Stencil Printing for Nerds

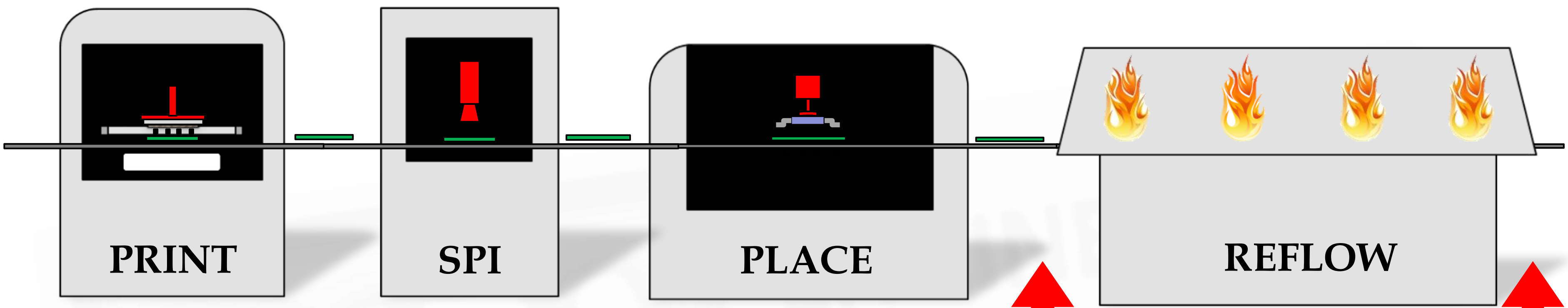
Chrys Shea

SHEA ENGINEERING SERVICES

Communicating Expertise

SHEA ENGINEERING SERVICES

Basic SMT Assembly Line



Deposits solder paste onto the PCB

Inspects solder paste to prevent defects

Places electronic components into "wet" solder paste deposits

Thermal process:

- *Activates fluxes*
- *Melts solder*
- *Makes solder joints*

Optional: Automated Optical Inspection (AOI) machine, which can identify placement or soldering defects, may be inserted before or after the reflow operation.

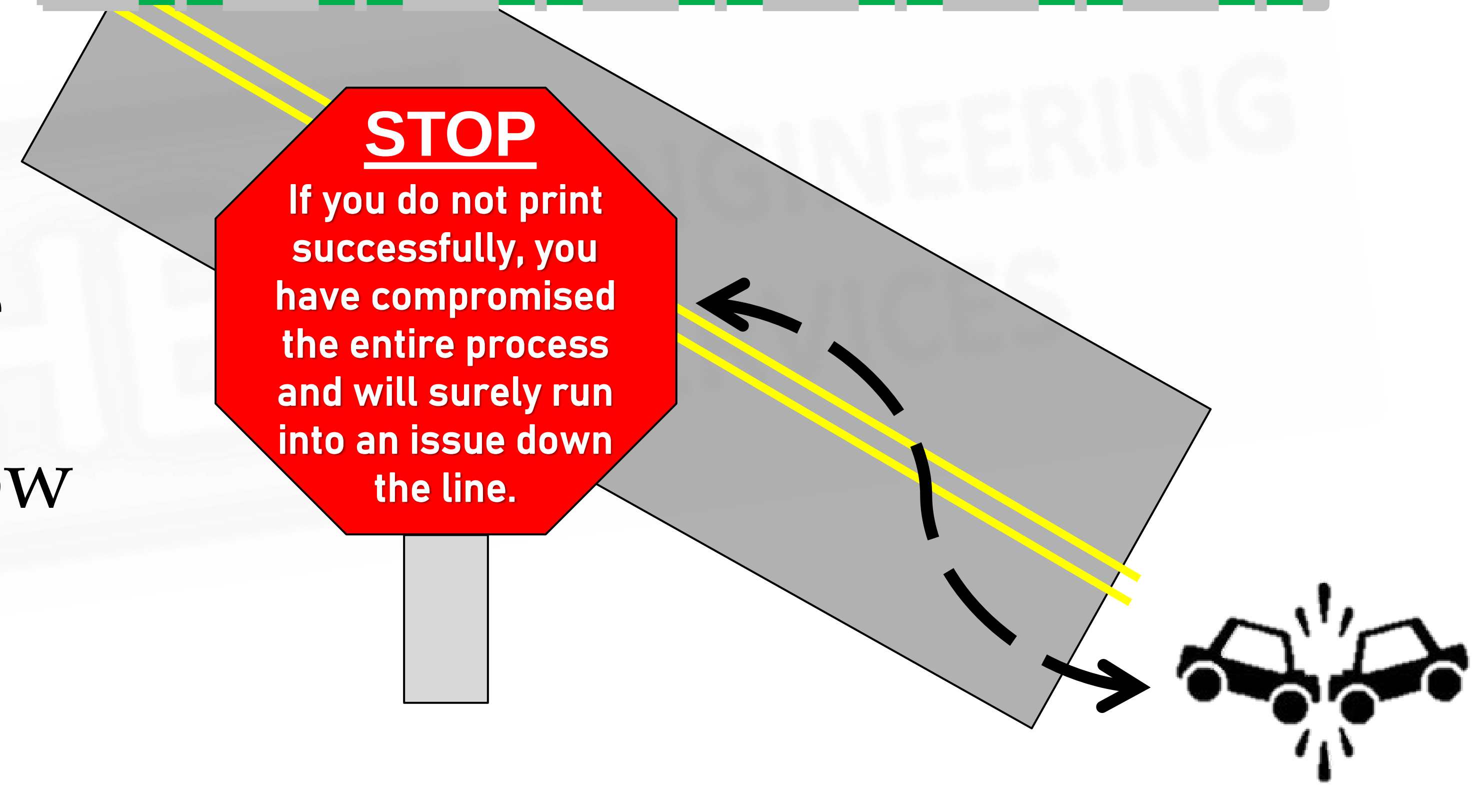
SHEA ENGINEERING SERVICES

Surface Mount Technology:

The Manufacturing Process

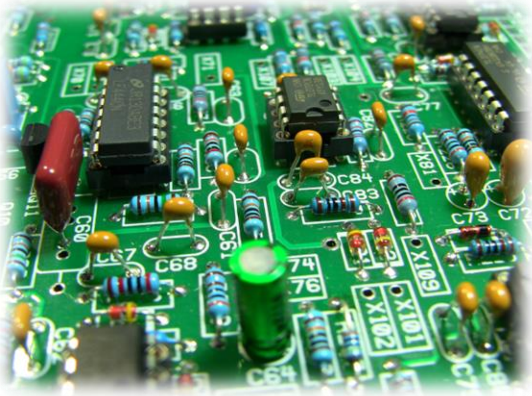
1. Print
2. SPI
3. Place
4. Reflow

Printing Solder Paste onto the Circuit Board is the first step in Surface Mount Technology Manufacturing.
It is where profits are made or lost in the SMT process.



SHEA ENGINEERING SERVICES

From T-Shirts to Circuit Boards



SHEA ENGINEERING SERVICES

What Do We Mean By Printing?

Depositing a design reduced from a screen or stencil onto a surface

A blade is pulled across the surface, pressing the print media through the stencil on the surface.

Image Source: University of Wisconsin Milwaukee
https://uwm.edu/studentinvolvement/event/screen-printing-on-t-shirts/2020-04-09/

The stencil and newly printed surface are then separated and *voila*!

Like Your Favorite T-Shirts!

SHEA ENGINEERING SERVICES

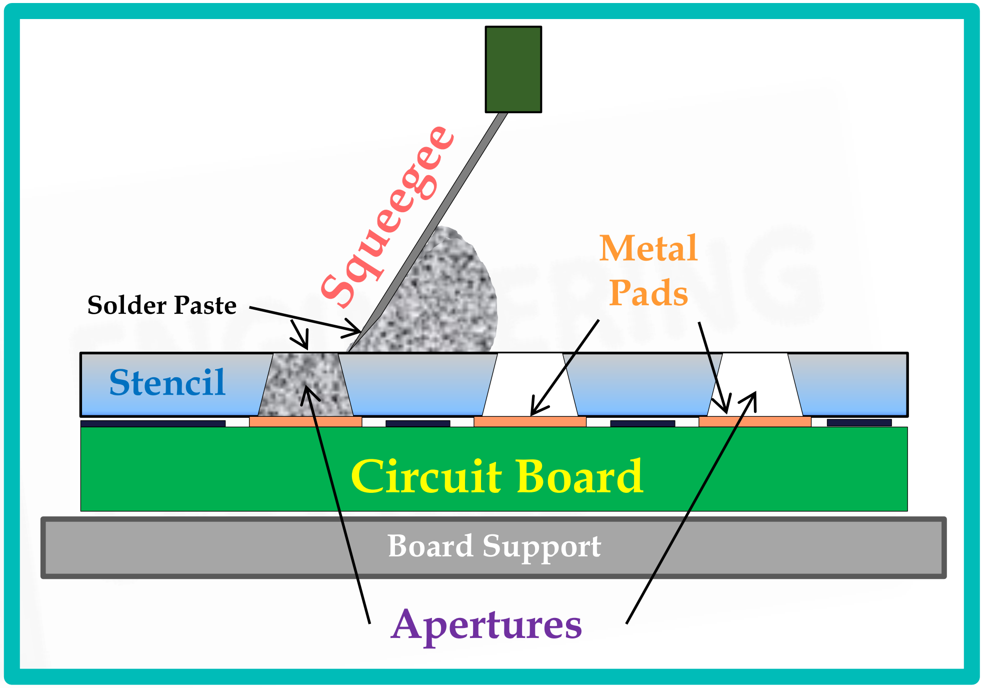
Printing Circuit Boards (Not T-Shirts)

Solder paste is printed onto a Circuit Board using a printer in order to create an *electrical and mechanical connection* between the board and the components placed on it.

Board Support is placed in the printer to prevent the circuit board from shifting or bending.

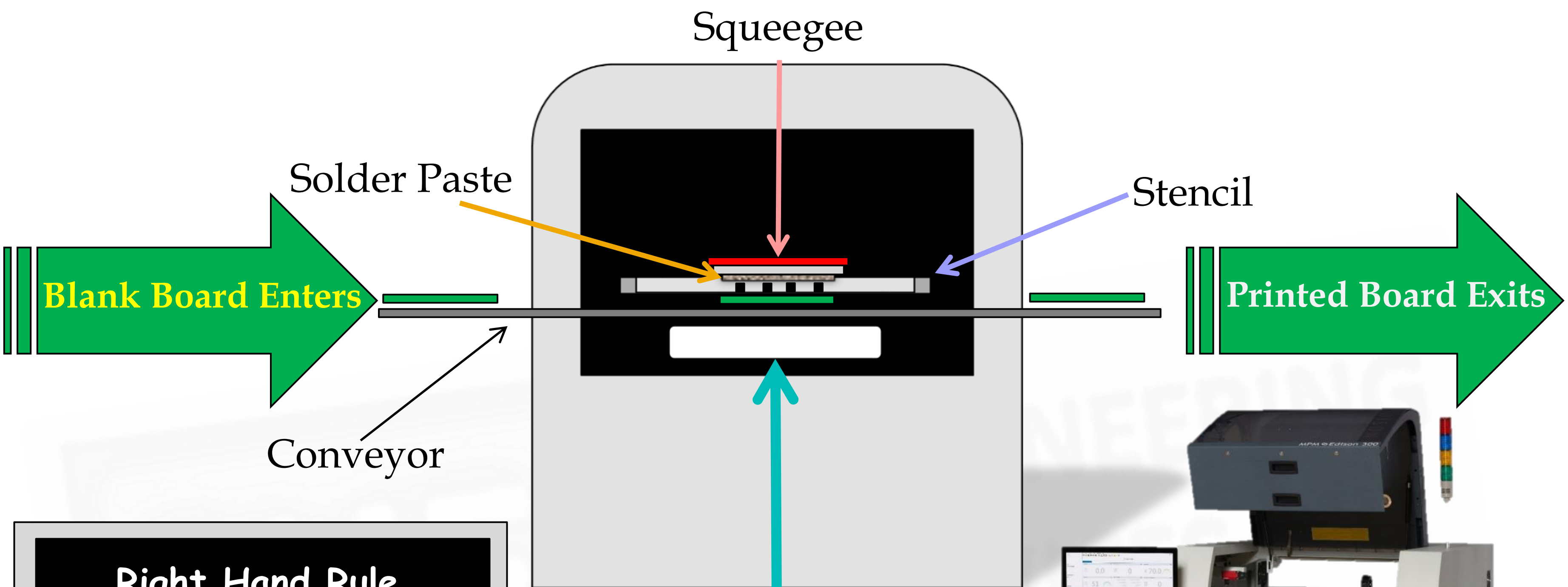
After the circuit board moves into the printer, it is raised so that the metal pads of the circuit board align with the apertures of the Stencil to ensure proper gasketing.

The squeegee then *rolls* the solder paste from front to rear or vice versa, printing the paste onto the circuit board through the stencil apertures. The board lowers and exits the machine and the under side of the stencil is periodically cleaned.



The goal of printing is to deposit the proper amount of solder paste at the proper location in the proper shape.

Not Like Your Office Printer



Board Support
Or
Under Stencil Cleaning
System

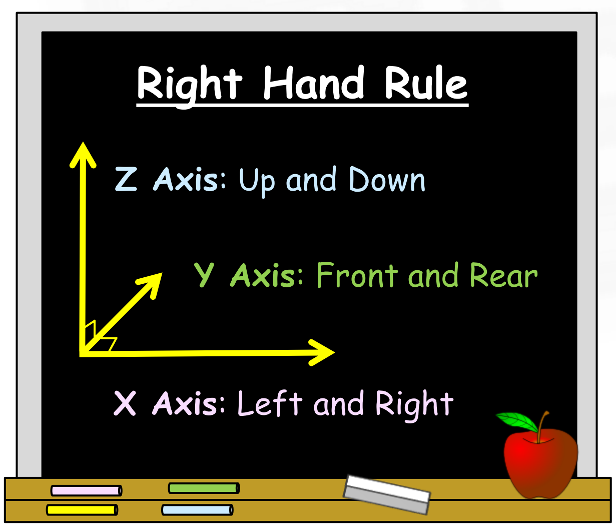
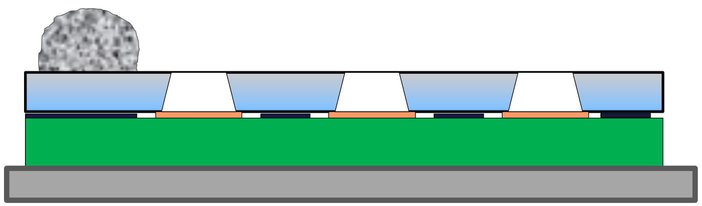


Image Courtesy of ITW EAE

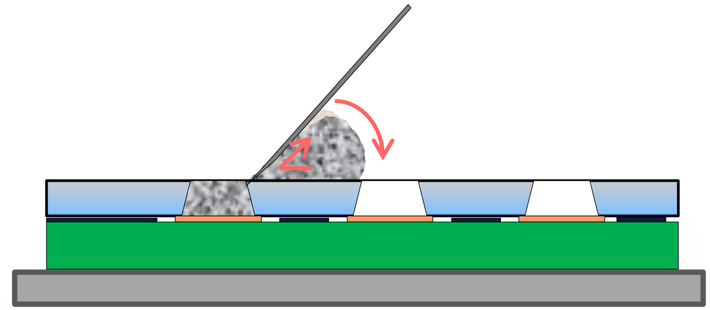
SHEA ENGINEERING SERVICES

What's Going On Inside There?

