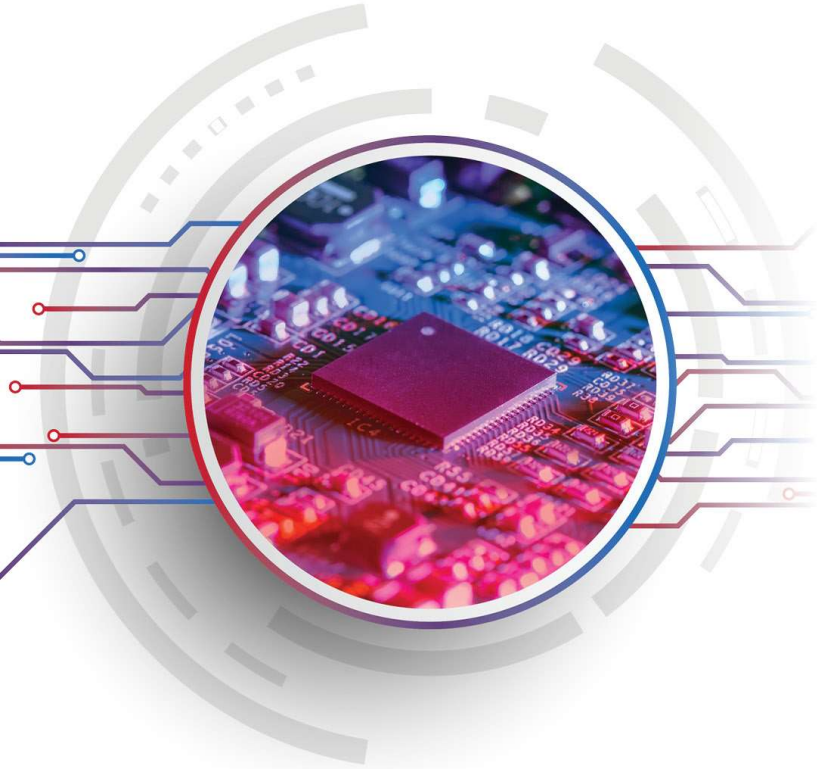


STENCIL BASICS

Module 3

Jan 2023 V2

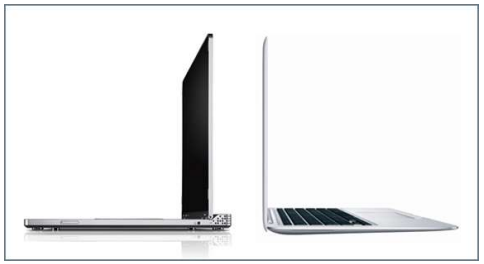
MacDermid Alpha ELECTRONICS SOLUTIONS

Module 3 - Stencil Basics

3.0 Introduction
3.1 What are stencils
3.2 Types of Stencils
3.3 Effect of stencils to process yield
3.3 Apertures & significance of the edge definition
3.4 Manipulation of aperture designs (in Gerber File)
3.5 Stencil Tension
3.6 Steel Squeegee Blades
3.7 How much solder paste must be printed
3.8 How thick are stencils
3.9 What is the stencil aperture ratio
3.10 How is the volume of an aperture calculated
3.11 How much solder paste is enough / how is it measured
3.12 Significance of bottom tooling
3.13 What is stencil misalignment
3.14 When are apertures increased in size
3.15 Describe what a stencil frame is
3.16 Describe universal frames
3.17 Describe what fiducials are
3.18 What do need to know to order stencils
3.19 State when stencils should be replaced
3.20 How should stencils be stored
3.21 Cleaning stencils
3.22 What are the most common problems with stencils

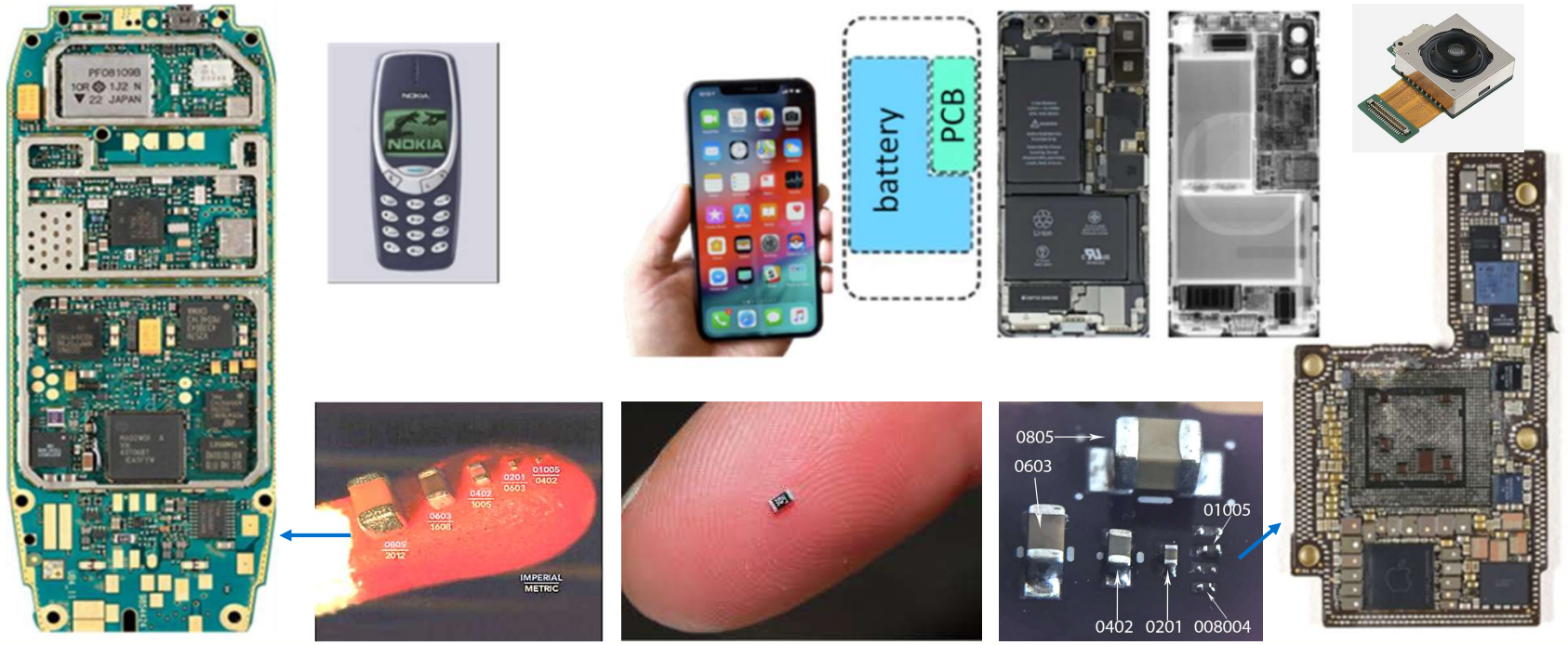
Stencil Basics

Evolution of Devices

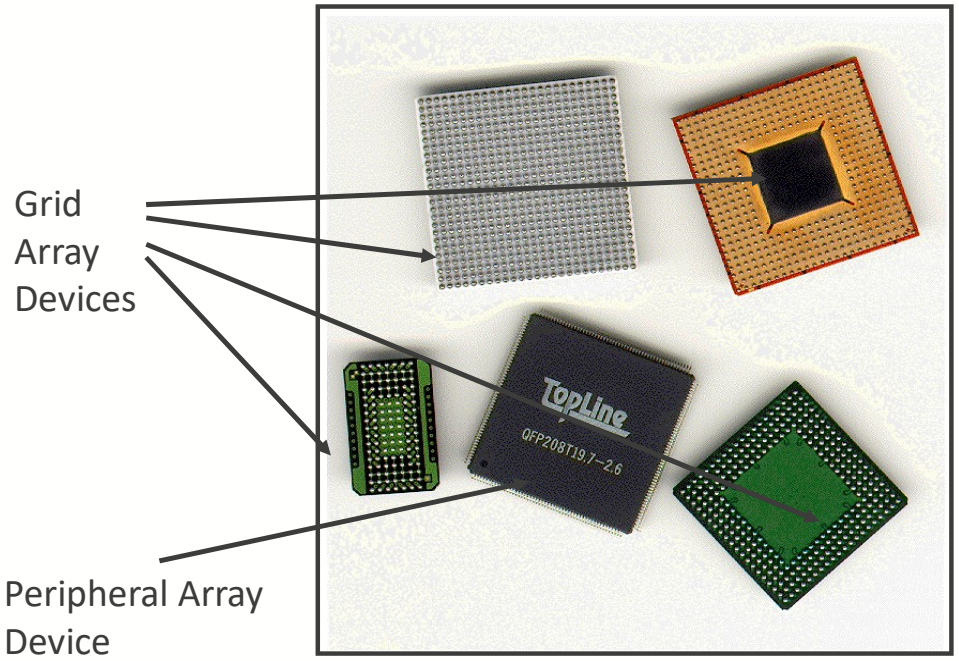


Smaller, Thinner, Lighter & Faster

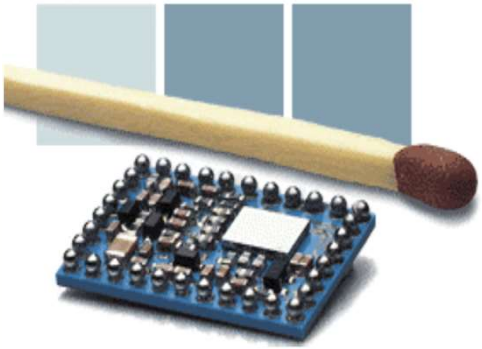
Components: Smaller & Thinner



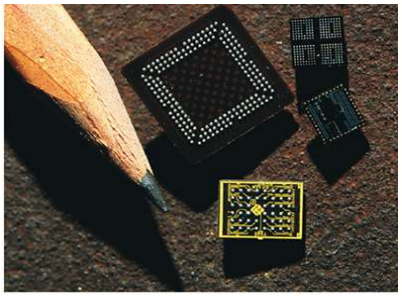
Components: Smaller & Thinner



PLCC's → QFP's → BGA's → CSP's



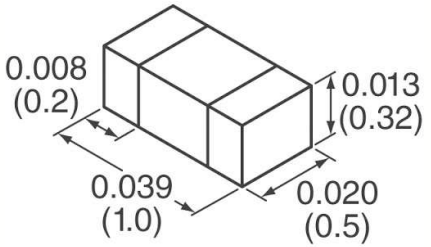
Bluetooth, MCM

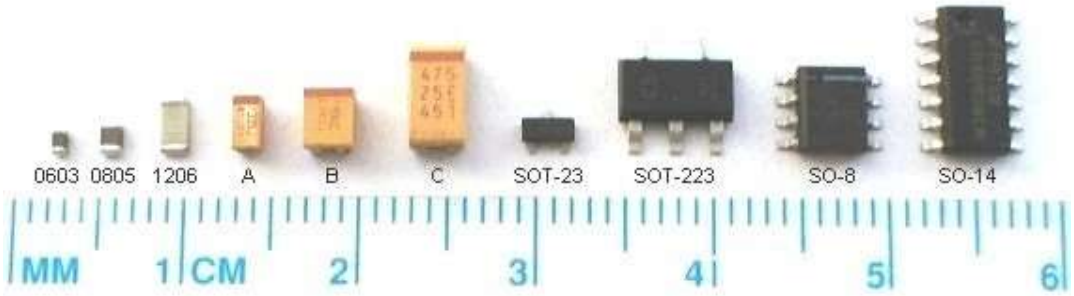


µBGA, CCGA, COB, etc.

Surface Mount Package Sizes

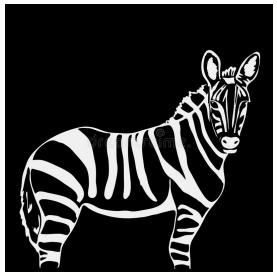
Package type	Size in inches	Size in mm
0201	0.024" × 0.012"	0.6 mm × 0.3 mm
0402	0.04" × 0.02"	1.0 mm × 0.5 mm
0603	0.063" × 0.031"	1.6 mm × 0.8 mm
0805	0.08" × 0.05"	2.0 mm × 1.25 mm
1206	0.126" × 0.063"	3.2 mm × 1.6 mm
1210	0.12" × 0.10"	3.2 mm × 2.6 mm
2020	0.20" × 0.20"	5.08 mm × 5.08 mm
2512	0.25" × 0.12"	6.35 mm × 3.0 mm





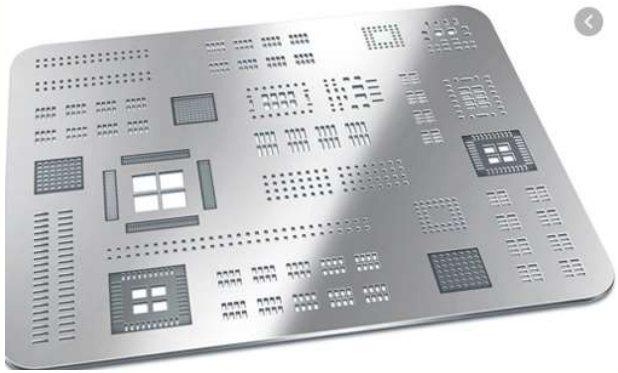
In general,
the stencil is a simple tool to make repetitive impression.

Is being commonly used by every one at least once in life time.



Importance of Stencils

- Devices have become Compact, Thinner, Lighter & Faster
- Assembled with small, thin packages over tightly spaced PCB traces
- Tightly spaced Foot prints (eg: Fine pitch BGAs & QFNs) demand for optimal & precise volume transfer of joining materials onto the PCB pads



- Well designed Laser-cut / Electroformed stencils are used predominantly to transfer solder paste onto the PCBs
- It is important to understand the role play of stencils in the electronics manufacturing

Functions of Stencils

Two Primary Functions in Surface Mount Assembly

1. The stencil is the gateway to an accurate and repeatable solder paste / encapsulant deposition onto a substrate (PCB)

2. As solder paste is printed through the stencil apertures, it forms deposits
   - that hold the components in place &
   - when reflowed, secure them to the substrate (PCB)

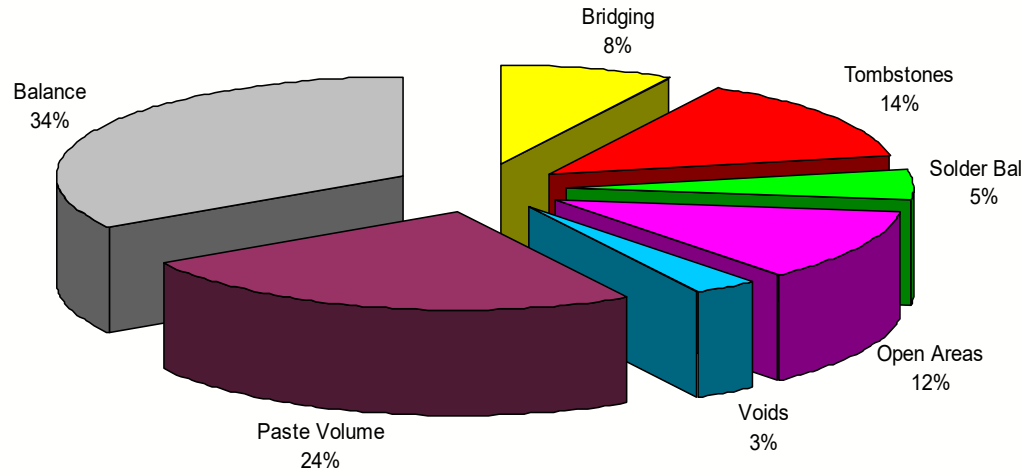
The stencil design
Its composition and thickness, the size and shape of its apertures ultimately determines the size, shape and positioning of the deposits. These are crucial to ensuring a high-yield assembly process .

http://www.qualiecocircuits.co.nz/

Assembly Process Defects – Linked to Stencil

Non Paste printing issues:

34%
- Missing parts
- PCB Issues
- Solder pad issues
- Component issues
- Component placement



66 % of the defects originate from paste printing

Printing the solder paste is the major cause for defects !

