

SMTA

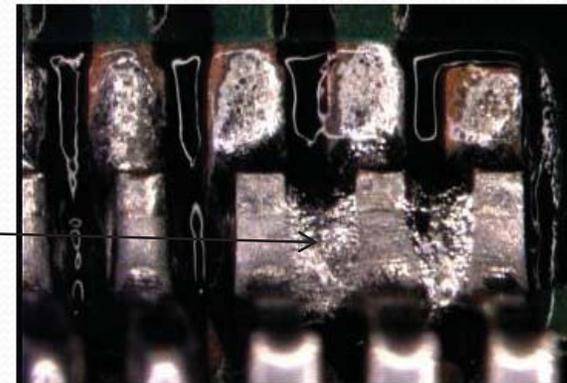
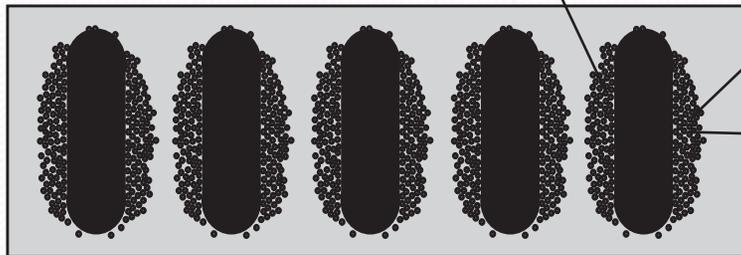
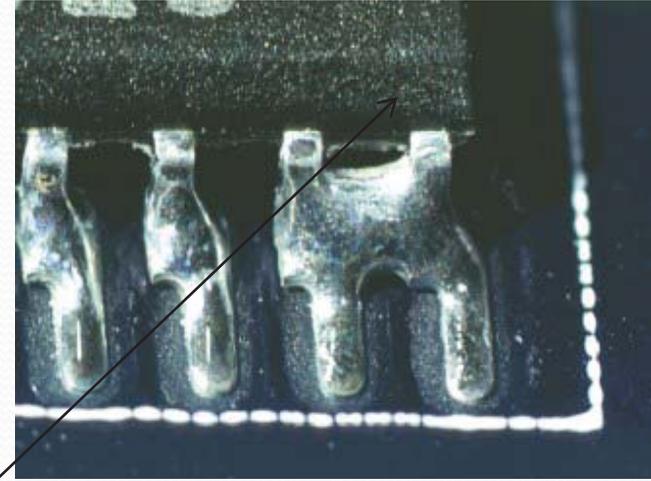
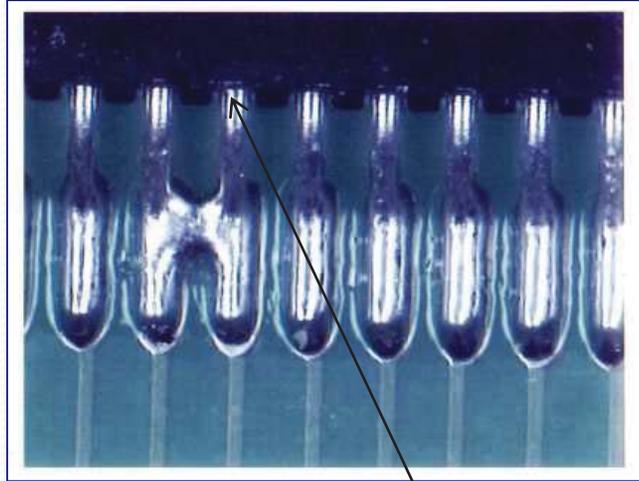
Surface Mount Technology Association

Stencil Printing Fundamentals

SMTA-Printing 101

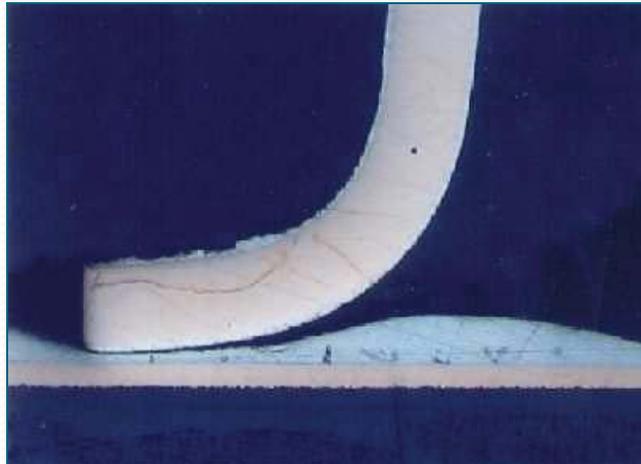
Examples of Print Defects

Short

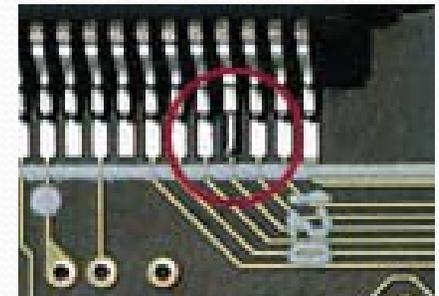
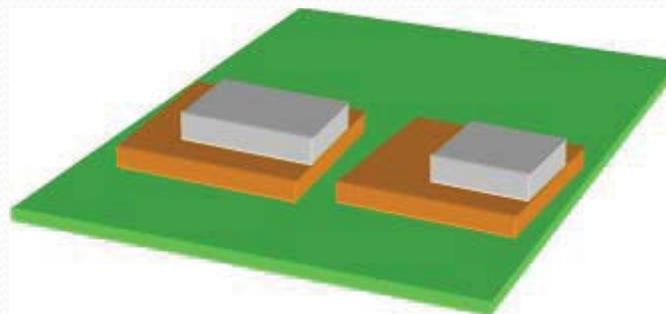
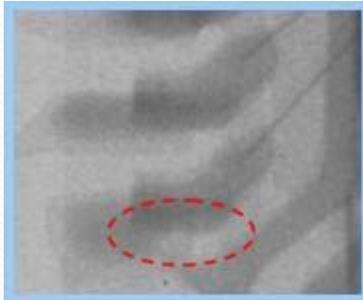


Print related causes: Excess paste, dirty stencil

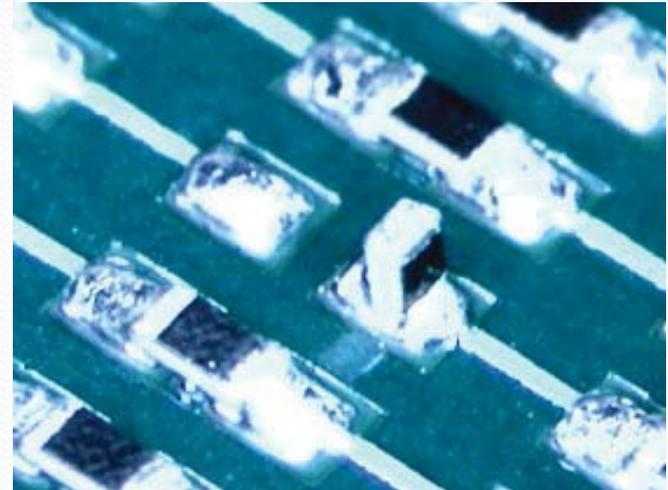
Open



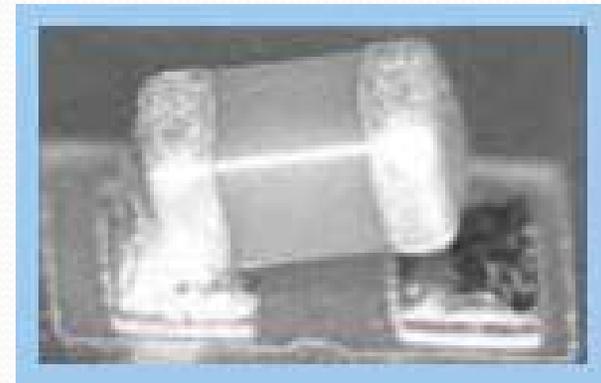
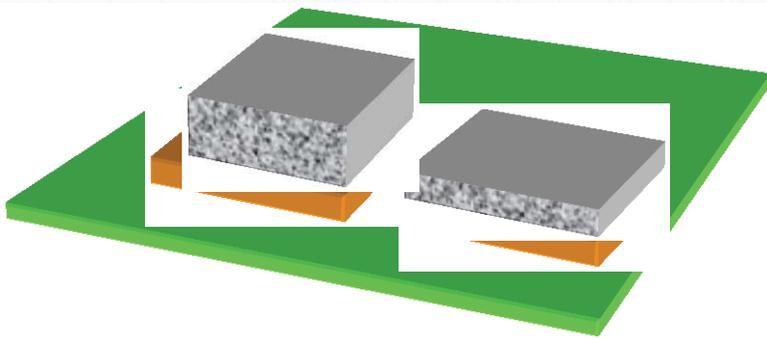
Print related cause: Lack of solder paste or inconsistency of solder paste



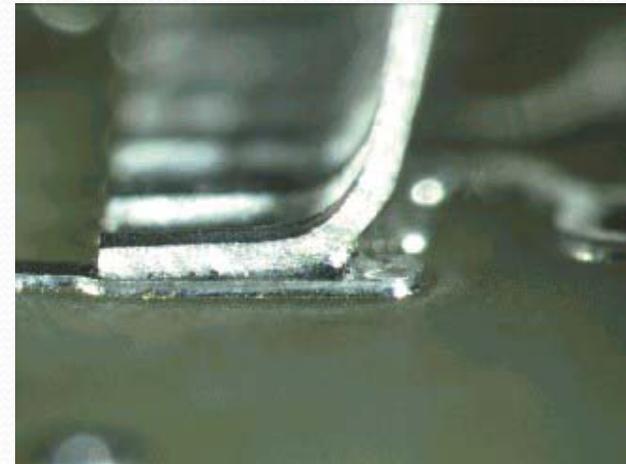
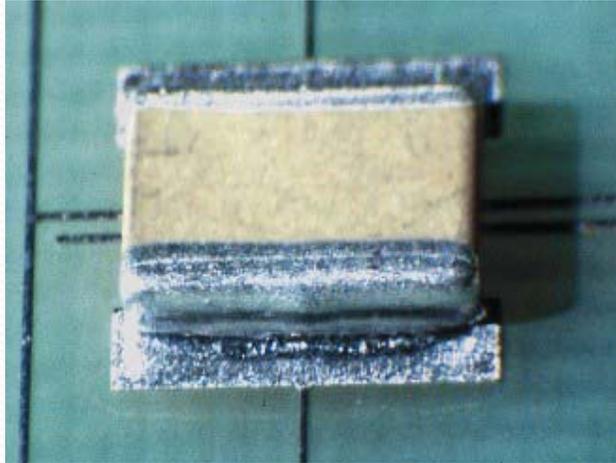
Tombstone



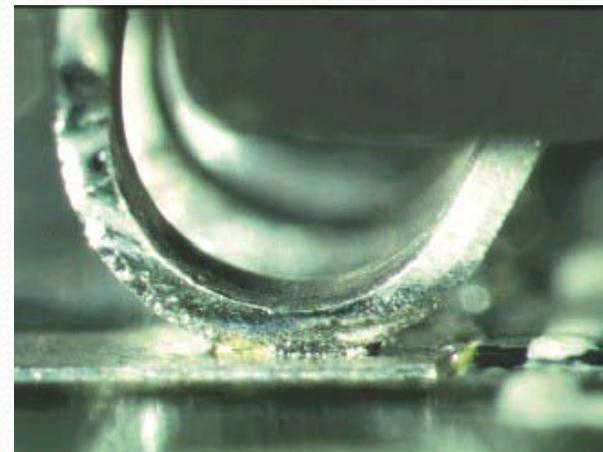
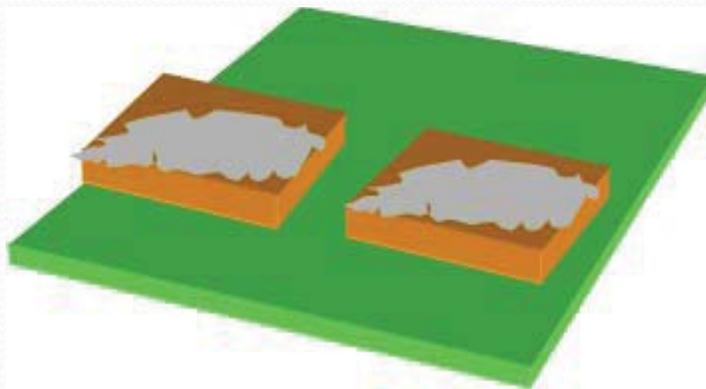
Print related cause: Inconsistency of solder paste



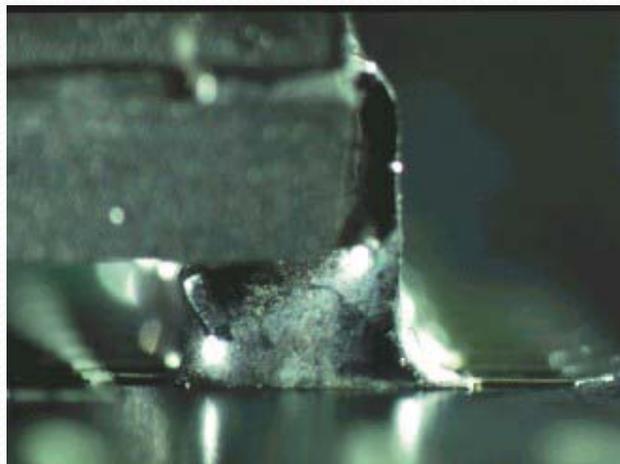
Poor solder Joint



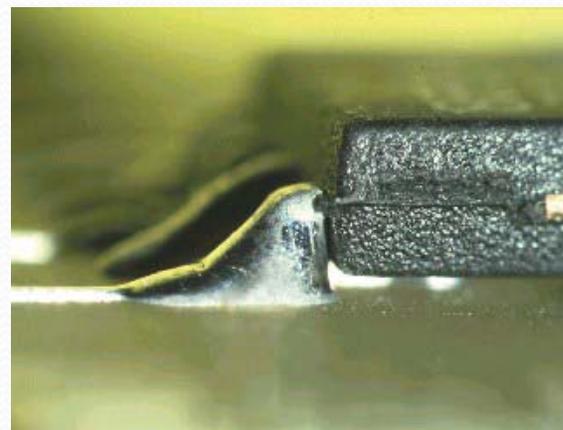
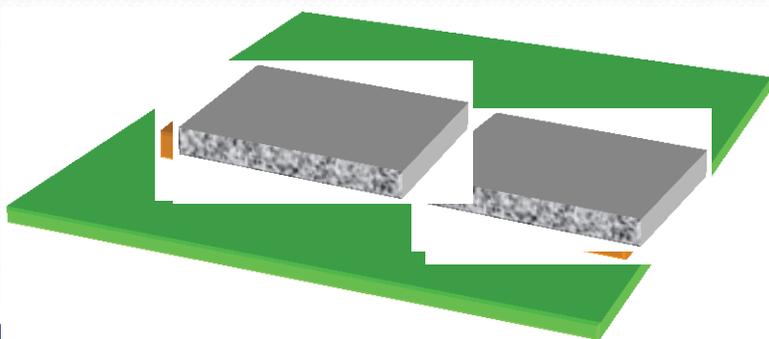
Print related cause: Lack of solder paste