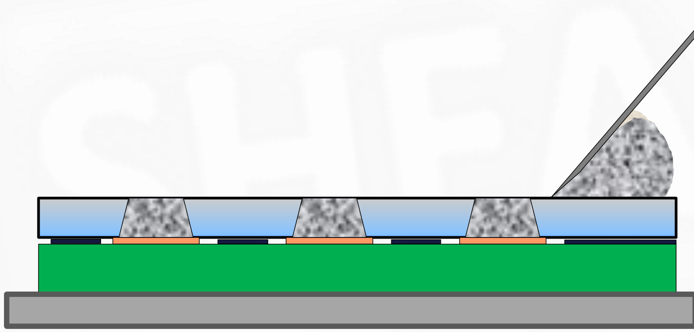
Stages of The Print Process



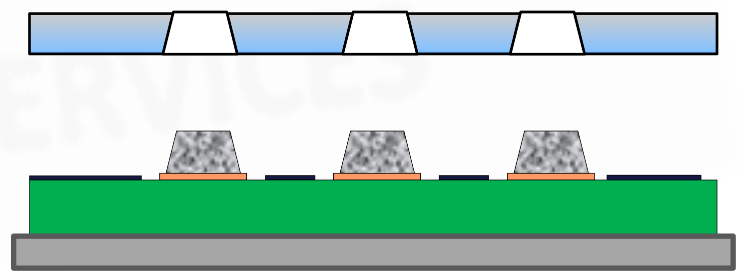
1) Alignment, gasketing of apertures to pads



2) Squeegee motion thins paste so it flows into apertures



3) Paste recovers; stiffens up



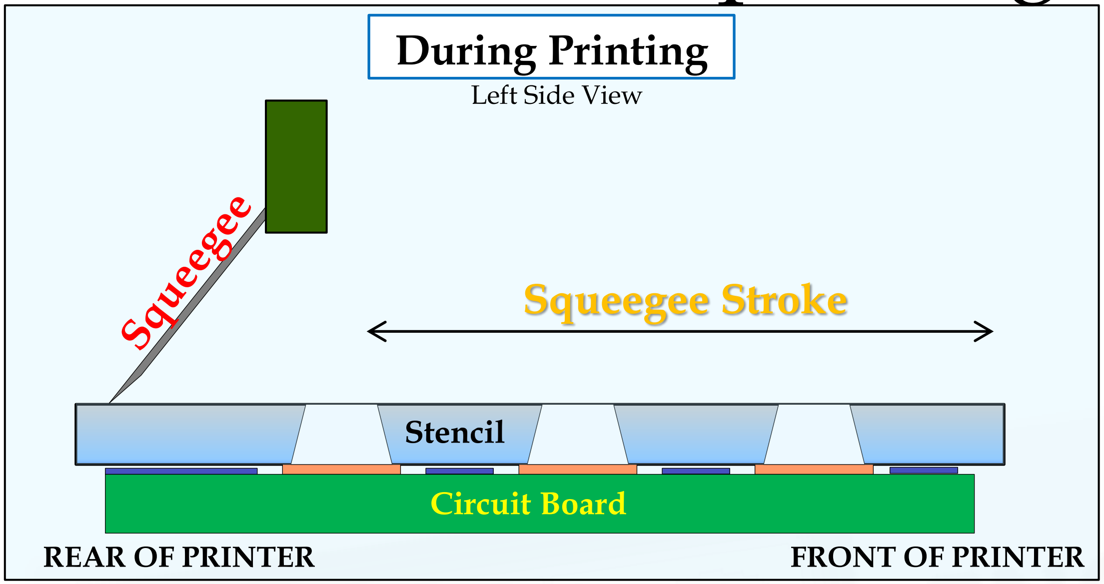
4) PCB (printed circuit board) separates from stencil; paste deposits release from stencil onto pads

SHEA ENGINEERING SERVICES

Talk Printing To Me

SHEA ENGINEERING SERVICES

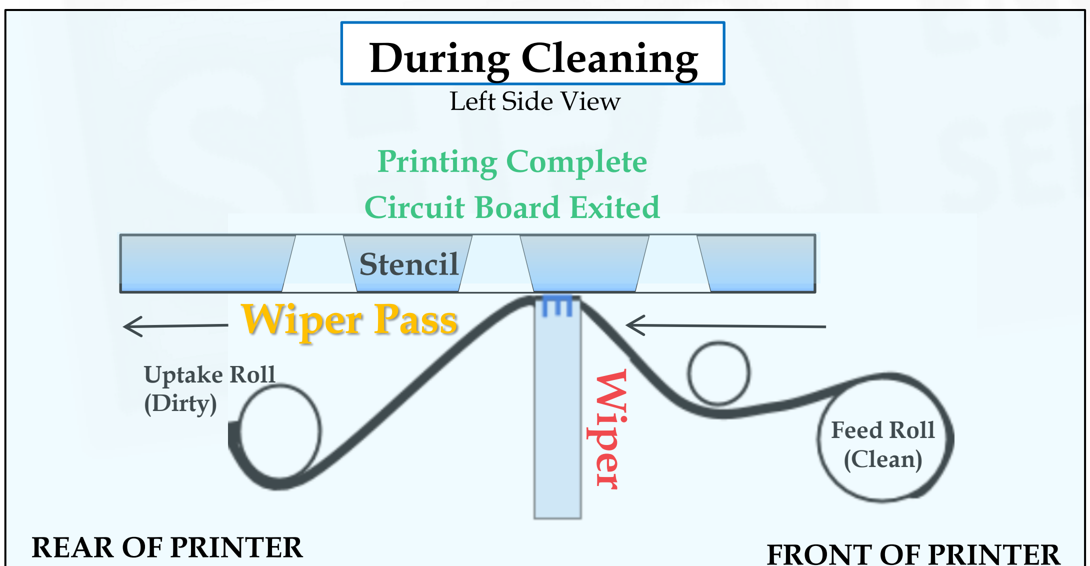
Basic Print/Wipe Language



Stencil Printing

- **The Squeegee moves across the top of the stencil in** strokes to deposit solder paste onto the circuit board
- Strokes may be performed Front to Rear and Rear to Front
- **Start with Rear to Front** so that you can see the paste roll

After printing, the circuit board exits the printer.



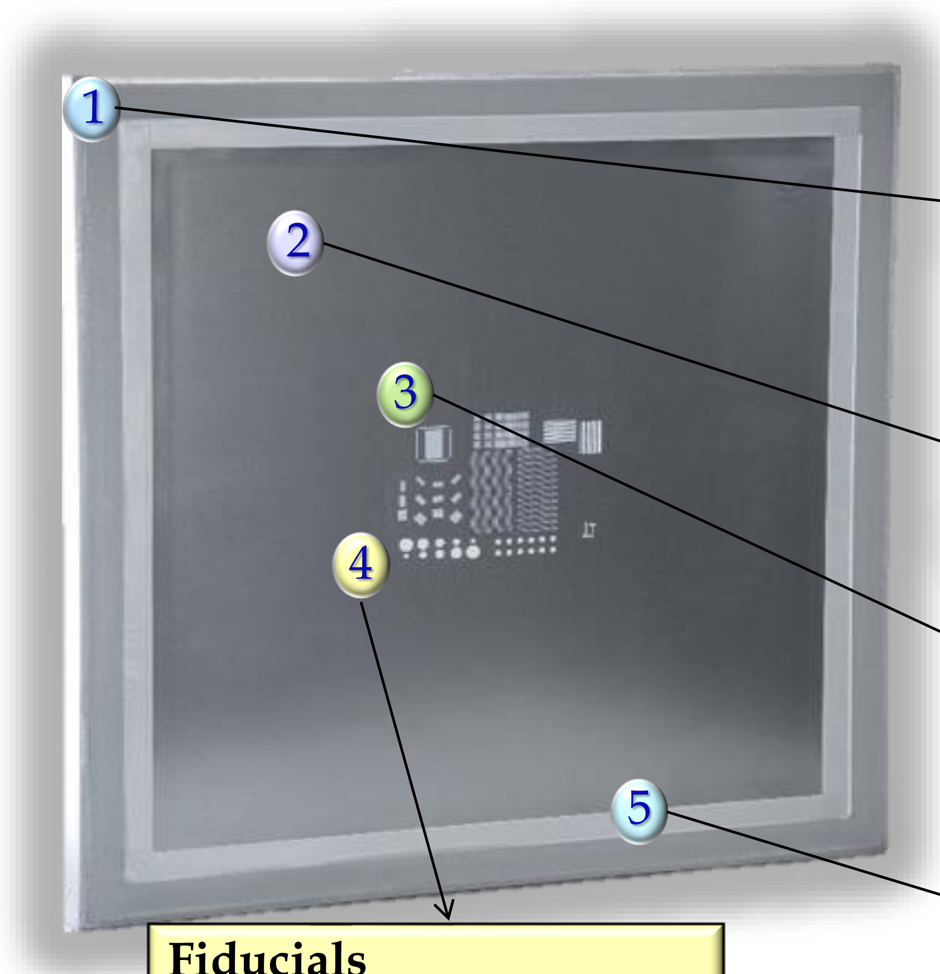
Under Stencil Cleaning

- **The Wiper moves under the stencil in** passes to remove residual solder paste from the side of the stencil that contacts the PCB.

Depending on the wiper system:

- Passes may be performed Front-to-Rear or Rear-to-Front
  - The order of wipe passes is known as the *Wipe Sequence*
- A single pass may perform a combined wet/dry wipe or vacuum function

Stencil Terminology



Frame
- Tube ~ 1.5in thick
- SpaceSaver ~ 0.5in thick
- ➢ "frameless" stencils utilize a common frame with interchangeable foils to save storage space

Foil
- Thickness determines Area Ratio
- Typically 4 or 5 mils

Apertures
- Laser cut preferred over E-Formed
- Laser cut provides better size, thickness, and location accuracy

Mesh
- Attaches foil to frame
- Sets the tension of the foil
- ➢ "frameless" stencils don't use mesh – they attach directly to the frame

Fiducials
- Small marks in the stencil which allow the printer to identify the stencil's position

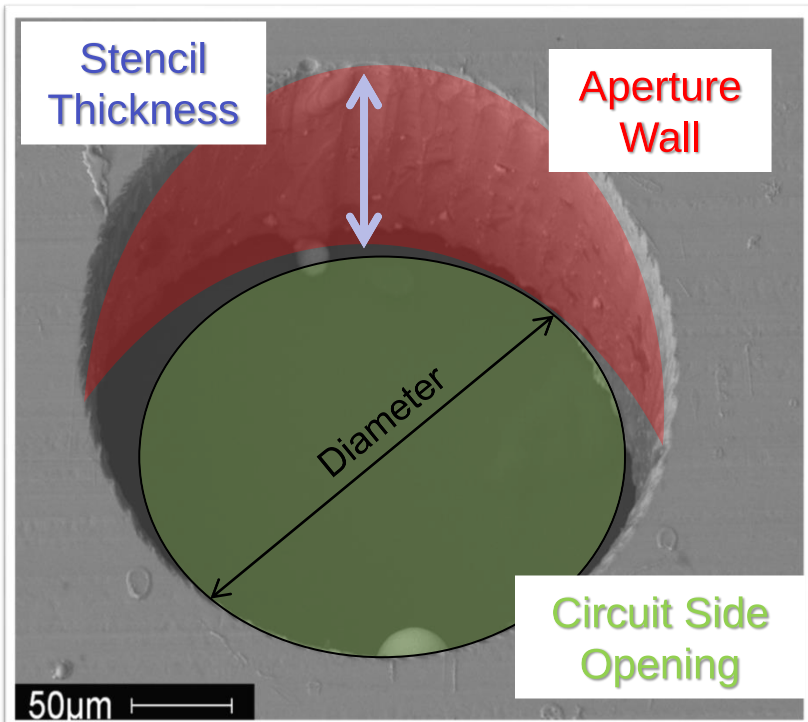
Image Courtesy of Henkel

SHEA ENGINEERING SERVICES

Area Ratio (AR)

Helps predict how much solder paste will be released from the aperture.

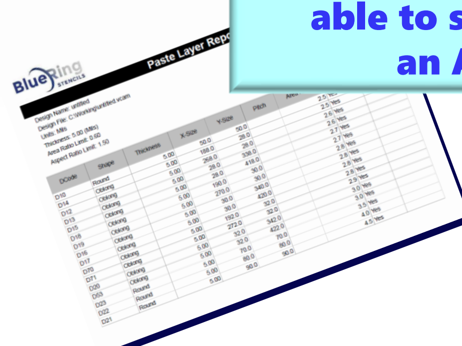
$$AR = \frac{\textit{Area of Circuit Side Opening}}{\textit{Area of Aperture Walls}}$$



See IPC-7525B for stencil design standards

Aperture Design	AR Formula
Square	$AR_{Square} = \frac{\textit{Side Length of Square}}{\textit{4 X Stencil Thickness}}$
Circle	$AR_{Circle} = \frac{\textit{Diameter of Circle}}{\textit{4 X Stencil Thickness}}$
Oblong, etc.	http://www.beamon.com/area-ratio-calculator/

Stencil Vendors should be able to supply you with an AR Report!

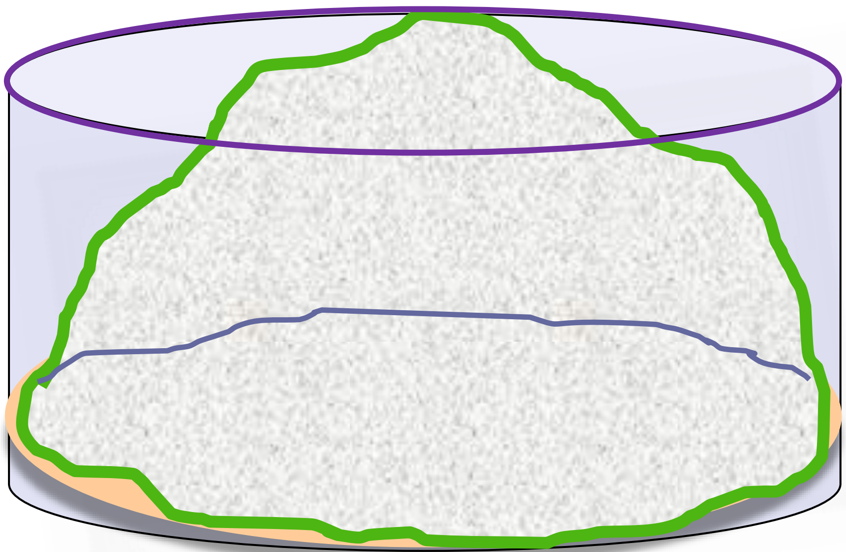


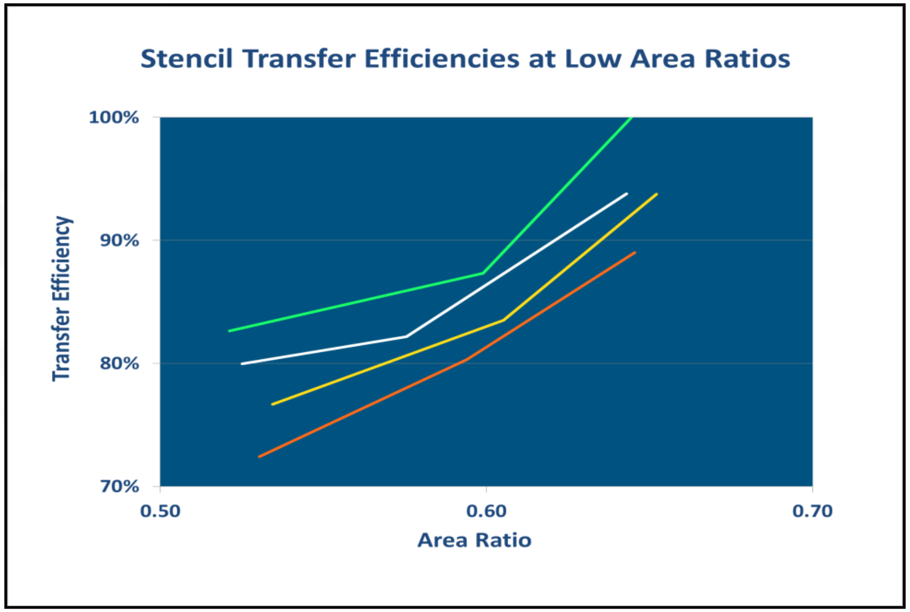
Tip
Keep AR >0.66 to make printing easier.

Transfer Efficiency, TE

Relates how much paste we want to print versus how much we actually printed.

$$\% \, TE = \frac{\textit{Volume of Paste Deposited}}{\textit{Volume of Stencil Aperture}} \text{ X } 100$$





- Minimum of 80% TE when possible
- *Minimizing Variation* is more important than maximizing transfer!

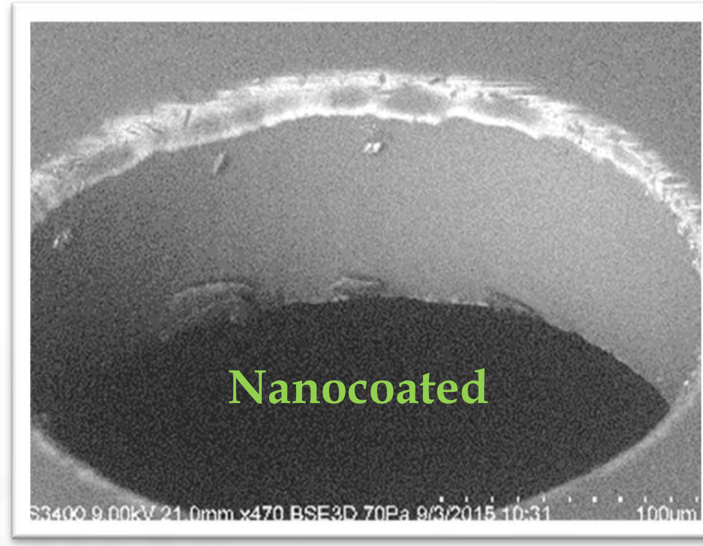
SHEA ENGINEERING SERVICES

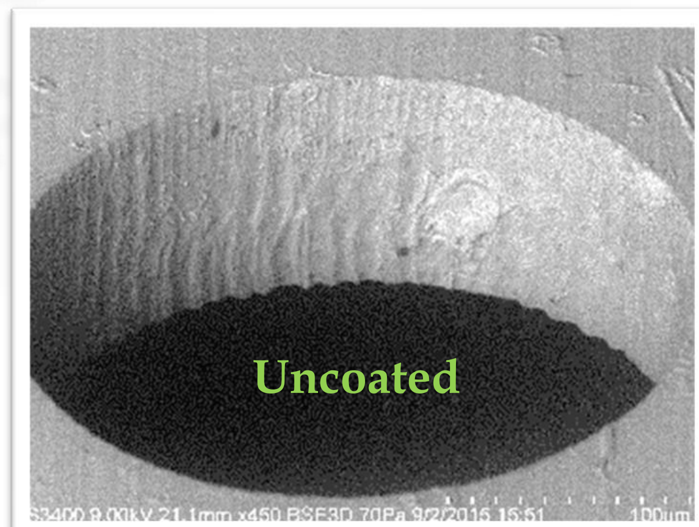
Easy Traveling

Nanocoating your stencils will help you avoid bumps in the road.