- Unpredicted volume and mis-placed paste deposit is the most common factor.
- Insufficient paste due to bad wetting of Lead-free solderpaste in combination with the large range of aperture sizes on mixed board populations.
- Also component placement problems do occur more and more.

Objective of Printing / Amount of Solder Paste

Solder Paste printing process with support of Stencil , expected to produce

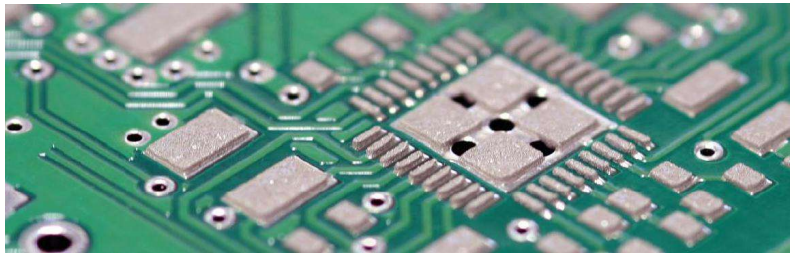
- *Square Edged*
- *Flat Topped*
- *Well Defined*

Solder Deposits (Prints)

Goal of the SMT Printing Process :

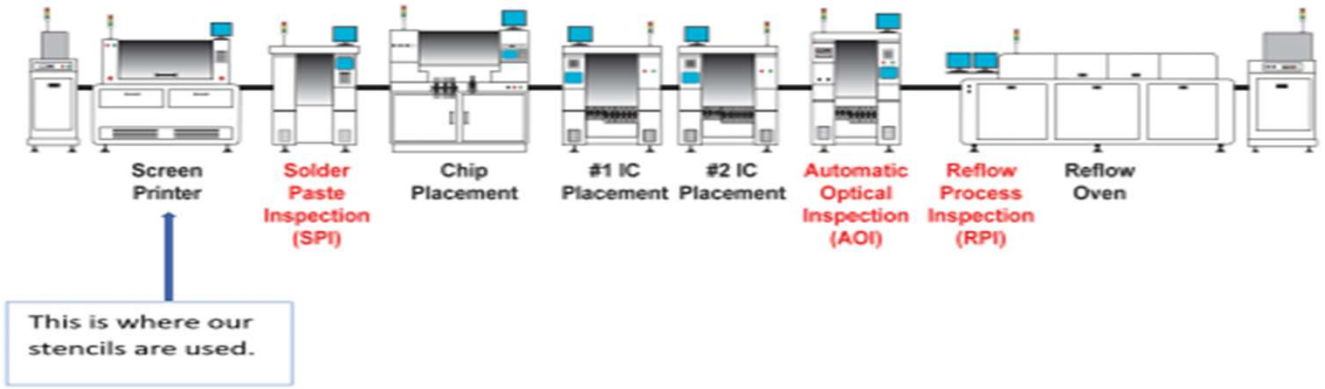
- *No Open Joints*
- *No Solder Bridges*
- *No Variations in Solder Volumes*

After the Component Reflow



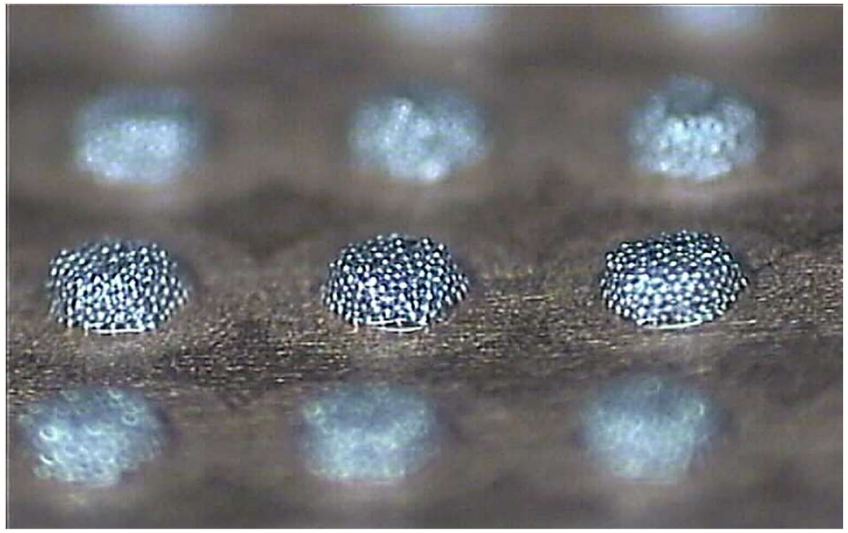
Why is the stencil so important?

- Stencil printing is the first step in the SMT assembly line.
- The stencil is a tool to apply the solder paste that will make every interconnection on the PCB robust
- It is often regarded as the most important element in the whole process.



The purpose of the stencil is to control position and size of the paste deposits on the PCB and can define the **Quality, Productivity, and Profitability** of the SMT line.

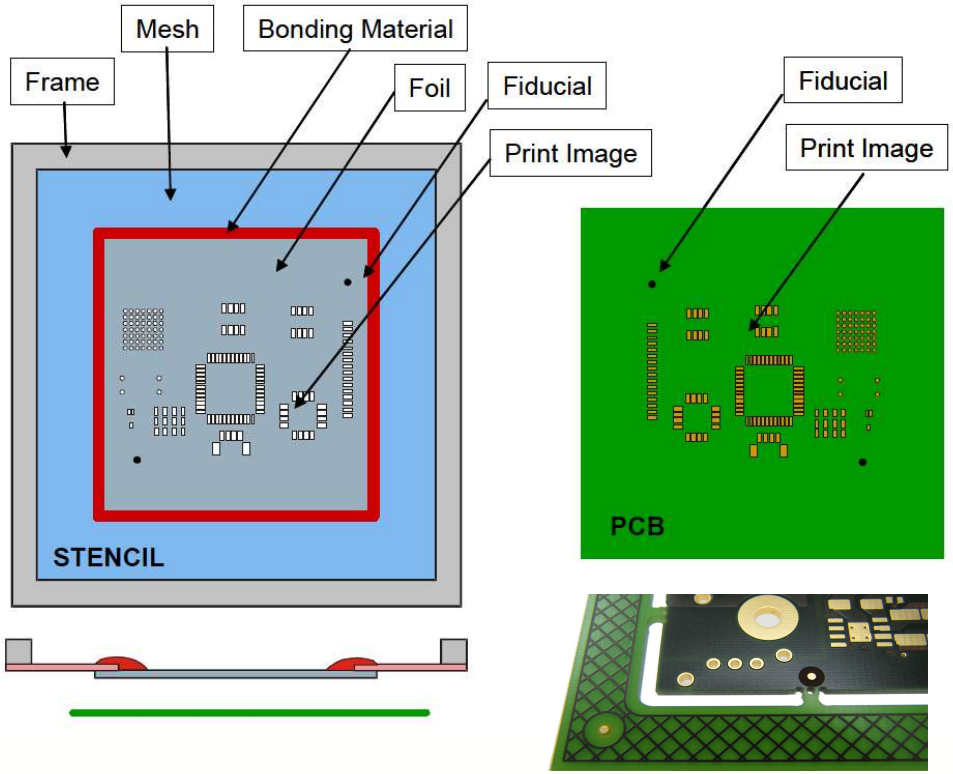
A Good Solder Paste Deposit

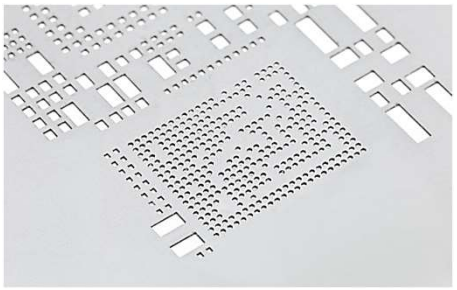


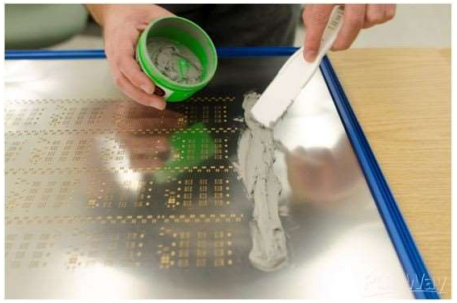
Needs a Consistent, Reliable and Reproducible process resulting in Maximum Yield

Right Solder paste, Stencil, Sgueegee and a good cleaning solvent combination will contribute to it.

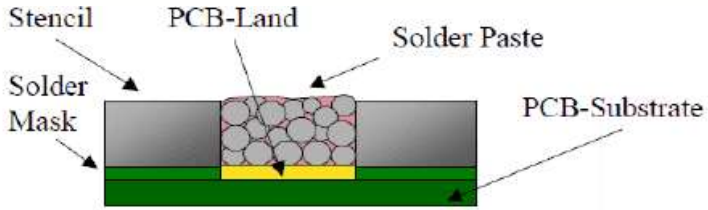
Stencil Terminology



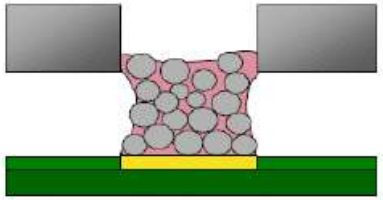




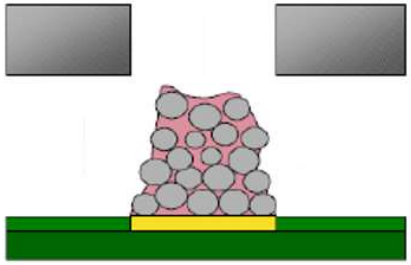
Paste Printing through Stencil



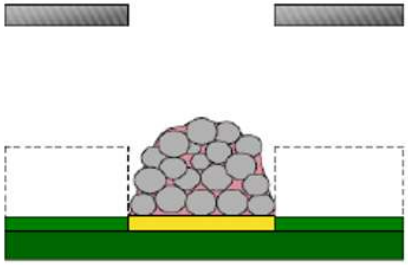
1. Aperture Filling



2. Stencil Separation

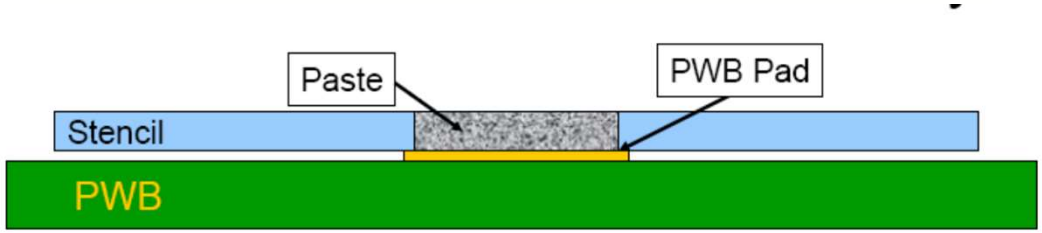


3. Paste Release

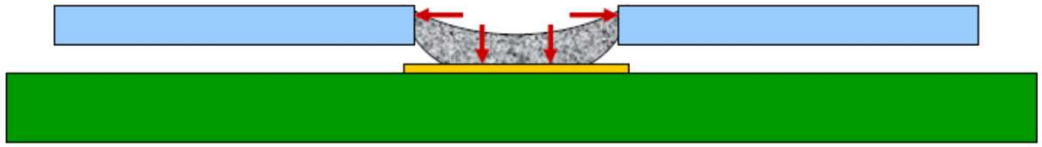


4. Solder Deposit

Fine Feature Printing



After the aperture is filled, the solder paste sets up and sticks to both the stencil walls and the pads.



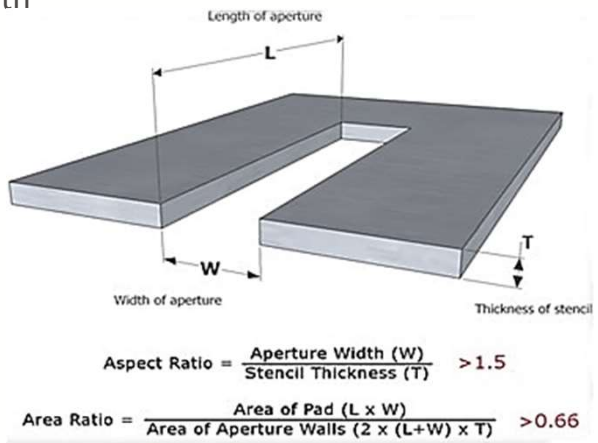
At separation, the forces holding the deposit to the pad must overcome the forces holding the deposit to the stencil walls

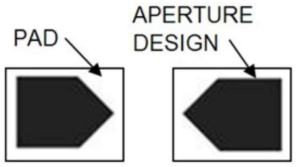
Stencil Technology

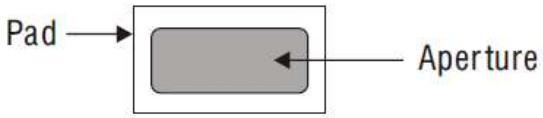
- Stencil technology evolved to Chemically Etched, laser-cut, and electroformed foils as printing requirements dictated performance enhancements.
- In the beginning, there were thick-film screens; however, the wire mesh weaving across the aperture openings impeded paste transfer.
- Therefore Chemically Etched stencils were provided for better open areas and worked well for SMT devices with lead pitches down to 0.8 mm.
- The Chemically Etched trapezoidal aperture, along with enabling post processes of electropolish and nickel plating, helped smooth aperture sidewalls, improving paste transfer.
- Laser stencils were used and they were introduced in mid – 90's, just in time to meet the print requirements of 0.65-mm-pitch SMT devices.
- Special Grade SS Mat'ls + Laser Cut + Electroform + Step UP/DN + Surface Coatings in combination used to deliver optimal transfer of paste onto the substrate as the demand for smaller, lighter, thinner devices drive tighter circuit designs / packages for fine pitch assembly

Stencil Design Aspects

- Design guide for stencil technology was determine by Aspect ratio, defined as aperture width (W) divided by stencil thickness (T) is greater than 1.5.
- With the help of this ratio good paste transfer is per formed.
- Aspect ratio is a good guide if the length of the aperture is greater (at least 5×) than the width.
- But again with the introduction of BGA's and QFN's aspect ratio has certain limitations.
- So new design guide was introduced known as
  Area ratio, defined as the area of the aperture opening divided by the area of aperture walls.
- The walls of the aperture are trying to hold the paste in the aperture, while the pad under the aperture opening tries to pull the paste away.