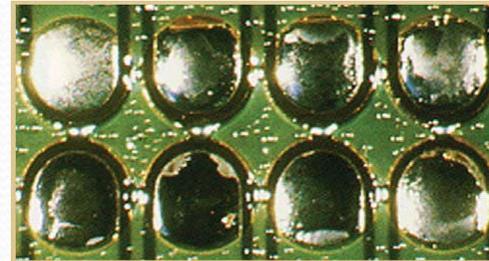
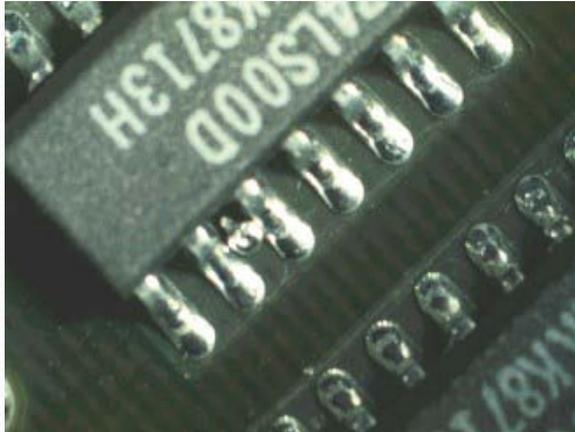
Poor solder Joint



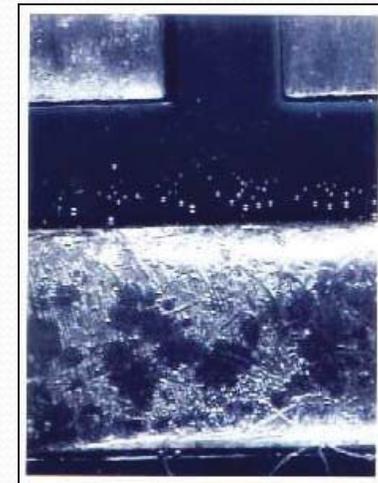
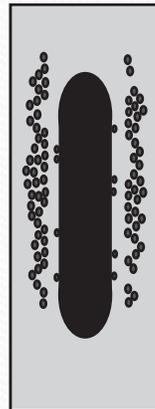
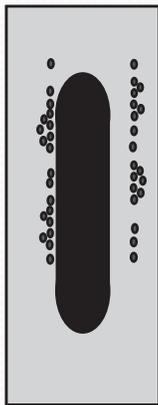
Print related cause: Excess solder paste



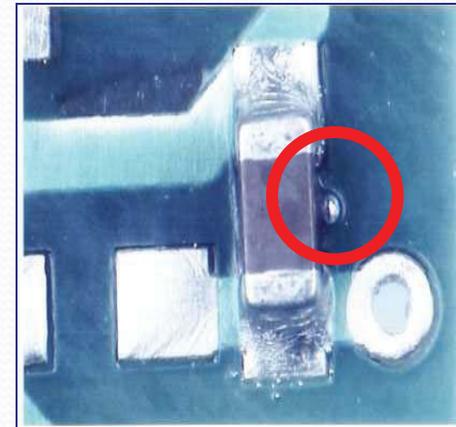
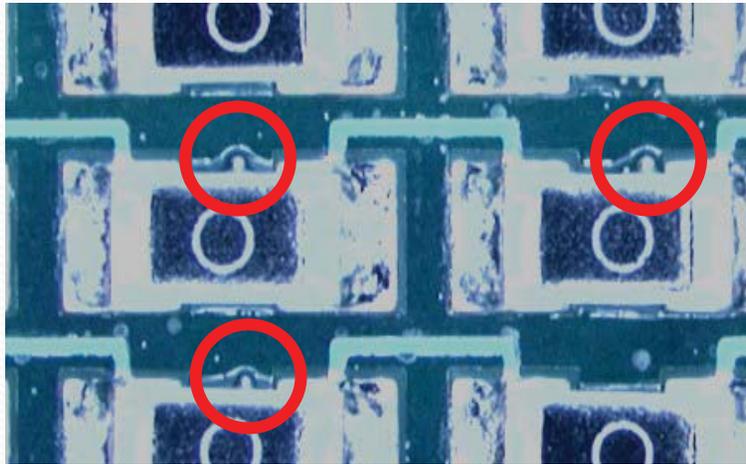
Solder ball



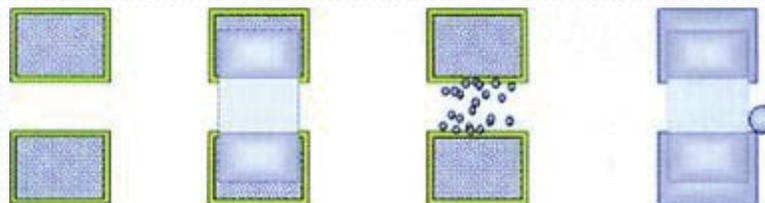
Print related cause: Dirty stencil



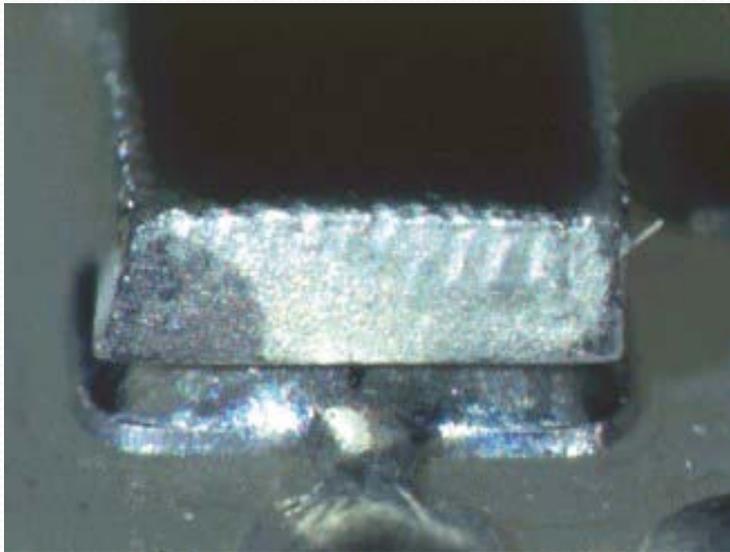
Mid-chip Solder Ball



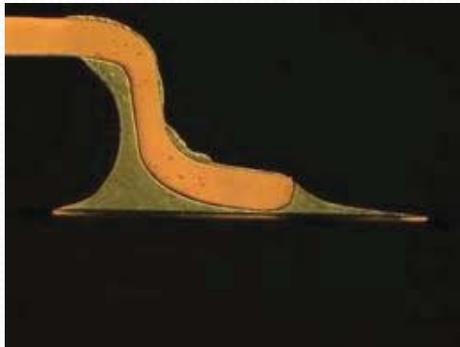
Print related cause: Poorly designed stencil



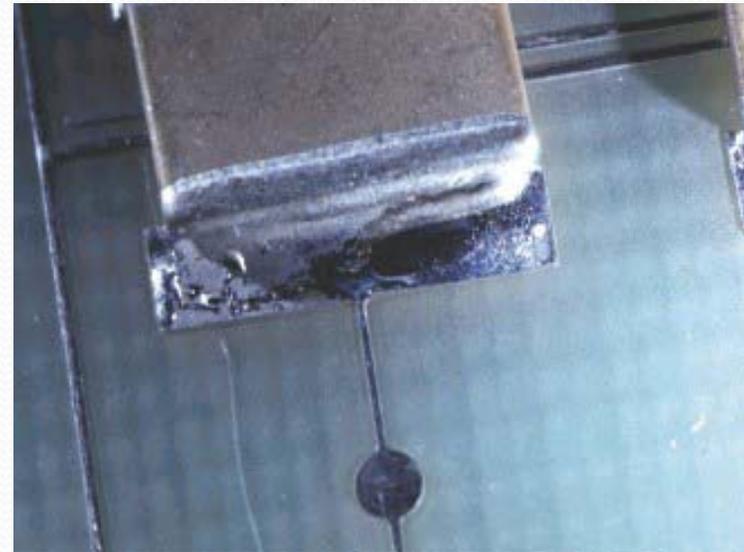
Non-wetting



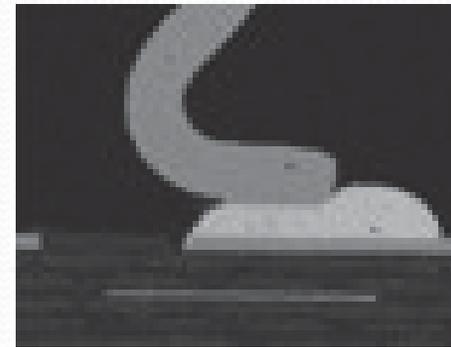
Good wetting



De-wetting



Bad wetting

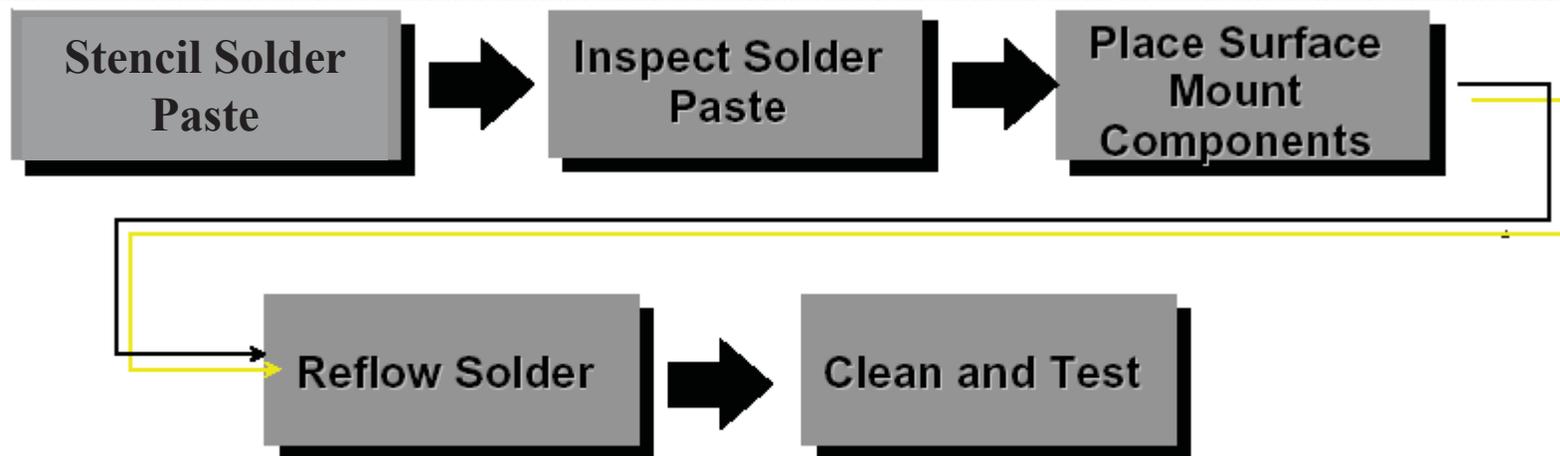
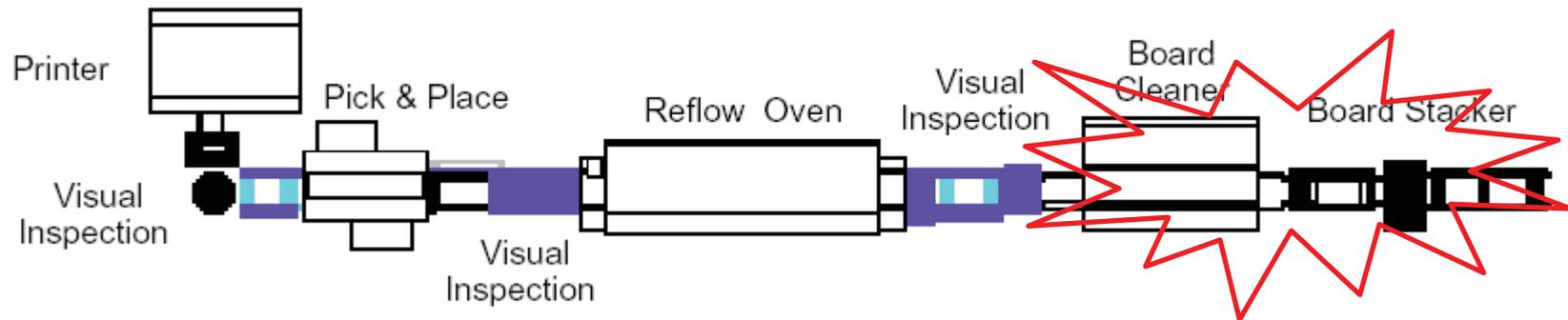


SMT Overview

Surface Mount Technology (SMT) refers to a specific type of electronics assembly where electronic components are attached to the surface of a substrate (typically a printed circuit board).

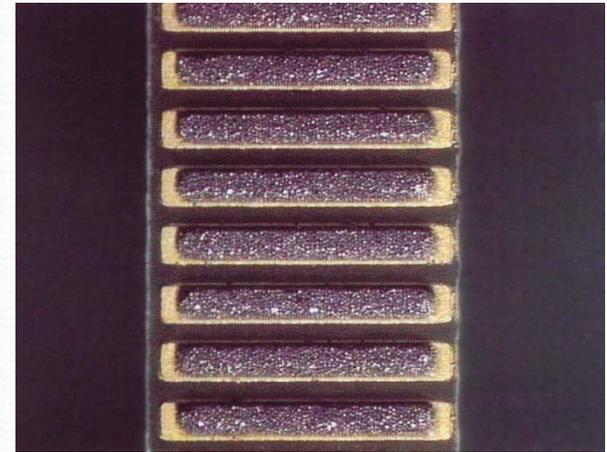
SMT is a modern alternative to traditional **thru-hole technology** where components were attached to substrates by leads that passed through holes in the substrate. Surface Mounted components require less space so SMT is helpful in product miniaturization.

