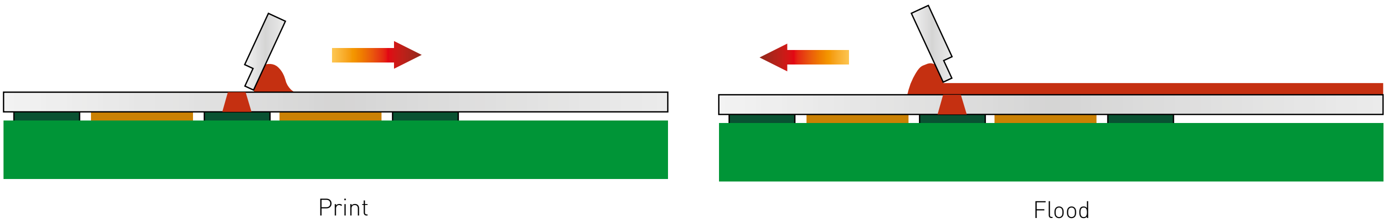


Figure 8.13 Print and flood

8.1.7 Stencil cleaning

As the SMAs mentioned do not contain particulate matter, cleaning is easy and the selection of effective stencil cleaning solutions will depend upon the stencil format, the SMA selected and the manufacturer's recommendations. Care must be taken when using mesh mounted stencils as some cleaning solutions, concentrations and temperatures required may attack or destroy the bonding agents.

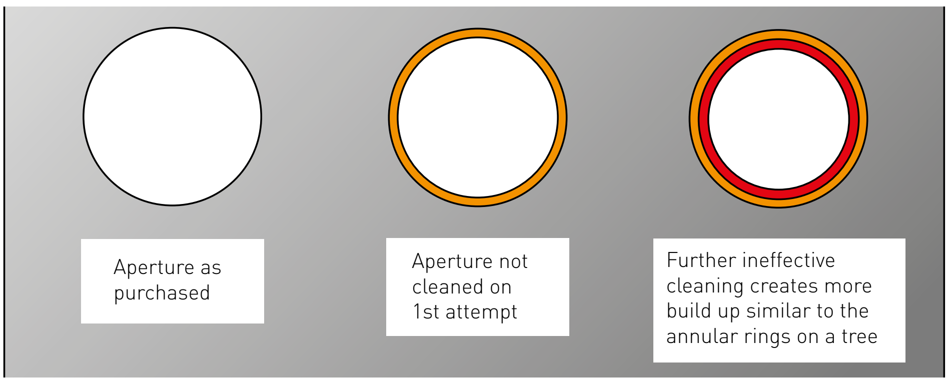


Figure 8.14 Stencil cleaning

8.2 Pin-in-hole reflow (PIHR)

This assembly technique is used extensively as an alternative for subsequent hand or wave soldering operations of leaded components - from crystal oscillators and headers to multi-row connectors.

As with all surface mount technology, successful end product quality relies on the accuracy and adequacy of the printed solder paste deposits. Optimised stencil design is an essential requirement to ensure the printing solution delivers successful PIHR results.

In its simplest form, the technique involves printing solder paste onto the printed circuit board for both surface mount devices and leaded components. With PIHR it is essential that sufficient paste volume is available to ensure the required solder fillets are formed on the top and bottom sides of the PCB and in the component holes.

Following insertion of leaded components and placement of surface mount devices the assembly is reflowed. Shorter product cycle-times, reductions in manufacturing costs and elimination of contamination by second operation flux deposits are the main benefits.

Figure 8.15 Pin-in-hole reflow (PIHR)

There are basic guidelines and considerations, which need to be followed to ensure success using this process.

8.2.1 PIHR criteria

8.2.1.1 Solder fillet appearance

Resultant PIHR solder fillets can have a flatter profile than those from hand or wave solder processes but with careful paste volume matching it is possible to create fillets that resemble the traditionally accepted profiles. One important consideration is that solder paste, although 88-92% metal by weight, is only approximately 50% by volume. This cannot be ignored when calculating the theoretical solder fillet volume required, since absence of paste volume before reflow will only lead to insufficient solder fillets on the finished assembly. Although the solder fillets may look different, their strength and electro-mechanical integrity is not affected.

8.2.1.2 Resist compatibility

Printing off pad is the first option available when trying to achieve greater solder volumes - similar to hand or wave soldered joints. It is necessary to print paste onto the solder resist surrounding the component pad as well as onto the pad itself and also into the hole (see figure 8.16 below). Increased surface area alone can, in some instances, provide the extra volume required.

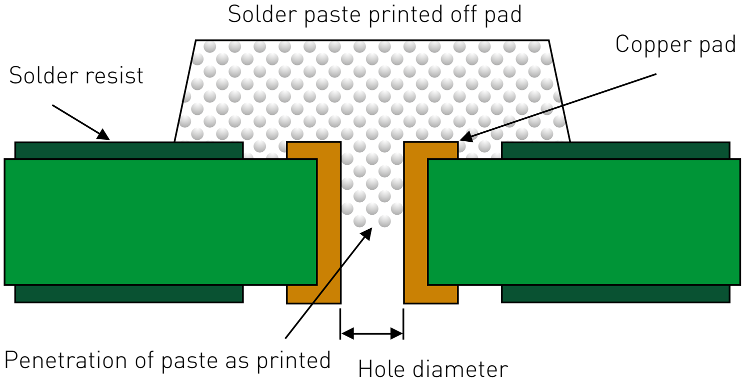


Figure 8.16 Resist compatibility

This method can be used providing the solder resist used on the PCB will facilitate the retraction of the paste deposits during reflow, without leaving isolated satellite solder balls behind.

A simple test utilising either a PCB or portion of salvage can be carried out (see figure 8.17)