Stencil Thickness

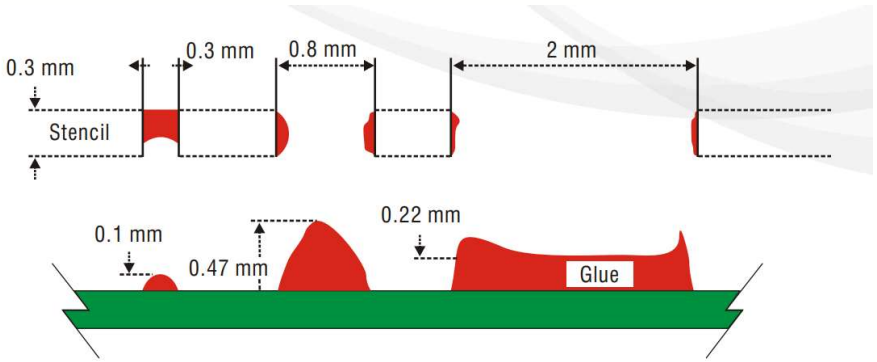
- Optimal paste transfer onto a PCB is impacted by the relationship that exists between ***the pad size, aperture opening and foil thickness.***

- While the aperture may be appropriately sized for a pad, a stencil that is either too thin or too thick may still cause less than optimal deposition of solder paste.

- This relationship is also known as ***"aspect."***
- Aspect is the difference in forces that either pull paste from an aperture and on to a pad or cause paste to be held within an aperture.
- These forces can be quantified and represented as a measurement called the ***Aspect Ratio.***
- In simple terms, for a paste to be adequately deposited on a pad, the paste surface tension must be stronger that the surface tension of the paste to the aperture wall.
- A broad set of rules has been adopted that help us design stencils with appropriate Aspect Ratios depending on the type of stencil ordered.
- It is important that the smallest aperture on the board be used for this calculation.

Stencil Thickness

- Stencil thickness is an important part of stencil design.
- Optimal paste deposition onto a PCB is impacted by
  - the relationship that exists between the pad size,
  - aperture opening and stencil thickness.
- The aperture may be appropriately sized for a pad, but a stencil that is either too thin or too thick may still affect paste deposition/ volume.
- Change in Stencil Thickness , affect Aspect Ratio & Area ratio of stencil which are main ratios for stencil designing.



Importance of ASPECT AND AREA RATIO

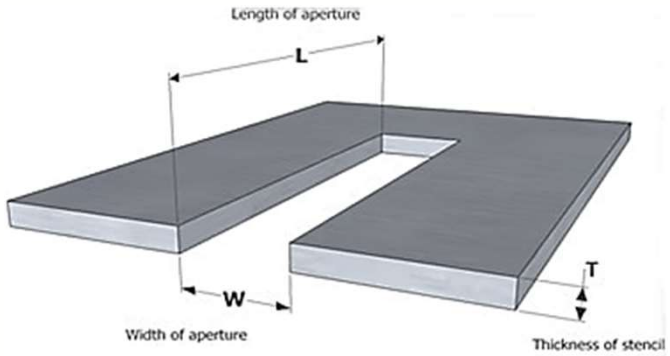
The ability of the paste to release from the inner aperture walls depends primarily on three major factors:

1. Area ratio/aspect ratio for stencil design

2. Aperture side wall geometry

3. Aperture wall smoothness

The first factor is aperture design-related while the other two factors are stencil technology related



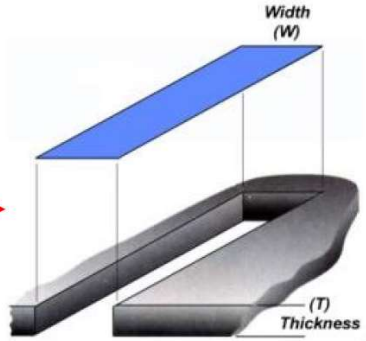
$$\text{Aspect Ratio} = \frac{\text{Aperture Width (W)}}{\text{Stencil Thickness (T)}} > 1.5$$

$$\text{Area Ratio} = \frac{\text{Area of Pad (L x W)}}{\text{Area of Aperture Walls (2 x (L+W) x T)}} > 0.66$$

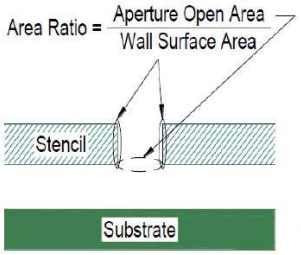
Aspect & Area Ratios

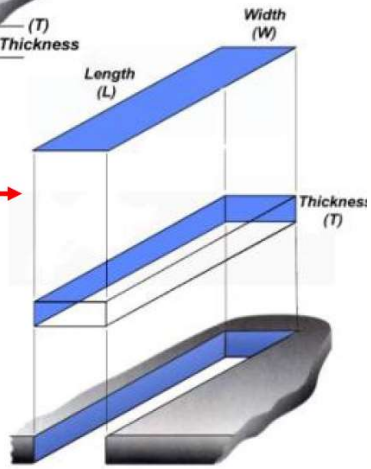
Empirical relationships between aperture size and thickness

- **Aspect Ratio**
  Width / Thickness (W/T)
  should be > 1.5



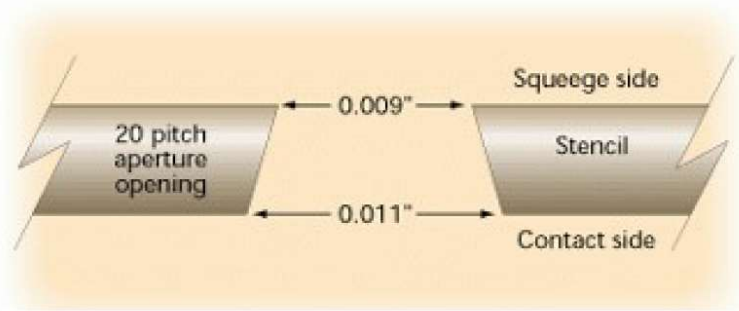
- **Area Ratio**
  (LW) / (2T(L+W))
  should be >0.66

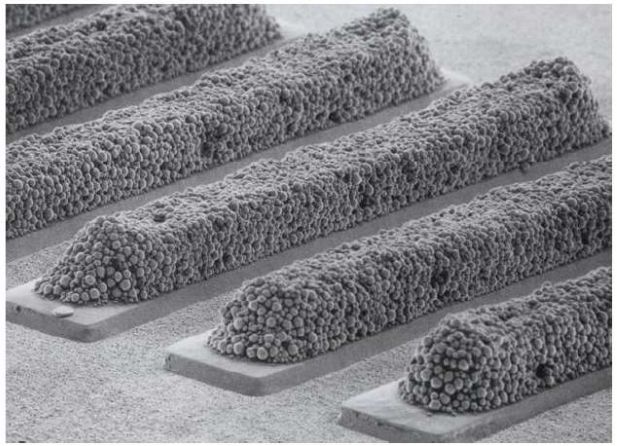




Trapezoidal Shaped Apertures

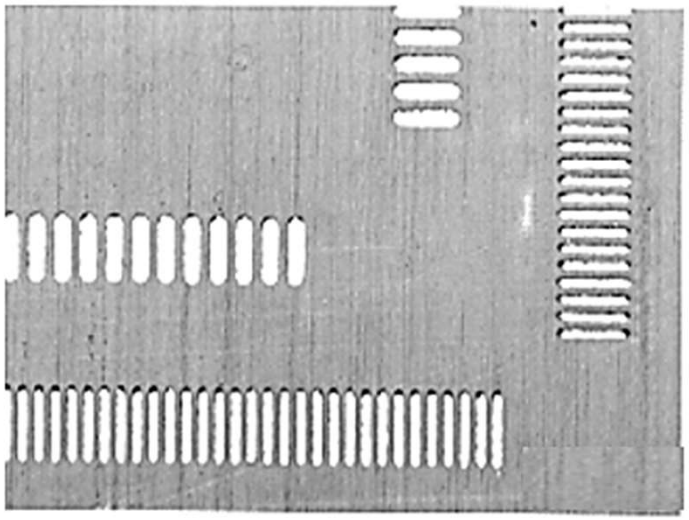
- Aperture openings which are 0.010 to 0.040 mm larger on the contact side (PCB side) than on the squeegee side
- Acceptable for components of 0.5 mm pitch and below
- Trapezoidal apertures, which enhances effective solder paste release, also form a “brick-like” deposit that assists firm component placement



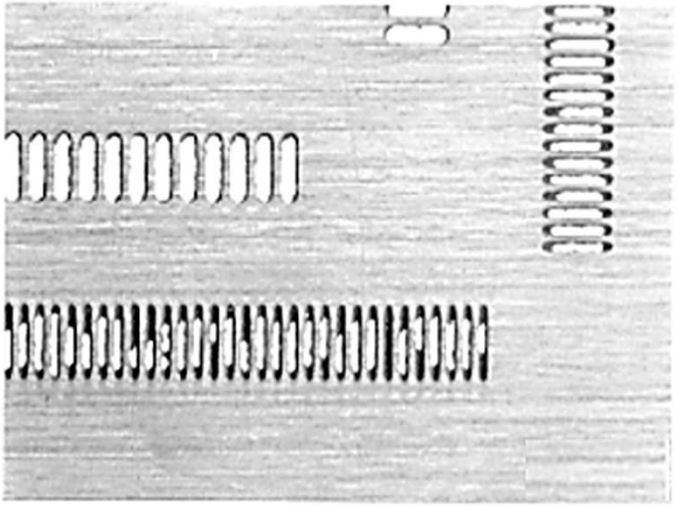


Effect of well designed Stencil Ratios

Effect of Area Ratio > 0.66 & Aspect Ration > 0.15



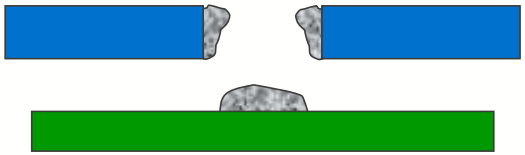
Effect of Area Ratio < 0.66 & Aspect Ration < 0.15



Effects of Well-Designed Area Ratio

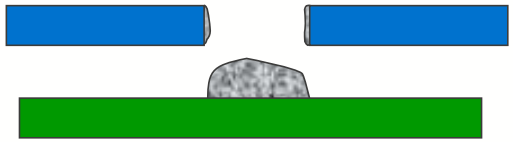
10 mil circle in 5 mil foil

AR = 0.5
TE = 22.3%
Aperture Volume = 393 mil^3
Deposit Volume = 88 mil^3



10 mil circle in 4 mil foil

AR = 0.63
TE = 49.4%
Aperture Volume = 314 mil^3
Deposit Volume = 155 mil^3



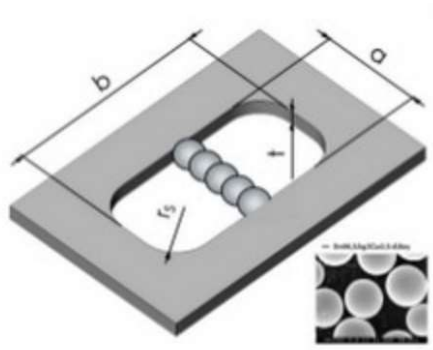
Reducing the stencil thickness by 20%

<u>increases</u> the deposit volume by 76%!

Paste Release Vs Particle Size

The release of solder paste from the apertures of the stencil is also affected by the particle size of the selected solder paste.

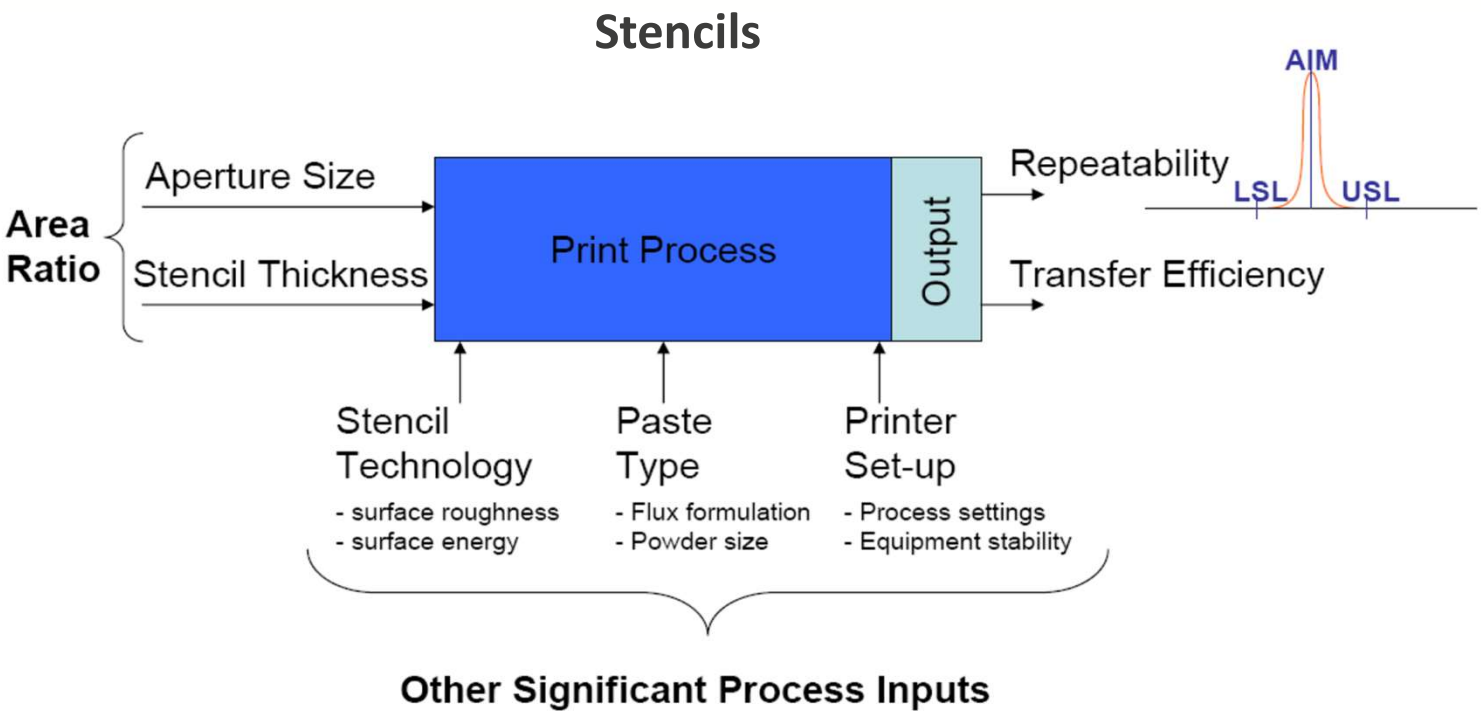
Particle size in microns	Particle type
75-45	2
45-25	3
38-20	4
25-15	5
15-5	6