Basic SMT Process Flow

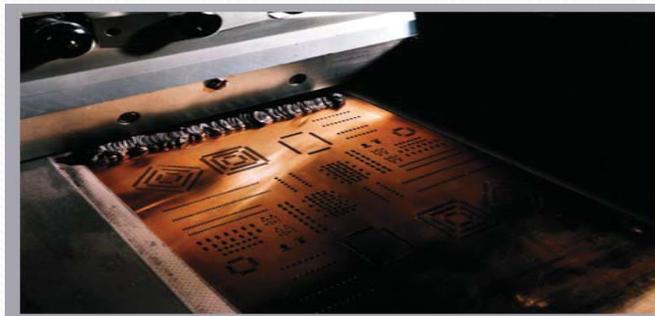


Critical Elements in SMT Assembly

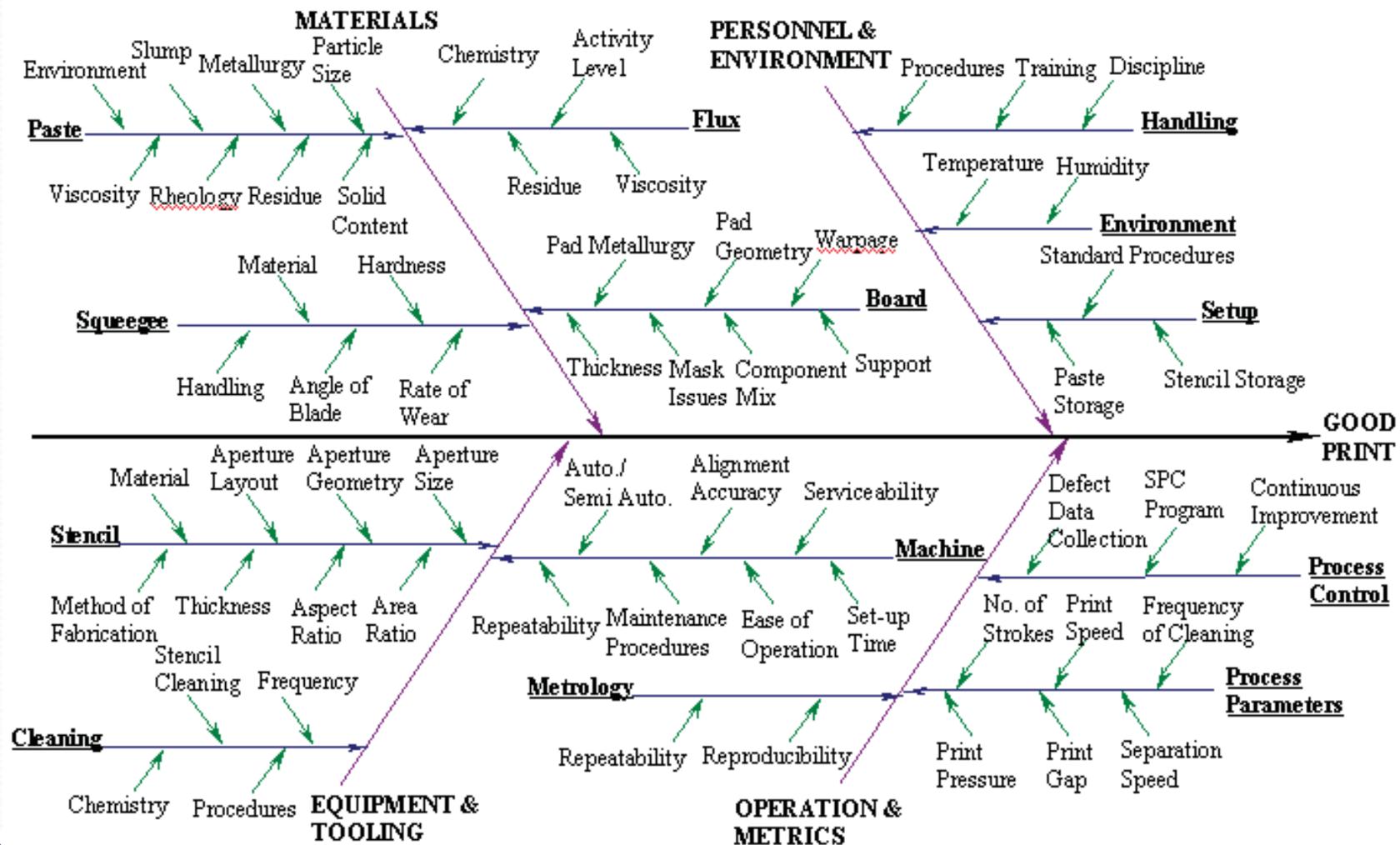
- Solderability (Board/Pad)
- Paste Selection
- **Printing process**
- Component Placement
- Reflow process
- Cleaning process (if required)



50-70% of soldering defects occur due to poor printing

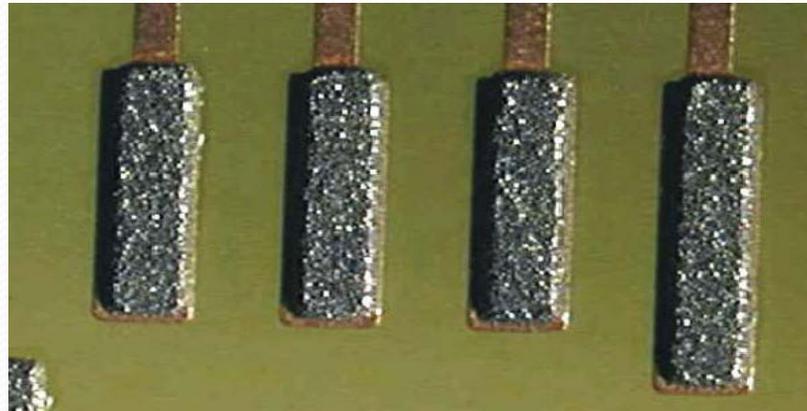


Factors Affecting the Printing Process



Key elements to successful Printing

- Paste
- Stencils
- Boards
- Blades
- Tooling
- Release
- Wiping
- Offsets



Perfect print every time!!!



Solder Paste

- Solder Paste is a homogeneous mixture of solder powder (metal) in a stable viscous flux vehicle which is used to join two metal surfaces upon heating.

- Two main components of Solder Paste are:

- Solder Powder:

- Forms a permanent metallurgical bond between two or more metallic surfaces

- Paste Flux:

- Provides 2 main functions.

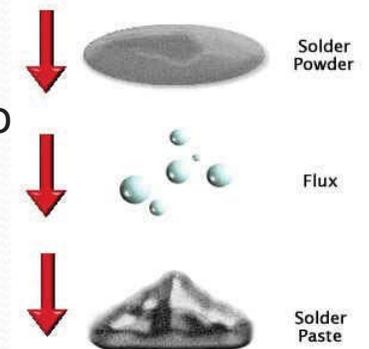
1. It suspends the powder to maintain the homogeneous mixture of the solder paste.
2. It chemically removes oxides from the components, the PCB pad, and the powder allowing a good metallic bond to form during reflow.

- 85 - 92% Metal by weight for printing applications (typical)

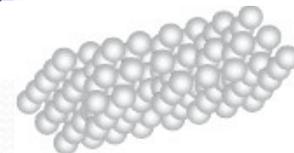
- 50% metal / 50% flux by volume

- 6 mil thick deposit leaves 3 mil of solder after reflow.

- Solder is typically packed in jars or cartridges



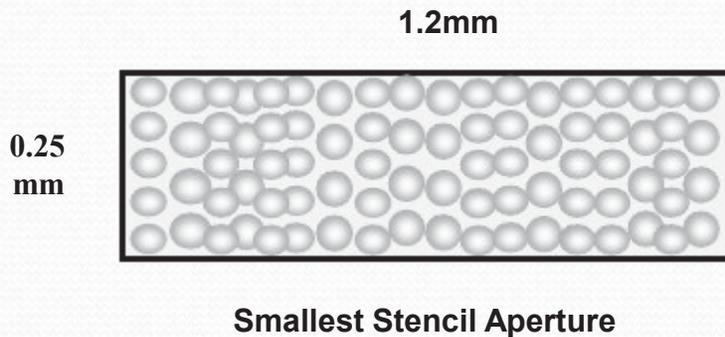
Solder Alloys



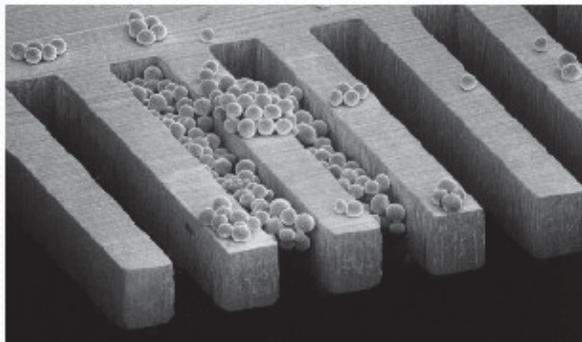
- Most Common Metal Mixture is 63% tin (Sn) and 37% lead (Pb)
 - Gold or Silver can be added as an anti-leaching agent
 - Sn62.5/Pb36.1/Ag1.4
- No-Lead alloys are becoming more common as a result of RoHS (Reduction of Hazardous Substances)
 - Sn96.5/Ag3.0/Cu0.5(SAC305)
 - Sn95.5/Ag4.0/Cu0.5(SAC405)
 - Sn98.5/Ag1.0/Cu0.5(SAC105)
- Different elemental mixes allow variation of reflow temperatures and physical properties

Particle Size

- The recommended particle size of the solder paste is determined by the smallest dimension of a stencil's aperture.



- Ideally there should be at least 4 to 5 particle diameters across the smallest stencil aperture
- Note:** Smaller size, increases surface area which can increase oxidation



Recommended Solder Powder Selections									
Device Type	Pitch		Aperture Width		Aperture Length		Maximum Particle Size		Solder Powder Classification
	mm	m il	mm	m il	mm	m il	mm	m il	
QFP	0.65	25.6	0.60	23.6	1.95	76.8	0.15	5.9	Type 1 or 2
QFP	0.50	19.7	0.30	11.8	1.45	57.1	0.08	3.0	Type 2 or 3
QFP	0.40	15.7	0.25	9.8	1.20	47.2	0.06	2.5	Type 3 or 4
QFP	0.30	11.8	0.20	7.9	1.20	47.2	0.05	2.0	Type 3 or 4
PLCC	1.25	49.2	0.60	23.6	1.95	76.8	0.15	5.9	Type 1 or 2
0402	N/A	N/A	0.45	17.7	0.60	23.6	0.11	4.4	Type 2 or 3
0201	N/A	N/A	0.23	9.1	0.35	13.8	0.06	2.3	Type 3 or 4

Fluxes

What is the Role of Flux?

- Flux: from the Latin *Fluxus*, to Flow
 - Aids in wetting by reducing surface tensions
 - Effective soldering requires that the solder “wet” the two surfaces to be joined
 - Removes & prevents oxidation
 - Oxides on the metal surfaces to be soldered will inhibit wetting
 - Additives vary by usage
 - There are hundreds, perhaps thousand of different flux formulations in use today.
 - The flux chemistry is the primary component of the solder paste that will determine its behavior for other solder pastes using the same metal powder

Paste Flux Types

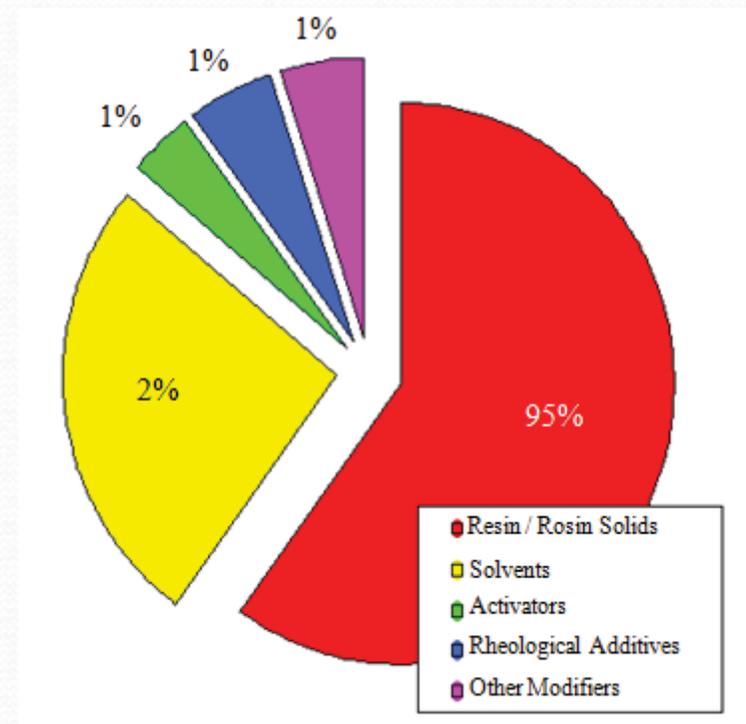
RMA:	Rosin Mildly Activated
RA:	Rosin Activated
WS/OA:	Water Soluble/Organic Acids
LR:	Low Residue/No Cleans

RMA's and RA's do not typically have to be cleaned, but as temperatures of the PCB's or the components increase towards the activation temperatures of the flux (approx. 150C), they may start to form halides or salts that can carry electricity and cause shorts. They may be cleaned using chemicals or water saponifiers. Water Soluble or OA's **MUST** be cleaned since the acids will eat away at the joints.

