



Considerations for Printing Lead-Free Solder Pastes

David Suraski

AIM

Cranston, Rhode Island USA

info@aimsolder.com

www.aimsolder.com

www.leadfree.com

Abstract



- It has been reported that the SMT printing process accounts for 75% of all SMT process-related defects. That said, attention must be paid to the printing process in order to ensure reliable, repeatable soldering results.
- As with many processes, SMT printing will require reexamination and process adjustment when lead-free soldering is implemented.



Introduction

- As compared to tin-lead solder pastes, lead-free pastes should exhibit similar features on the stencil and many equipment set points should transition well.
- However, implementation of lead-free solder paste does necessitate some critical, often overlooked adjustments, as well as provides an opportunity to review and fine-tune several key printing parameters.

Paste Handling



- Shelf-lives with lead-free pastes may be reduced as compared to tin/lead, and storage conditions may be slightly more stringent.
 - Alloy dependent.
 - Some pastes require freezing.
- However, in general, the same rules as with tin/lead apply.
 - Prevent/minimize exposure to heat and humidity.
 - Allow paste to come to room temp before using.
 - Do not mix old and new paste.

Stencil Apertures



- Many manufacturers currently use reduced aperture-to-pad ratios to prevent bridging and solder beading.
- Due to differences in the solderability characteristics of lead-free circuit board finishes and the inability of lead-free solders to spread as well as tin-lead, reduced stencil apertures may need to be opened up back to a 1:1 aperture-to-pad ratio.
- This ratio should not result in bridging because density differences between lead-free and tin-lead solder pastes results in less slump with lead-free pastes.

Cycle Times



- Some lead-free pastes have shown a propensity to stick to squeegee blades after the print cycle.
- This is a result of the different densities of tin-lead and lead-free alloys.
- To combat this, print cycle times may need to be slowed.
- By slowing the print cycle time, any solder paste sticking to the squeegee blades should fall back onto the stencil prior to the next print stroke.

Squeegee Speed



- Squeegee speed is the traverse speed of the squeegee during the print cycle. These typically range around 1 to 3" per second, with many printers capable of 0.2 to 8" per second.
- When setting up a printing process, the print cycle should first be adjusted to an on-demand cycle rate, where the printer is printing “just in time” for the placement equipment.

Squeegee Speed



- The faster the squeegee speed is, the higher the squeegee pressure generally must be.
- As squeegee speed and pressure increase, so does the amount of friction generated at the squeegee/stencil interface.
- This can promote greater paste shear, possibly resulting in slumping, under stencil bleed-out, and pad-to-pad bridging.

Squeegee Pressure



- Squeegee pressure is the downward pressure exerted by the squeegee blade onto the stencil surface during the print cycle.
- A typical starting point for squeegee pressure for a lead-free solder paste is 1.5 to 2 lbs. of pressure per linear inch of printable area.

Squeegee Pressure



- Squeegee pressure should be adjusted to just high enough to achieve a good, clean, topside wipe of the stencil surface. Leaving paste behind on the stencil surface can promote poor aperture release, torn prints, insufficient solder coverage and premature paste dry out.

Separation Distance



- Separation distance is an adjustable distance to which the printed circuit board (PCB) and stencil separate at a controlled speed at the completion of the print stroke.
- Working in conjunction with separation speed, this controlled parting continues until the separation distance set point has been reached, at which time the speed of PCB and stencil separation increases to its maximum.
- This function allows for controlled clearance of the solder paste from the stencil apertures, providing for more uniform and repeatable paste deposits.

Separation Distance



- Separation distance should be enough to allow all deposited paste to clear the stencil apertures prior to the increase in separation speed, which occurs when the separation distance set point has been reached.
- In general, the greater the separation distance and slower the separation speed, the more reliable and repeatable the paste deposition.



Separation Speed

- An adjustable velocity that, in conjunction with separation distance, works to control post-print separation of PCB and stencil.
- At the completion of the print stroke, and after any squeegee up or separation delay, the separation of PCB and stencil begin at a controlled speed defined by separation speed.
- This controlled speed and separation continues until the pre-set separation distance has been reached, at which time separation speed increases to its maximum.
- As with separation distance, this parameter is designed to aid the separation of solder paste from the stencil apertures.
- In general, the slower the separation speed the more reliable and repeatable the paste deposition.
- *Too fast of a separation speed can result in dog ears / peaking / wicking, clogged stencil apertures and poor paste coverage.*

Snap-Off



- Snap-off is the programmable distance between the topside of the PCB and the bottom side of the stencil.
- If equipment is calibrated correctly, board thickness is properly set, and snap-off is set to zero, this should result in on-contact printing. This setting allows for full “gasketing” of the stencil to the component pads during the print cycle, thus preventing bridging related to paste bleed-out, even on ultra-fine-pitch components.
- On-contact printing also provides for a more uniform paste deposition and more consistent paste height.