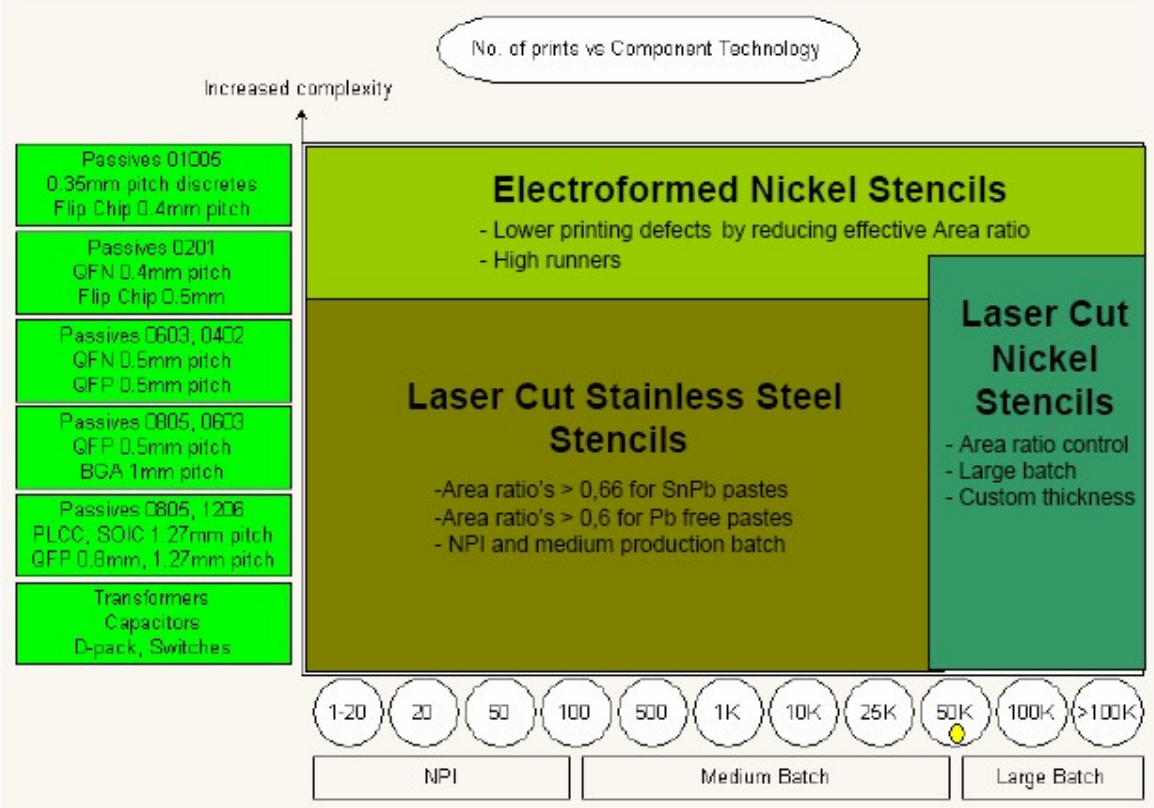
5 Solder Ball Rule

There is a rule of thumb which says ideally a minimum of 5 solder particles should span the width of the smallest aperture.

Fine Feature Printing



Technology selector



Controllable Elements

On Design:

- Aperture Design
- Aperture Reduction
- Component PAD Geometric

On the Stencil

- De-Burring Process
- Stencil Material
- Laser Cutting Parameters

On the Performance of the Stencil

- DfM Analysis Skill
- Manufacturing Experience

Types of Stencils

SMT stencils are made of robust engineered materials and can be manufactured in different ways.
The aperture can be formed on the stencil in any of the following ways:

Electroforming:

- The nickel is electroformed in this process to produce a stencil.
- It is also referred to as EFAB. There are high initial costs involved in the process.
- The process also requires high processing times than other manufacturing methods.

Laser Cutting:

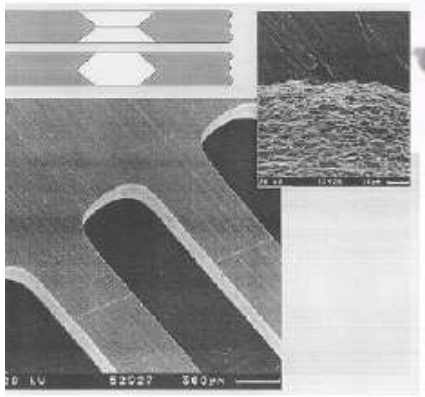
- The openings / apertures on the Stencil are created on a stainless steel sheet using a laser beam.
- This stencil provides a high degree of precision.
- Laser cutting produces fine and accurate results, which is why it is favoured over the other processes mentioned here.

Chemical Etching:

- As the name suggests, an aperture is created through chemical etching.
- The costs involved in the process are low;
- However, the results are much inferior to laser cutting process.

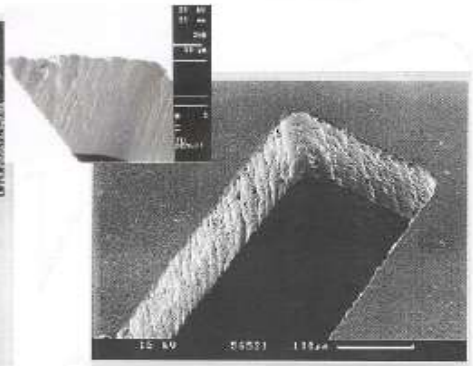
Types of Stencils

Chemically Etched



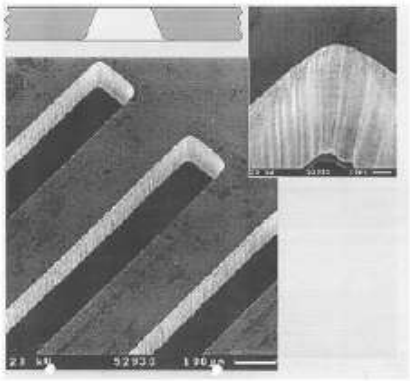
- Lowest cost
- Not suited for fine pitch applications

Laser



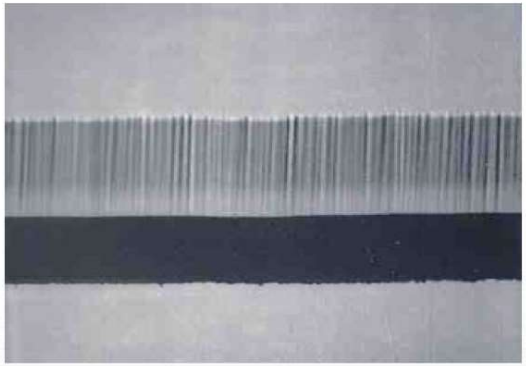
- Very high processing accuracy (dim. Stability)
- Suited for fine pitch applications
- Rough aperture walls
- Cost depending of number of apertures

Electroformed



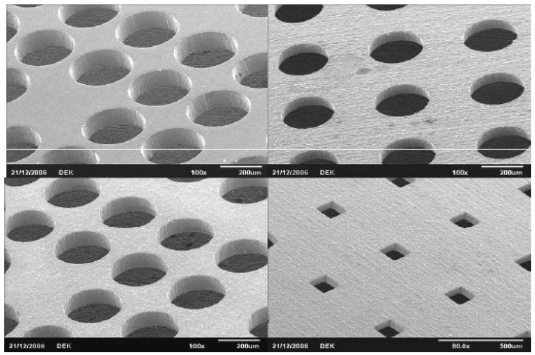
- Very smooth aperture side walls
- Suited for ultra fine pitch applications
- High processing precision
- High initial cost

Electroform vs Laser

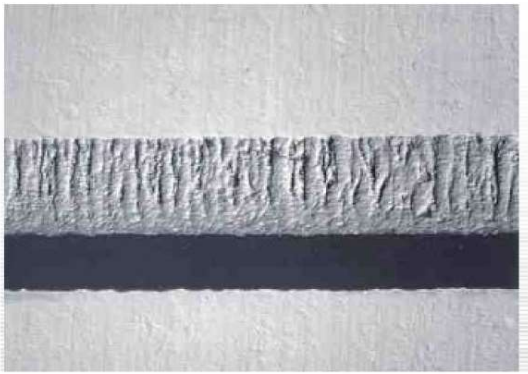


Electroform

E-Form Stencil SEM Image



Laser



Undergo Deburring Process

Technology Comparison

Technology	Accurate apertures	High number of holes	Long production runs	Fast delivery	Paste yield	Price/ performance
Etched	-	+++	+	++	+	+
Laser*	+++	+++	+++	+++	+++	+++
Electroformed	+++	+++	+++	+	+++	+++

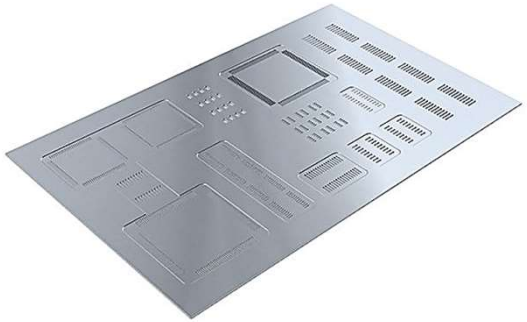
Framed & Frameless Stencils

Framed Stencils

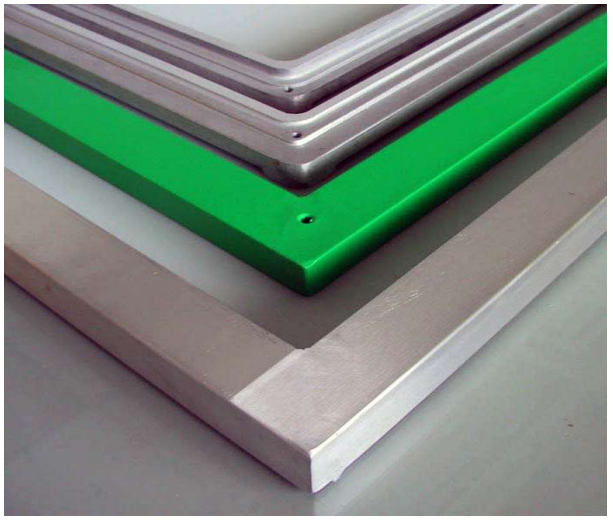
These laser-cut solder paste stencils are permanently mounted on the frame where the border mesh provides the necessary tension to the stencil foil. These are designed for mass production.

Frameless Stencils

These stencils are not required to be permanently mounted on the outer frame. These low-cost stencils are recommended for prototype PCB assembly or short runs.



Stencil Frames



Stencil Frames

- Different types of frames such as: light aluminium, cast aluminium, and stainless steel are available.
- Cast aluminium is used exclusively when a very high-mesh tension is required and high tensile strengths are necessary.
- The screen or stencil printing machine usually determines the frame size requirement.
- Framed stencils possess high durability, stable handling and high tensile strength in comparison to tensioning systems.

Standard Aluminium

Advantages
- stable handling
- longevity and durability
- high tensile strength
- low net weight

Cast Aluminium

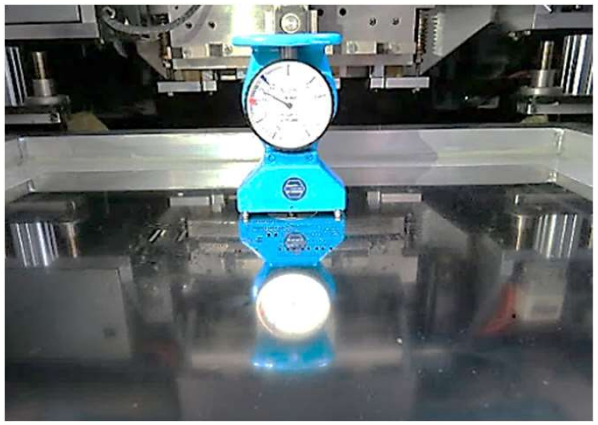
Advantages
- high stability
- high consistency
- high mesh tension

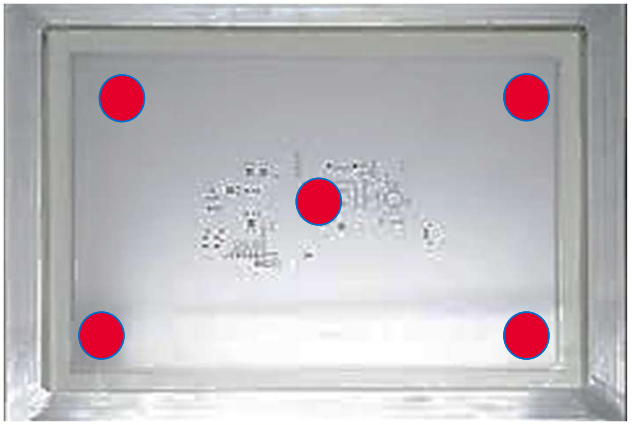
Stainless Steel

Advantages
- higher stability
- higher mesh tension possible
- fast cleaning

Stencil Tension

- Print performance with the mesh-mounted stencil degrades as print cycles increase and tension decreases,
- Mechanically tensioned stencils show much better results.
- Tension consistency from print to print is critical, particularly for miniaturized, high-density assemblies.
- A loss of tension will likely result in poor print outcomes.