Paste Flux Ingredients

- Rosin/Resin
 - Solder flow properties, activity
- Solvent
 - Dissolves rosin/resin and activator
- Activators
 - Oxide scrubbing and cleaning
- Rheological Additives
 - Rheological agents, viscosity modifiers to keep the solder powder suspended in the mixture
- Other Modifiers
 - Wetting agents, tackifiers and coloring agents

FLUX COMPONENTS BY PERCENTAGE



Solder Paste Characterization

- Viscosity Profile
- Slump
- Tack Profile
- Reflow Profile
- Solder Balling Potential
- Printability
- Storage Stability



Chemistry differences in solder pastes

The flux and powder chemical and physical properties will impact the following:

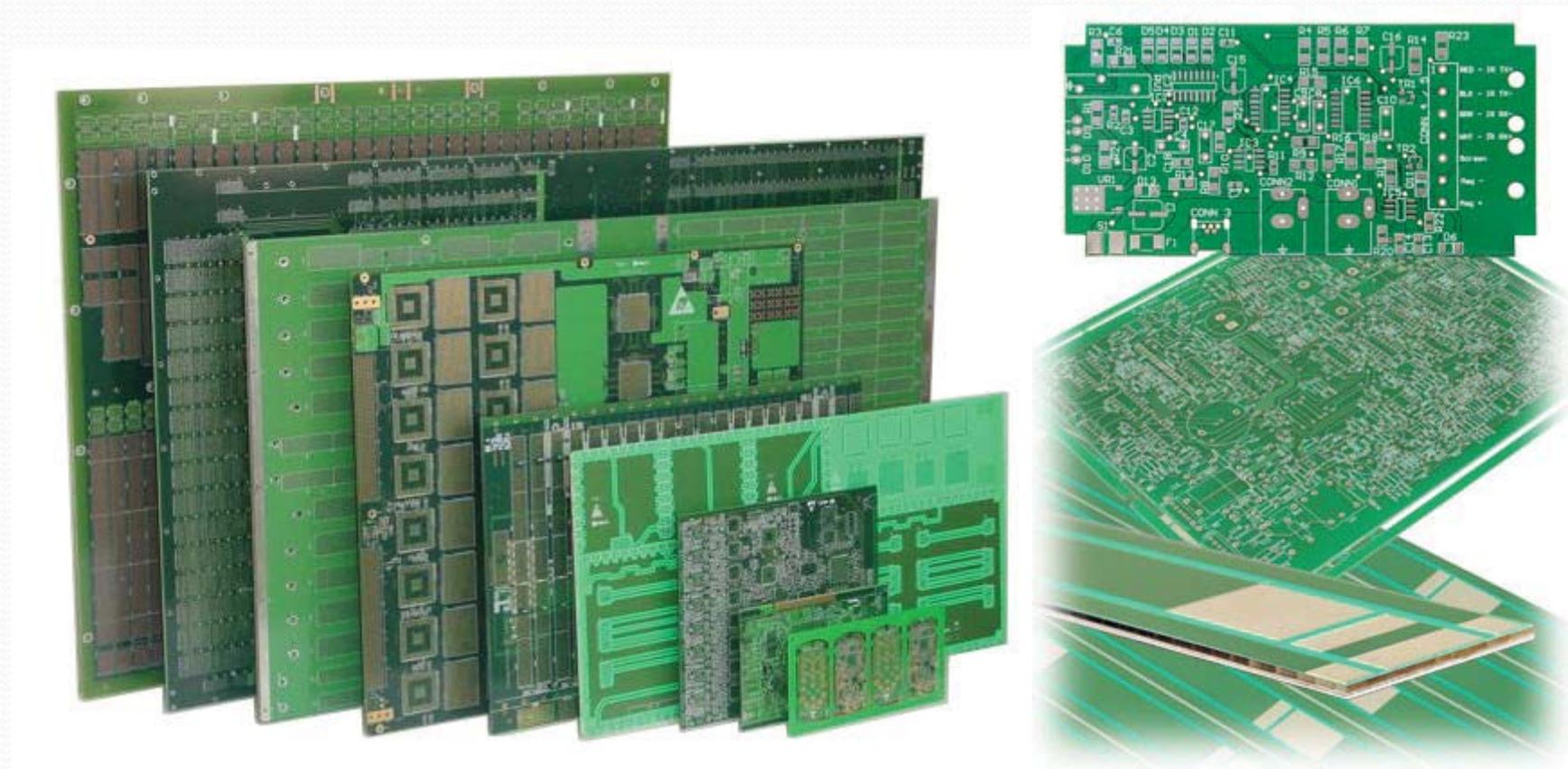
- Print definition and consistency
- Resistance to temperature and humidity
- Slump behavior after print
- Initial tack behavior
- Tack life after print
- Flux residue removal of stencil and misprints

Note: Not all solder pastes are alike, for best print results ask for the above data

Solder Paste: Handling & Storage

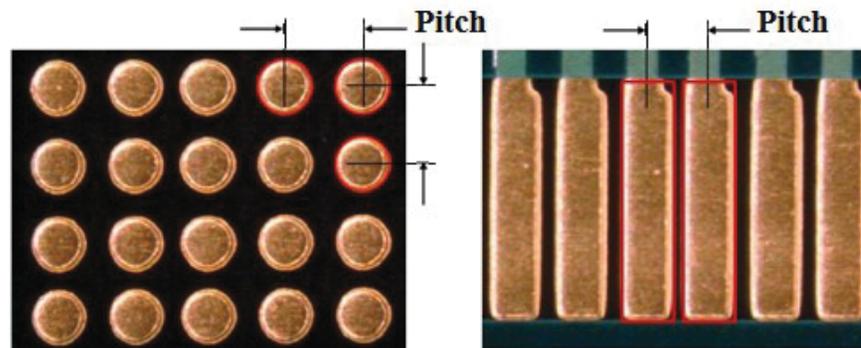
- Solder paste should be refrigerated when not being used.
- Keep the jar closed when not in use.
- Solder paste can only be reused if it has been out on the stencil for less than one hour. It should never be reused twice.
 - Always refer to the solder paste suppliers specification for details
- Solder paste should be thawed out for at least 2 hours before using (must be at room temperature prior to use).
- Stir thoroughly before using, or if using an automatic dispenser, the paste should be kneaded as necessary.
- Never heat up solder paste to help it thaw.
- Always use protective gloves when handling solder paste and wash hands prior to eating, drinking, or smoking.

Substrate-PCB



PCB (Printed Circuit Board)

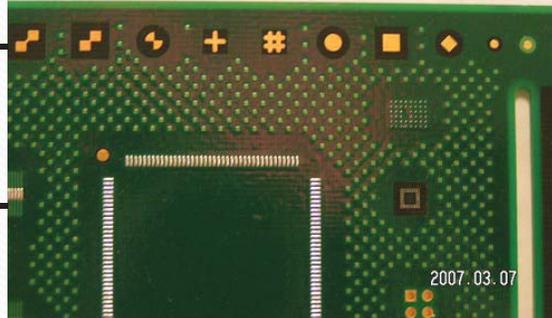
- The Printed Circuit Board (PCB) is the first thing to consider when looking at the stencil printing process
- The finest pitch device on a PCB typically dictates the overall process
 - The finer the pitch, the narrower the process window



- PCB Should have good rigidity, minimal routing
- PCB should have minimal warpage.
- Fiducials located on “Breakaways” of panelized boards may decrease accuracy

Fiducial Marks

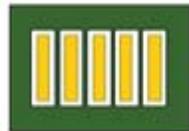
- Board level fiducial marks
 - Used on every board to adjust the location of the stencil in the printer to align with the component pads on the printed wiring boards.



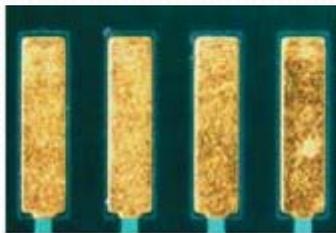
- Component level fiducial marks
 - Used on boards with complex components for very precise alignment during the component placement (pick and place) process.

PCB Solder Mask

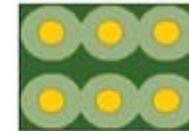
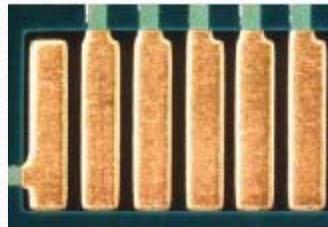
- Solder mask is a coating on the PCB, used to prevent solder adhesion between pads or in other undesirable locations.
 - Pad defined solder mask (between the pads) is common for Fine Pitch devices
 - “Windowed” solder mask is around a particular row of pads



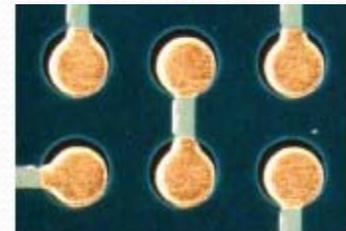
Pad Defined Solder Mask



Windowed Solder Mask



Resist Defined Pads

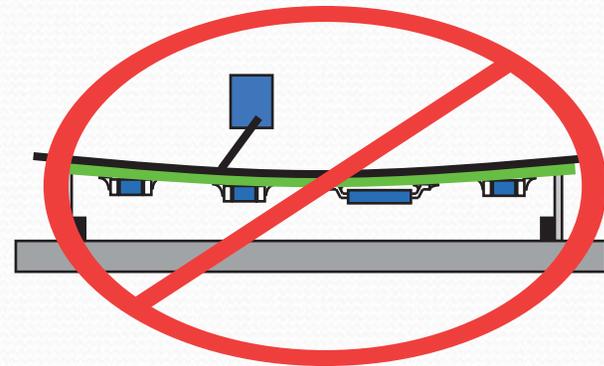
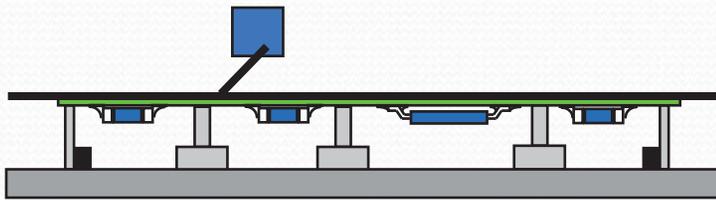


Board Support



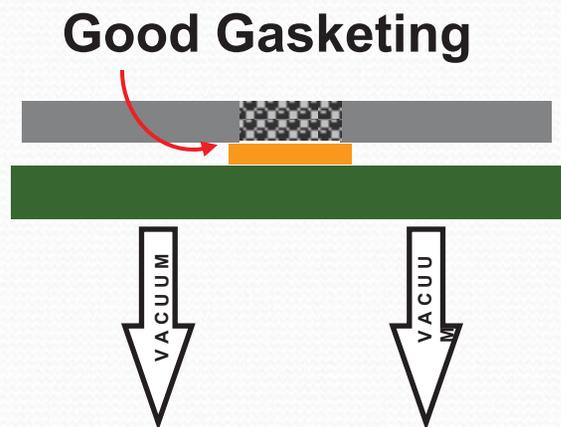
Board Support

- It is VERY important to secure and support the board during the printing process
- Any movement of the board in the X, Y, and/or Z Axis will cause printing defects
 - X and Y Axis are secured by snuggers and vacuum
 - Use caution with vacuum fixturing for thru hole printing
- Poor board support can negate (eliminate) all the good work done in stencil design

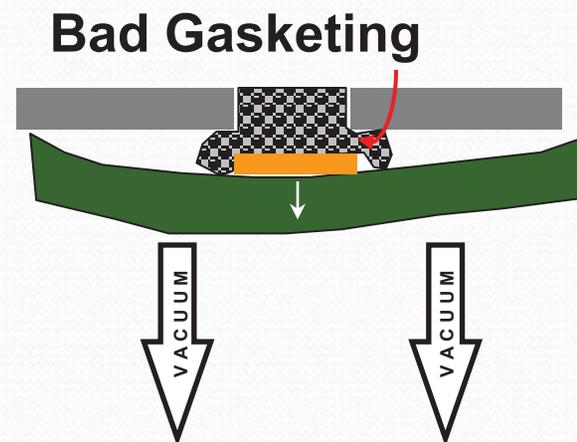


Board Support

- Inadequate board support can cause a bad “gasket” between the Board and Stencil
 - Board warpage can also cause this

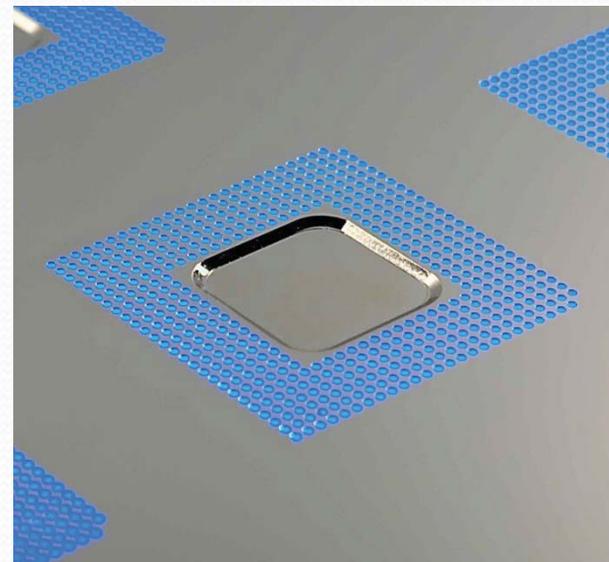
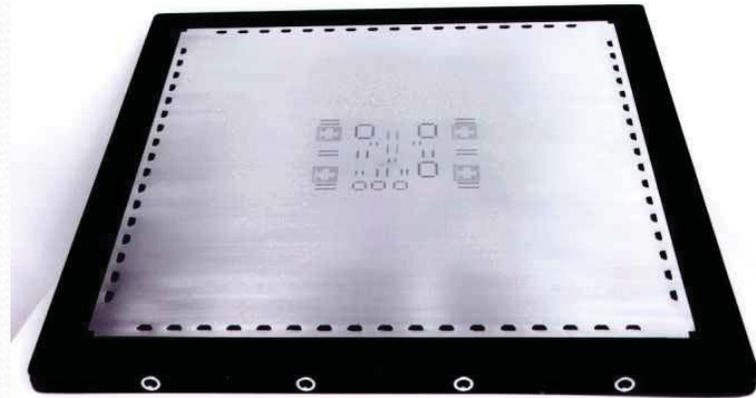
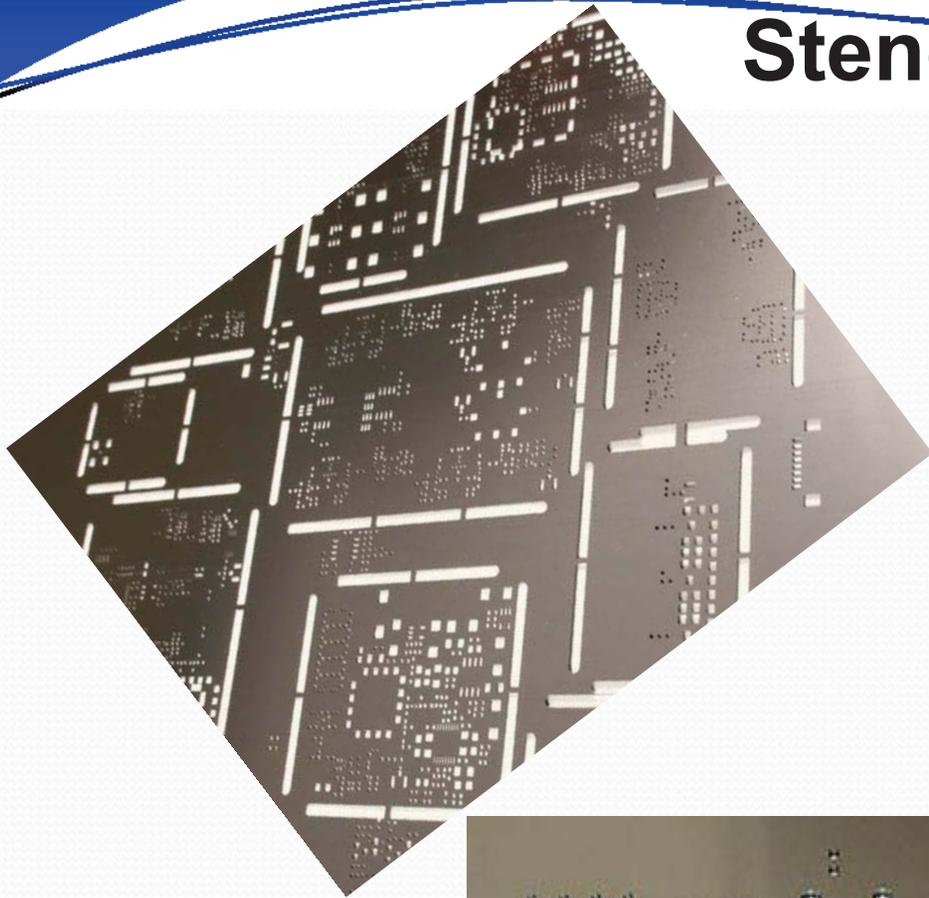