The nanocoating shown:

- Fills the valleys in the cuts and smoothes the walls
- Repels solder paste
- Improves Print Quality:
  - Less adhesion between paste and wall
  - Less surface area of wall
  - Results in higher volumes, lower variation and better shapes
  - Especially effective at AR <0.66

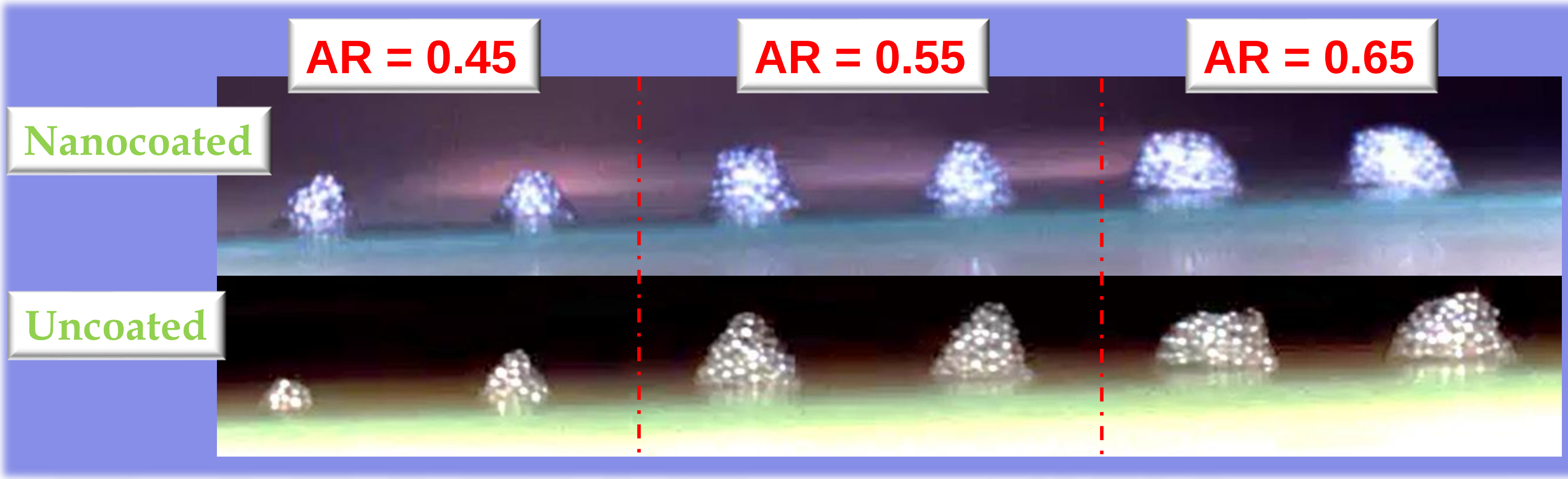




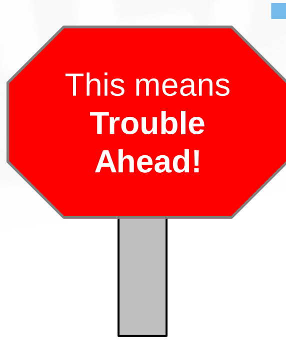
SHEA ENGINEERING SERVICES

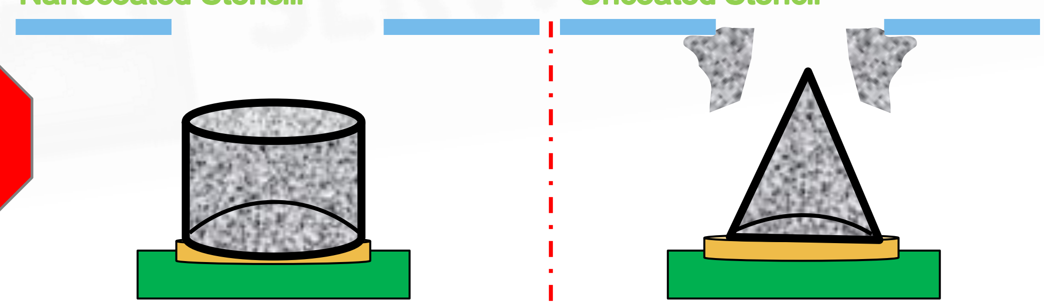
See For Yourself!

Deposits Released From Various Stencils



The uncoated stencil released solder paste deposits in shapes reminiscent of a Hershey Kiss.



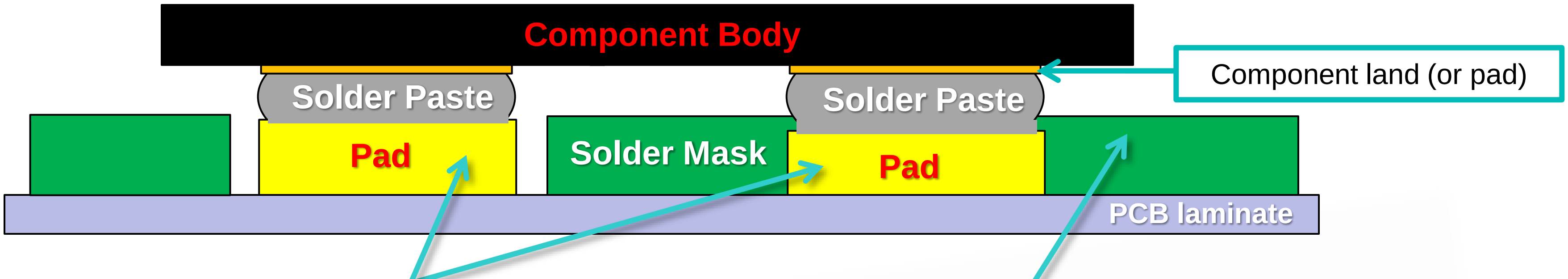


✓ More like a puck

✗ Less like a cone

SHEA ENGINEERING SERVICES

Circuit Board Language

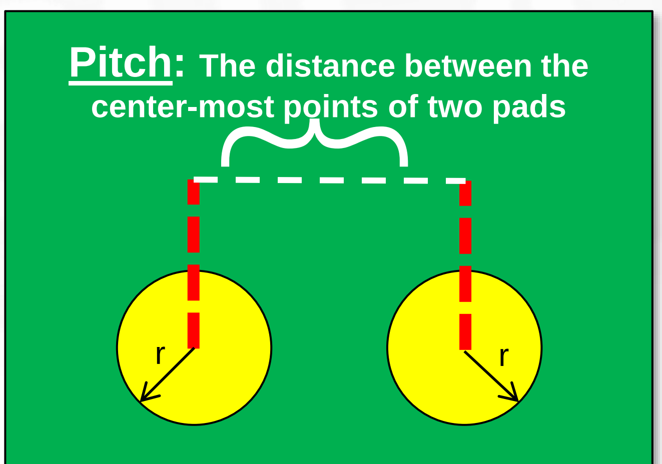


PCB Pads

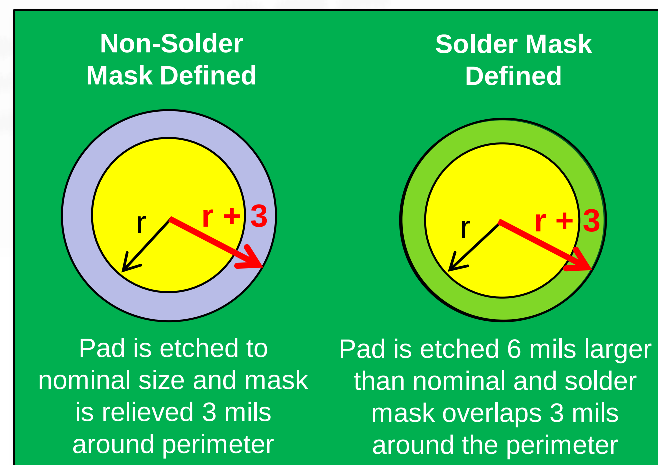
Etched in copper during PCB fabrication, these create the interconnects between the components and the PCB or other components

Solder Mask

A very thin layer of dielectric material that protects traces from oxidation, accidental solder wicking or solder bridging

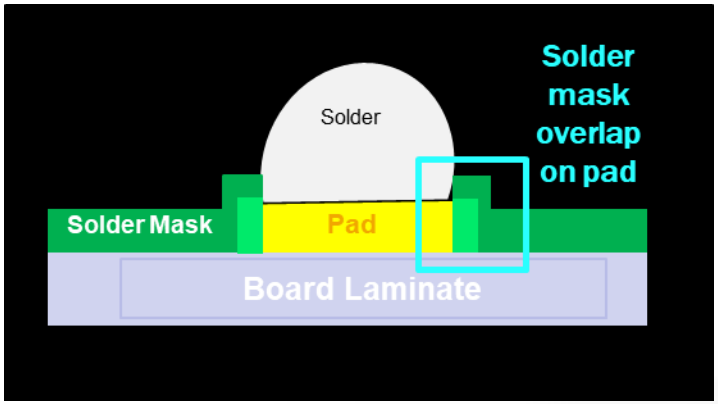


Fine Pitch =< 25 mils



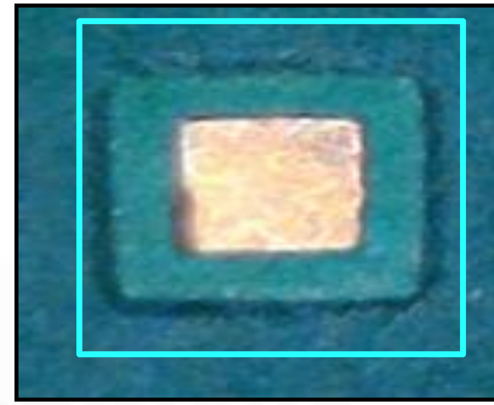
Pad Definition

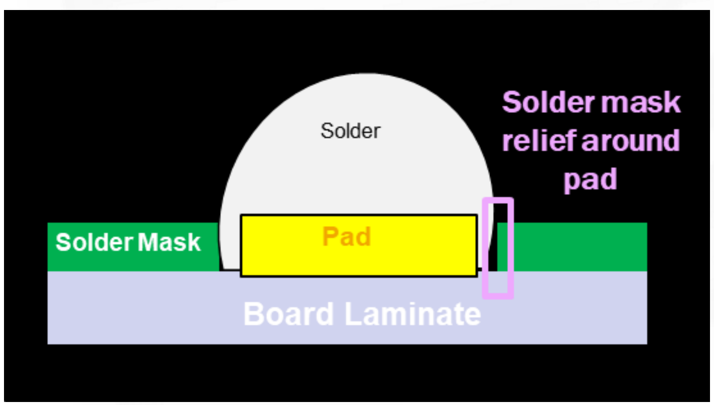
PCB Pad Definition



Solder Mask Defined (SMD)

- Pad is larger than nominal size: *easier to etch (DFM)*
- Solder mask overlapping on all sides creates gasketing surface: *easier to print (DFA)*





Non-Solder Mask Defined (NSMD or Copper Defined)

- Pad is nominal size: *challenging to etch below 8mil*
- Undersized pads present gasketing issues: *harder to print*
- *“Wrap around” of solder on sides of pad improves strength*

Why do we care about pad definition?

Because it seriously impacts printability, as the data will soon show…

SHEA ENGINEERING SERVICES

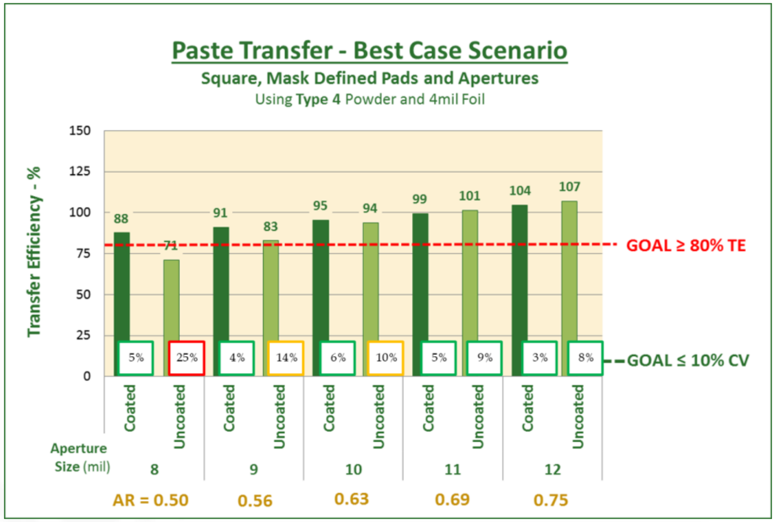
Not Nanocoating Your Stencil

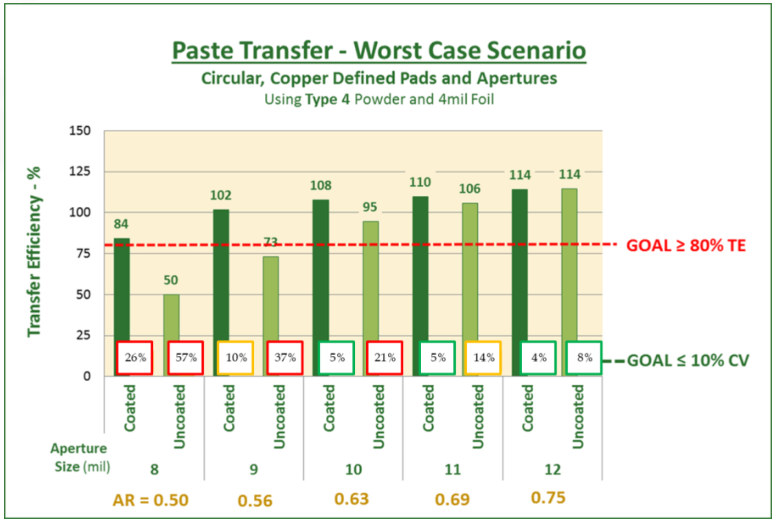
and wondering why your yields are bad….

Is like grabbing a dish from the oven without a mitt and wondering why your hand got burnt.