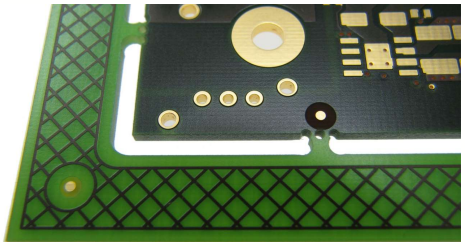


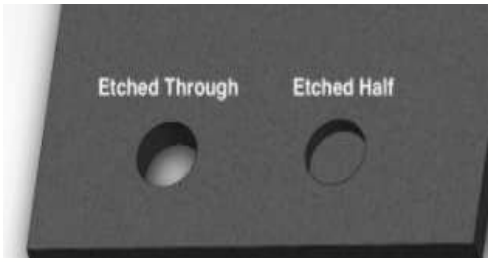


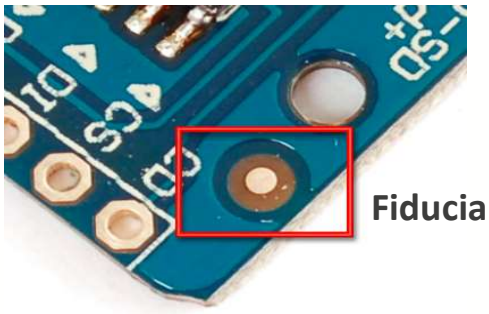
Tension Measurement Locations

Fiducials – PCB & Stencil Accurate Alignment

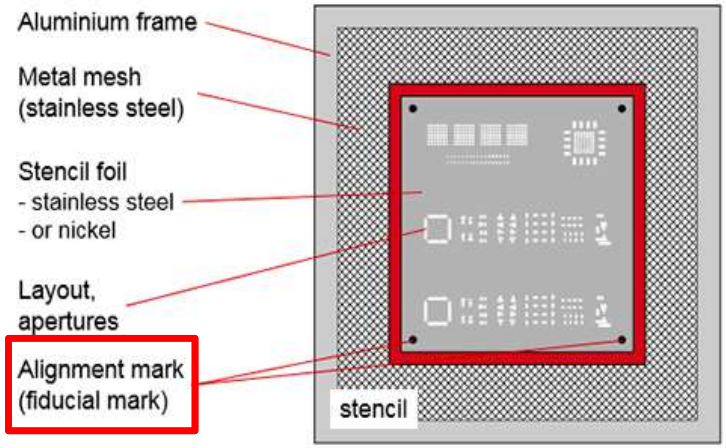
- Fiducials are alignment marks consisting of a 1mm circle surrounded by a 2mm clearance.
- Screen printers are equipped with a vision system,
- Fiducials half cut (into the thickness) of the stencil will allow for the printer to automatically align the board with the stencil.







Fiducial



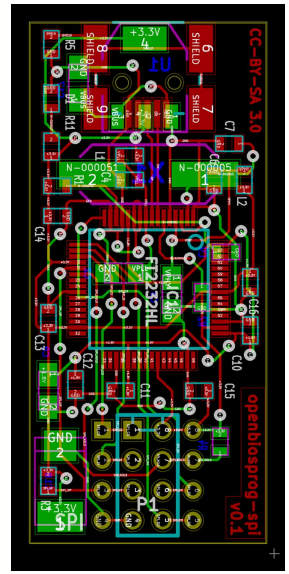
Stencil / PCB Bottom Support

- An important factor to ensure the PCB is held flat against the stencil during the printing process.
- If the PCB is not fully supported it can lead to printing defects such as a poor paste deposit and smudging.
- PCB supports are generally supplied with printing machines
- They have fixed height and have programmable positions to ensure a consistent process.
- Adaptable PCB supports available of varying designs
- They mould themselves to the PCB and are useful for double sided assemblies.

Stencil / PCB Bottom Support

Gerber: Input Require to make stencil

- A “ Gerber file format” is a de facto standard for PCB-designed data storage or transfer.
- Gerber file format is an open ASCII vector format for representing 2D binary images.
- Describes and communicates the constituents of a PCB image like
  - the number of copper layers, NC Drill files,
  - solder masks layers, Stencils layout, and many other such attributes.
- Gerber files also act as input files to PCB printing devices like photo-plotters and Automated Optical Inspection (AOI) machines to print or compare circuit board images.



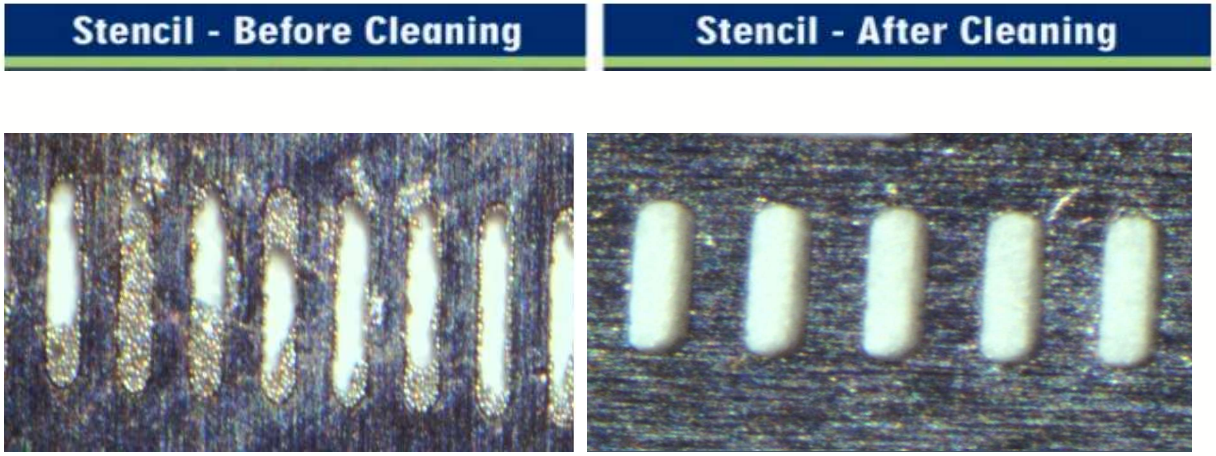
Stencil Storage

Stencils and screens are a large investment; storing them properly will protect their value and prolong the usable life while providing organized, easy access.

Specially designed storage racks are available in market or end user can also order customized design as per there requirement.

Stencil Cleaning

- The stencil must be cleaned regularly during use which can be done either manually or automatically.
- Many of the automatic printing machines have a system that can be set to clean the stencil after a fixed number of prints using lint-free material.
- The system performs two functions, the first being the cleaning of the underside of the stencil to stop smudging, and the second is the cleaning of the apertures using vacuum to stop blockages.



Stencil Cleaning

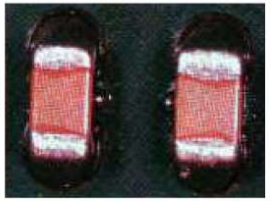
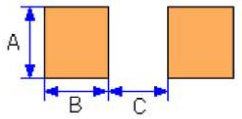
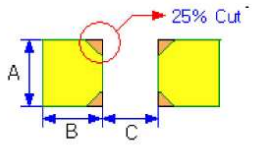
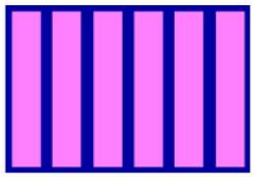
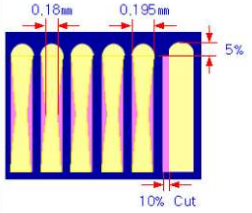
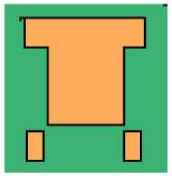
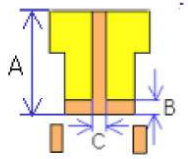
Manual Cleaning

Recommended for Low throughput

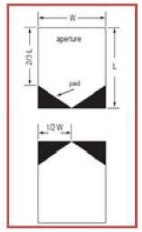
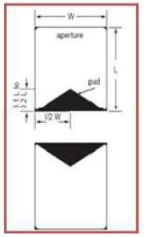
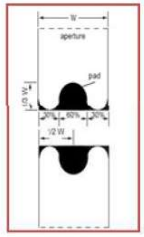
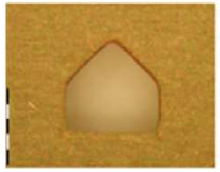
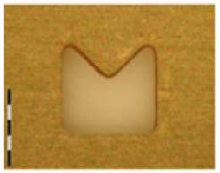
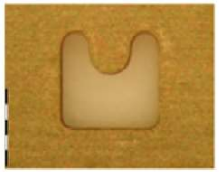
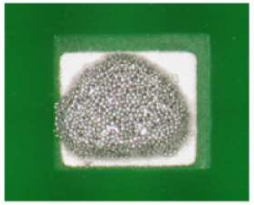
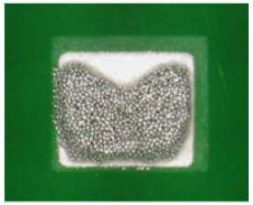
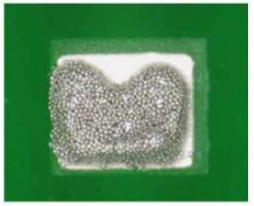
Auto Cleaning

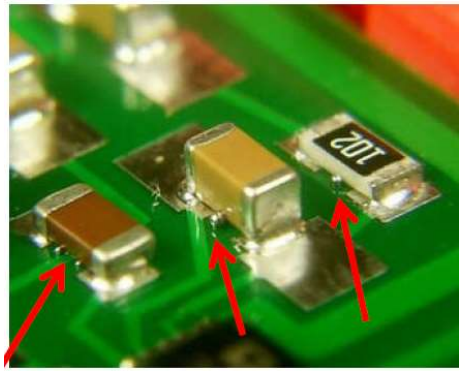
Recommended for High throughput

DFM on Aperture Designs (Pre Eng)

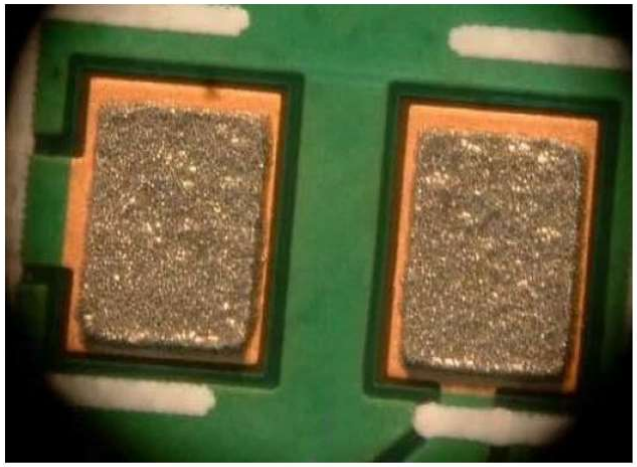
Defect	Photo	Stencil Design	After DfM Design	Result
Mid Chip Solder Balling				-R/C Chip -Medial 25% Cut -Solder Ball prevention
Solder Bridging at QFN Pads				--Centre PAD Cut : Hourglass shape application - → Inside 0.18㎜, outside 0.195㎜ -Dummy PAD : Medial 10% Cut Hourglass shape DNA -Reverence (the way) 5% extension -A Round (Oblong) outside application
Slitting of Heat Sink area				A : CAD PAD width B : 1/4 Cutting work C : Mask reinforcing band width (1.0 ㎜) D:Outside : 0.3㎜ extension

Pad Designs for Passive Components

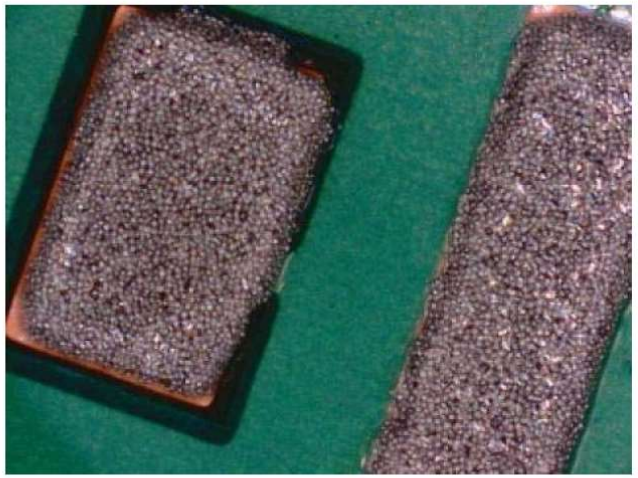
Design	Home Plate	Inverse Home Plate	Rounded Inverse Home Plate
CAD Data			
Stencil Output			
After Print			



Leaded Vs Lead-Free



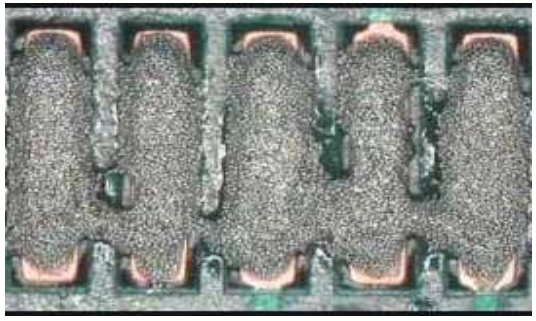
Normal Paste = Stencil Aperture Reduction

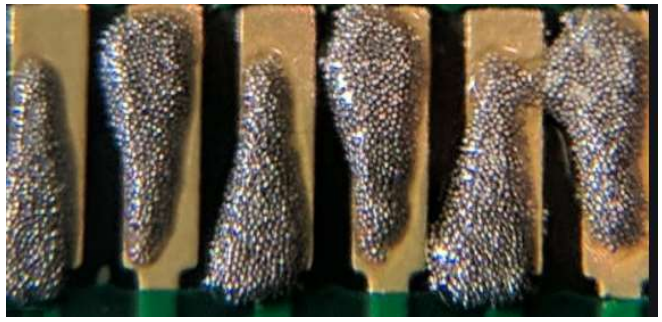


Lead Free = Full Pad Printing

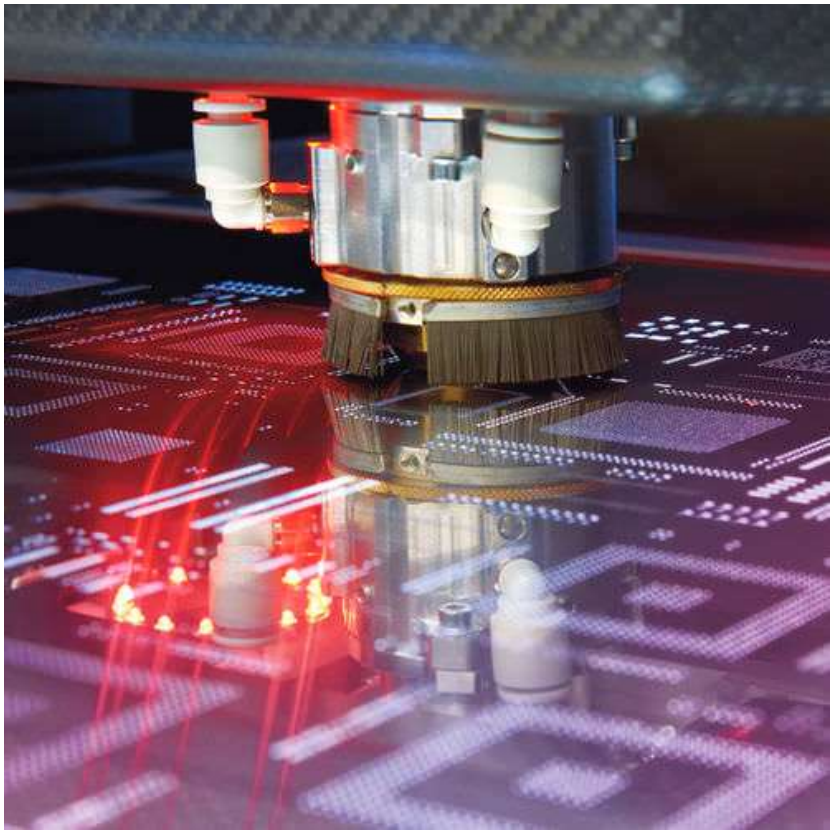
Common Problems with Stencil

Stencil Parameter	Problem
Thickness	A stencil that is too thick will cause a solder bridge short.
	A stencil that is too thin will cause an insufficient solder to be applied.
Aperture Size	When the stencil aperature size is too big, a solder bridge short can occur.
	When the stencil aperature size is too small, and insufficient solder paste will be applied.
Aperture Shape	It is best to use a circular-shaped stencil aperture design. Its size should be slightly smaller than the PCB pad size, preventing a bridging defect during reflow.