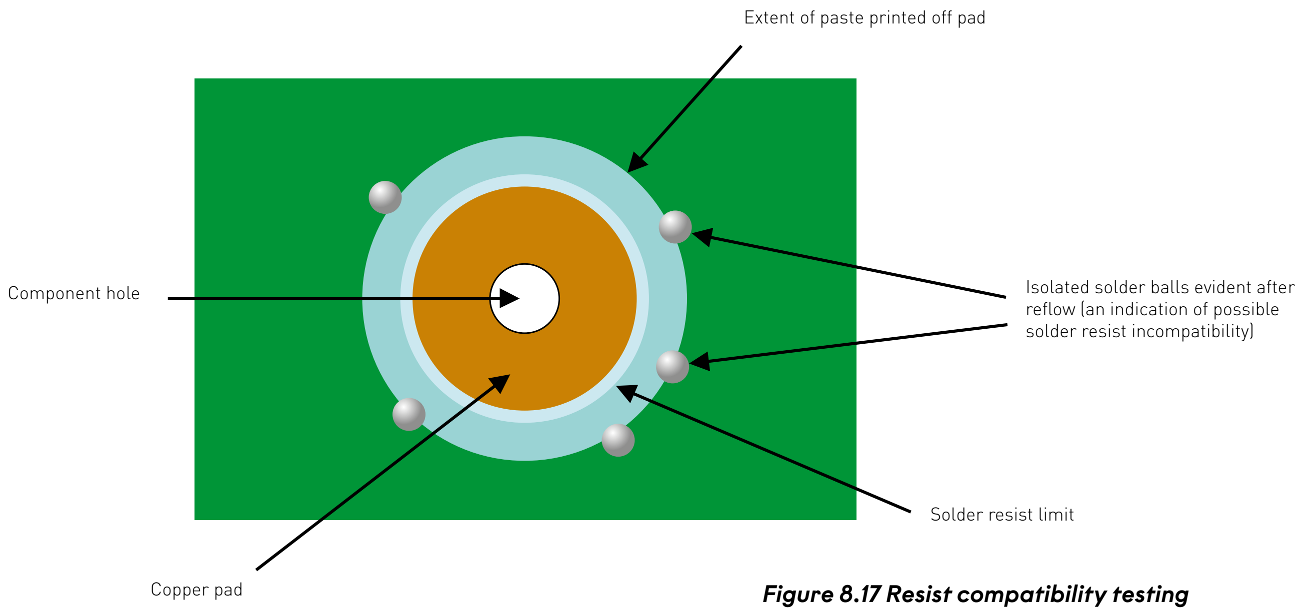


Figure 8.17 Resist compatibility testing

8.2.1.3 Penetration of paste

This depends largely on the squeegee speed and pressure used. Too quick and the penetration will be restricted, too high and excessive paste may result. In fact, it isn't necessary to completely fill the holes with paste, as the insertion of the component lead will displace a portion of solder paste. Under normal printing conditions with 4-6 kgs pressure and 10-25 mm/second printing speed, a penetration of between 45-85 % can be expected.

Varying these two parameters or overprinting can increase paste penetration. Achieving an average 60% solder paste penetration will mean a paste depth of nearly 1.0mm on a PCB 1.6mm thick. To achieve this, a reduction in speed is required from that generally accepted for surface mounting printing where expected paste height is usually 0.150-0.200 mm.

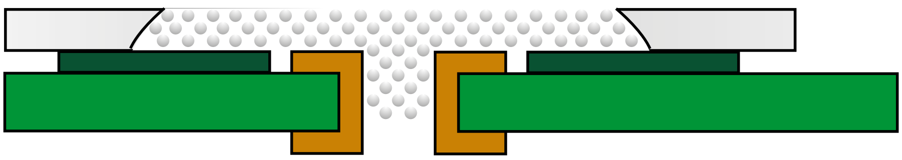


Figure 8.18 Penetration of paste

8.2.1.4 Component lead length/projection

Lead projection should be kept to between 1.2-1.5mm when using single thickness stencils with a maximum of 2.0-2.5mm possible where increased paste volumes are available using multi-level stencils.

The component lead will displace a percentage of the paste upon insertion with longer pins carrying paste further away from contributing to the bottom fillet formation.

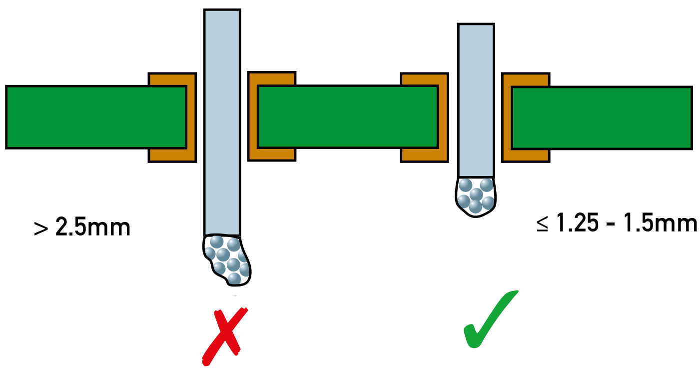


Figure 8.19 Lead projection

8.2.1.5 Auto insertion / manual insertion

Care should be taken to ensure insertion is carried out as accurately as possible to avoid disturbing or merging adjacent paste deposits. This is true for both hand and machine insertion. For machine insertion particular attention should be focused on the ability of the machine to hold or grip the component and on the design or suitability of the component body itself.

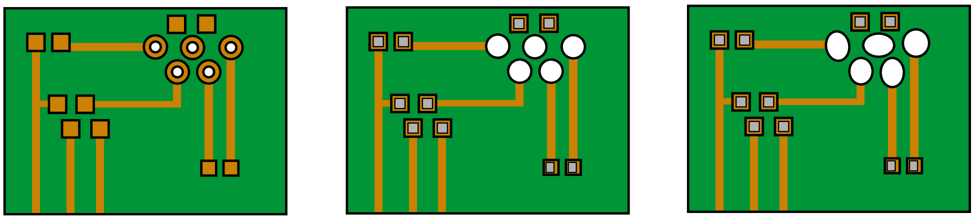


Figure 8.20 Comparison of bare PCB to printed with paste for PIHR

There is a noticeable difference between the paste deposits for surface mount devices with the general reduction in size from the original pad size and the expanded deposits associated with PIHR requirements. With manual insertion it is necessary for the operator to fully acquaint themselves with the board layout relevant to the component holes beneath.

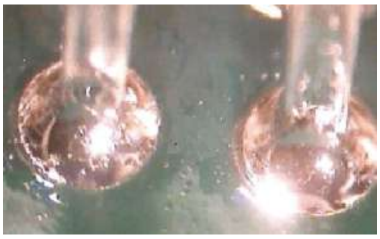
Good hand and eye coordination are required to consistently place 96-way connectors, but this can be achieved with practice.

8.2.2 PIHR component selection

Connectors selected for PIHR should be made of a material capable of surviving reflow without distortion, melting or cracking. During reflow the connector body will experience temperatures of between 215-230°C for a period up to one minute. To ensure the connector will survive; it is a good idea to select from a range that manufacturers qualify to temperatures 20-30°C above peak reflow temperature.

Although it may be possible to adjust the reflow profile to accommodate existing SMT components without damaging any plastic components, this technique has limited effectiveness.

The solder paste reflows and is attracted towards the hottest points, namely the PCB pads and component leads. Extended deposits retract to form the required solder fillets.



Left: Figure 8.21 Paste as printed

Right: Figure 8.22 Resultant solder fillets post reflow

8.2.2.1 Standoff height/relative position

Suitable components will have built in protrusions on their underside to maintain a standoff distance.

Printed paste deposits should not be closer than 0.100 mm (0.004") from the underside of the component body to avoid possible contamination of the body with the flux and solder spheres in the solder paste.

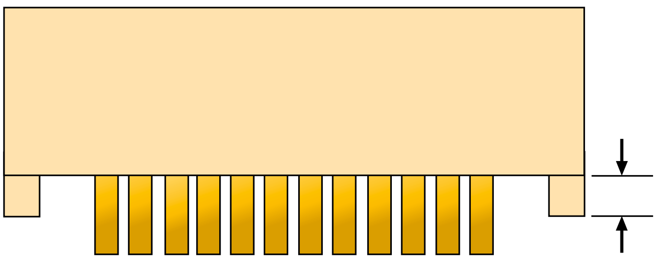


Figure 8.23 Standoff height

Select components with standoffs of 0.250-0.325mm (0.010"-0.015") to avoid contamination and provide inspection access for the top solder fillets.