Nanocoating improves your process at ARs <0.70

(Just do it)





Data courtesy of AIM Solders

SHEA ENGINEERING SERVICES

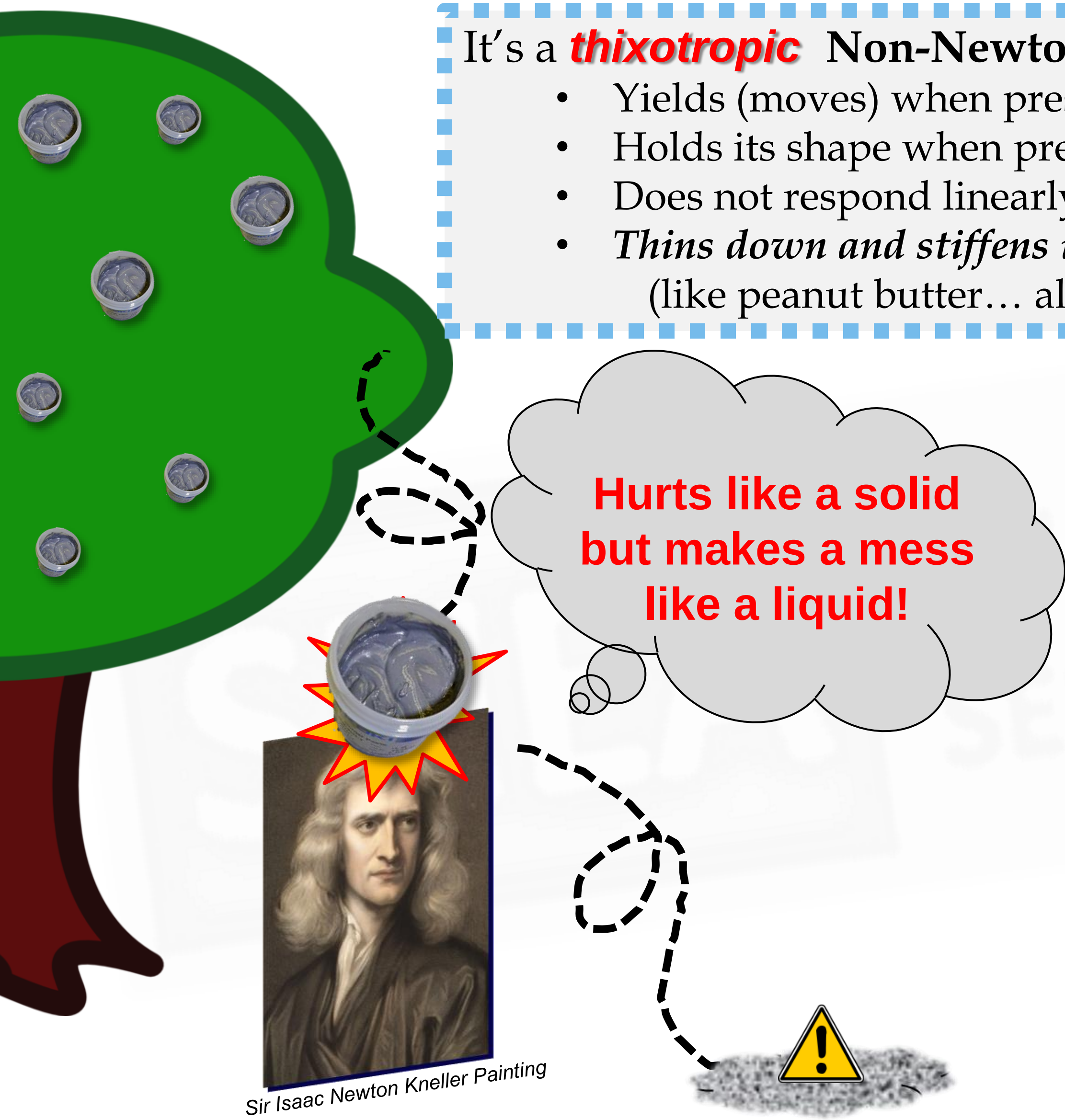
A Constant State of Flux

SHEA ENGINEERING SERVICES

Solder Paste: The Mysterious Material

It's a ***thixotropic*** **Non-Newtonian** fluid:

- Yields (moves) when pressure is applied to it
- Holds its shape when pressure is not applied
- Does not respond linearly to shear
- *Thins down and stiffens up*
  (like peanut butter… also messy and sticks to the knife)



Sir Isaac Newton Kneller Painting

Key Terms

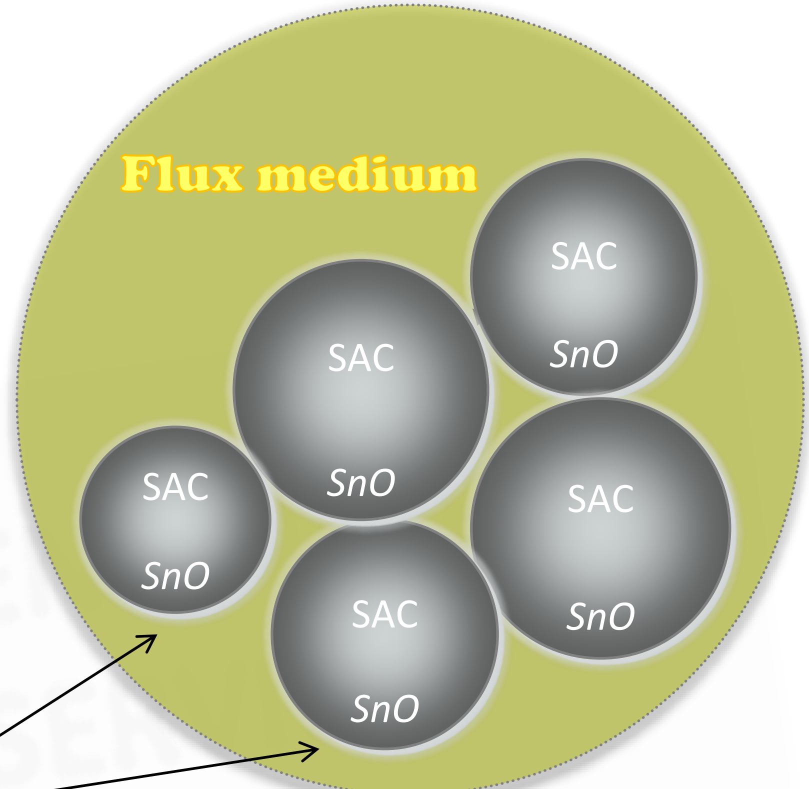
Viscosity – how easily a fluid flows under pressure

Rheology – how the viscosity of a fluid changes as the pressure changes

What is Solder Paste?

A suspension of tiny solder particles in flux medium.

All particles have an oxide shell which helps prevent clumping.



<u>Flux to Metal Ratio</u>

50:50 by Volume

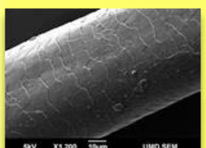
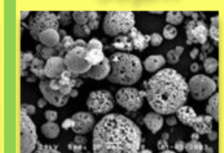
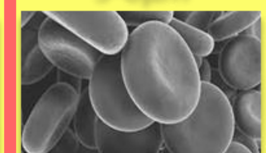
90:10 by Mass

Illustration Courtesy of Henkel

SHEA ENGINEERING SERVICES

Solder Powder Particle Size Classification

Most Common (Type 3) — **Least Common** (Types 7–8)

Equivalent	Human Hair ~50µm	Volcanic Ash ~30µm	Wire bond ~20µm	Plant cell ~10µm	Red blood cell ~5-6µm	Photosensitive cell ~2µm
						
Paste Type	Type 3	Type 4	Type 5	Type 6	Type 7	Type 8
Particle Size	45-25µm	38-20µm	25-15µm	15-5µm	11-2µm	<10µm

Largest Particle Size
Smallest Type Number

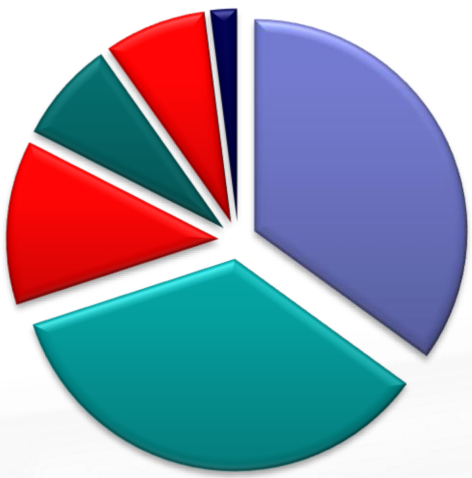
⟷

Smallest Particle Size
Largest Type Number

Smaller particles have higher oxide-to-solder ratios which make them more difficult to reflow.

Images Courtesy of Henkel

What the Flux?



Flux Recipe

- Solvents ~35%
- Resins/Rosins ~35%
- Activators 10-15%
- Surfactants 5-10%
- Thixotropes 5-10%
- Trade Secrets <2%

Image Courtesy of Henkel

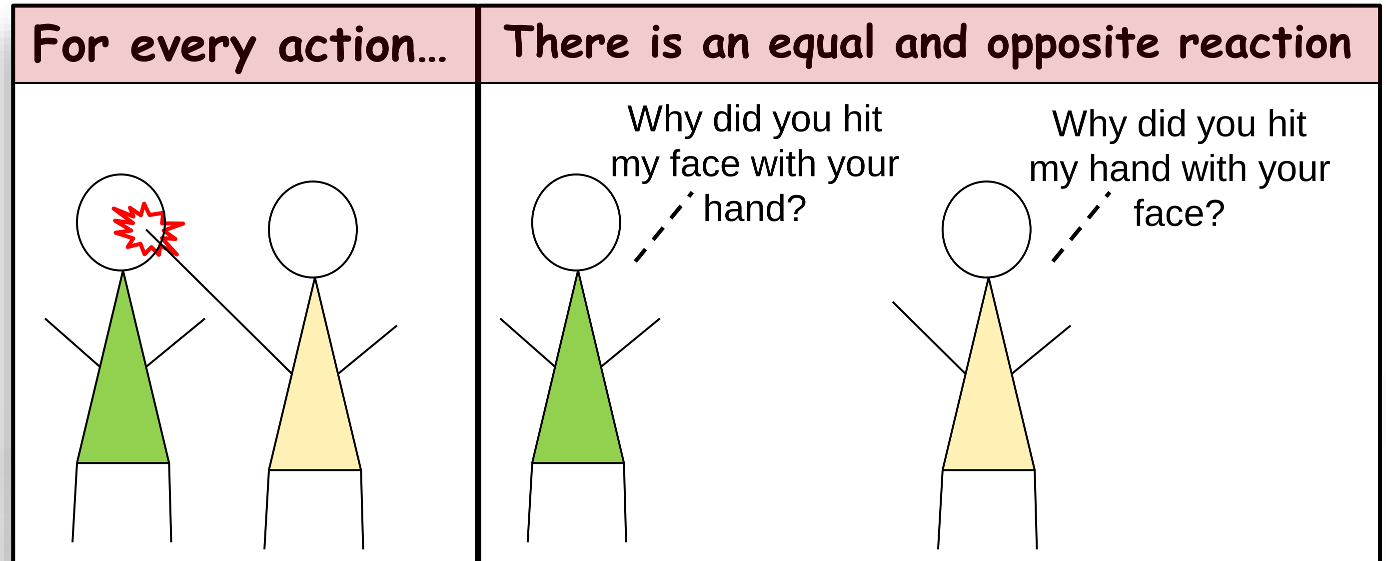
Solvents: Maintain ingredients in a uniform solution
Resins/Rosins: Remove oxides, aid in joint formation, provide reliability
Activators: “Clean” the passivation layer of metals to prevent corrosion
Surfactants: Reduce surface tension of paste to aid in wetting
Thixotropes: Modify the flow of the solder during printing
Trade Secrets: The world may never know

SHEA ENGINEERING SERVICES

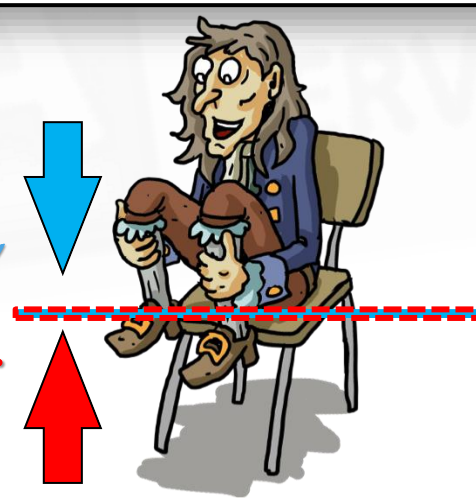
Don't Blow a Gasket!

SHEA ENGINEERING SERVICES

Newton's Third Law:



As Sir Isaac sits on the chair,



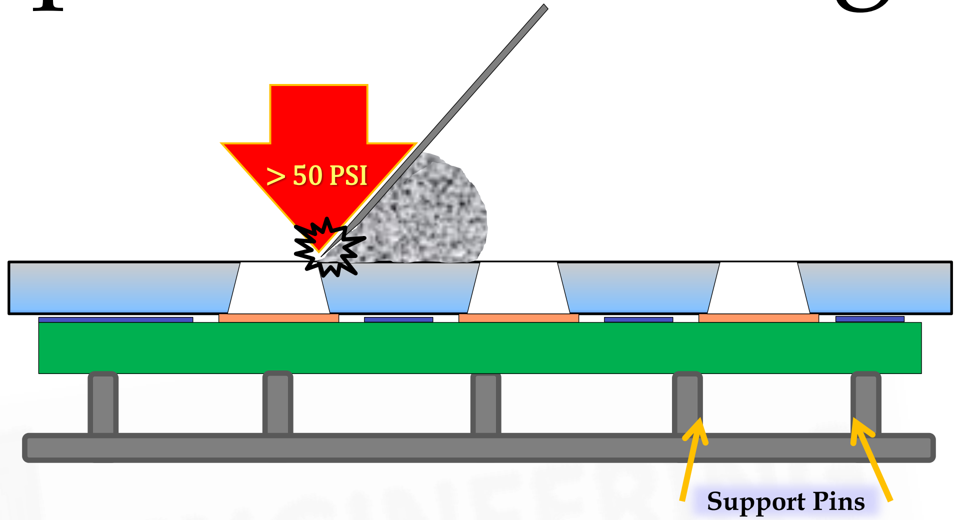
The chair supports him.

That's a law we can all agree on!

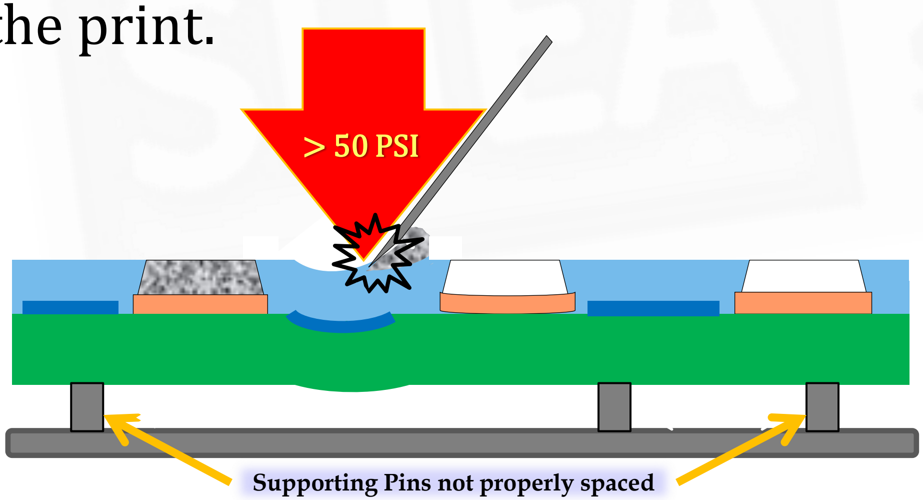
SHEA ENGINEERING SERVICES

Same Thing Applies To Printing

The squeegee exerts substantial pressure at its point of contact.



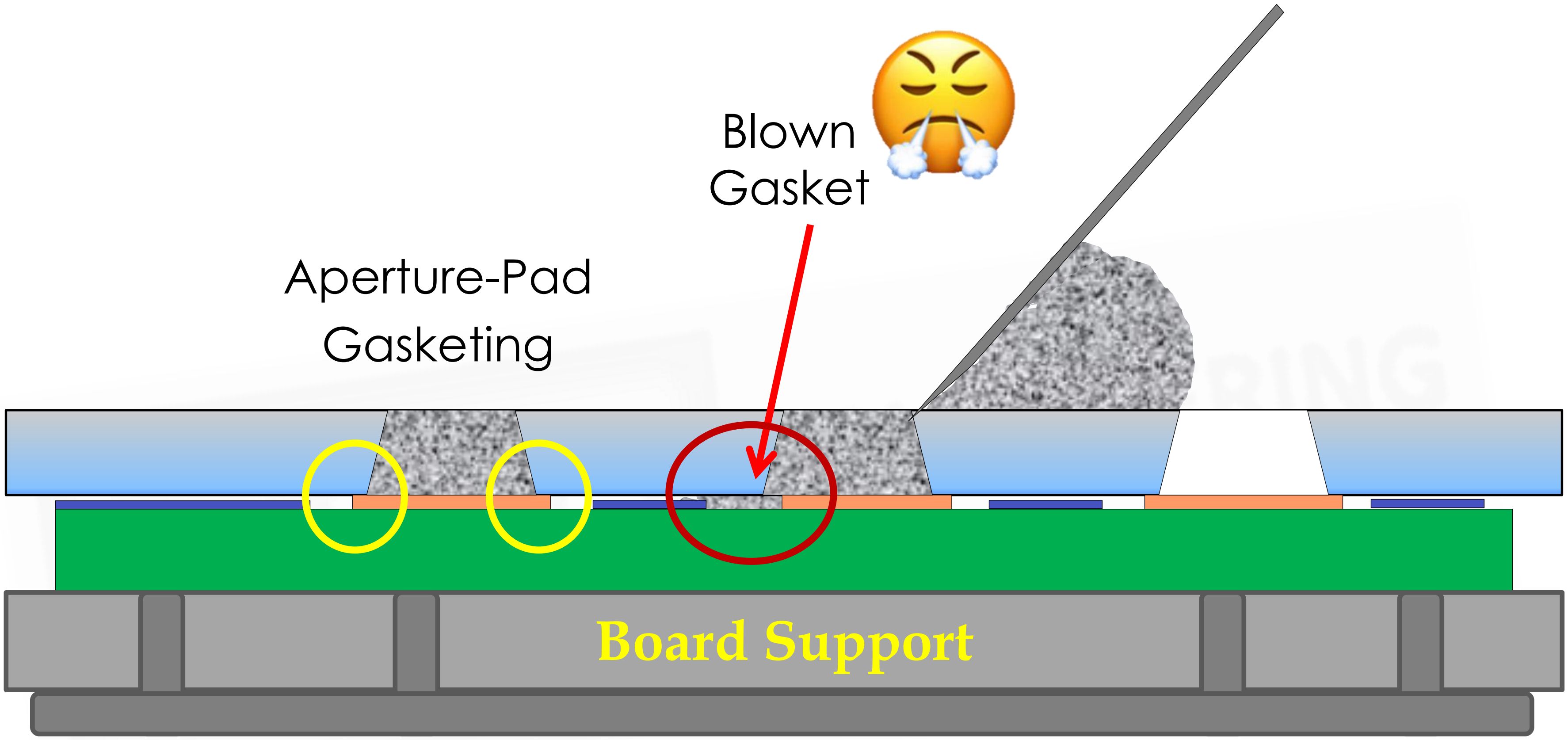
Without *proper* **Board Support** the pressure of the squeegee causes the stencil and board to deflect and snapback at different rates, smearing the print.