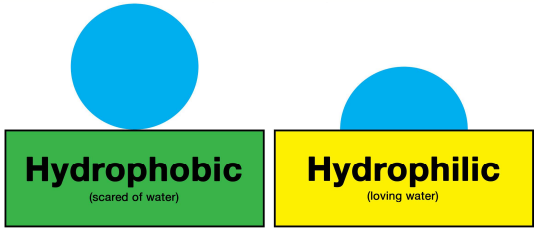
Laser Cutting Machine




Nano Stencils

Nano Coatings

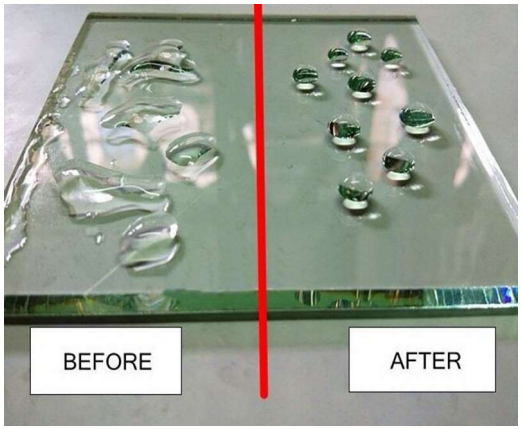
- Nano coatings are ultra thin layers or chemical structures
- Thickness of such coatings is in the range of 1-100 nanometers and few 1-2 Microns
- Applied to surfaces by a variety of methods and applied to a wide range of substrates
- They chemically bond with non-porous surfaces.
- E.g.: Coating thickness of paint used in the auto industry is about 125 microns or 125K NM
- These surface can be converted as water-resistance (hydrophobic) and oil-resistance (oleophobic) as well as Fluxophobic.
- Purpose of nano-coating is to cause the solder paste to de-wet and to release from the stencil

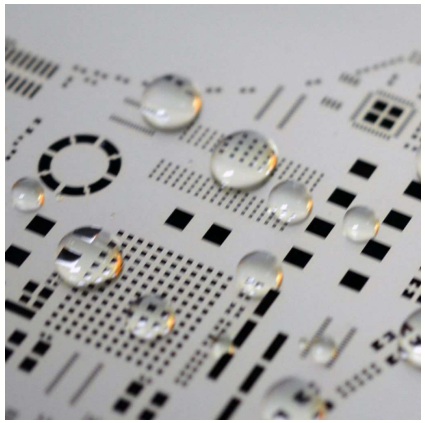


Water-Break test – a test for degree of wettability

The water-break test is a simple method to **check that a metal surface is clean**. A few drops of distilled water are placed on the surface, or alternatively the sample can be drawn from a bath of water. If the water does not break into droplets, then the surface is free from contamination.

Natural Hydrophobic Surfaces

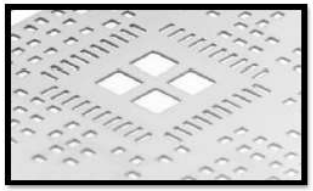




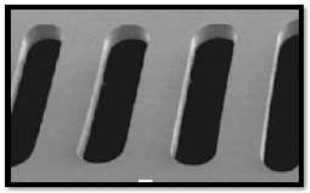
Ref: Tecan nanocoating

Performance Measurement

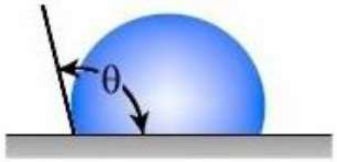
- **SURFACE FUNCTION**
  - **Contact angle**
  - Underside cleaning
  - Bridging

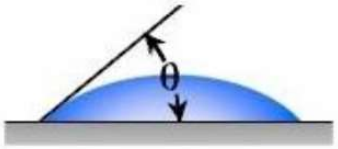


- **APERTURE FUNCTION**
  - Solder paste release
  - Transfer efficiency



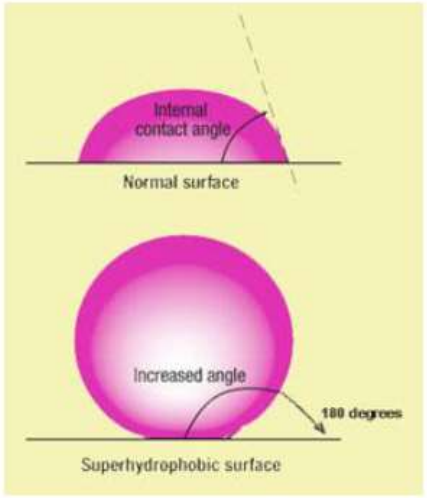
Hydrophobic Surface		Hydrophilic Surface
High	**Contact Angle**	Low
Poor	**Adhesiveness**	Good
Poor	**Wettability**	Good
Low	**Surface Energy**	High



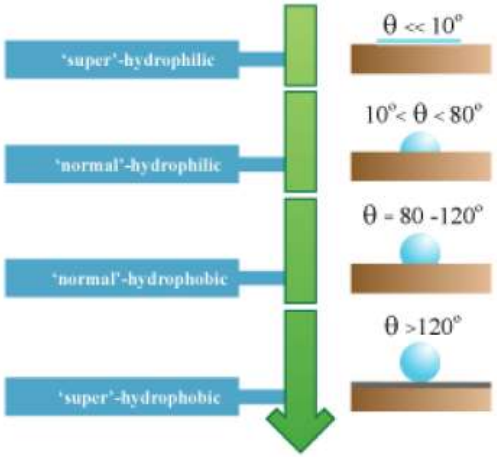


Ref: Tecan nanocoating

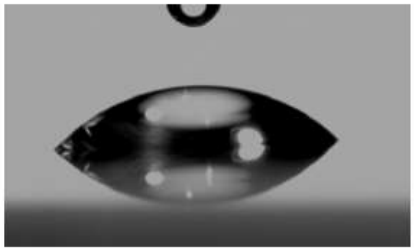
Contact Angle Examples & Images



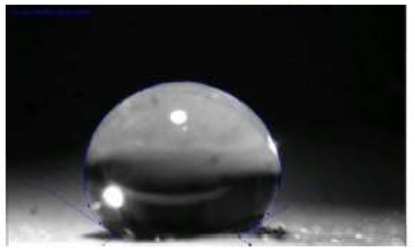
#1 - Contact Angle Examples



#2 - Contact Angle Characteristics



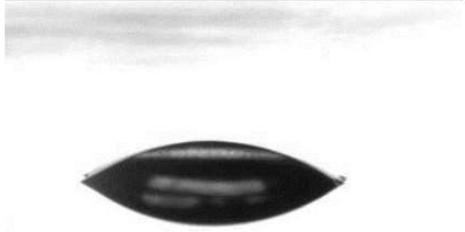
#3 - Water on Clean Glass



#4 Water on Lotus Leaf 146°

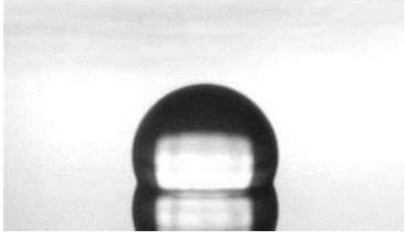
Contact Angle of Nano Coated Surface

Uncoated stencil



Contact angle: 48,1°
Surface energy: 53,3 mN/m

Nano-coated stencil

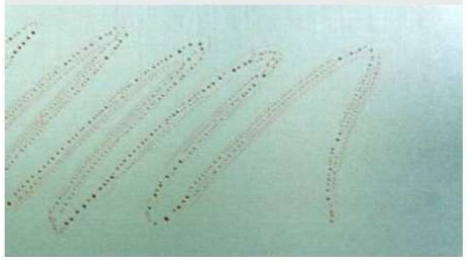


Contact angle: 110,2°
Surface energy: 15,2 mN/m

Uncoated stencil

Good wetting on the stencil surface

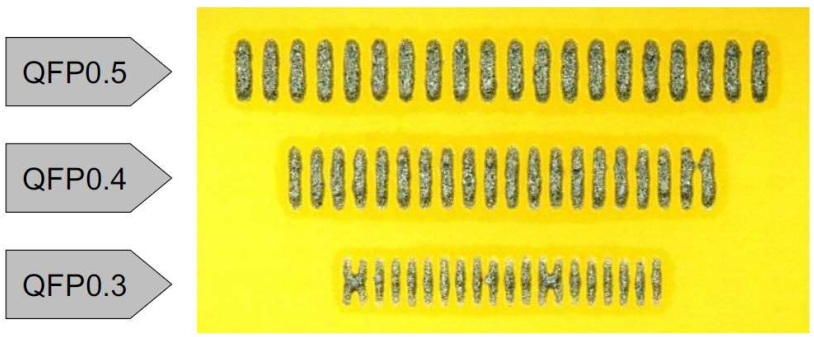
Nano-coated stencil



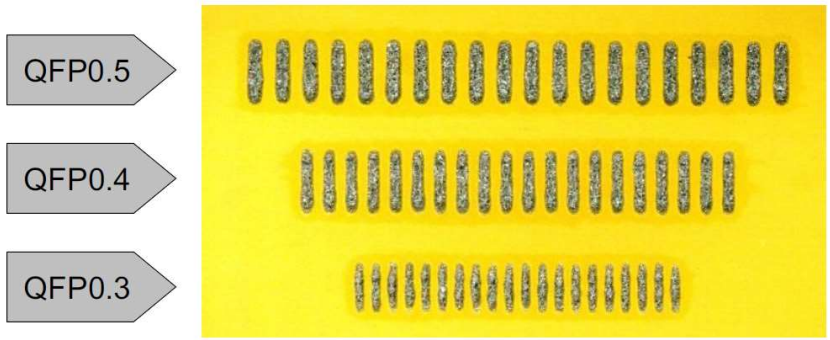
No wetting on the stencil surface

Print Results Regular Vs Nano Coated

Print Results of QFP after the 5th Print
Using an Uncoated Stencil (Area 1)



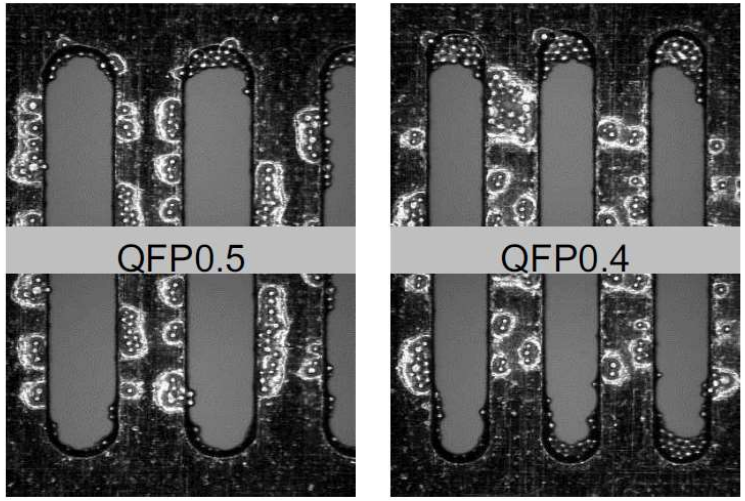
Print Results of QFP after the 5th Print
Using a Nano-Coated Stencil (Area 1)



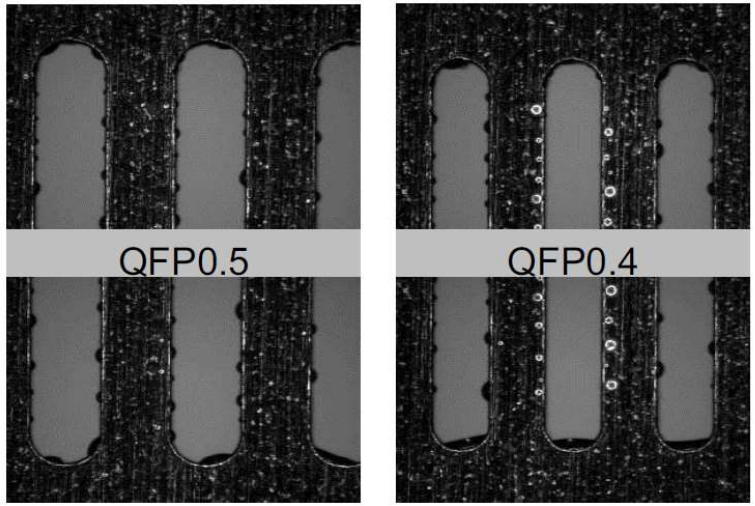
LaserJob Inc.
Eric Moen

B/side Stencil - Regular Vs Nano Coated

PCB Side of the Uncoated Stencil after the 5th Print – Detail (Area 1)

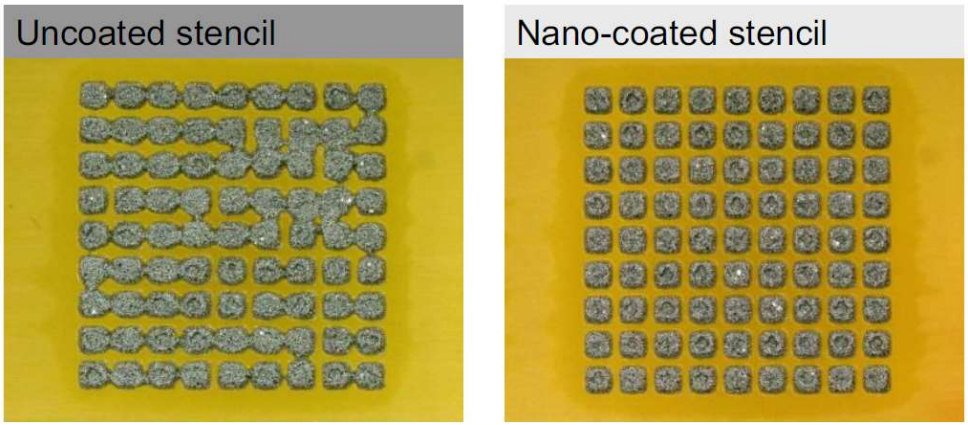


PCB Side of the Nano-Coated Stencil after the 5th Print – Detail (Area 1)