Even on simple single row headers, the relative position of the standoff can limit the paste dimensions applied, to the point where paste volume is not sufficient to form the fillets required. Selection of an alternative component with in-line standoffs that replaces the crown standoffs (figure 8.25), allows greater flexibility to maximise paste volume.

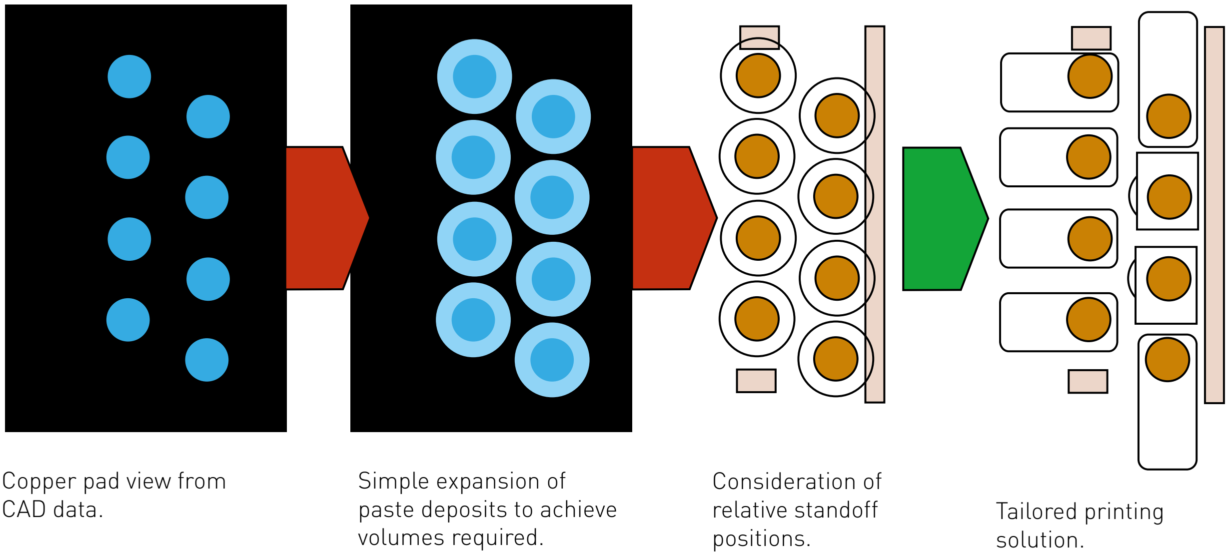


Figure 8.24 Paste deposit expansion

Standoff positions cannot be ignored; they can severely limit the overall result when their presence impacts on the stencil design, restricting expansion of the printed paste deposits and hence paste volume available.

Paste deposits don't have to be regular shapes to provide the volumes required (See figure 8.26)

Expansion should be limited to no more than 2.5-3.0 times the copper pad diameter in any direction. Ignoring this can create problems where the solder may not fully retract on reflow, leaving large solder balls behind between component terminations.

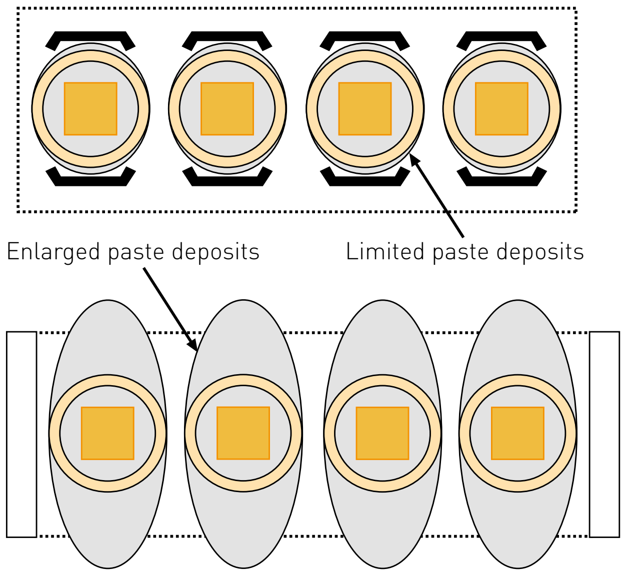


Figure 8.25 Limited vs enlarged paste deposits

8.2.3 Pin type

Round, square and oblong pins can all be used, providing a minimum clearance of approximately 0.250mm (0.010") exists between the pin and the inside edge of the component holes all around. Too small and insertion becomes difficult, too large and there may be problems ensuring 100% fill of the holes. When using square or oblong pins, the hole size should be calculated on the largest dimension plus the required clearance.

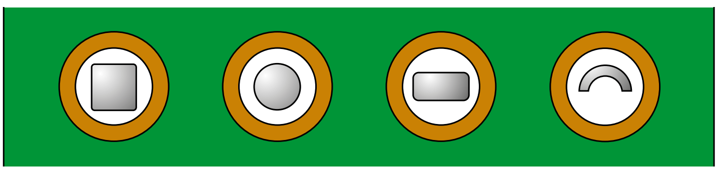


Figure 8.26 Sufficient clearance surrounding the pin is essential to ensure easy insertion

8.2.4 Stencil aperture designs

Solder paste, although a complex formulation, follows simple rules upon reflow having similarity to fine pitch printing where paste deposits should be distinct / isolated to avoid bridging and shorts.

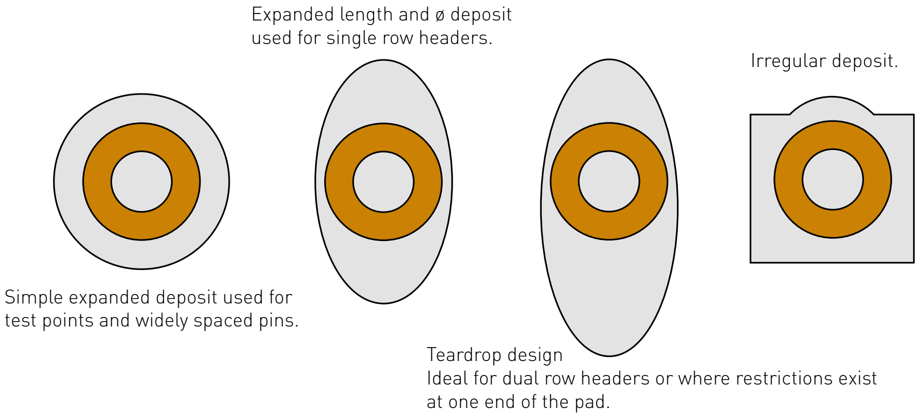


Figure 8.27 Stencil aperture designs

Using PIHR, bridged deposits would leave the paste confused and unsure where to go causing fillet variability or large isolated solder balls.