Good Support



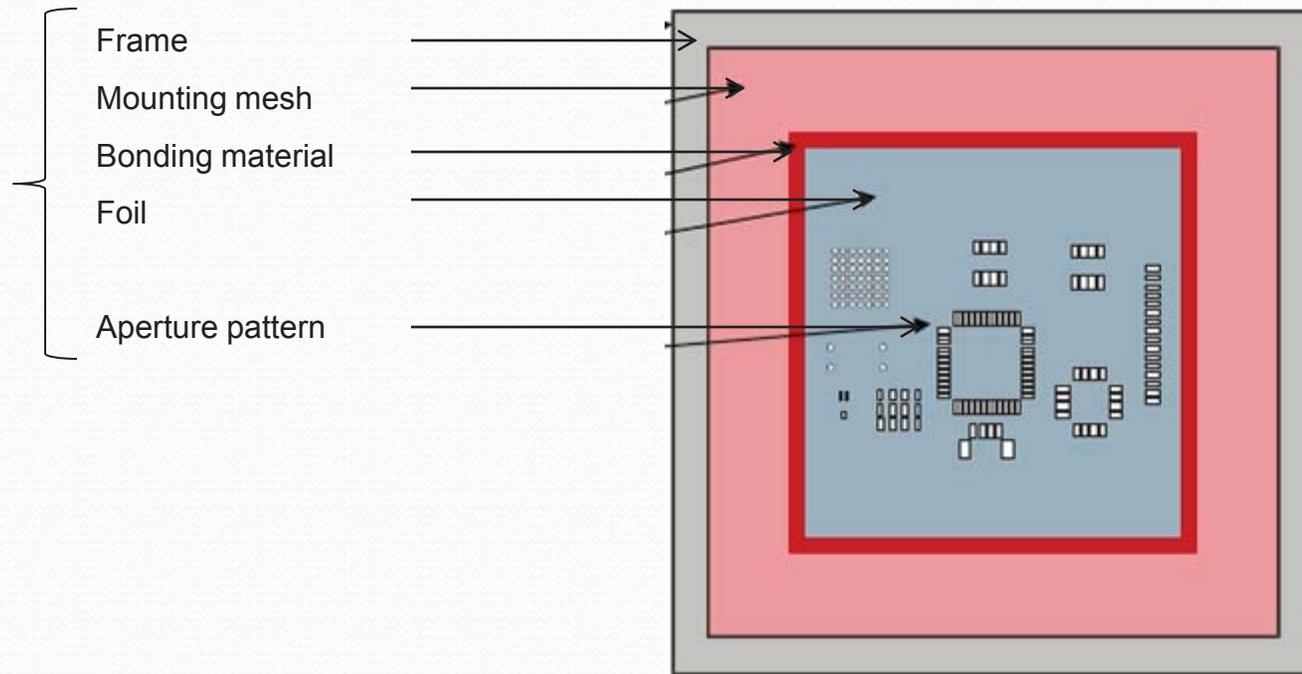
Bad Support

Stencil



Stencils Basic Components

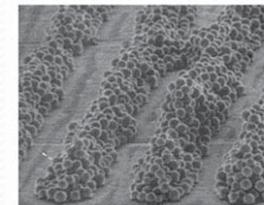
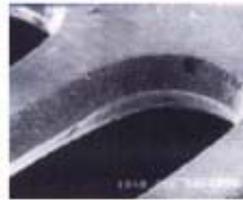
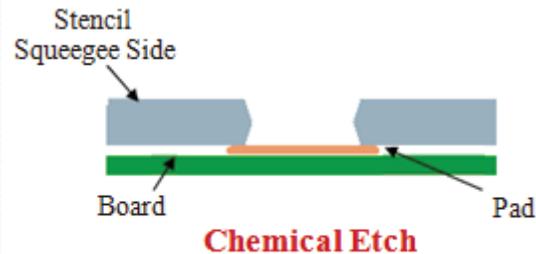
Basic components



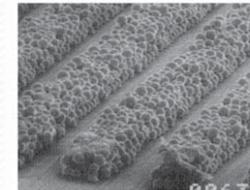
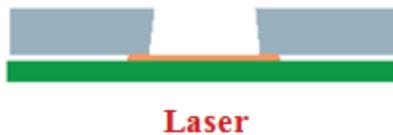
Squeegee side



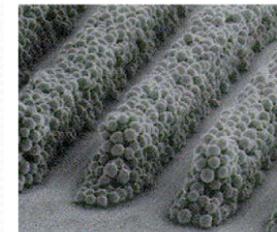
Stencil Manufacture



Aperture tapered from both sides to center of foil. Wall is usually smooth



Tapered aperture with more textured walls. Electro-polished for smoothness.



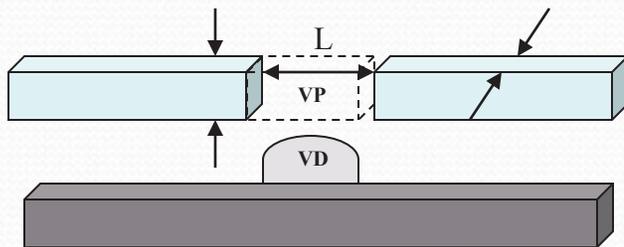
Tapered aperture with extremely smooth wall.

Design Rules For Stencils

- Aspect Ratio Rule
 - Ensures that the aperture will allow the printed material to release onto the substrate.
 - Aspect Ratio - $W / t > 1.5$



- Area Ratio Rule
 - Rule of Thumb for good paste release:
 - Area Ratio > 0.66



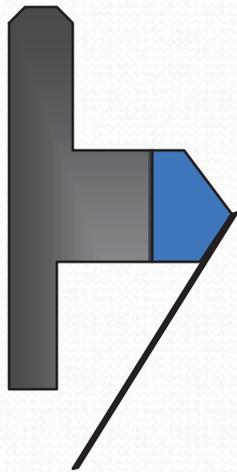
$$AR = \frac{\text{Pad area}}{\text{Wall area}} = \frac{L \times W}{2(L \times T) + 2(W \times T)}$$

Stencil Handling and Storage

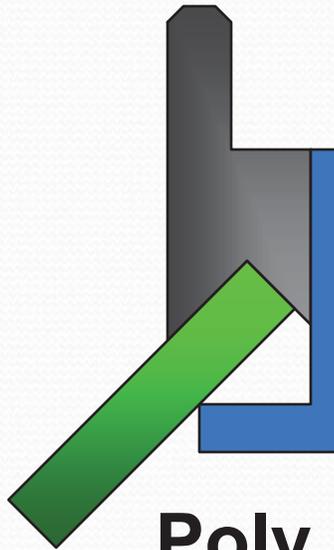
- Properly clean stencils after use
- Store stencils in the designated area after cleaning
 - Never store a dirty stencil
 - Do not leave stencils on benches, machines, the floor, etc. They should be only in the machine being used, being cleaned, or in the storage area
- Always inspect stencils for damage or wear prior to use. Never use damaged or worn stencils
- Clearly mark each stencil with the job or product number

Squeegee Blades

- Metal Trailing Edge
- Polyurethane Trailing Edge
- Diamond Cut



Metal



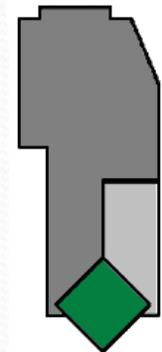
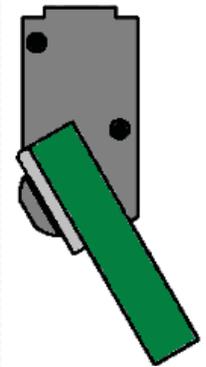
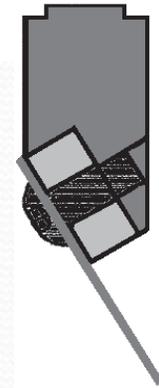
Poly



D-Cut

Squeegee Types

- Metal Trailing Edge Blades
 - Recommended for Ultra Fine Pitch and Fine Pitch stencil printing
 - Reduces effect of Scooping or Scavenging
 - Longer life than a plastic blade design
 - Susceptible to edge damage due to mishandling
- Polyurethane Trailing Edge Blades
 - Lower initial cost than metal trailing edge designs
 - Recommended for stepped stencil applications
 - Allows the use of different materials for specialized processes
 - Can vary paste deposit by initiating scooping
- Diamond Shaped Blades
 - Most commonly used for flux, ink, or epoxy printing
 - Four sided diamond shape allows for bi-directional printing
 - Printing edge always at optimal angle relative to stencil
 - Allows the use of different materials for specialized processes



Squeegee Blade: Handling & Storage

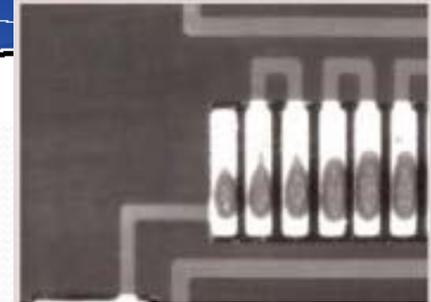
- Properly clean squeegee blades after use
- Store squeegee blades in the designated area after cleaning
 - Never store a squeegee blade until it is cleaned
 - Do not leave squeegee blades on benches, machines, the floor, etc. They should only be in the machine, being cleaned, or in the storage area.
- Always inspect squeegee blades for damage or wear prior to use. Never use damaged or worn squeegee blades.

Printing Equipment

- Modern sophisticated solder paste printing equipment offer a number of features for process automation to minimize process cycle time and
 - Accuracy, repeatability, and inspection capability to maximize process quality.
 - Vision System for alignment and post print inspection
 - Programmable Squeegee Head
 - Automatic Paste Dispenser
 - Automatic Stencil Wiper with Vacuum
 - Ease and Repeatability of Setup
 - Repeatable, Stable Board Support
 - Post Print Verification and SPC Data Collection



Vision System



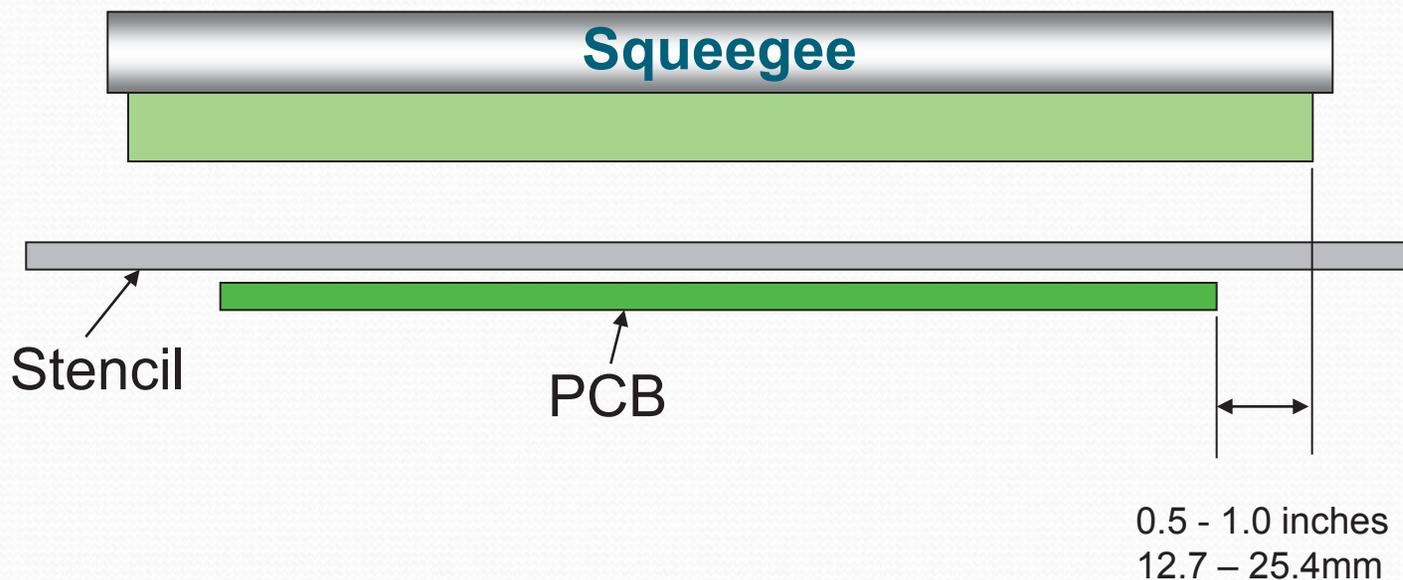
- Look Up / Look Down
 - “Looks at” fiducial marks on printed circuit board and fiducial marks on stencil
 - Gives machine ability to fine align each board to stencil.
- High Speed to enhance Cycle Time.
- High degree of repeatability (+/- 0.0005”)
- Standard mode and fine pitch mode
 - Standard Mode: System will look at stencil fiducials once and board fiducials for each board
 - Fine Pitch Mode: System will look at the stencil fiducials and board fiducials for each board