LaserJob Inc.
Eric Moen

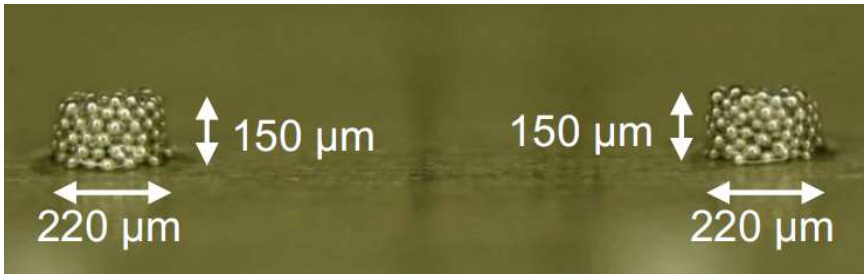
Deposits Printed with a Nano-Coated Stencil



(pitch: 500 µm) after the 5th print (Area 3)

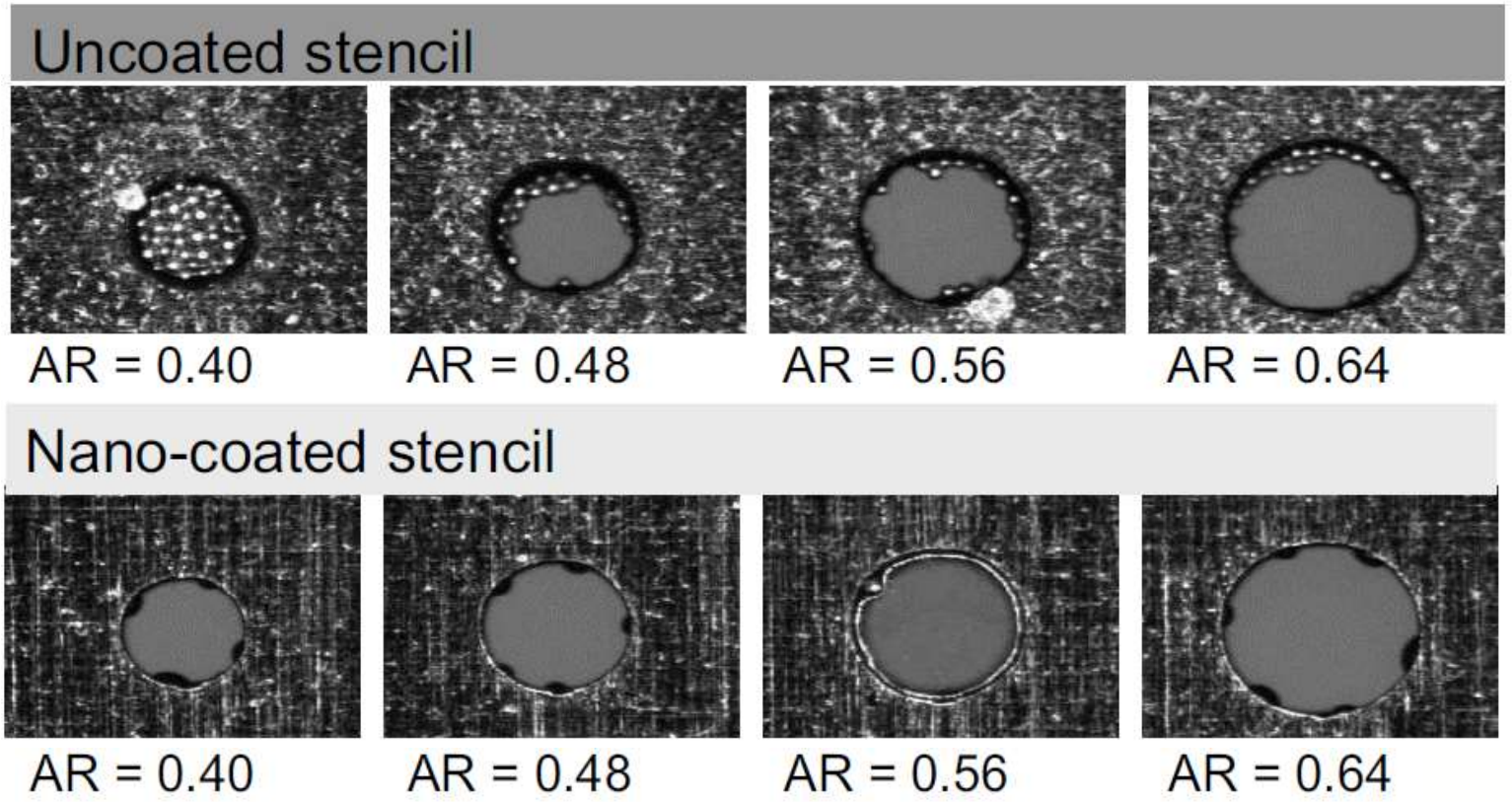
Print Results of 400 µm Squares

Solder paste : Type 4
Stencil thickness : 150 µm
Area ratio of 0.36



LaserJob Inc.
Eric Moen

Optical Inspection of the Stencil Apertures After the Print

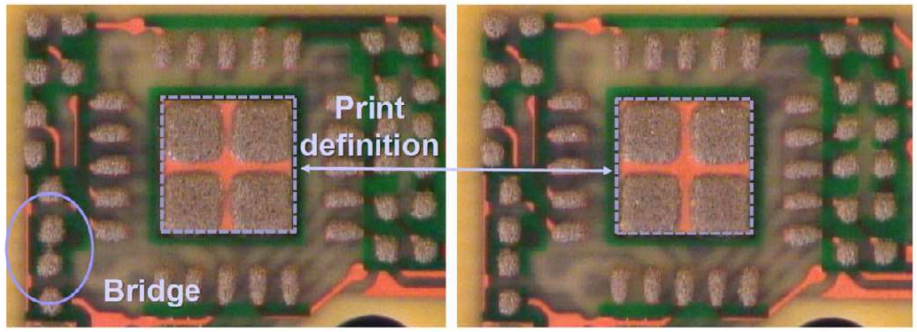


LaserJob Inc.
Eric Moen

Nano Coating Benefits

- **Better quality printing**

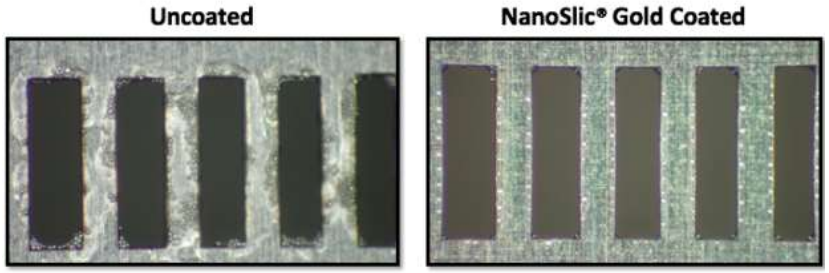
  - Because the flux is repelled from the aperture walls by nanocoating, there is a reduction in bridging,



Results from stencil with no nanocoating

Results from nanocoated stencil

1. Reduced need for underside cleaning
2. Reduced bridging
3. Improved solder paste release
4. Improved yield



Ref: Tecan nanocoating

Can Nano Coatings Really Improve Stencil Performance? – A study

Nano-Coating Thickness
Nano-coating thickness varies by supplier (Table 1). These thickness values were taken from supplier literature and were not measured directly.

Table 1: Coating Thickness

Coating	Thickness
Coating A	1000 – 2000 nm (1 – 2 microns)
Coating B	2 – 4 nm
Coating C	2 – 4 nm
Coating D	2000 – 4000 nm (2 – 4 microns)

Coatings A and D are 500 to 1000 times thicker than coatings B and C respectfully. Coatings A and D are applied using a spray and cure process by the stencil supplier. Coatings B and C are applied by wipe, either by the stencil supplier or by the user. Coatings A and D both have a tint and are visible on the bottom of the stencil and on the aperture walls. Coatings B and C are clear and are not visible on the stencil.

Surface Function – Contact Angle
Contact angle measurements were made multiple times and average values are reported here (Table 2).

Table 2: Contact Angle on Nano-coatings

Coating	Contact Angle DI water (deg)	Contact Angle n-Hexadecane (deg)
Coating A	103	60
Coating B*	101	66
Coating C*	109	70
Coating D	105	64
Uncoated (U)	54	9

*Inconsistent performance from lot to lot.

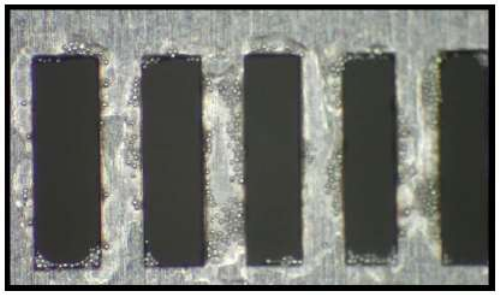


Figure 2: Bottom Side of an Uncoated Stencil After 20 Prints

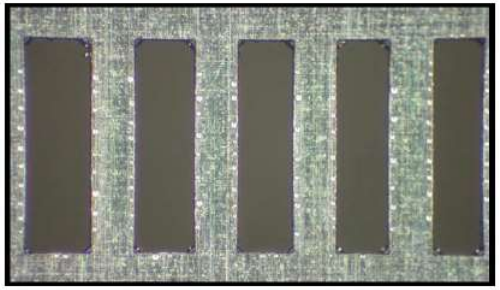


Figure 3: Bottom Side of a Nano-Coated Stencil After 20 Prints

The uncoated stencil shows solder paste adhering between the apertures (Figure 2). After 20 prints, solder paste is not present on the nano-coated stencil bottom (Figure 3). All of the nano-coatings tested (A, B, C, and D) displayed the same performance in this test.

Table 4: Summary of Return on Investment

Item	Cost Savings
Print cycle time improvement	2 boards printed per minute instead of 1
Cleaning material savings	$0.18 – $0.20 per board
Solder paste waste reduction	$0.04 - $0.07 per board
Yield improvement	Savings inestimable
Rework reduction	Savings inestimable
If nano-coating costs $40	ROI is 150 to 180 boards

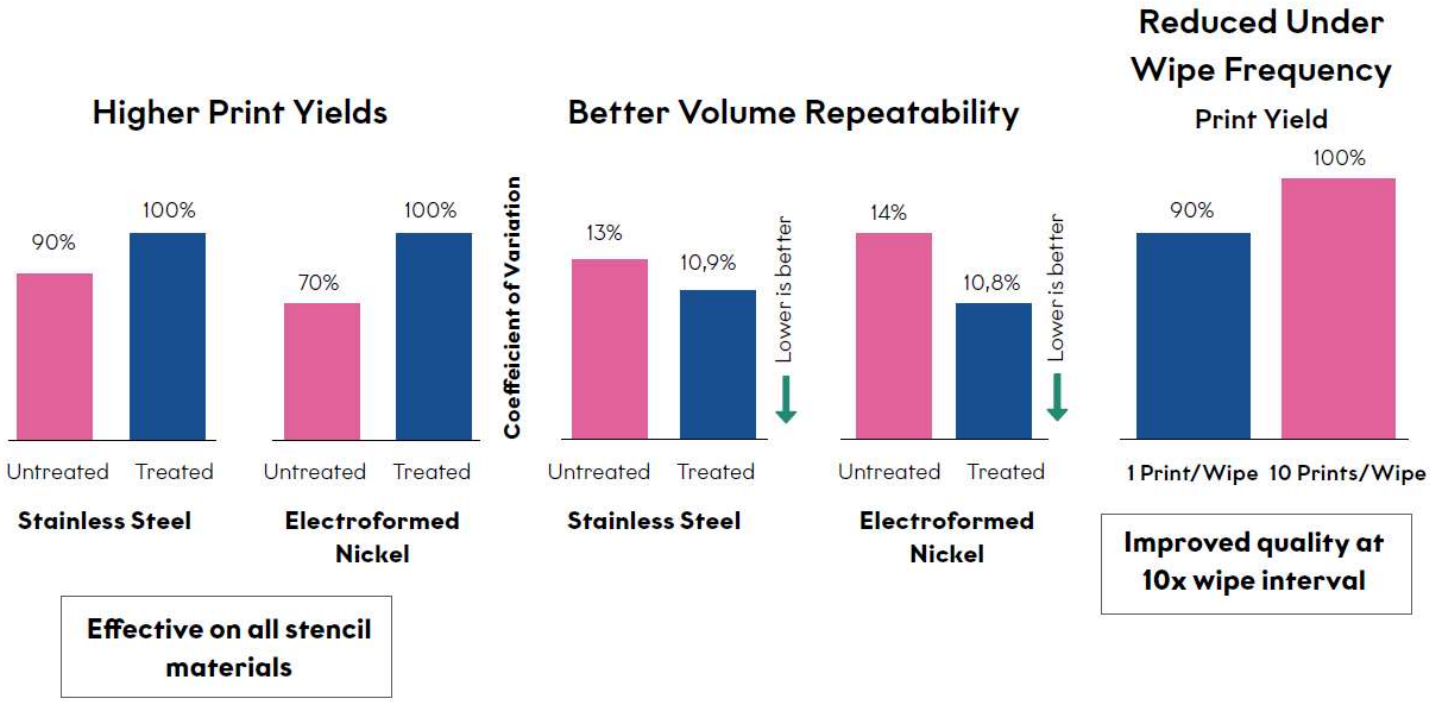
Table 5: Benefits of Nano-Coatings

Benefit	Nano-Coatings
Underside cleaning improvement	All coatings A, B, C, D
Bridging improvement	All coatings A, B, C, D
Transfer efficiency increase	Coatings A and D
Visible on the stencil	Coatings A and D
Re-apply by the user	Coatings B and C

Table 6: Negative Impact of Nano-Coatings

Negative Impact	Nano-Coatings
Coating wears through abrasion	Coatings B and C
Coating wear not visible	Coatings B and C
Transfer efficiency decreased	Coatings B and C

Print Definition Improvement

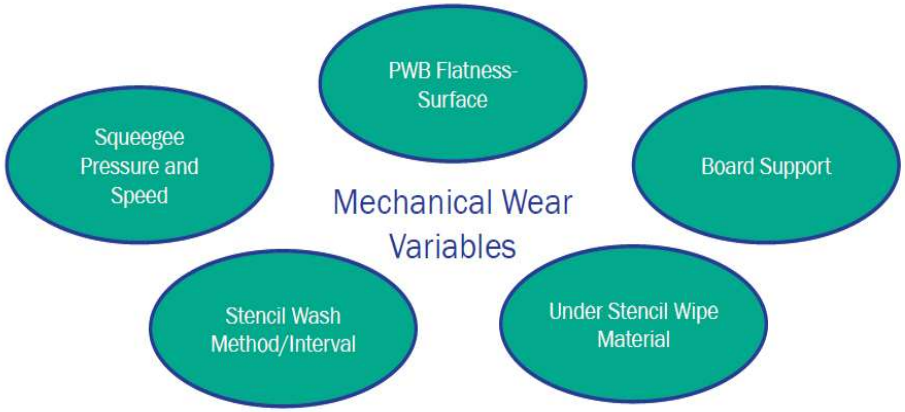


Improved Productivity – Reduced Costs

Ref: Tecan nanocoating

How long these coatings last?