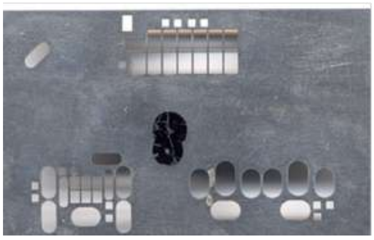


Figure 8.28 Typical elongated PIHR stencil aperture designs

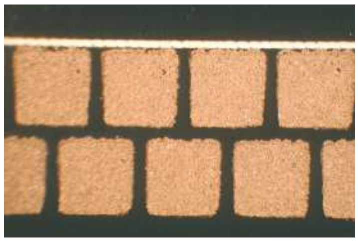


Figure 8.29 Resultant paste deposits

8.2.4.1 Volume is the result of area and thickness

When simply increasing the surface area of a printed deposit does not deliver the volume required, or the gap between adjacent deposits becomes smaller than 0.250 mm (0.010"), a different approach has to be adopted. Since PIHR combines existing surface mount technology with leaded components and those leaded components require more paste, a stencil with thicker areas is required.

Multi-level stencils offer distinct thickness levels on the same stencil, which can be tailored to produce the extra volumes required for PIHR applications whilst maintaining the appropriate stencil thickness for the surface mount devices, including fine pitch components.

8.2.4.2 Multi-level stencils / reactivity of squeegee blades

When using multi-level stencils, the squeegee solution must provide the flexibility and small reaction times required in both the ascent to and descent from the raised areas. Most surface mount assemblies are densely populated and as such provide only limited clearance between, for example, a large connector and a row of 0402 components. Recessed or mobile edged squeegee blades are able to offer improved control and consistency of multi-height printed deposits.

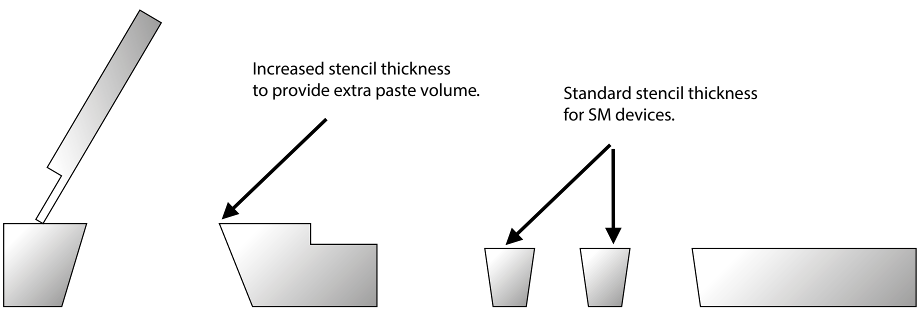


Figure 8.30 Recessed edged squeegee blades and multi-level stencils

8.2.5 Reflow reflector shields

Reflow reflector shields (see rework chapter) can be used to maintain existing reflow profiles whilst using non-reflow capable connectors. The shield reflects a large proportion of the heat from the sensitive component body whilst enabling reflow of the soldered terminations.

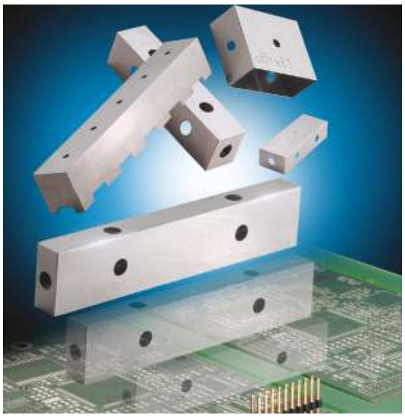


Figure 8.31 Reflow reflector shields

8.2.6 PIHR results

In this section we examine PIHR results with the following components:

- Molex header
- 25-way D-type socket
- 3-pin header
- 6-pin header
- 2-pin header with uncropped legs
- 20-pin DIL socket

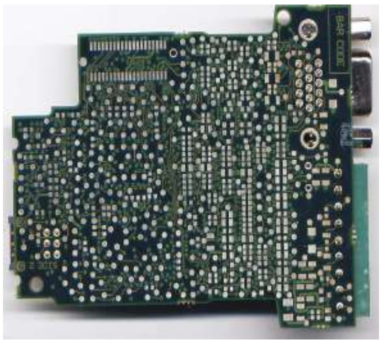


Figure 8.32 Typical PIHR PCB assembly

8.2.6.1 Molex header

Body moulding has been removed to examine the resultant solder fillets.

Figure 8.33 Top view

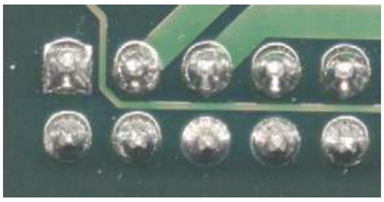


Figure 8.34 Bottom view

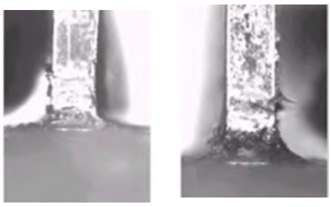


Figure 8.35 Top filets

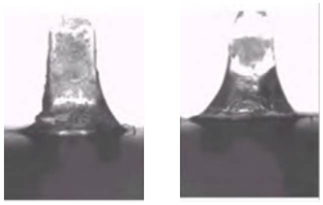


Figure 8.36 Bottom fillets

This component requires a reflow reflector shield to prevent meltdown. As you can see, the solder fillets have a high wetting angle and the paste has retracted from its printed position without leaving isolated satellite balls.

8.2.6.2 25-way D-type socket

This component also requires a reflow reflector shield to prevent meltdown. Again, the solder fillets have a good wetting angle and there is no evidence of isolated satellite balls.

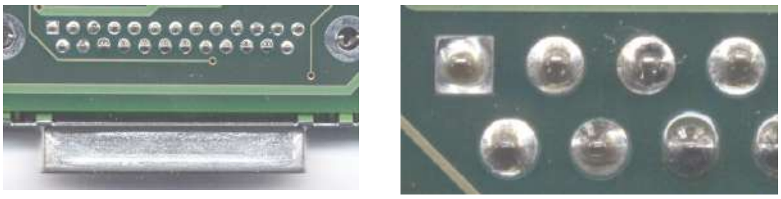


Figure 8.37 25-way D-type socket

8.2.6.3 3-pin header

Distortion of the plastic body has affected the pin positions.

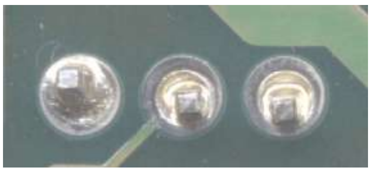


Figure 8.38 3-pin header

8.2.6.4 6-pin header

This component was reflowed using a reflow reflector shield. The volume of paste delivered was good, but, this component is not very well supported and during reflow it dipped slightly towards the PCB. One answer to this might be to dispense a glue to maintain its position.