Insufficient Board Support Can Cause:

- **Gasketing issues**
- **Paste smearing**
- **Wet bridging**
- **Deposits that are too tall**

Alignment and Gasketing



SHEA ENGINEERING SERVICES

#1

Reason for print defects, and therefore,

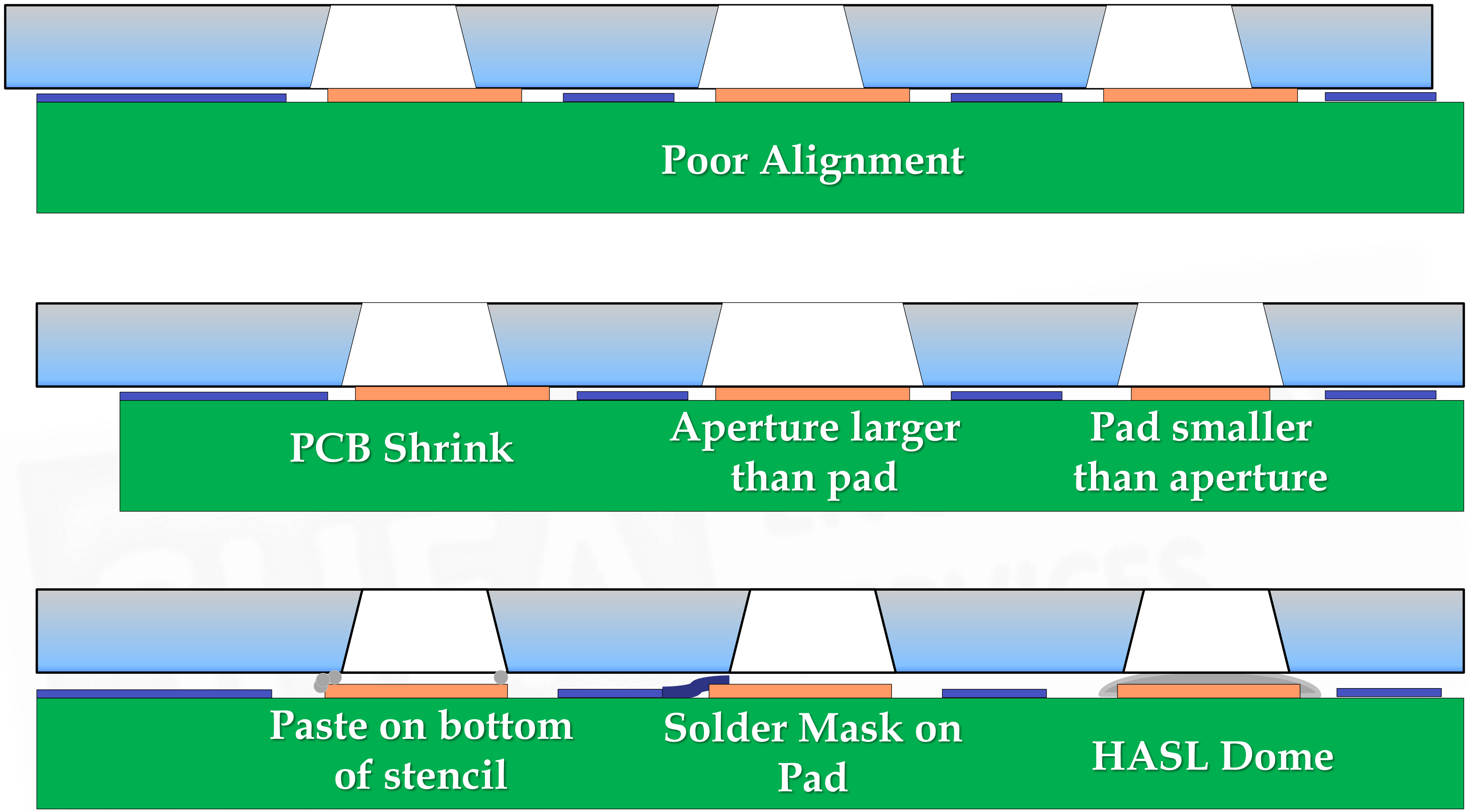
SOLDER DEFECTS:

BAD GASKETING

Between the stencil and the PCB pad

SHEA ENGINEERING SERVICES

Common Causes of Poor Gasketing



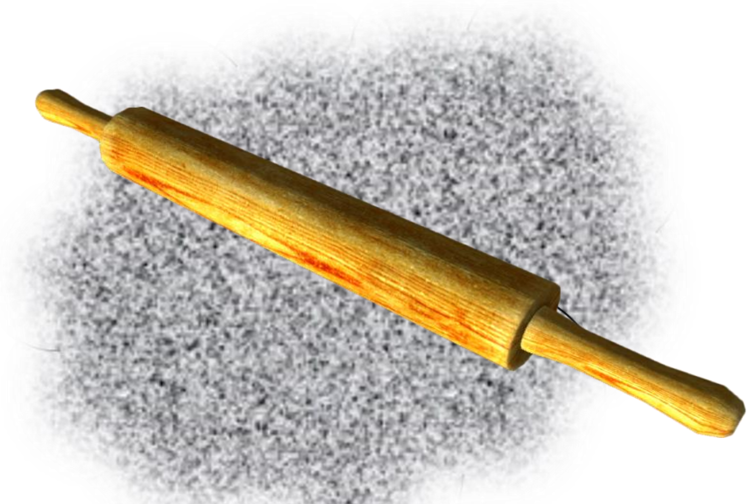
<u>Other common causes:</u>

- Labels
- Silkscreen
- PCB Clamps

HASL: Hot Air Solder Level
a PCB finish that covers the copper pad with solder

SHEA ENGINEERING SERVICES

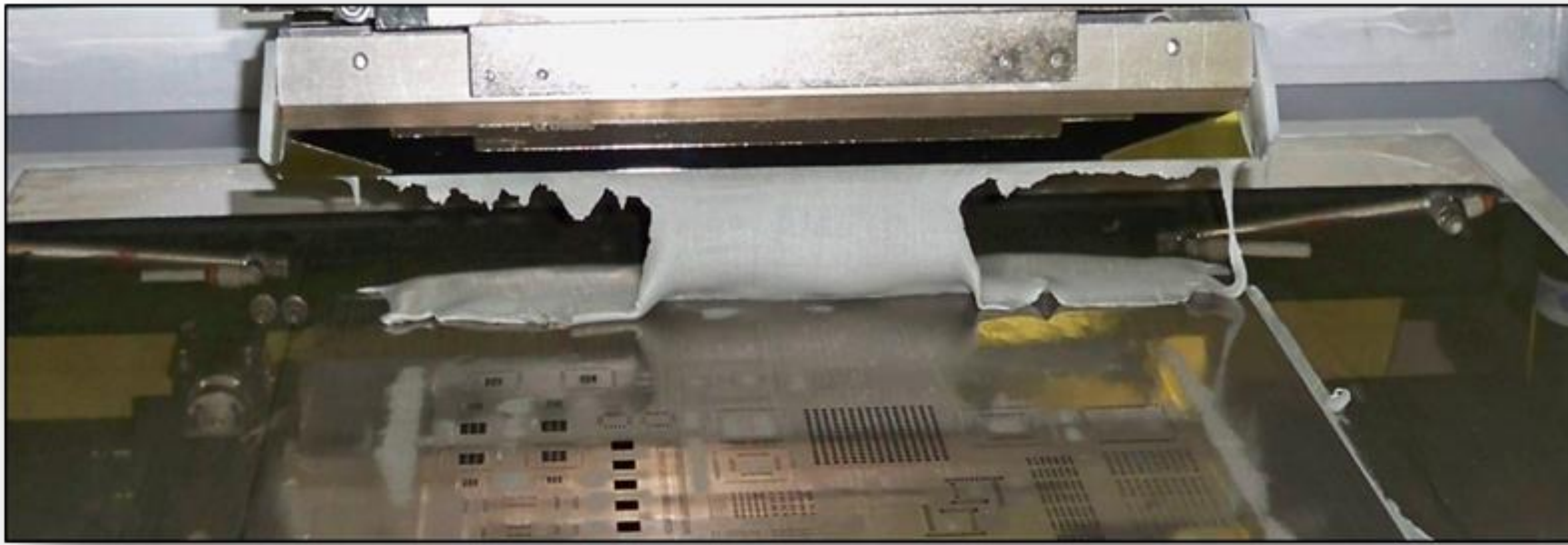
Let's Get Rolling



SHEA ENGINEERING SERVICES

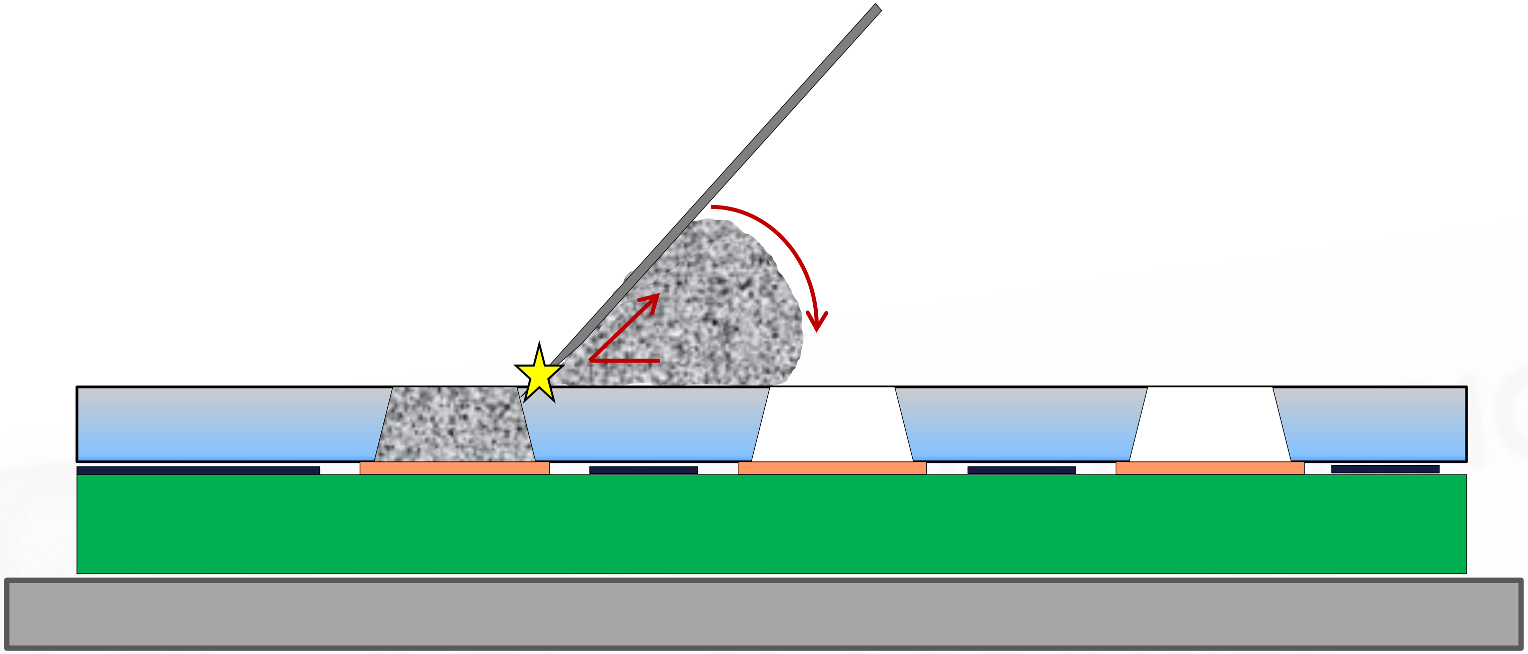
Things Are Getting Technical

The Mechanics of the Printing Process



SHEA ENGINEERING SERVICES

Rolling Fills the Apertures



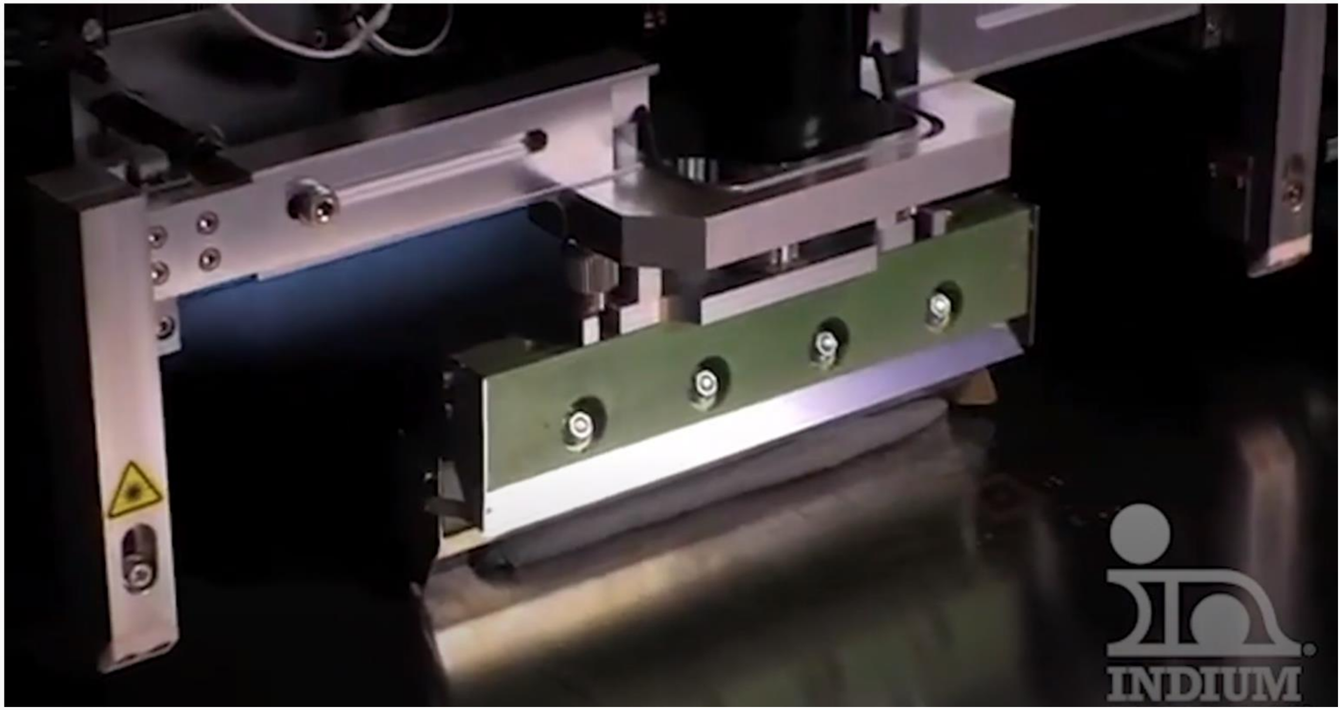
- Squeegee motion shears and thins paste so it can flow into apertures
- Paste picks up *angular momentum* as it rolls and the sharp turn at the ☆ *vertex* of the squeegee and the stencil creates a local area of high pressure that fills the apertures

ENGINEERING SERVICES

This is How We Roll…

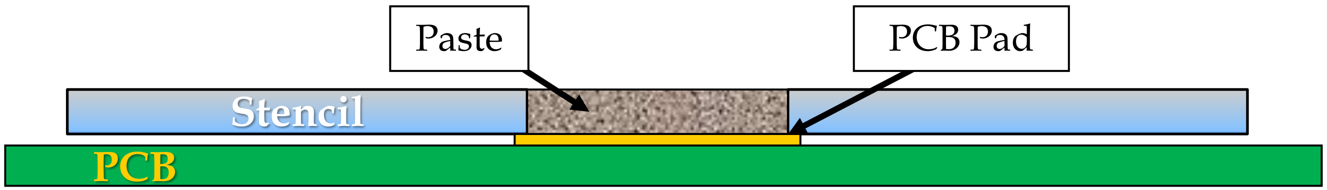
L👀k for:

- Paste rolling, not skidding
- A smooth, shiny bead without bubbles or tears as it rolls
- Paste gently dropping off the squeegee at the end of the stroke
- A “clean sweep” across the top of the stencil
- Minimal accumulation of paste outside the print area on the stencil

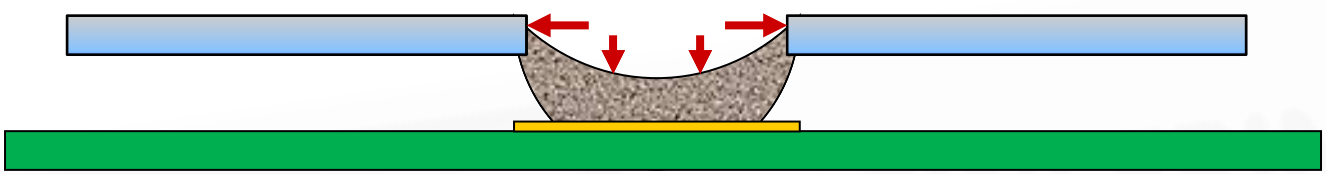


https://www.youtube.com/watch?v=Bw6Lt-XLklw

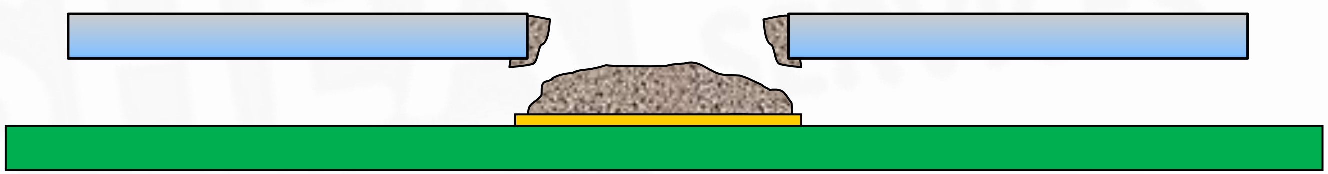
Solder Paste Release from Stencil



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



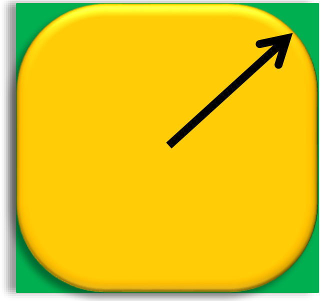
Depending on area ratio, a portion of the paste will release to the PCB, while some will stay in the aperture. Some paste may also stick to the bottom of the stencil due to stringing, bad gasketing or "pump out"

The smaller the AR, the lower the TE

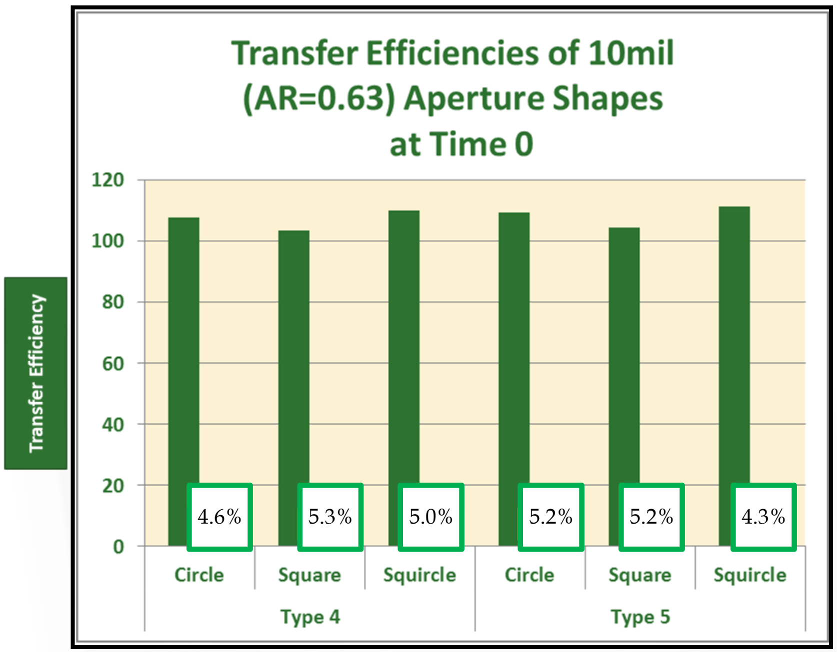
SHEA ENGINEERING SERVICES

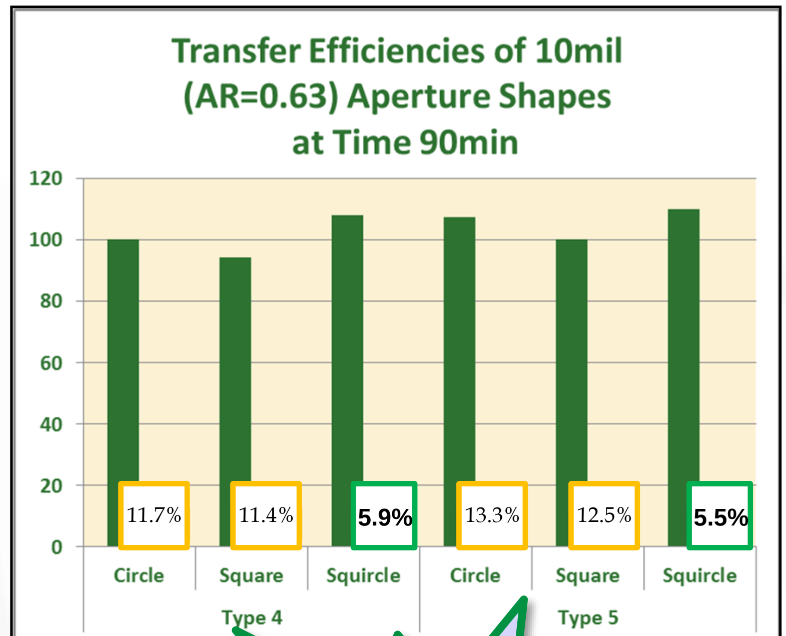
Round Your Corners

“Squircle”



Square with rounded corners





Coefficient of Variation (CV):
The standard deviation divided by the mean.

- <10 % is great
- 10-15 % is ok
- >15 % is unacceptable

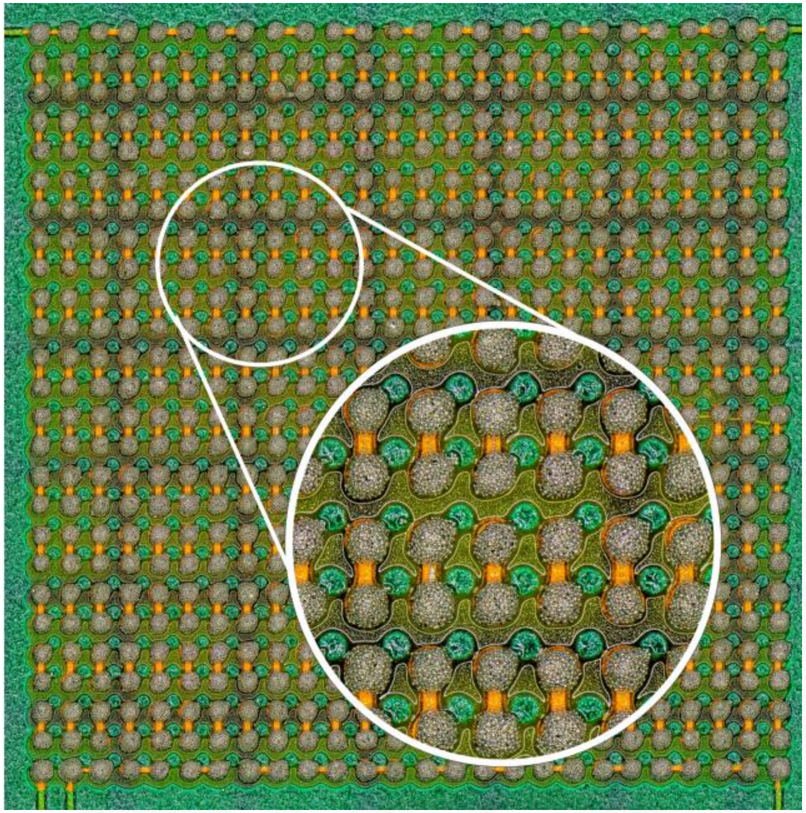
Squircle prints like a champ, even after a 90-minute pause!

Data courtesy of AIM Solders

SHEA ENGINEERING SERVICES

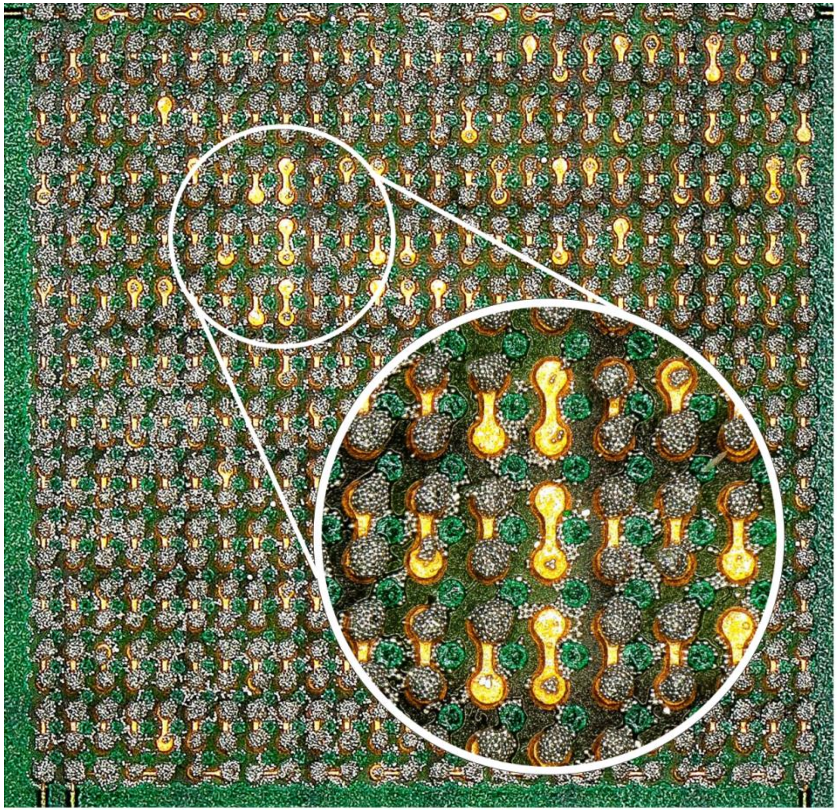
Got 5 Minutes?

The process you wish you ran…



VS

The process you currently run

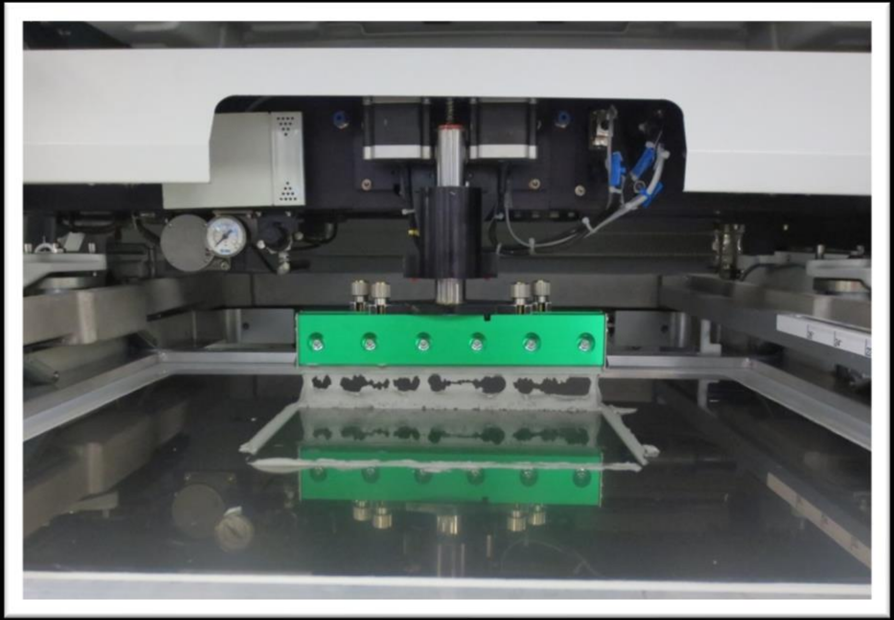


Images Courtesy of Henkel

SHEA ENGINEERING SERVICES

Overall System Check

- Takes 5-10 minutes
- Easy to incorporate with techs, skilled operators



Catches the problem at least 80% of the time

SHEA ENGINEERING SERVICES

Overall System Check

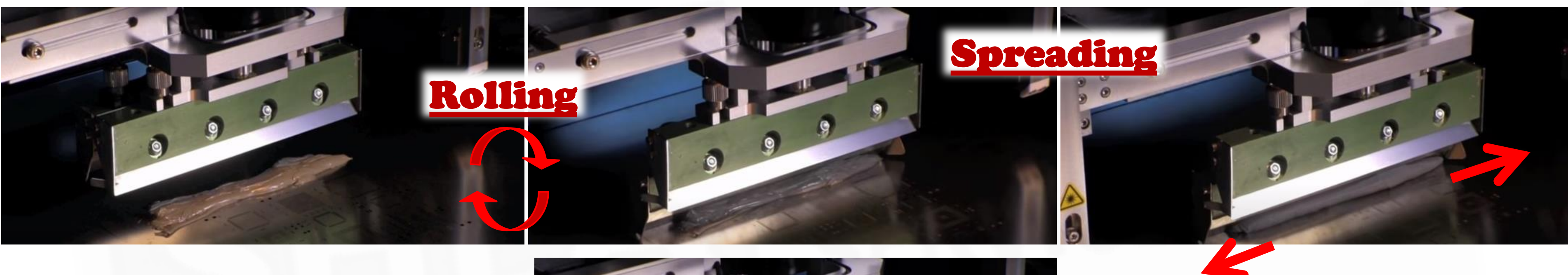
Knead the paste, wipe the stencil, print a board, observe

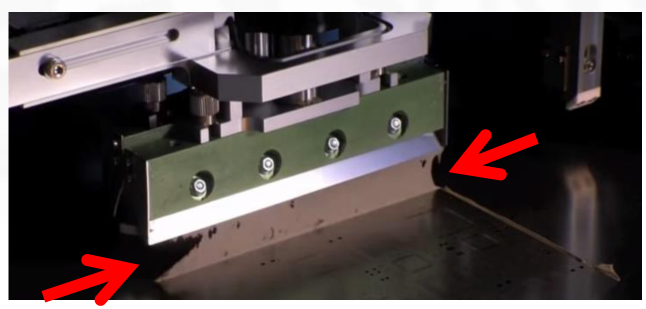
Is the right amount on stencil?

- Paste bead should be about 1.5 cm diameter (5/8")
- Bead size affects fill pressure

Does it roll over the surface of the stencil and release cleanly from squeegee blade?

- If no, replace it with fresh solder paste
- Check temperature and consistency





Images Courtesy of Indium Corperation

Curtaining off the squeegee

https://www.youtube.com/watch?v=Bw6Lt-XLklw

SHEA ENGINEERING SERVICES

Check the Tooling

Remove the stencil

Inspect the stencil

- Physical damage
- Paste/debris in apertures
- Worn out or dirty fiducials
- Rips or tears in mounting mesh
- Cracks or chips in nanocoating
- Use "Sharpie" style marker to check surface energy of coating

Inspect the squeegees

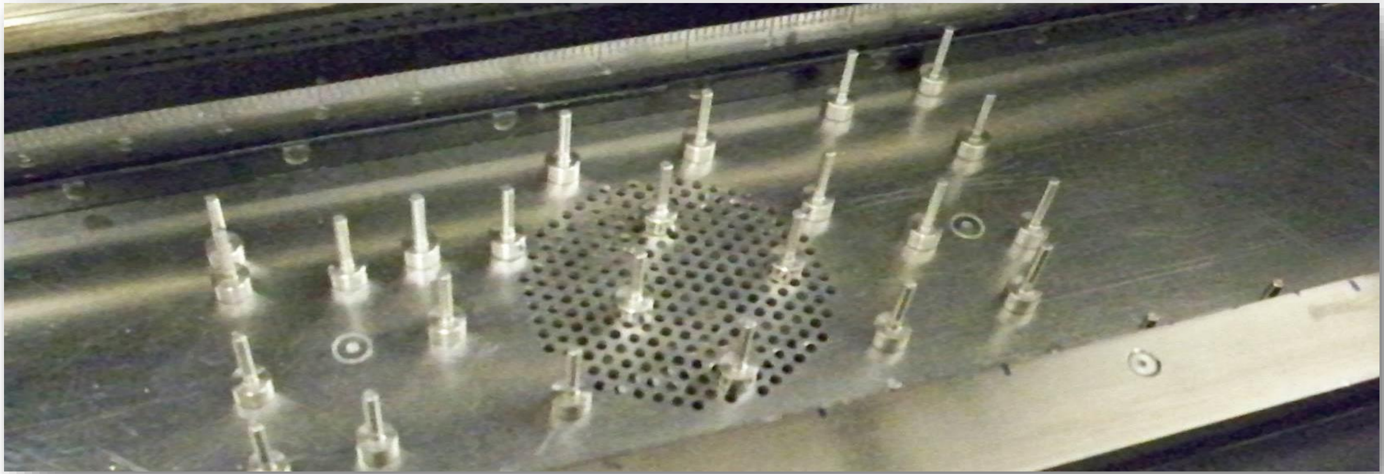
- Damage or dings, angle

Inspect the board support

- Is dried paste interfering with PCB seating in tooling?