Printing Parameters

- Squeegee length
- Squeegee pressure
- Print force
- Squeegee speed
- Squeegee angle
- Snap-off vs contact printing
- Paste Kneading
- Stencil wiping frequency

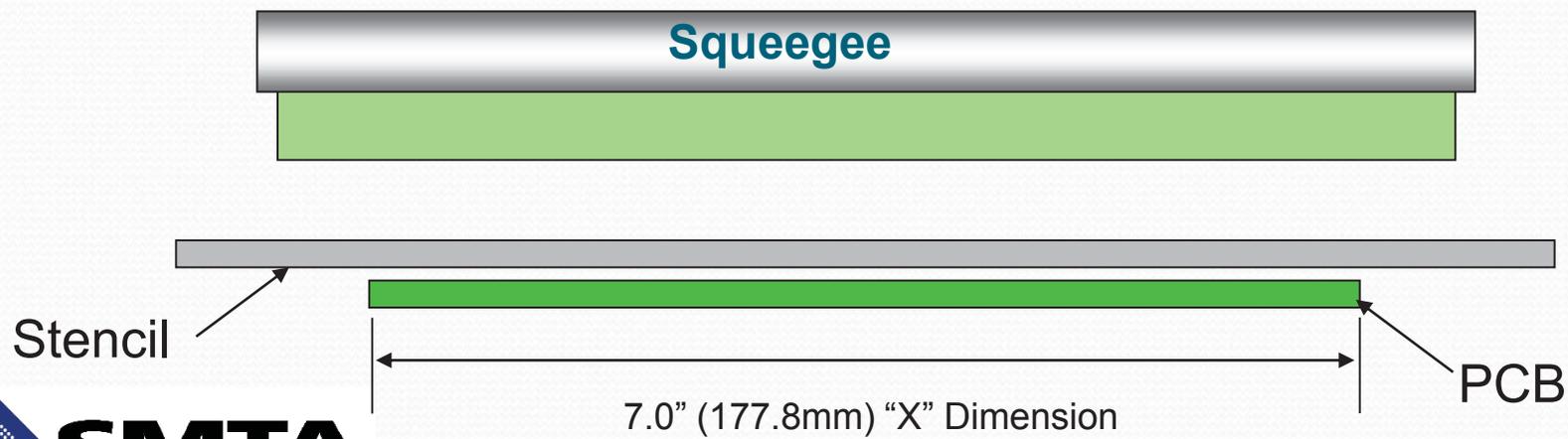
Squeegee Length

- Ideal Squeegee blade length should be 1 - 2 inches (25.4 – 50.8mm) longer than the PCB
 - Extending 0.5 to 1.0 inches (12.7 – 25.4mm) beyond each end of the PCB.



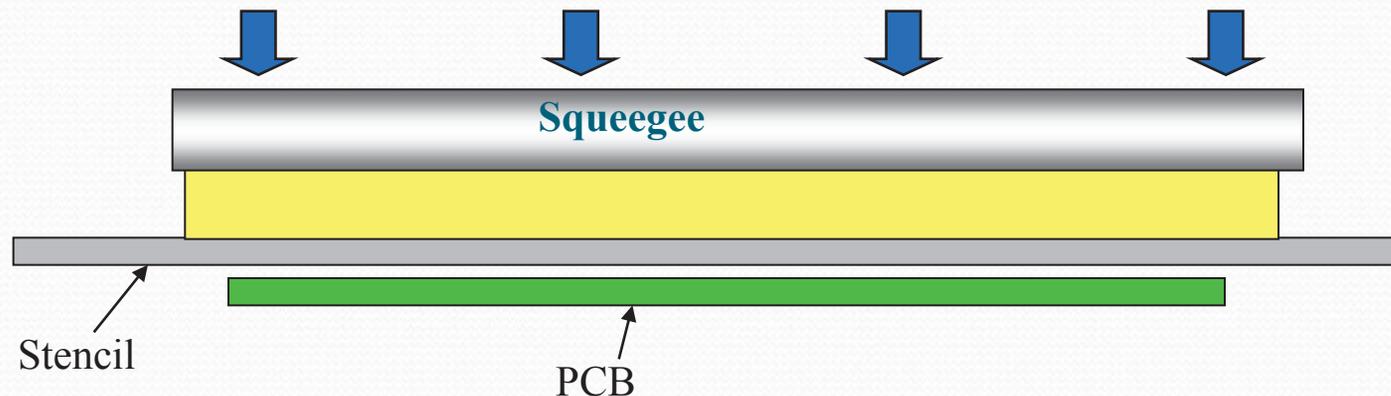
Squeegee Pressure

- Ideal squeegee pressure should be just enough to consistently wipe the topside of the stencil clean
- Approximately 1 - 1.5 lbs per inch (0.018kg – 0.026kg per mm) of the PCB “X” dimension is a good starting reference
 - Example, 7 inch PCB “X”:
 - (PCB “X” dim) * 1lb = 7lbs of force
 - (PCB “X” dim) * 0.018kg = 3.2kgs of force



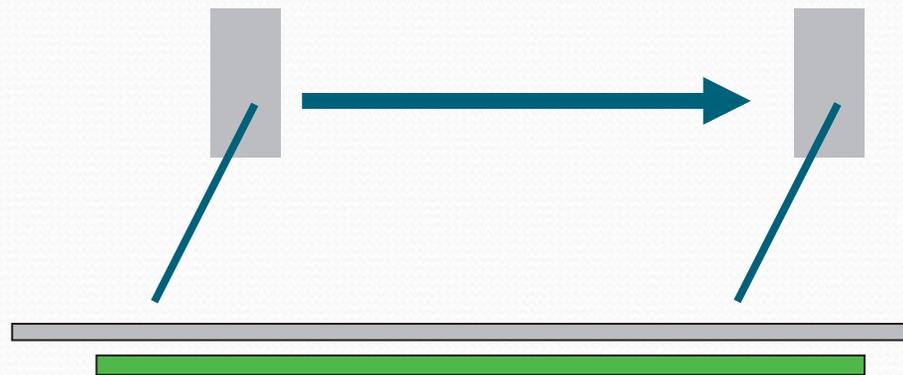
Print Force

- The print force is the amount of pressure applied by the squeegee to the stencil during the print process
 - You want to use just enough to get a clean stencil surface



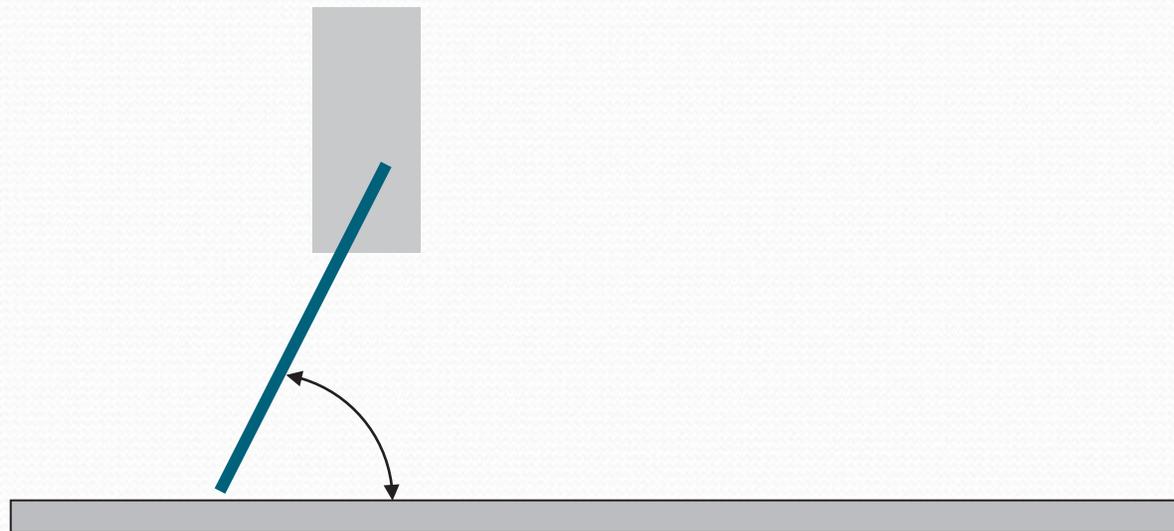
Squeegee Speed

- Speed depends upon paste type, blade type and type of components
 - Process' with finer pitch devices can have slower print speeds than large scale packaging, using the same paste and squeegee blades
 - Always refer to the solder paste specification for maximum print speed.



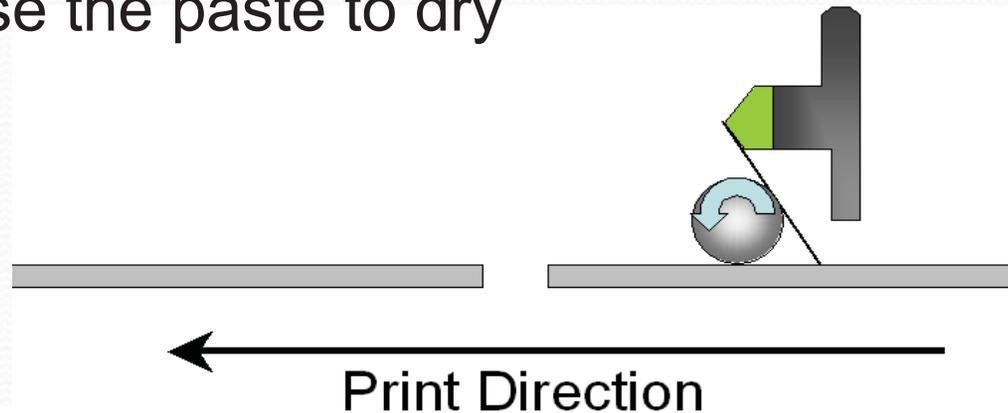
Squeegee Attack Angle

- Attack angle is the angle of the squeegee in respect to the stencil
 - MPM recommends 45 degrees at the solder paste specified print speed
 - Consult the machine vendor for recommendations



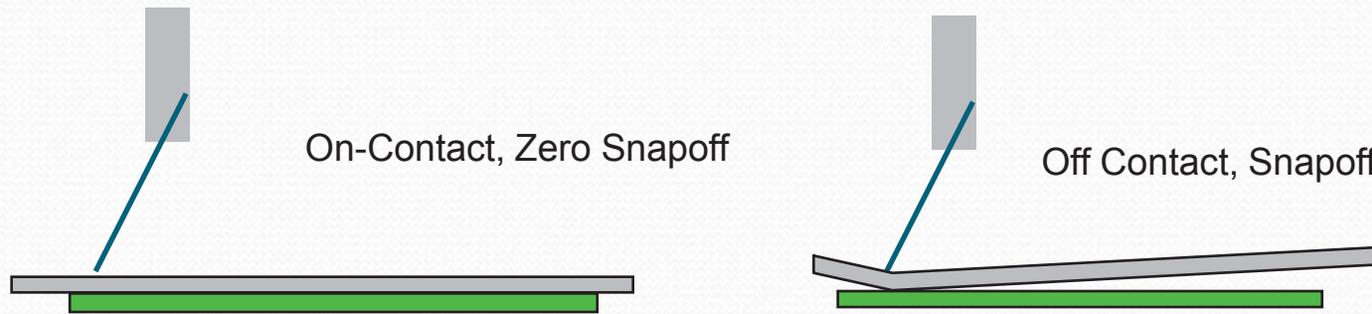
Paste Roll Diameter

- Keep the roll of paste about .5" to .6" (1.5 cm) diameter at all times
 - Automatic machine paste dispensers can help maintain consistent quantities
 - If an operator is responsible for paste replenishment insure that they are not applying too much paste
- Too much paste on the stencil will prohibit the paste rolling and cause the paste to dry



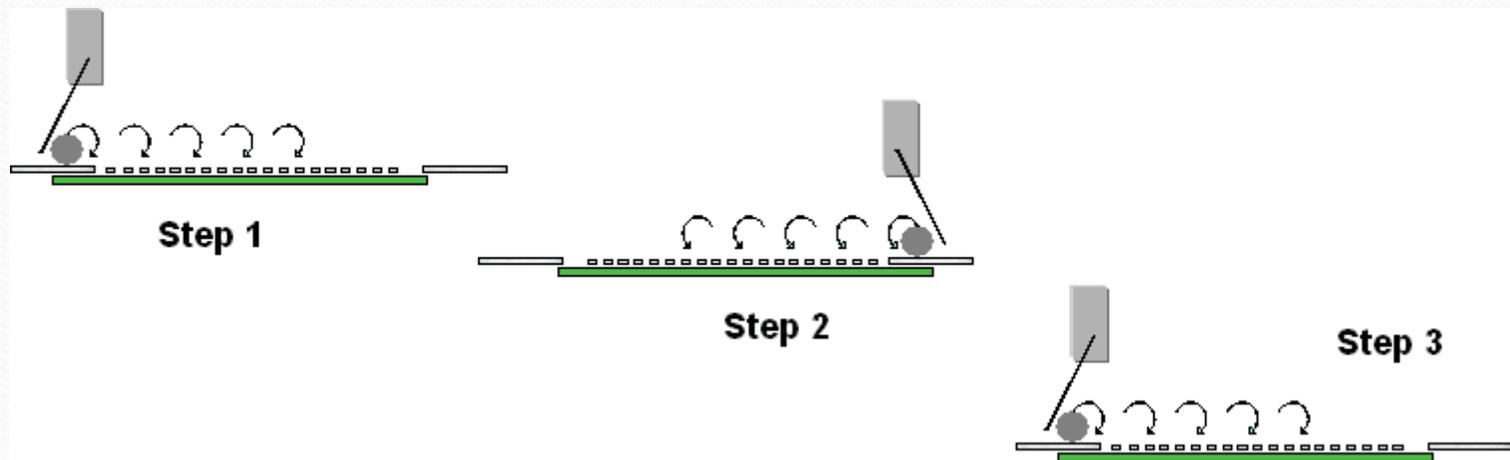
Snap-off Vs. On-Contact Printing

- After the stencil and PWB are aligned to each other, they are brought together for printing.
 - **Contact printing is recommended.**
 - Some process' may use a small gap (snapoff) between the stencil and PWB



Paste Kneading

- Some automatic printers will automatically mix the solder paste, after pauses, to ensure a good working viscosity on the next board print
 - This is commonly called 'kneading' the paste
 - Pause time before kneading is paste dependent

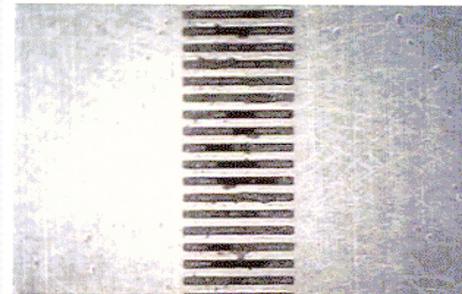
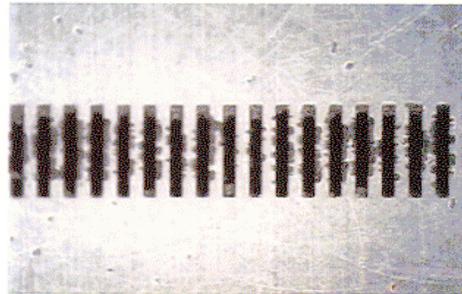


Enclosed Print Heads

- Advantages:
 - Protects solder paste from environment
 - Reduce wasted solder paste
 - Protects operator from solder paste
 - Keeps process clean
 - Increase printing speed (solder paste dependent)
 - Reduces process change over time
- Issue
 - Solder paste behavior in enclosed print heads
- Examples:
 - MPM Rheometric Pump Head
 - DEK ProFlow

Wiper

- First line of defense to eliminating defects
- Wiping the stencil eliminates “bleed-out” that can cause solder balls and bridging
- Vacuuming cleans out the apertures that can cause opens or insufficient deposits
- Solvent wiping can eliminate flux build-up that can effect release, lubricate the paste during vacuuming and wiping
- Frequency is a direct collation how well the board, stencil and paste are interacting within the print process.