Figure 8.39 6-pin header

8.2.6.5 2-pin inductor with uncropped legs

These were reflowed without the need for shields. Since their lead projection is significant the solder fillets are depleted by the paste residing on the ends of the component (it is too far away to retract back to the pads). A possible solution would be to crop before insertion.

Figure 8.40 2-pin inductor with uncropped legs

8.2.6.6 20-pin DIL socket

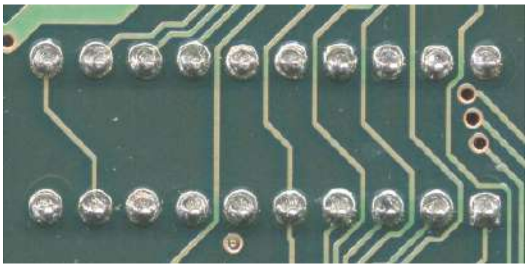


Figure 8.41 20-pin DIL socket

This component was reflowed using a reflow reflector shield. The volume of paste delivered was good, there are well-formed solder fillets and no solder ball contamination. As this component has turned pin legs which are larger on the PCB side it is not a good idea to force the socket down into the solder paste as solder bridges may form.

Sockets with turned pins can effectively squash paste into adjacent deposits. Increased separation between deposits avoids bridging resulting from insertion.

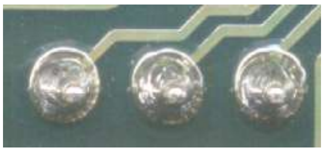
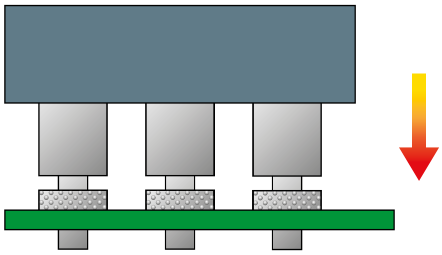


Figure 8.42 Sockets with turned pins



8.2.7 PIHR summary

PIHR is a technique that utilises existing production equipment, personnel and processes and can be employed successfully provided the basic guidelines are followed. The calculation and provision of the appropriate volume of solder paste to deliver the solder fillets required, is by far the most important determining factor to the success of the technique.

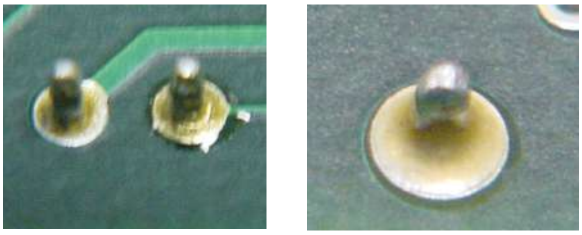


Figure 8.43 PIHR solder fillets seen from the underside of the PCB

Tecan's CAD engineers calculate the appropriate printed paste volume to fulfil the solder fillet requirements and will design the stencil apertures and create any local thickness increases necessary to ensure your process is successful. The use of x-ray equipment, although not essential, will however enable a better understanding of:

- The volume of solder fillet achieved
- Whether the fillet has formed with voids

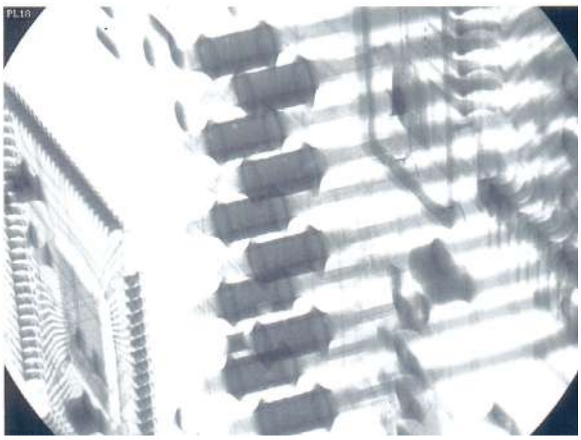
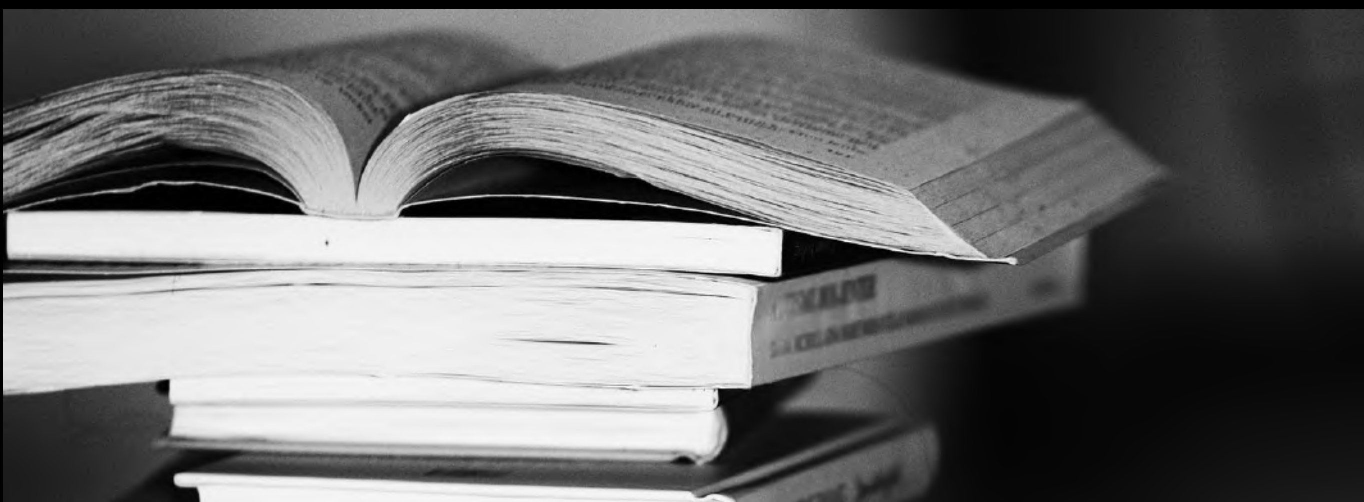


Figure 8.44 X-ray showing good hole fill with minimal voids

9.0 glossary

an explanation of terminology



9.0 Glossary of terms

Active temperature: The ratio of the actual temperature to the melting temperature of solder.

Activator: A chemical added to flux to aid wetting of solder deposits.

Adhesive bond: The physical connections between the mesh and metal stencil or frame.

Angstrom: Unit used to define the wavelength of light, ultra violet energy and x-rays; one angstrom is equivalent to 10-1 nanometres (10-10m).

Anisotropic adhesive: A material with low concentration of conductive (usually silver) particles to permit electrical conduction in the Z axis only.

Aperture listing: List of "D" Codes (definition codes) specifying the shape and size of apertures required. An essential element of electronic data transfer.

Aperture modification: The alteration/reduction of aperture dimensions to accommodate excess solder paste deposit and possible variations in PCB pad dimensions.