Shuttle a board into position

- Tap or press on top to verify support
- Check for movement in X & Y



SHEA ENGINEERING SERVICES

Check Alignment and Setup

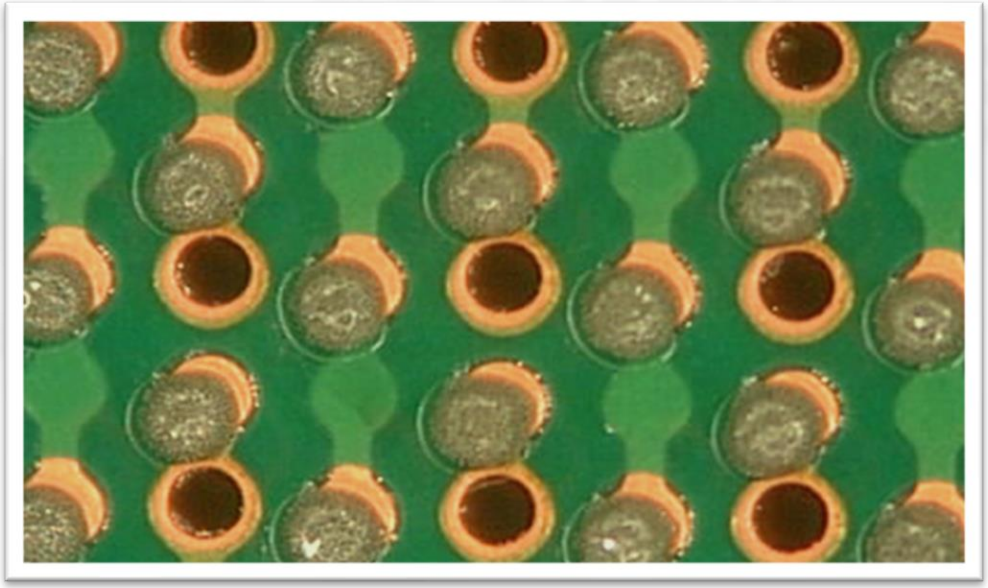
Reinstall the stencil

Check the alignment

- Watch the process, including the vision finding the fiducials
- Confirm alignment
- Check contact between stencil & board

Recheck Print Parameters

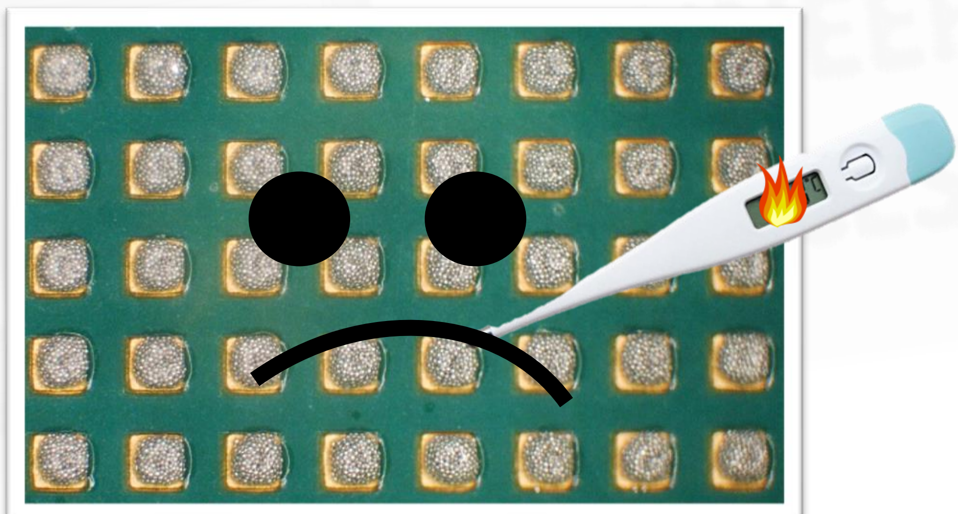
- Speed, Pressure
- Snap off Delay, Speed and Distance
- Do they make sense?
- Reference internal documentation or tech data sheets



SHEA ENGINEERING SERVICES

Overall System Check

- Takes 5-10 minutes
- Easy to incorporate with techs, skilled operators
- Catches the problem at least 80% of the time



If root cause is not found, investigate the specific symptoms…

SHEA ENGINEERING SERVICES

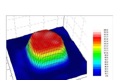
5 Minutes Is Up!
Still Having Printing Issues?

Solder Paste Print Defects

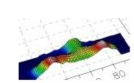
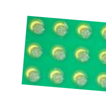
This is for when you've given the 5-Minute Overall System Check a try, as well as one of your colleagues. Then you've given it another go and you still don't know what is going wrong…

Introducing:

- Solder Bridges
- Poor Print Definition
  - Peaks or "Dog Ears"
- Insufficient Solder Volumes
  - Poor Aperture Fill
  - Poor Aperture Release
- Poor Gasketing
- Poor Alignment

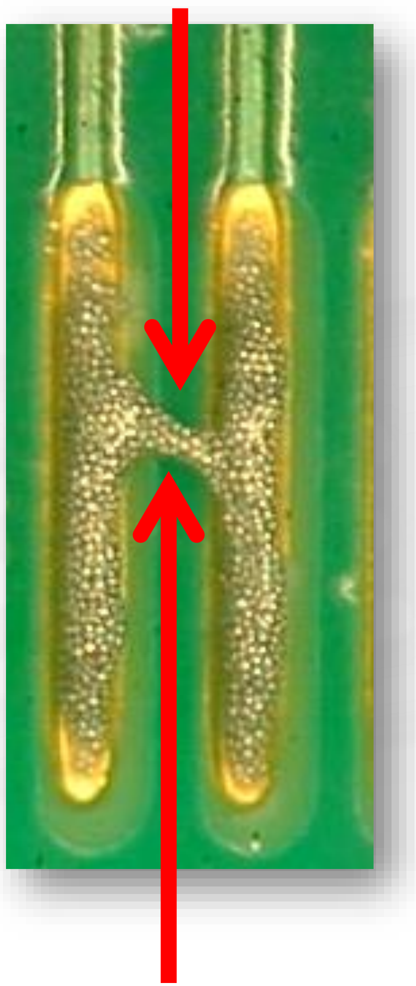


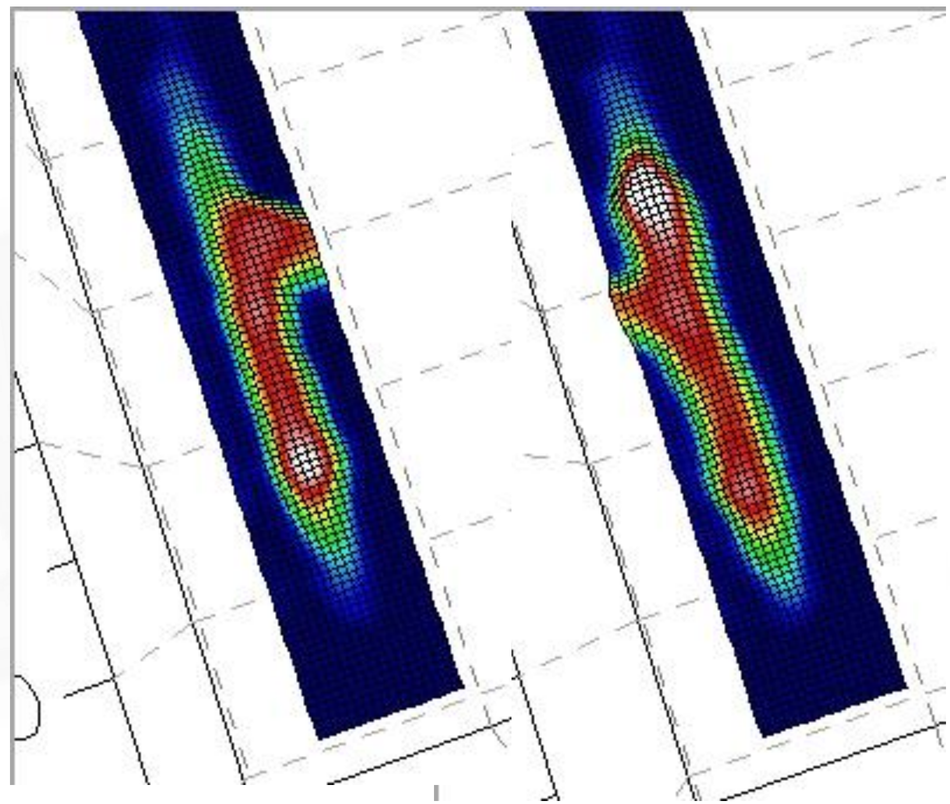



SHEA ENGINEERING SERVICES

Solder Bridges

When 2 solder paste deposits connect unintentionally





SHEA ENGINEERING SERVICES

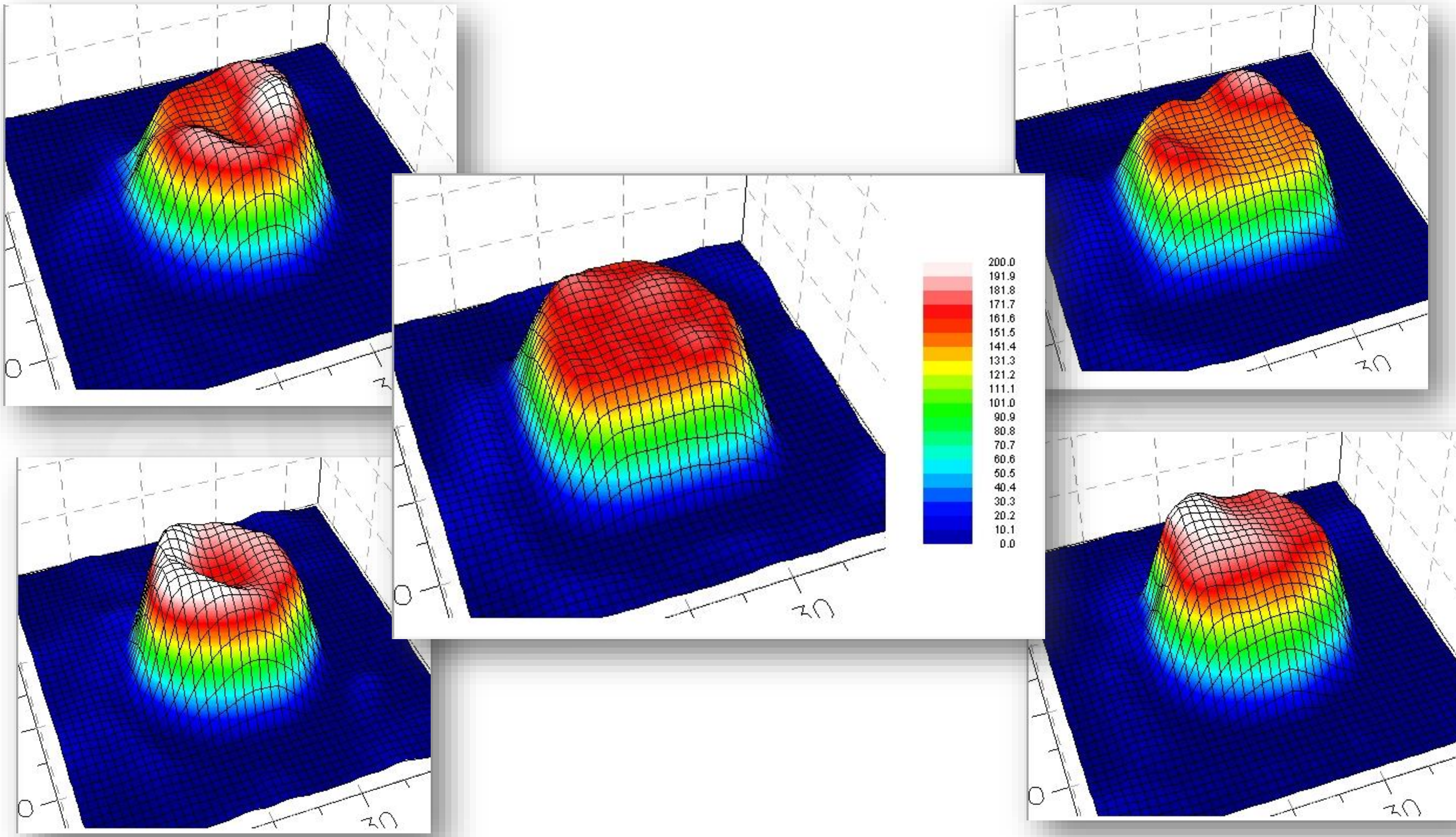
Solder Bridges

If you suspect…	Then investigate:
Bad Gasketing	See slides on possible reasons for bad gasketing. Check board support
Residual paste from previous print	Stencil cleaning parameters Increase wipe frequency
Separation speed (too fast or too slow)	Increase or decrease separation speed - Different pastes have different optimums and its usually one or the other – no middle
Squeegee pressure too high	Decreasing the force. Most pastes work well with 1 – 1.25 lb/in force
Too much paste	Check bead on stencil. ½ - ¾ inch is typical (the diameter of a dime or nickel)
Paste is too warm	Compare working temperature and tech data sheet. If printer is getting hot inside, check exhaust fans

SHEA ENGINEERING SERVICES

Poor Print Definition - Peaks and "Dog Ears"

When a section of a solder deposit is much taller than the rest of the deposit.

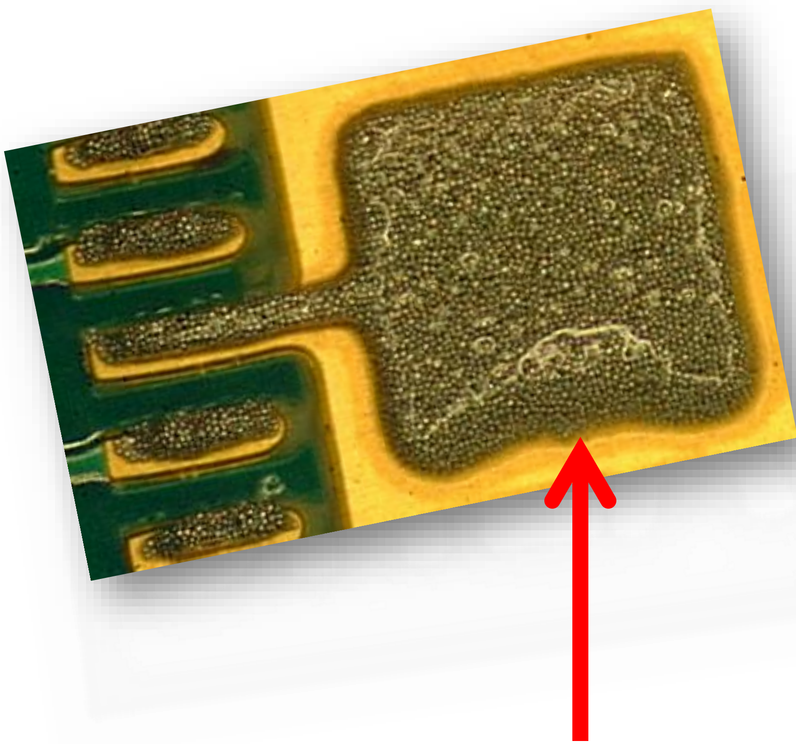


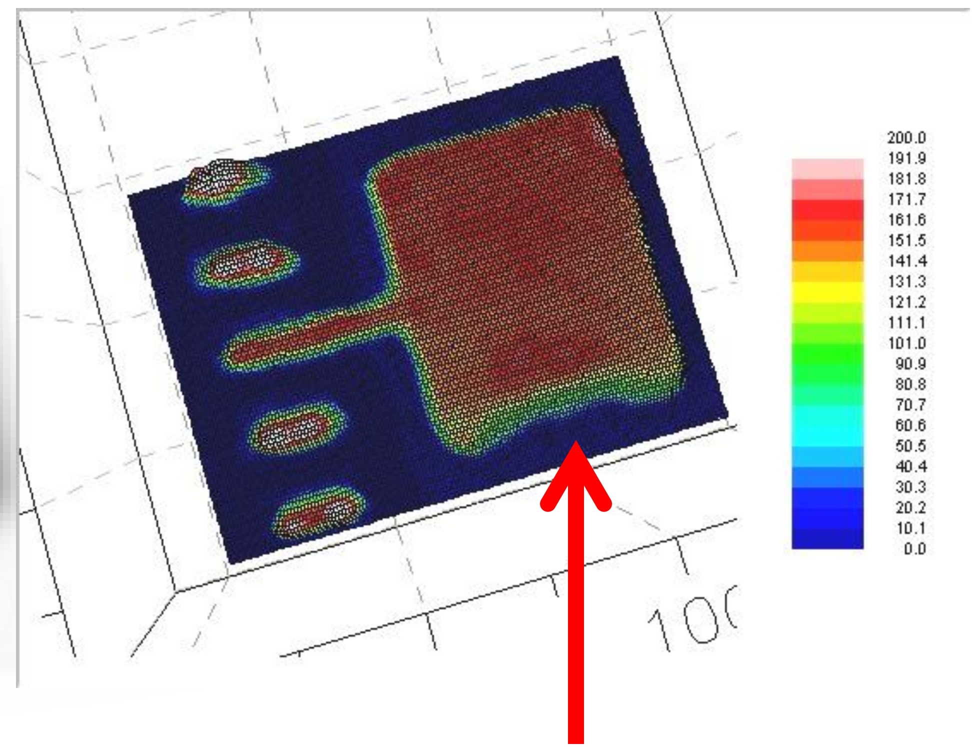
Poor Print Definition

If you suspect…	Then investigate:
Bad Gasketing	See possible reasons for bad gasketing Check board support
Separation speed (too fast or too slow)	Increase or decrease separation speed - Different pastes have different optimums and its usually one or the other – no middle
Residual paste from previous print	Stencil cleaning parameters Increase wipe frequency
Misalignment	See section on alignment
Squeegee pressure too high or too low	Adjust force. Most pastes work well with 1 – 1.25 lb/in.
Paste is too warm	Check temperature and tech data sheet

Insufficients - Poor Aperture Fill

When a section of a solder deposit is absent.