Solvents

- Solvent chemistry has been focused on the application of automatic wipers in printers
- Delivery systems have been improved for a calculated and repeatable application
- A flammable liquid is defined as a material that has a flash point of 100 F
- Choose a material that gets close to this FP for good evaporative properties
- Check for residue (Mirror test) that can effect printing
- Always Start the wipe Profile with Solvent first – followed by a Vacuum then a Dry wipe

Determining Wiper Frequency

- Print 2-3 boards
- Jog vision system out to inspect stencil apertures.
- Repeat this until you see squeeze out.
- Subtract 1 - 2 prints from the determined number of prints for your wipe frequency.

Process Development and Control

- Every printing process is unique (different) using a different combination of inputs and process parameters. Only formal experimentation and studies can optimize any particular printing process
- Use formal Designed Experiments (DOE's) and statistical studies to identify and quantify critical operating parameters (speed, pressure, downstop, etc.).
- Use the same analytical approach in evaluating and selecting materials (solder paste, printed circuit boards, etc.) and tools (stencils, support tooling, squeegees, etc.).
- Once operating parameters are established, implement Statistical Process Control (SPC) and lock operators out of adjusting any parameters.

SPC vs. Inspection “Prevention vs. Reaction”

- Why use Statistical Process Control (SPC) instead of 100% inspection?
 - Inspection is expensive and not totally reliable. Even 100% inspection can miss about 20% or more of the defects. If inspection finds a defect, it has already cost money to produce and repair, and may cause more before it is corrected.
 - SPC monitors the process, not the product. Therefore, if a process is kept “in control”, the process will indicate when corrective action is required before the faults occur.

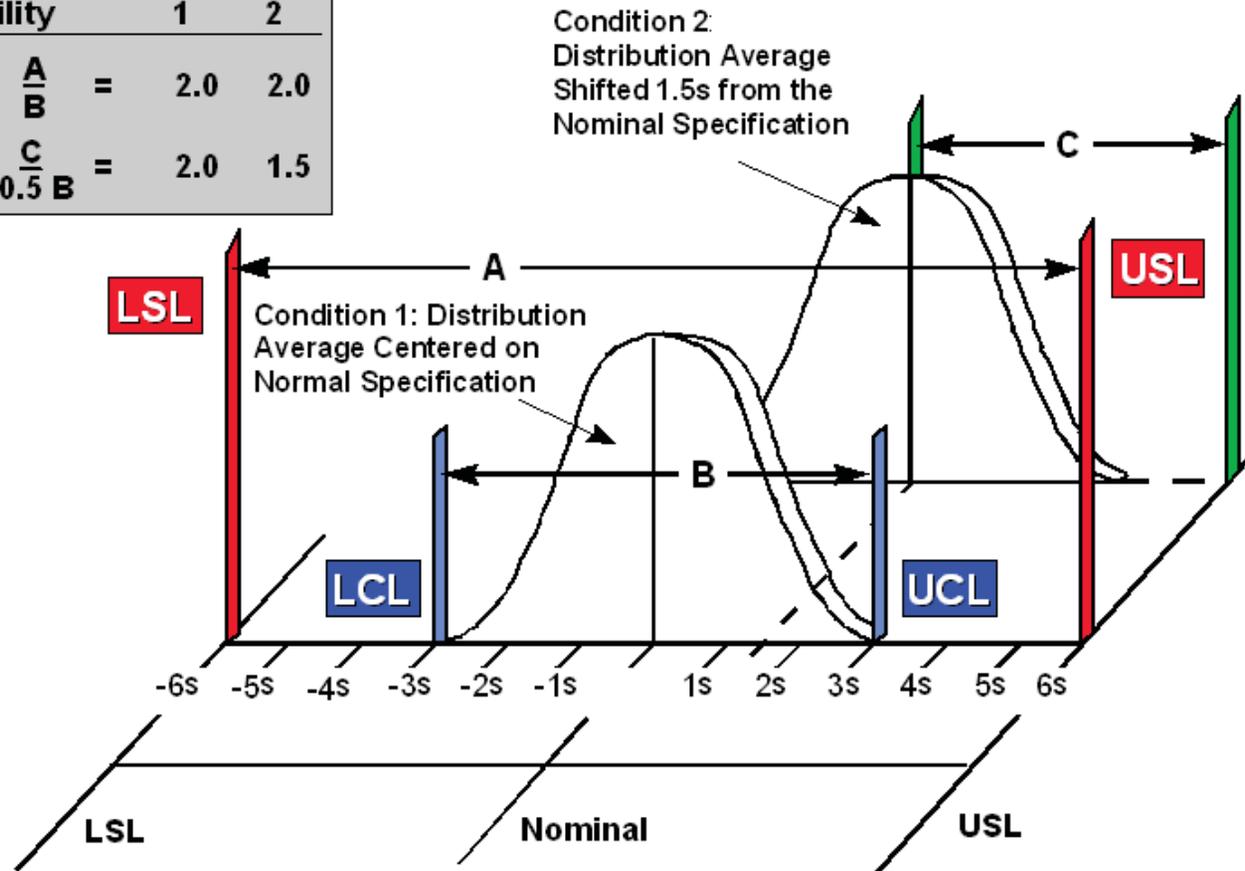
The Screen printing Process-Critical Success Factors

This chart identifies the key variables that must be quantified and controlled to produce the optimum printing process. All values are suggested values.

Solder paste Viscosity	By Paste Type	Viscometer	Xbar, R Control Charts
Stencil Aperture Width	Dependent upon stencil Mfgr	Micro view system	Xbar, R Control Charts
Facility temperature & Humidity	By paste type	Temp/Humidity gage	Xbar, R Control Charts
Time solder paste sits on stencil	10 to 15 min's (paste dependent)	Equipment's paste knead or double print next board	Equipment's process software
Stencil wash: IPA-%flux on board	From supplier	Omega meter	Process log
Stencil wash: Saponifier-Tritation	From supplier	Titration kit from supplier	Process log
Stencil quality (Dents, cleanliness, wear & tear)	Not applicable	Visual inspection	Process log & audits
Support tooling (Placement, cleanliness, wear & tear)	Not applicable	Visual inspection	Process log & audits
OUTPUT CHARACTERISTICS			
Produced by machine or operation			
Solder paste height	8 mil+/_ 2 mils 6 mils -1, +2 mils	3D SPI system	Xbar, R Control Charts
Solder paste coverage	+/- 10% Target	Equipment 2D system	Equipment's 2D error detection system
Process defects & errors	Not applicable	Factory reporting system, trend charts arts	U-Control chart by process, Pereto charts
Solder paste volume	+/- 30% Target	3D SPI system	Xbar, R Control Charts
PREVENTIVE MAINTENANCE CHARACTERISTICS			
Height of support nest plate to base	Screen printer mfgr	Height gage	PM logs & audits
Stencil holder to base parallelism	Screen printer mfgr	Height gage	PM logs & audits
Force applied to right/left side of print head	Screen printer mfgr	Calib to 90 Degrees	PM logs & audits
Printer cleanliness (nest, stencil, base, height sensor, etc.)	Not applicable	Visual inspection	PM logs & audits
Squeegee blade (dents and angle)	Not applicable	Visual inspection	Process log & audits
OPERATOR DEPENDENT CHARACTERISTICS			
Frequency of stencil wiping	Every 6 to 8 boards (paste dependent)	Not applicable	Process audits or automatic wiping
solder paste handling from regrigeration1	Ambient for 24 hrs	Not applicable	Process log, date/tie stamp paste container
OTHER CHARACTERISTICS			
Operator certification program	Not applicable	Not applicable	Written and/or observation tests

Process Capability

Capability	Condition	
	1	2
$C_p = \frac{A}{B} =$	2.0	2.0
$C_{pk} = \frac{C}{0.5 B} =$	2.0	1.5



Process Capability Cp and Cpk

Printing Environment

- Adequate environmental conditions are a necessity for good printing (refer to solder paste specs).
 - Temperature
 - **Maintain the operating condition of the paste (typically 68 to 78 F, 20 to 25.6 C).**
 - **Operating outside of these settings may cause slumping or excessive drying out of the paste.**
 - Humidity
 - **Maintain approximately 50% (40% to 60%)**
 - **Less than 50% will result in the paste drying out (the volatiles will evaporate).**
 - **Greater than 50% will cause the paste to absorb moisture in the air and cause outgassing, resulting in voids and solder balls.**

Printing Environment

- Adequate environmental conditions are a necessity for good printing (refer to solder paste specs).
 - Air Flow
 - **Keep air flow in the print area to a minimum. Increased air flow will drive off the volatiles in the paste resulting in poor printing and reflow.**
 - Cleanliness
 - **Keep the printer, stencils & support tooling clean. Solder paste buildup in the worknest or other moving areas will decrease the effectiveness of the board support and the repeatability of the machine.**
 - **Keep the vision system clean. If solder paste or other material get on the camera, it will not “see” the fiducial marks.**

KEEP THE PROCESS CLEAN

Some Helpful Hints For Good Printing

- Insure paste is within shelf-life limits
- Store in a cool, dry place normally 5-10°C
- Examine tubes and syringes for flux separation prior to use, discard if separation is noted
- Print initially over “Mylar” film several times to check co-planarity
- Insure paste is rolling over stencil, not dragging
- Insure that misprinted boards are thoroughly cleaned prior to re-use, wiping is not sufficient

Some Helpful Hints For Good Printing

- Examine several boards under 10 -20X magnification for definition, smearing or bleeding
- Note squeegee speeds, lower speeds (typically 1-2"/sec.) generally give better paste roll and print definition
- If top stencil smearing occurs try increasing squeegee pressure (typical range of 0.5 - 2 kg/in of squeegee), bleeding of paste on underside, decrease squeegee pressure

Some Helpful Hints For Good Printing

- An aperture to pad ratio of less than 1 is recommended, reducing bridging and solder balling issues
- If printing is stopped for more than 1 hour, cover paste on stencil with plastic wrap, remove dry paste from squeegee and stencil before resuming printing
- Avoid the re-use of old paste, drying impacts print quality and increases defect rates

Post Print Verification

- To ensure a quality print, Post Print Inspection can catch defects such as solder bridges, no solder, excessive solder, misprinted solder, and insufficient paste heights.
 - Two Dimensional (2D)
 - Measure solder paste PWB pad coverage
 - Three Dimensional (3D)
 - Measures solder paste volume
 - Keep process In control
 - Prevent defects from “escaping” the printing process
- BGA's may require 100% inspection
 - Solder joints cannot be seen by inspectors

Post Print Verification

- Post print inspections can be part of printing equipment or a separate machine
 - There are many post print inspection systems available
- The primary issues in determining the correct post print inspection system for your process are:
 - Two Dimensional (2D) or Three Dimensional (3D)?
 - Cycle time
 - How much do you want to inspect in the allotted time?
 - What you want to inspect
 - Can the inspection system “see” what you want to inspect?
 - Data collection required
 - How much and what data do you want supplied by the inspections system?

Defect Prevention.....but first the Objectives of Solder Paste Printing

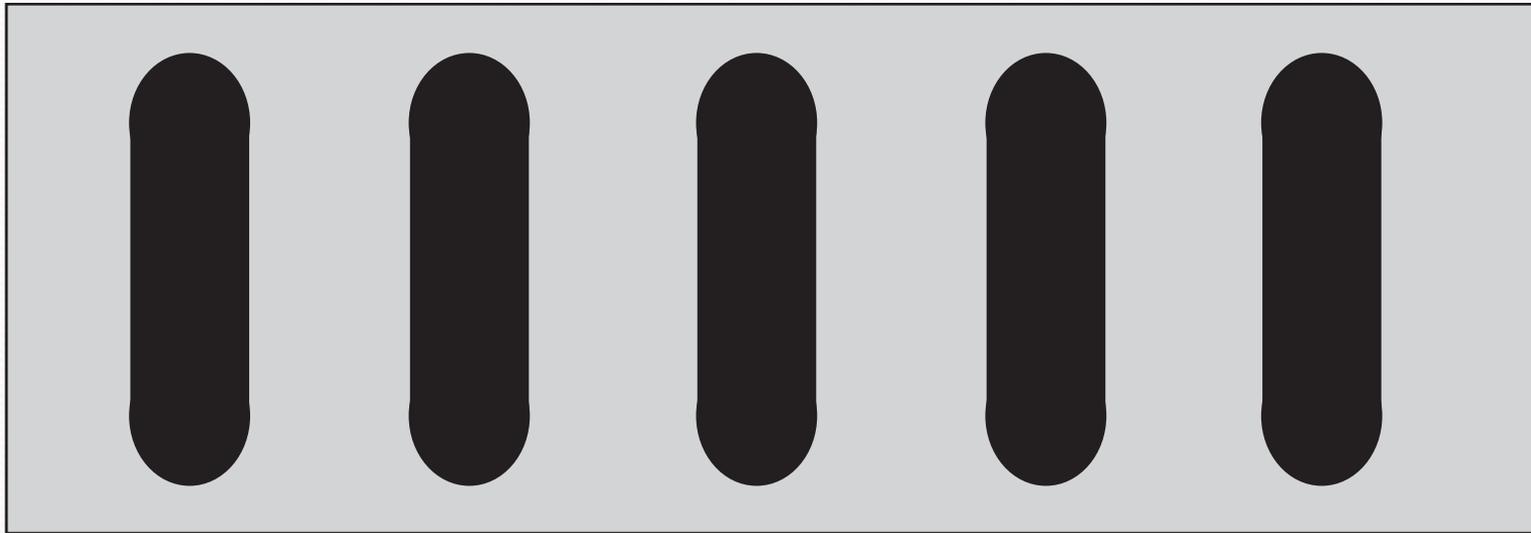
- Deposit appropriate amount of solder paste consistently in the proper location.
- Control volume of deposit.
- Avoid bridges.
- Perform printing in acceptable time.