- Nano-coatings provide printing benefits
- But they last as long as the coating is functional
- Mechanical and Chemical damage occur to the nano-coatings during use
- Nano-coatings are nano-meters to microns thick and will not last forever
- Nano-coating life can be extended



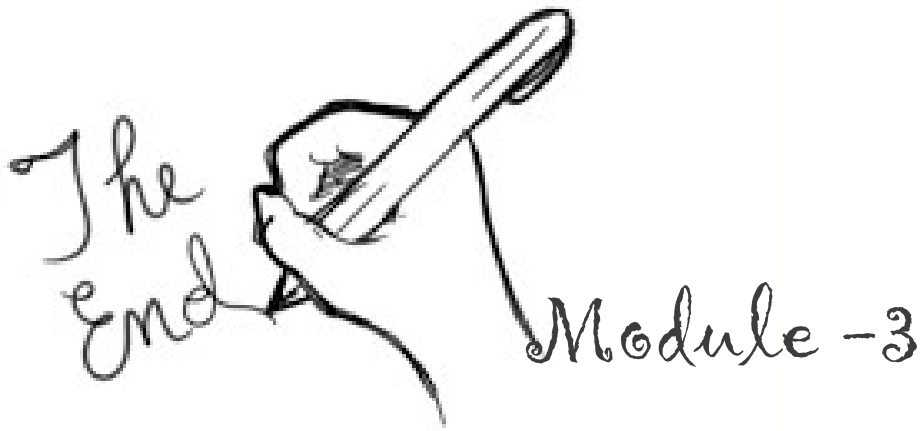
Nano-coatings – factors to consider

- What is the smallest aperture and pitch on the stencil and what is its thickness?
  - In the case of a large board with large apertures, the benefit of nanocoating is insignificant.
  - Stencil performance can be considerably improved - with fine pitches and small apertures.

- How many prints are you using the stencil for?
  - Involving High volume mass production runs
  - nanocoating significantly reduces the frequency of cleaning cycles.

Summary

- Area ratios of 0.40 are printable due to the reduced adhesion between solder paste and stencil surface
- The cleaning frequency of the pcb side of the stencil can be reduced dramatically
- The risk of solder paste bridges at QFP structures is reduced significantly
- Solder paste deposits printed with a nano-coated stencil are more uniform
- As potential savings in cleaning materials, solder paste waste, yield improvements and avoidance of rework.
- The nano-coating of stencils improves the printing performances and sets a milestone in stencil technology
- The cost to apply most nano-coatings is negligible
- The user should be aware of the benefits and negative impacts when deciding to use a nano-coating.



ANY QUESTIONS?

Back up slides

Nano Coating

1. Reduce Defects (Bridging, Insufficient, Solder Balls)

- Reduces the ability of flux (solder paste) to stick to apertures and bottom side of stencil
- Improves the ability of the USC to clean the stencil
- Improves the Gasketing of the stencil to the PCB
- Greater first pass yield

Plane Eye View
No Coating

Plane Eye View
After NANO Coating

2. Increases Efficiency

- Reduces the frequency of cleaning
- Allows time for more production or SPI

3. Reduces Cost

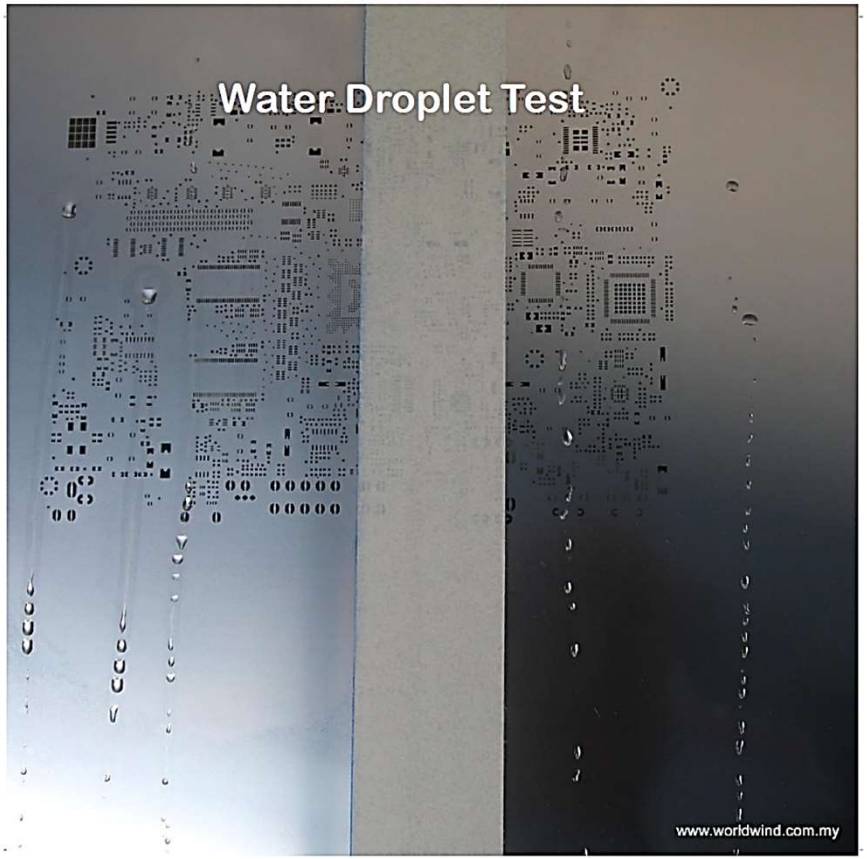
- Less cleaning uses less USC fabric and solvent
- Less expensive vs. competitors products
- Easy to apply product with just a wipe
- Can be applied by stencil manufacture or end user to new or existing stencils
- Permanent bond but can be reapplied if needed
- It is chemically inert when dry so no possible interaction with paste

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
- This can reduce the number of cleaning cycles required during the paste printing process
- Found to have improved paste transfer efficiency for fine pitch apertures
- Measurable advantages from the first print out of the stencil
- Can be applied on the production line to previously used stencils.

Water Drop Test for Nano Coated

Nano Coated

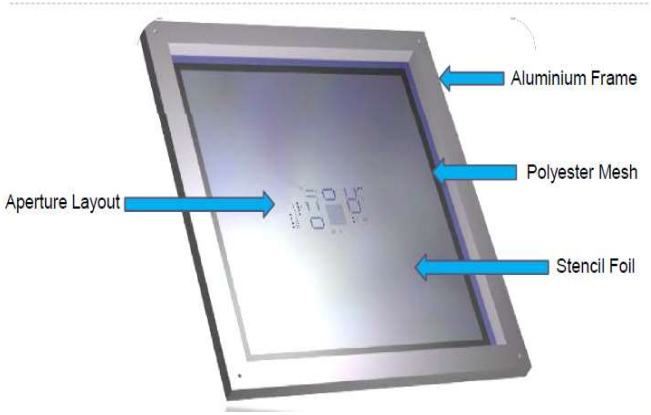


Not Coated

- hydrophobic nano-coating that, when applied to the bottom-side of the stencil foil and stencil aperture walls, minimizes solder paste's ability to stick to the foil surface. Up to a 10X increase in the number of prints before cleaning the stencil is possible as well as successful printing at lower surface area ratios.
- The permanent nano-coating is extremely durable and compatible with all stencil cleaning equipment, as well as the majority of stencil cleaning solutions on the market. Lower standard deviations, higher print repeatability and minimized rework are just some of the tremendous benefits of the Nano-ProTek coating. The addition of DEK's Nano-ProTek coating to Fine Line Stencil's highly successful UltraSlic™ and Slic™ stencil technology further increases the performance gap between it and any other stencil technologies available today.
- Features:
  - Can be applied at room temperature
  - No training requirement
  - Forms a permanent bond
  - Can be reapplied at any time; will only stick to area not already covered
  - As safe as IPA
  - REACH compliant
  - Compatible with stainless steel or nickel stencils
  - Will coat up to a 29" x 29" stencil
- Benefits:
  - Unprecedented price-point
  - Enhances paste release

What is SMT Stencil

- The SMT stencil is typically a sheet of 40um – 300µm thick SS foil (1.5 mil to 6 mil)
- The apertures are formed using laser according to the solder pads on the printed circuit board
- Stencils are used for transferring solder paste or SMD Adhesive on to a PCB.
- The accuracy of placement onto the PCB is critical. Hence alignment has to be 1:1 (Stencil Vs PCB)
- In addition to the above stencil ensures formation of properly sized and shaped deposits.



The Stencil foil is tensioned and fixed to the frame by polyester mesh. The Tension of the Stencil Foil is around 40N/cm (± 5N/cm)

The Purpose of Stencil

- The surface mount assembly process uses a stencil as a gateway to an accurate and repeatable solder paste deposition.
- A stencil is a thin sheet or foil of engineered stainless steel with a circuit pattern or foot print cut into it using a Laser / Chemical Etching
- It matches the positional pattern of surface mount devices (SMD) on the printed circuit board (PCB) for which the stencil is to be used.
- After accurately positioning and matching the stencil over the PCB, a metal squeegee forces solder paste through the apertures of the stencil to form deposits on the PCB for holding SMDs in place.
- The solder paste deposits, when passed through the reflow oven, melt and secure the SMDs to the PCB.

Stencil Ratios

The ability of the paste to release from the inner aperture walls depends primarily on three major factors:

1. Area ratio & Aspect ratio for stencil design
2. Aperture side wall geometry
3. Aperture wall smoothness.

The generally accepted design guideline for acceptable paste release is
> 0.66 for the Area ratio and
> 1.5 for the Aspect ratio.

- When the stencil separates from the PCB, paste release encounters a competing process
- This again creates doubt, will it transfer to the pad on the substrate or will it stick to the side aperture walls?
- The solution for this is, the area of the pad is to be greater than two-thirds of the area of the inside aperture wall
- This will facilitate the paste to achieve 80 percent or better paste release.

AREA RATIO > 0.66 & ASPECT RATIO > 1.5

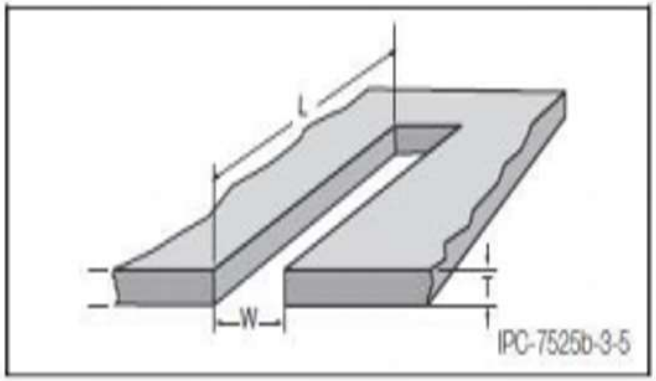
Stencil Design is Important

- Reconcile Paste Requirements
  - Most Difficult to Print
  - Minimize Post Reflow Rework Requirements
- Maximize Area Ratio – Know Error Sources
  - Powder Type - Packs
  - Flux Chemistry – Releases
  - Decrease Foil Thickness – Increases TE
- Step and Stepless Step Stencils – Watch Keep Outs
- Reverse Taper
- Preforms – Tape and Reel
- Stencil Scaling

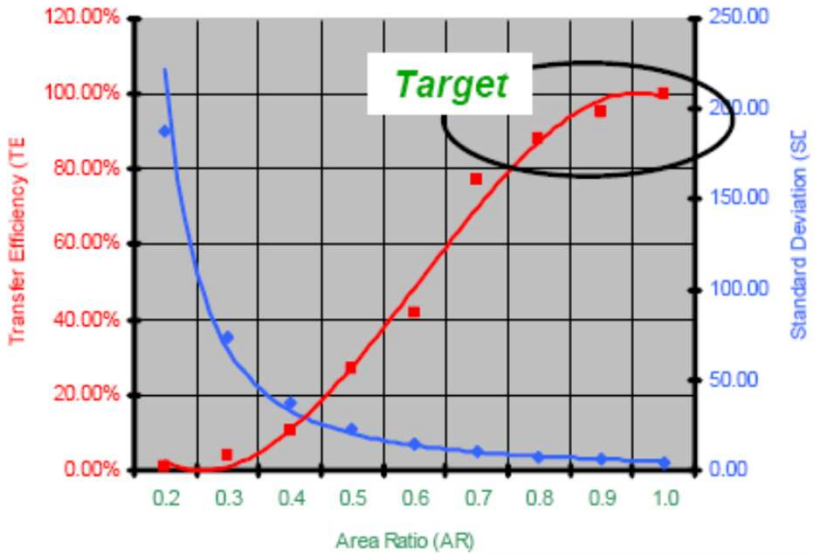
Stencil Ratios

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

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



Science Behind Printing



Area Ratio = Pad Area > 0,66
Wall Area

Transfer Efficiency = Volume Deposit
Volume Aperture

The higher the Area Ratio,
the higher the Transfer Efficiency,
the lower the Standard Deviation

- **Generally, all the equipment used for stencil laser cutting recognize GERBER format as its operation format**.
- Therefore, by making GERBER file, in a point of fact, you provide originally well-defined and clear task for the machine.
- In short, this is a “native language” for a manufacturer. Speaking the same language with him you eliminate a case when you are not understood by anybody because all the data is represented in the file itself.

Gerber® data describes the file format that provides a language for communicating with the photo plotting system to produce a tool for chemically etched stencils. It is also used to produce the laser cut or electroformed stencils. While the actual data format may vary from file to file depending on the software package or designer, the data format commonly used by photo plotter and laser equipment is known as Gerber.

Stencil Replacement

Paste deposition that is too small is the most common factor affecting Ultra-Fine pitch printing. The stencil is the prime element in determining the application of the exact amount of material in the exact location. Therefore it is important that a stencil is considered as a precision engineered tool that is vital to the overall process performance and should be cared for as such. Most stencils will not be used to the end of their stencil life.

Major reasons for stencil replacement are:

- Design Revision (layout change)
- Process Optimization (corrective action)
- Damage (handling)
- Wear & Tear (use)

Monitoring stencil life is critical to the overall print process performance & consistency.

Material	Estimated Life Span*
Hard Nickel Stencil	100K
Stainless Steel Full Hardness Stencil	50K

Electroformed Example

E-Form Stencil SEM Image