Seen in large apertures and in small ones that are in high density areas

SHEA ENGINEERING SERVICES

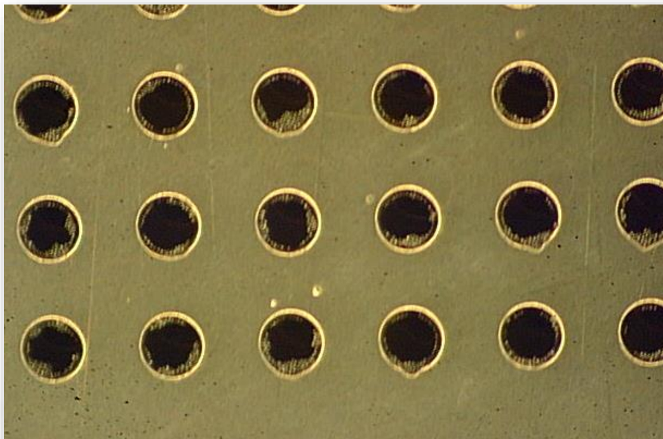
Poor Aperture Fill

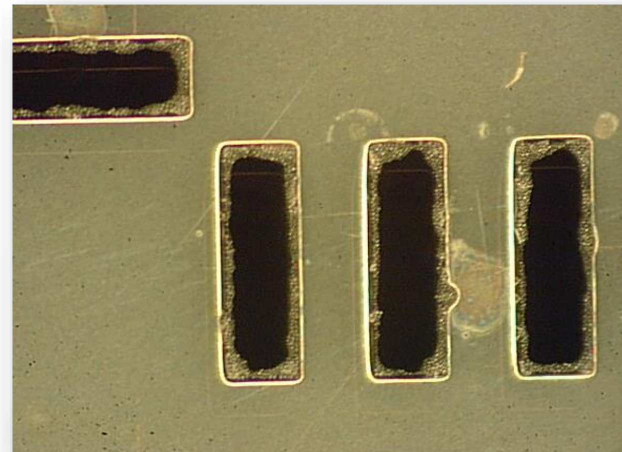
If you suspect…	Then investigate:
Pause in printing raised paste viscosity	Knead 4 -10 strokes. Clean board used for kneading
Squeegee speed too high or too low	Check print speed
Squeegee pressure too low	Increase the force. Most pastes work well with 1 – 1.25 lb/in.
Not enough paste on stencil	Check bead on stencil. ½ - ¾ inch is typical
Paste is too cold	Check temperature and tech data sheet
Paste sticking to squeegee blade	Check bead on stencil. ½ - ¾ inch is typical. Check paste temperature
Squeegee worn or damaged	Inspect blades and replace if necessary

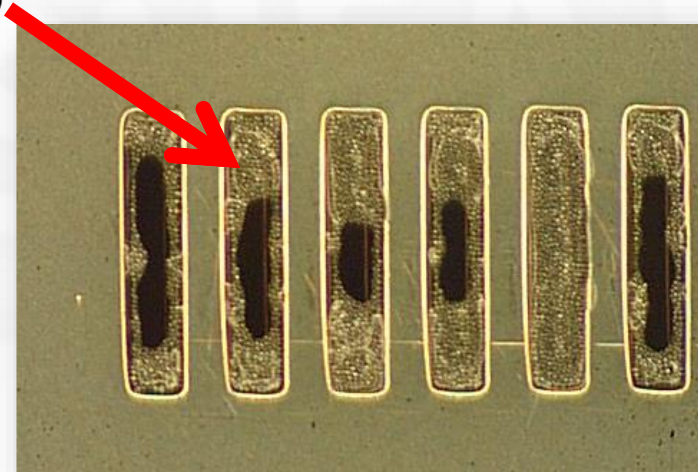
SHEA ENGINEERING SERVICES

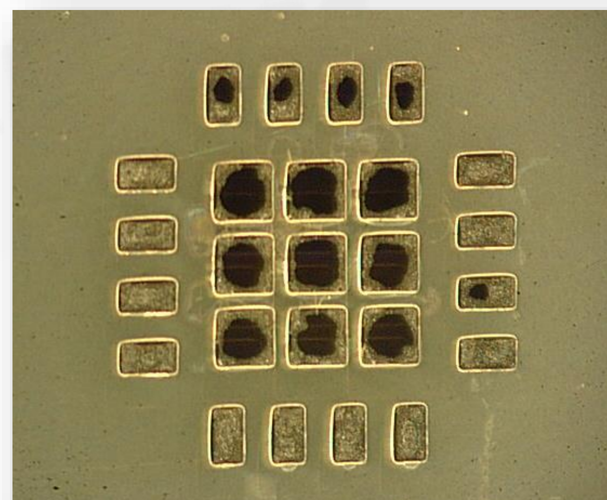
Insufficients - Poor Aperture Release

When solder paste remains in the stencil between prints.









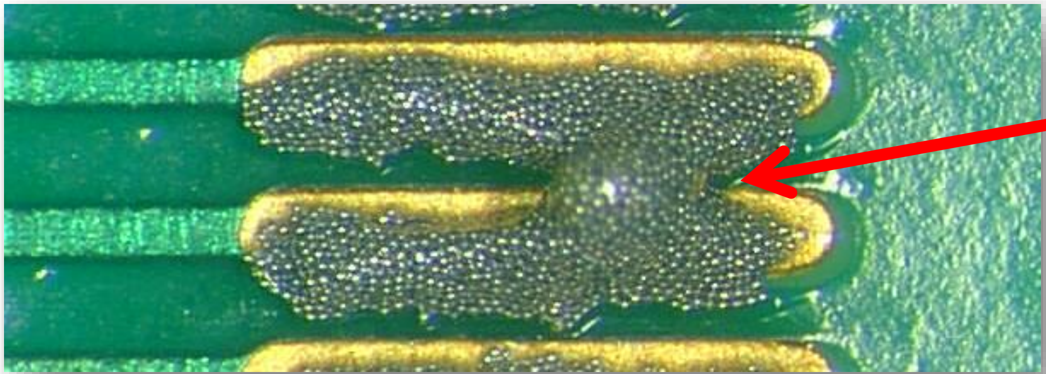
SHEA ENGINEERING SERVICES

Poor Aperture Release

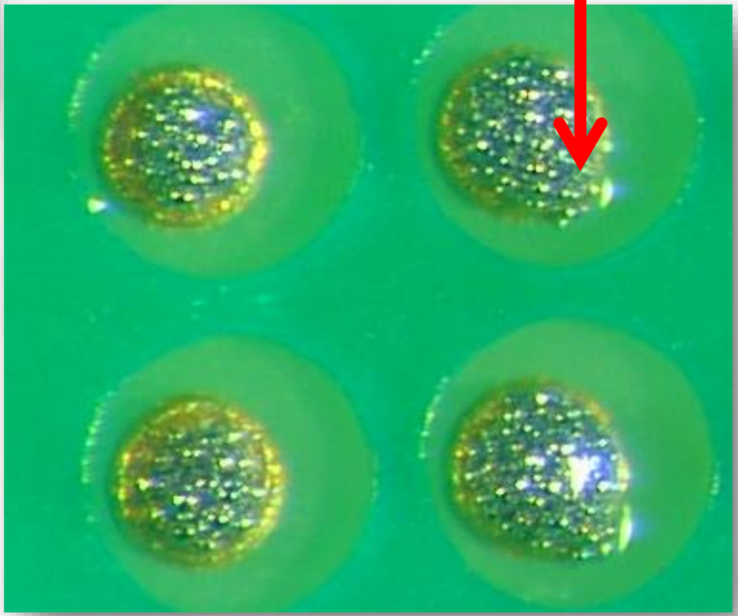
If you suspect…	Then investigate:
Pause in printing raised paste viscosity	Knead 4 -10 strokes. Clean board used for kneading
Residual paste building up in apertures	Check stencil cleaning parameters, increase frequency, clean after down time
Paste is too cold	Check temperature and tech data sheet
Squeegee pressure too low	Increasing the force. Most pastes work well with 1 – 1.25 lb/in

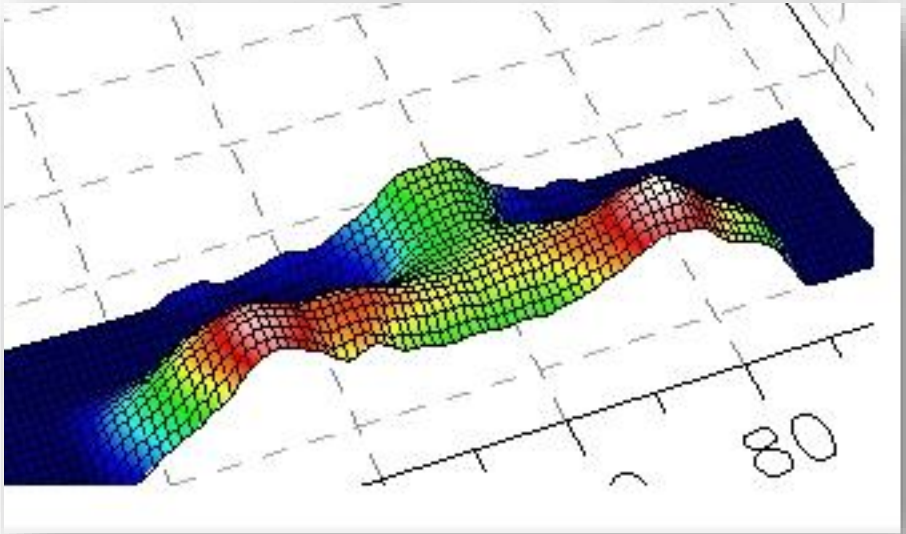
Poor Gasketing

When the stencil does not create a seal with the circuit board.



Solder paste that squeezed under the stencil during printing





SHEA ENGINEERING SERVICES

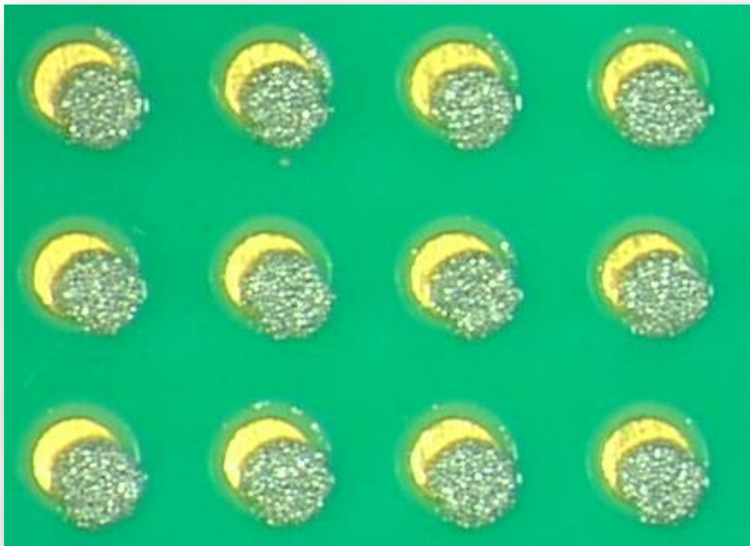
Poor Gasketing

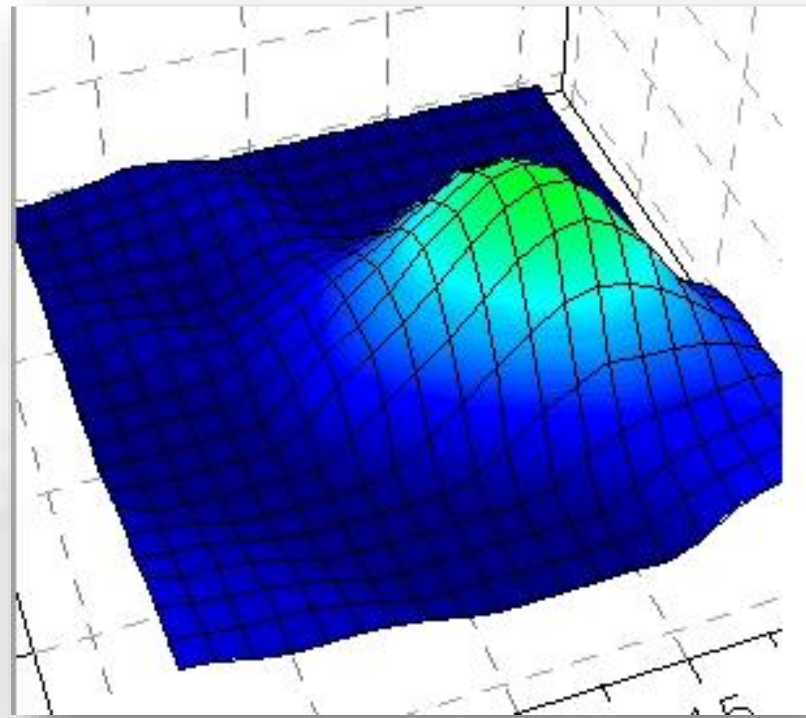
If you suspect…	Then investigate:
Board Support	Check (clean or improve) PWB support
Bad alignment	See section on alignment
Solder mask higher than pads	Check solder mask height and compare to specification
Stencil apertures larger than PWB pads	Measure and compare to specification
Hot Air Solder Level finish creates uneven printing surface	More planar, non-HASL finishes. Consult with PWB vendor on improving doming effect of HASL process.
Labels, inks, or other surface features prevent stencil from seating on PWB	Proximity of features to defects. Consider changing locations of those features or half-etching the bottom of the stencil to accommodate them.

SHEA ENGINEERING SERVICES

Poor Alignment

When the solder deposit is not printed exactly where it's supposed to.





SHEA ENGINEERING SERVICES

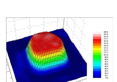
Poor Alignment

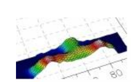
If you suspect…	Then investigate:
Board Support	Check (clean or improve) PWB support
Printer alignment error	Check printer fiducial reading routine. Watch fiducial find on screen
Stencil mesh torn or tension too loose	Check for stencil movement at beginning of print stroke
PWB or stencil positional error	Corner-to-corner alignment of apertures and pads.
PWB shrink or stretch	Corner-to-corner alignment. If alignment cannot be achieved, stencil can be scaled to compensate for PWB error.

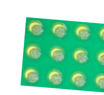
SHEA ENGINEERING SERVICES

Summary - Troubleshooting

- **Understand the key elements in the solder paste printing process**
- **Maintain control of the process**
  - It's where most of the rework comes from
  - It's where the money is in SMT!
- **When problems arise, first do the 5-Minute Overall System Check**
  - 80% chance that you resolve the problem
- **If specific defects continue to occur, follow logical troubleshooting guidelines**







SHEA ENGINEERING SERVICES

Thank You!

Contact info:

Chrys Shea
Chrys@SheaEngineering.com

SHEA ENGINEERING SERVICES

Communicating Expertise