Causes of Wet Bridging

- Improper stencil design
- Inadequate or infrequent underside stencil wiping
- Excess solder volume
- Poor gasketing of stencil to pad

These conditions lead to “squeeze out” of solder paste onto the underside of the stencil which causes bridging.

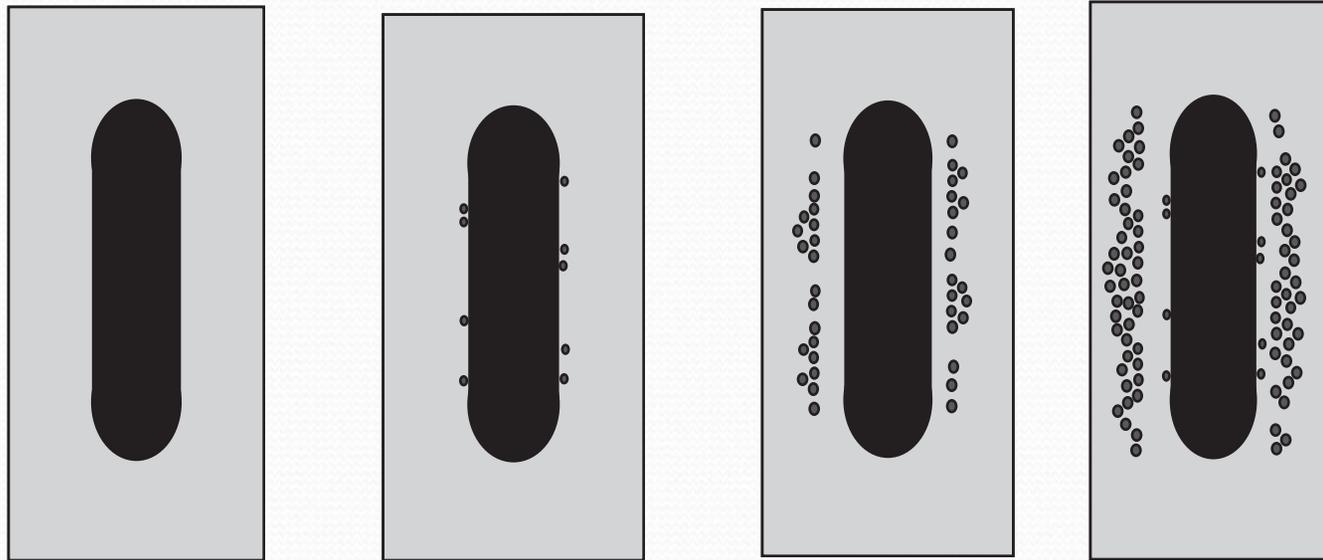
Squeeze Out



Board side of the stencil

Automatic stencil wiping can remove squeeze out before bridging occurs.

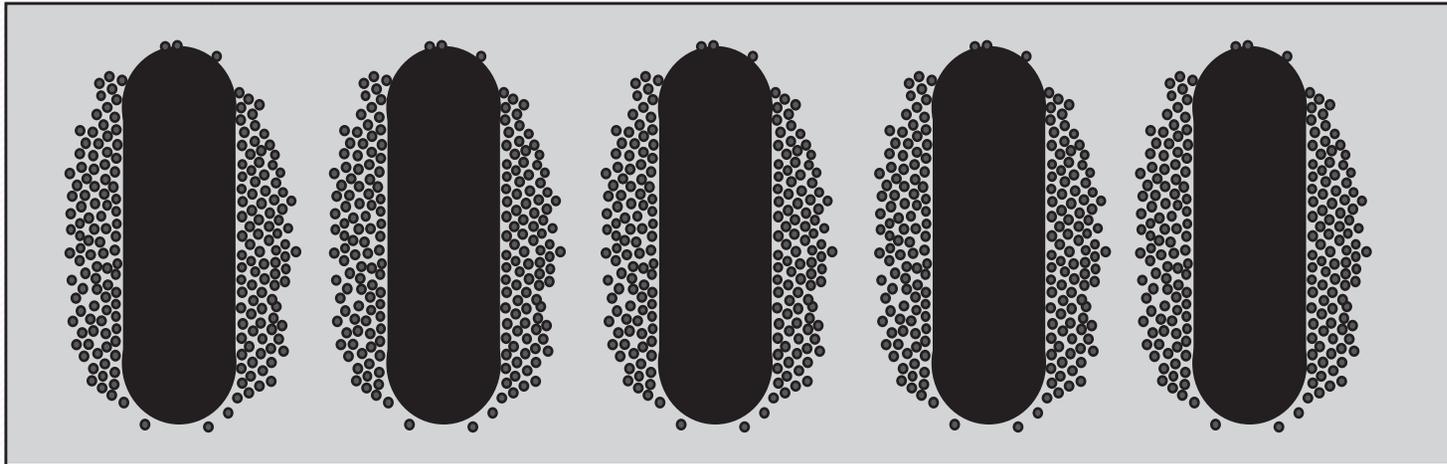
Squeeze Out



Board side of the stencil

Squeeze out refers to solder particles that have “squeezed out” onto the underside of the stencil during printing.

Squeeze Out



Board side of the stencil

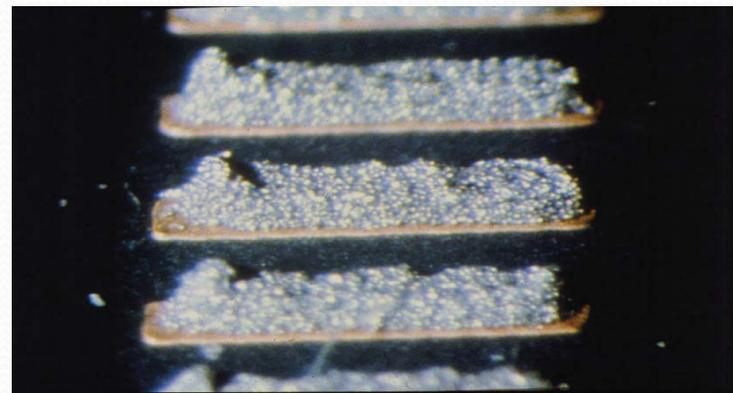
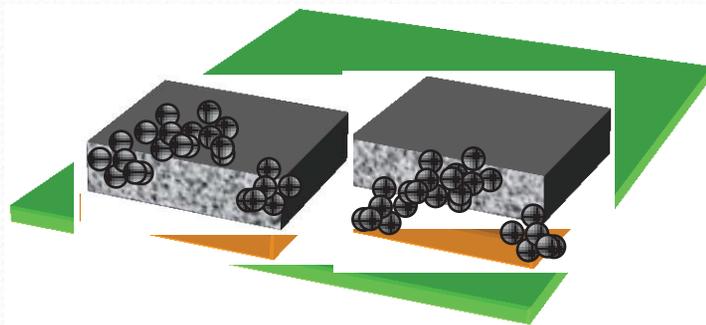
As Squeeze out starts closing the gap between fine pitch apertures, bridging may occur.

Causes of Wet Bridging

- The speed at which squeeze out accumulates is dependent on
 - Size of aperture relative to pad
 - Gasketing of stencil to pad

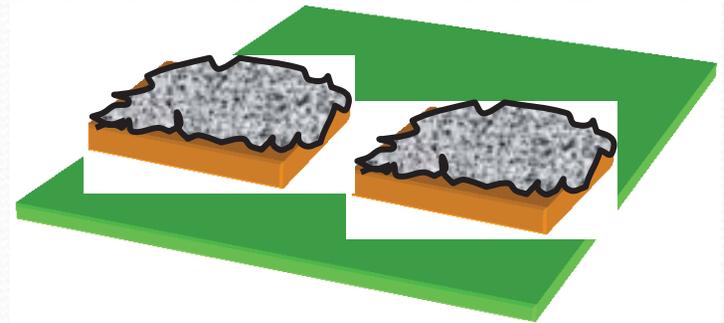
Peaking / 'Dog Ears'

- Stencil snapoff too large.
- Temperature / Humidity too high.
- Excess solder paste clinging to bottom of stencil / Clean.
- Excess squeegee pressure



Insufficient Depositions

- Major Causes
 - Poor aperture design
 - Improper print parameters
 - Paste in poor condition
 - Pauses in printing cycle
 - Improperly cleaned apertures



Insufficient Depositions

Stencil design for the prevention of insufficient depositions

- Aspect Ratio of 1.5 or greater and/or Area Ratio of .6 or greater
- Smooth inner wall / trapezoidal shape
 - Laser cut or electroformed for fine pitch
- Electropolished chemically etched stencils

Insufficient Depositions

Critical Print Parameters for prevention of Insufficient Depositions

- Squeegee pressure high enough to wipe the stencil clean (but not much higher)
- Polyurethane blades (too soft)
- Adequate board support
- Appropriate squeegee speed (paste must roll)
- Worn or damaged stencil

Insufficient Depositions

Solder Paste Management for prevention of Insufficient Depositions

- Do not reuse paste if possible
- Do not exceed paste stencil life
- Always use the oldest solder paste first
- Paste should be thoroughly thawed and mixed before using (thaw at room temperature)
- Clean stencil apertures after pauses in printing

Consistency in Paste Volume

Major Concerns

- Squeegee blade type (Polyurethane)
- Aperture geometry
- Board/Stencil separation
- Board orientation
- Board support

Misaligned Print

Major Concerns

- Vision system not “seeing” fiducial marks
 - Solder paste/material on cameras
 - Objects too close to fiducial marks (uniqueness)
- Not using half etched fiducials
- No fiducial marks on board/stencil
 - Using pads on board @ apertures on stencil for fiducials
- Insufficient support/snugging tooling
 - Board moving during printing
 - Conveyor not adjusted correctly
- Printed circuit board stretch
- Stencil stretch

Terms & Definitions

- Alignment - (also *registration*) - The process of registering two targets to each other, i.e., the application or deposit of the solder paste is directly on top of the lands.
- Alloy - a substance (solder) composed of two or more metals (tin and lead) fused together
- Bridging - the result of solder paste which has bled, slumped or misprinted to form an unwanted connection between two or more pads or lands.
- Coplanarity - the measurement of leads in respect to each other, in the same plane.
- Delta T- Temperature differential between points
- Durometer - a measure of hardness or material density on polyurethane squeegee blades (i.e., 70, 80, 90). The higher the durometer number, the harder (more dense) the material

Terms & Definitions

- Downstop - the downward distance (.075) the squeegee is allowed to travel beyond the board surface during the print stroke
- Eutectic Point - the ratio of tin (Sn) and lead (Pb) that produces the lowest temperature that an alloy (solder) can exist as a liquid. It is also the temperature that the alloy changes directly from a liquid to a solid
- Fiducial - the [from Latin. fiducia, from fido, to trust.] accepted as a fixed basis of reference or comparison. Thus, fiducial point, fiducial mark.
- Fillet - a concave surface formed in the reflowed solder between the component lead the pad (land)
- Flux - a component of the solder paste which provides a chemical cleaning action (removal of oxidation) of the metal (pads/lands) during reflow operations. Lowers the surface tension of the solder during reflow operations and helps promote the formation of good solder fillets.

Terms & Definitions

- Inert (non-reactive) - In reflow, nitrogen is inert as it displaces air from the process
- Land - (or *pad*) the metal conductive surface on the printed wiring board on to which the component lead is attached or connected with solder.
- Lead - the leg (or arm) of the component that attached to the printed wiring board's land (or pad) during solder reflow operations
- PCB (or PWB) - printed circuit board
- Pb - the designation for lead
- PWB (or PCB) - printed wiring board

Terms & Definitions

- Reflow - a process in which the solder paste is heated to the point where it melts and where *wetting*, *wicking* and surface tension allow the solder to flow thereby forming a soldered connection between the component lead the PWB's pad (*land*)
- Registration (see alignment)
- Solder mask - a protective coating (usually a epoxy-based resin or a UV curable photopolymer) that is applied to both sides of the printed wiring board (except for the pads/lands) prior to component insertion and wave solder. Prevents etch runs (circuit lines) from bridging and shorting during wave solder operations.

Terms & Definitions

- Snap-off - the action of the stencil during off-contact screen printing. During a screen printing stroke, the down-force of the squeegee deflects the stencil down to the surface of the PWB, and as the squeegee travels during the print stroke, the stencil begins to lift (or snap) from the surface of the PWB.
- Sn - the designation for tin
- Spikes - peaks of solder paste deposited onto the pad caused from incorrect machine set-up.
- Solder - an alloy usually consisting of tin and lead fused together as a result of extreme heat.
- SMT - Surface Mount Technology

Terms & Definitions

- TAL - Time Above Liquidus or melting point, typically 30~60 seconds for 63sn/37pb solder paste
- Thru-hole (or plated-thru hole) - a drilled hole in the printed wiring board that electrically connects its conductive layers
- Viscosity - the density or thickness of the solder paste. Has a direct affect on solder paste deposition during screen printing. Measured in centipoise. The higher centipoise number, the thicker the paste.
- Wetting - the formation of a relatively uniform, unbroken and adherent film of solder to the PWB.
- Wicking - The action that causes molten solder to move from one area to another by wetting the surface of solderable metal