- Global modification: Expressed, for example, as either 25 microns or 0.001" per edge reduction or as a percentage reduction across the entire image area.
- Local or Specific Modifications: Alteration to width and/or length of an aperture on a specified device.

BGA: Ball grid array. Integrated circuit package having all its electrical/mechanical terminations on the underside. Main advantages include a larger number of pin outs at coarser pitch than conventional PLCCs on a size for size basis.

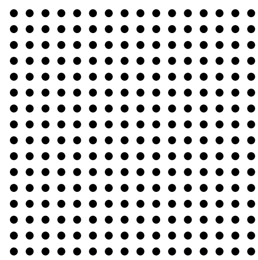


Figure 9.1 Ball grid array

Bi-directional tension (BDT): Where stencil tension is actuated from two axes, similar to that of a meshed stencil, as opposed to the single axis tensioning where the stencil is tensioned across two opposing ends, in the same direction as squeegee travel.

Bridging: The occurrence of solder forming an electrical path inadvertently through two adjacent, normally isolated soldered pads.

CAD: Computer aided design.

Chemical polishing: A chemical process used to smooth the side walls of apertures to assist paste release.

Coining: Impressions created in the stencil after using excessive squeegee pressures or through general misuse.

Coplanarity: A measure of the distance between the highest and lowest pins of a device when placed on a flat plane.

Deposit height: The average height of the printed deposits.

Deposition: The process of applying solder pasted by means of horizontal travel and pressure through a screen or aperture onto a base material.

De-soldering: Removing solder from a joint in order to effect a repair. Methods include wicking, pulse vacuum (solder sucker), localised heat and pull (hot air and laser) and solder extraction.

Differential etching: The double-sided etching process modified to achieve smaller apertures on the PCB side and sloping aperture walls. See trapezoidally-formed apertures).

Figure 9.2 Differential etching

Digitisation: The conversion of feature locations of a printed circuit board design to its digital representation in X-Y coordinates.

DIL: Dual in line – an electronic device having leads in line on two opposing sides.

Durometer: A measure of shore hardness of polyurethane squeegee materials.

Electro chemical polishing: The stencil is used as an anode when it is immersed in an electrolyte solution, electrical current flows removing metal and smoothing the side walls of the apertures to assist paste release.

Epoxy: A polymer thermosetting resin able to form a chemical bond to plastic and metal surfaces.

Etch factor: The ratio of etched depth to the lateral etch, or undercut. The proper definition of etch factor must include the required specification for sidewall condition. .

Eutectic: An alloy having a melting point lower than any of its constituent metals.

Fiducial: A specific mark in the artwork or design, which can be laser marked, or etched into a stencil to mimic corresponding features on the PCB and used by machine vision systems to verify layout orientation and location.

- Image fiducial: Global fiducial marks on a multiple PCB fabrication panel located within the perimeter.
- Local fiducial: A fiducial mark on the PCB designed around specific high accuracy requirements for component placement.

Filmwork: See phototooling.

Fine pitch technology: A surface mount assembly technique with component lead pitch down to 0.500mm (0.020"). See also ultra-fine pitch.

Flux: A material used to promote fusion or joining of metals in soldering, welding or smelting. A wide range of rosin fluxes are available for soldering electrical/electronic components.

Flux activity: The cleaning and wetting of metal surface, during heating, caused by the flux.

Flux residue: Particles of flux materials remaining on the PCB after soldering and cooling.

Flux R: Rosin non-activated.

Flux RA: Rosin activated.

Flux RMA: An organic acid of mildly activated rosin, containing no halogens, dissolved in isopropyl alcohol, which attacks and removes oxides from the metals to be soldered. Used in no clean paste.

Flux RSA: Rosin super activated

Gerber data: Electronic transfer of data possible as tabular information on aperture selection and operating commands based on dimensions in X and Y coordinates. Used in directing a photo plotter or laser cutting machine.

Gull wing device: Electronic device with end terminations resembling the shape of gull's wings.

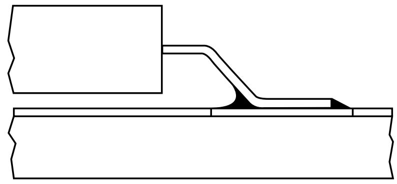


Figure 9.5 Gull wing device

Heel crack: A crack in the solder fillet across the lead bond width in the heel area.

Hydroscopic: A material that is capable of absorbing moisture from the air.

Image location:

- Centred Image: Placement of the PCB and hence the printing pads, in the centre of the stencil.

- Offset image: Many printers have one rail (front or rear) fixed permanently in position with the other rail moving to accommodate PCBs of larger or smaller dimensions. The edge of the PCB to be printed must always be obvious from the stencil data to ensure the image is positioned correctly.

J wing/J leaded device: Electronic device with end terminations in the shape of the letter J.

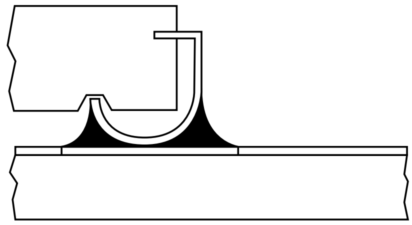


Figure 9.6 J leaded device

Laser-cut stencils: A beam of focused laser light vaporises the metal to produce the apertures. Resultant dross and burrs on the non-squeegee side of the stencil must be removed before the stencil can be used for printing. Fine or ultra-fine pitch stencils may require nanocoating or chemical/ electrochemical polishing.

Laser-formed stencils: a combination of electroformed nickel material with the apertures created using precision laser cutting. Resultant stencils offer good paste release and accuracy.

Laser plotter: A machine used to produce phototools from electronic data input to the highest quality at speed using laser technologies. Features include high resolution and repeatable accuracy.

LCCC: Leadless ceramic chip carriers.

Liquidus: The temperature of a metal or alloy at which it is completely liquid (molten).

MELF: Metal electrical face, describing cylindrical passive components that require special handling for placement.

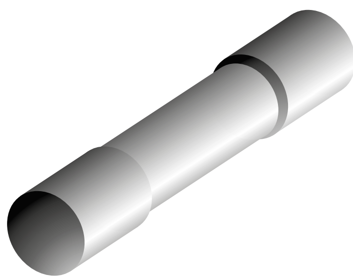


Figure 9.7 MELF

Meshed stencils: A combination of metal stencil element and supporting mesh structure to provide tension. The stencil is suspended from the aluminium printing frame by the mesh (polyester or stainless steel).

Metal content: Defines the ratio of solder powder to the total composition based on weight and expressed in percentage e.g. 90%.

Micron: Also known as micrometer. One thousandth of a millimetre (10-6 metres).

Mixed technology: Component mounting approach that utilises both plated through hole and surface mount on the same assembly.

Multi-level stencils: Stencils that are manufactured by a combination of etching and laser-cutting which can achieve both recessed and prominent surfaces used for multi height deposits of solder paste of glue on the same printing stroke.

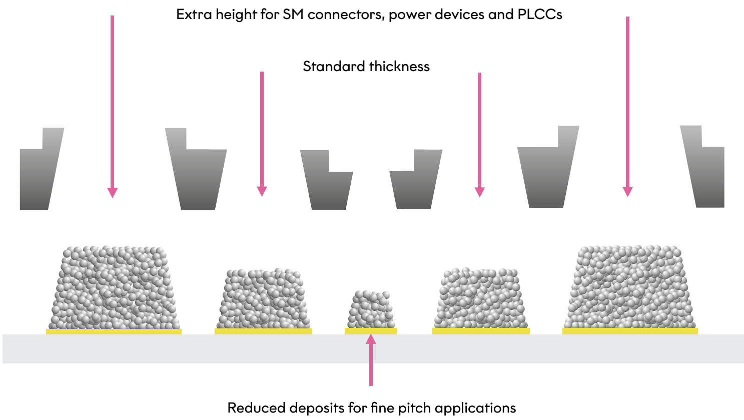


Figure 9.8 Multi-level stencils

Nano coating: With a thickness of no more than 1-100 nanometers, nanocoatings are ultra-thin layers or chemical structures that are applied to surfaces by a variety of methods and applied to a wide range of substrates and chemically bond with non-porous surfaces.

Nitrogen inert soldering: Mass soldering within a nitrogen gas atmosphere to limit the effects that further oxidation could have on the soldering process.

Occluded apertures: In traditional emulsion screens the presence of mesh filaments in screen openings limits the passage of paste material so that fine pitch printing is far more difficult than with the stencil process. Typically, only 50% of pad volume is printed.

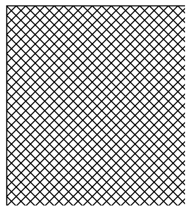


Figure 9.9 Occluded apertures

On contact printing: Used for surface mount stencil printing. Intimate contact is maintained between the non-squeegee side of the stencil and the substrate until print separation.

Pad: The portion of conductive pattern on a printed circuit designated for the mounting or attachment of components.

Pallet: Support tooling used for the transport and holding of thin or flexible PCBs during printing, placement or reflow processes.

Particle size: Definition of the range of diameters used in the manufacture of solder paste powder.

- Type 2: 75μm - 53μm
- Type 3: 53μm - 38μm
- Type 4: 38μm - 20μm

PCB: Printed circuit board. A laminated construction with an insulating base material featuring copper cladding on one or more sides. Materials used:

- FR2 (phenolic resin laminated paper widely used in consumer electronics)
- FR3 (epoxy resin laminated paper for better thermal stability and electrical properties)
- FR4 (increased dimensional stability for consumer, industrial and military applications)
- Polyimide where harsh environments prevail e.g. space/military applications

Photo etching: This process is used for the production of multi-level surface mount stencils. A blank of metal is cleaned and coated with a photo-resist. The coated sheet is then exposed to UV light through the photo master when exposure takes place. The unexposed areas are developed away, removing the resist, leaving the metal bare where etching occurs. Etching solution is sprayed at pressure onto the top and bottom surfaces removing the unwanted metal extremely accurately.

Photo resist: An emulsion applied to a metal blank which is sensitive to portions of the electromagnetic spectrum which when exposed and developed masks portions of the metal substrate with a high degree of integrity.

Photo tooling: A term generically applied to the entire group of photographic products used to produce photo etched stencils.

Photo tool stability: In order to maintain the definition and tolerance of an image a temperature of 20oc and humidity of 50% should be constant. Failure to observe the above may cause the film to shrink or expand with the obvious resultant inaccuracies.

PIHR Pin-in-hole reflow: PIHR involves printing solder paste onto the printed circuit board for both surface mount devices and leaded components. With PIHR it is essential that sufficient paste volume is available to ensure the required solder fillets are formed on the top and bottom sides of the PCB and in the component holes. Following insertion of leaded components and placement of surface mount devices the assembly is reflowed.

Figure 9.10 PIHR Pin-in-hole reflow

Pitch: The measurement in microns or thousandths of an inch between the centres of two adjacent apertures on the same device.

PLCC: Plastic leaded chip carriers.

Print area: The area within the stencil/screen frame available for printing deposits. Most dual squeegee printing machines require a distance of approximately 40mm to be added at each end of the print stroke for the separation of the front and back squeegee blades. On meshed stencils a further encroachment into the print areas is required to secure the stencil to the mesh.

Printhead: The assembly in a printer that houses and controls the squeegee blade(s).

Print speed: The rate at which the squeegee blade moves across the stencil during the print cycle. Typical speeds range between 10 – 150mm/sec (0.4 to 6.0 in/sec). Optimum speed will depend on the pattern to be printed in terms of the finest pitch, image orientation and printing area.

Print separation speed also known as peel-away speed: The rate of PCB separation after printing, expressed in mm/sec ("/sec).

Pumping, also known as bleeding: Excessive squeegee blade pressures force solder paste through the apertures breaking the natural gasket seal between stencil and PCB pads. The resultant contamination of the underside of the stencil requires immediate stencil cleaning to prevent subsequent transference of paste to adjacent gaps between pads.

QFP: Quad flat package.

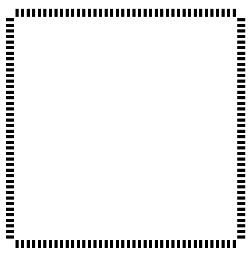


Figure 9.11 Quad flat package

Quick change solutions: Stencils are located into position and tension is achieved without the need to permanently fix the stencil to the frame with either mesh or adhesives.

Reflow soldering: The joining of components to the PCB by placing in solder paste, followed by heating in an oven until the solder fuses and then cooling to form joints.

Residue: Any visual or measurable form of process related contamination.

Rheology: Describes the flow of a liquid, as with solder paste or epoxy, or its viscosity and surface tension properties.

Scavenging/Skimping: caused in the majority of instances by employing polyurethane squeegee blades. The pressure used to ensure the blade seals to the stencil surface often takes the leading edge of blade into apertures thereby removing the top of the expected paste deposits (particularly evident with larger apertures).

Screen printing: Actually a derivative of the ancient art of stenciling. Whereas stencils have no means of retaining isolated centres in place (such as the centres of the number 8), the screen mesh, coated with a photo-patterned emulsion, can accommodate all such features. This process does result in occluded apertures.

Shadowing: When infrared soldering component bodies block energy to adjacent areas, resulting in insufficient solder flow.

Side walls: An aperture should always be considered as a base of XY complete with 4 side walls (2(XZ) and 2(YZ) where Z is the stencil thickness. For superior paste release characteristics at fine pitch the side walls should be as smooth as possible. See Wall base ratio.

Skew: The misalignment of a component from its target position.

Slump: Spreading of the printed deposit (solder paste, surface mount adhesive or conductive adhesive) form its original position, before soldering or curing. Too much slump results in the loss of printed definition and bridging may result.

Snap-back: The return to normal of a stencil after being deflected by the squeegee moving across the surface of stencil and substrate.

Snap-off: The height the stencil is set above the board for 'off-contact' printing, which determines the amount of deflection that occurs during the snap-back action of the stencil. Snap-off distance is typically in the range of 0.6 – 3mm (0.024" – 0.120"). On contact printing would have a zero snap-off distance.

SOIC: Small outline integrated circuit.

Solder: Generally described as fusible alloys with liquidus temperatures below 400oC (750oF). Common alloys include; Sn/Pb, Sn/Ag, Sn/Sb, Sn/In, Sn/Bi, Sn/Pb/Ag, Sn/Pb/Bi, Sn/Pb/In, Pb/In, Pb/Ag and Pb/Sb. As a result of ROHS / WEEE legislation, lead-free alloys have been introduced. The generally accepted replacement alloy is Sn/Ag/Cu. Proportions of each element may vary from brand to brand but liquidus occurs at approximately 240°C.

Solder ball: Small spheres of solder that break away from the designated solder pads but remain on the board after wave or reflow soldering. These are sources of electrical shorts.

Solder balling: sometimes known as mid chip solder beading, a defect which occurs as paste which is initially under the component is squeezed out during final reflow. Most commonly caused by paste being present in excess amounts under the inert component body, which when reflowed is thrown out from under the chip. This problem can be overcome by ensuring paste and profile compatibility.

Solder bump: Solder spheres bonded to contact areas or pads of devices and used for face-down bonding as in BGA, micro-BGA and flip-chip devices.

Solder physical properties: Solder can be made in various physical forms including bars, ingots, wire, powder, perform, sheet, balls and paste. In addition to the elemental compositions and physical forms, the performance of solder materials is generally determined by the specific land pattern to which it is applies.

Solder powder: Small regular graded particles of solder (see above).

Solder mask also known as solder resist: A coating applied over selected parts of a PCB that acts as a mask to prevent solder flow/encroachment in the covered areas.

SOT: Small outline transistor.

Squeegee: A rubber, polyurethane or metal blade that wipes across the stencil to push the solder paste or adhesive through the stencil apertures and onto the PCB below.

Squeegee blade attack angle: The angle formed between the stencil and the squeegee blade during the squeegee travel. Most commonly 60° to the stencil surface but sometimes amended to 45° to improve penetration for PIHR applications.

Squeegee direction: Direction of squeegee or print stroke, often dictates placement of an image on the stencil.

Squeegee pressure: The pressure exerted on the stencil during the print cycle, typically in the range of one to six pounds per squeegee inch. Excessive pressure can cause pumping of paste which requires continual under stencil cleaning.

Squeegee side: Top surface of the stencil – that which is in contact with the squeegee blade.

- Non-squeegee side: Bottom side of the stencil – that which is contact with the PCB to be printed.

Squeegee stroke: The total effective length of squeegee blade travel over the stencil, expressed in mm/inches.

Statistical process control (SPC): The use of statistical techniques to analyse a process or its output to determine any variation from a benchmark and to take appropriate action to restore statistical control where necessary.

Stencil: A thin sheet of material with a printed circuit image formed into/through it. Among the many materials that have been used for stencil printing are: Beryllium copper, brass, molybdenum, nickel silver and polycarbonate. The most common materials in use today are stainless steel, nickel and fine grain steel.

Stencil clogging: The contamination or blocking of stencil apertures by 'dead paste' or solder particles.

Stenciling: Common method for applying solder paste to PCBs for surface mount assembly, this process provides high throughput and good precision as the apertures are completely open and unobstructed.

Stencil thickness: Selection of the correct thickness of the stencil required for a printing application depends upon achieving the necessary component solder volume requirements. The following considerations should be made:

- The finest pitch present
- Adjustment of aperture sizes to suit
- The ability to compromise paste volumes between finer pitch and larger components
- Selection and achievement of multi-height deposits

Step and repeat: panelisation details required to accurately position multiple images on the same PCB/stencil.

Step-down stencil: A stencil of varying thickness (less than the original material thickness) used to accommodate coarse and fine pitch deposits on the same print cycle.

Figure 9.12 Step-down stencil

Step up stencil: for use where a greater volume and height of solder deposit is needed to add volume for designed components -particularly suitable for through hole reflow solder.

Surface insulation resistance: The electrical resistance of an insulation material between a pair of conductors - used to determine the state of cleanliness.

Tack time: The useable life of the solder paste before and after printing. Generally the longer the retention period the greater the time allowed between printing and placement without the risk of component loss occurring during reflow.

Tension: Meshed stencils require supporting fabric tension to be adjusted to approximately 36 – 40 newtons per cm2 before bonding to ensure stencils fixed on the mesh present a flat surface when printing.

Test coupon: A portion of functioning circuitry used exclusively for testing the completed PCB (positioned for easy access).

Thermal cycling: A method to induce accelerated stress on components by heating and cooling in an air circulating oven to verify reliability.

Tombstoning: A soldering defect in which a chip component has pulled into a vertical plane with only one termination soldered.

Ultra-fine pitch printing: Printing where the component pitch is less than 0.500mm (0.020").

Vector printing: Where either the image or the squeegee approach is at 45o. Enables the speed of printing to be optimised whilst still filling the apertures with solder. Improved pad to pad print consistency with QFPs and PLCCs (four-sided devices) where usually the rectangular apertures are both parallel to, and at 90o to the direction of squeegee travel and printed results can be variable.

Viscosity: A measure of the force needed to overcome the resistance to flow.

Vision recognition systems: A CCD camera-based system used within both SMT printing and placement machines to ensure 'closest match' between target positions and printed/placed positions. The image data recorded by CCD cameras is evaluated using digital image processing technology.

Voids: Incomplete volumes of paste sometimes caused by entrapment of air with the progression of paste and squeegee. Paste should roll not skid the stencil surface to ensure adequate/correct filling of apertures.

Wall/base ratio: For the finest apertures, the correct selection of thickness of the stencil is important, as, when the wall area of the aperture increases significantly compared to the pad area, the paste may tend to stick to the walls rather than the PCB pad. The relationship between the surface areas of the base of the aperture to the surface area of the 4 walls dictates the release characteristics of the paste deposit. For the best paste deposits the base should always exceed the 4 wall surface area.

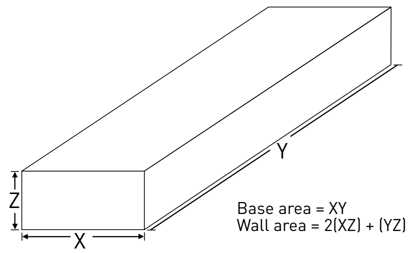


Figure 9.13 Wall/base ratio

- Ideal Ratio: = 1.5 : 1.0
- Acceptable Ratio: = 1.15 : 1.0

Wave soldering: The technique of bringing a PCB in contact with a continuous wave of molten solder to complete a mass formation of joints.

'Wendy house' apertures: A modification to the stencil apertures for discrete components that remove solder paste from the non-wettable faces of the components.

Wettability: The degree to which a metal surface will accept the flow of solder as permitted by its freedom of oxides.