Stencil Cleaning

- Stencils should be cleaned frequently enough to ensure total removal of any bottom side residues, but not so infrequently as to allow the same residues to dry or cake on, making their removal much more difficult.
- Some applications may require cleaning after every board printed, while others may be able to clean the stencil a surprising one time per shift, or even not at all.
- Regardless of how often you do clean your stencil, one sure method of determining whether or not the job is getting done is to remove the stencil and visually inspect the bottom side.

Cleaning Solutions



- It is recommended that you first attempt to perform dry wipes when using an automated under stencil wipe.
- If a dry wipe doesn't do the job, then an appropriate stencil cleaning solution is recommended.
- A good cleaning solution should facilitate the removal of under stencil residues without damaging the solder paste it contacts.
- Any foreign substance entering a paste's chemistry is considered a contaminant and could do irreparable harm.
- Only materials that are 100% compatible should be used.

Lead-Free Paste Printing Requirements



- Easy to achieve clean stencil wipe
- Do not clog apertures, even on fine-pitch applications
- Prints high-speeds without slumping or insufficients
- Long stencil life
- Compatible with enclosed paste deposition systems
- Long tack time for batch operations

Lead-Free Vs. Tin-Lead Solder Paste Comparison

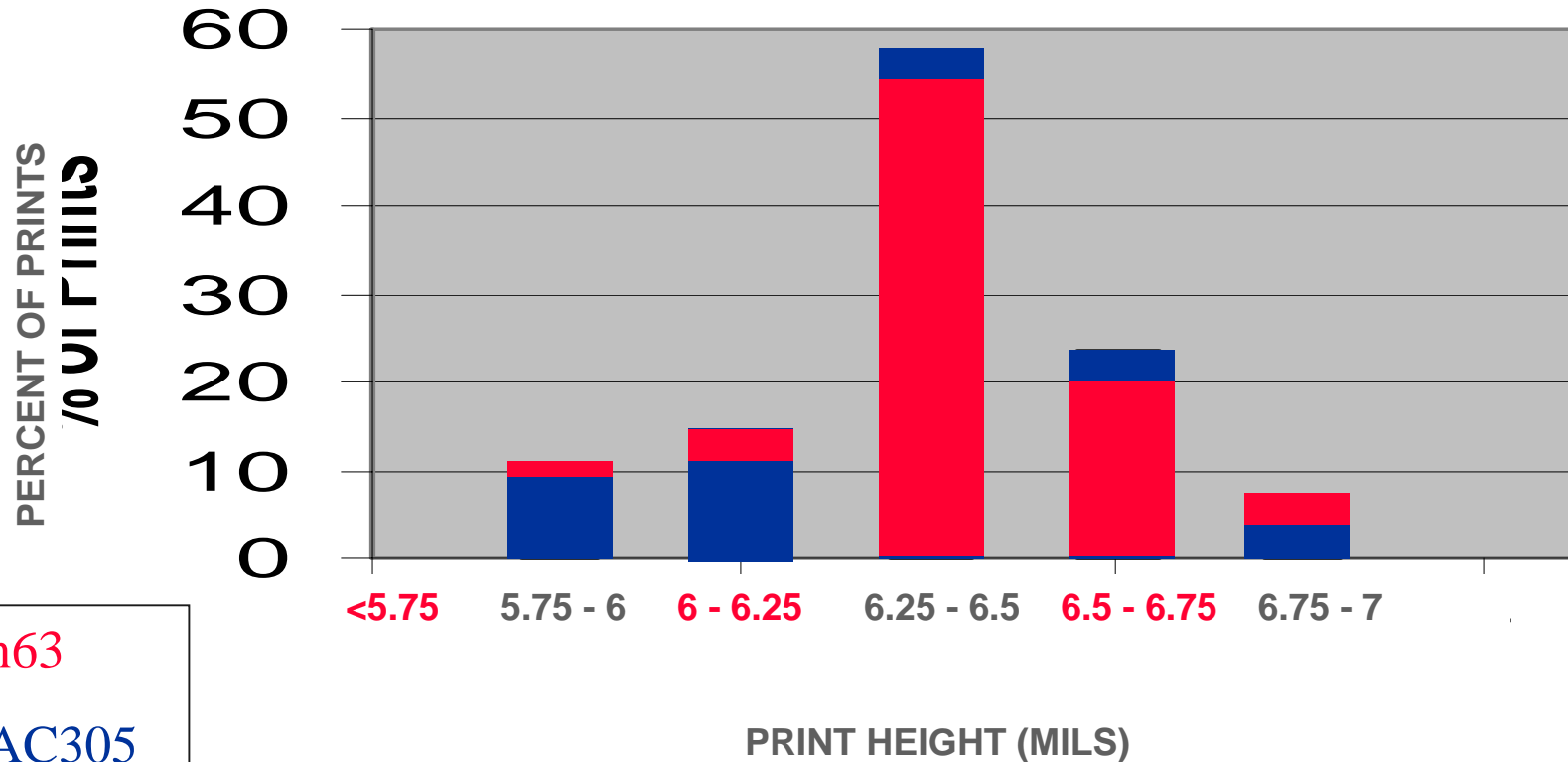


- Same flux chemistry used in conjunction with Sn63 and SAC305.
- Metal load optimized for alloy.

Print Height Consistency Analysis

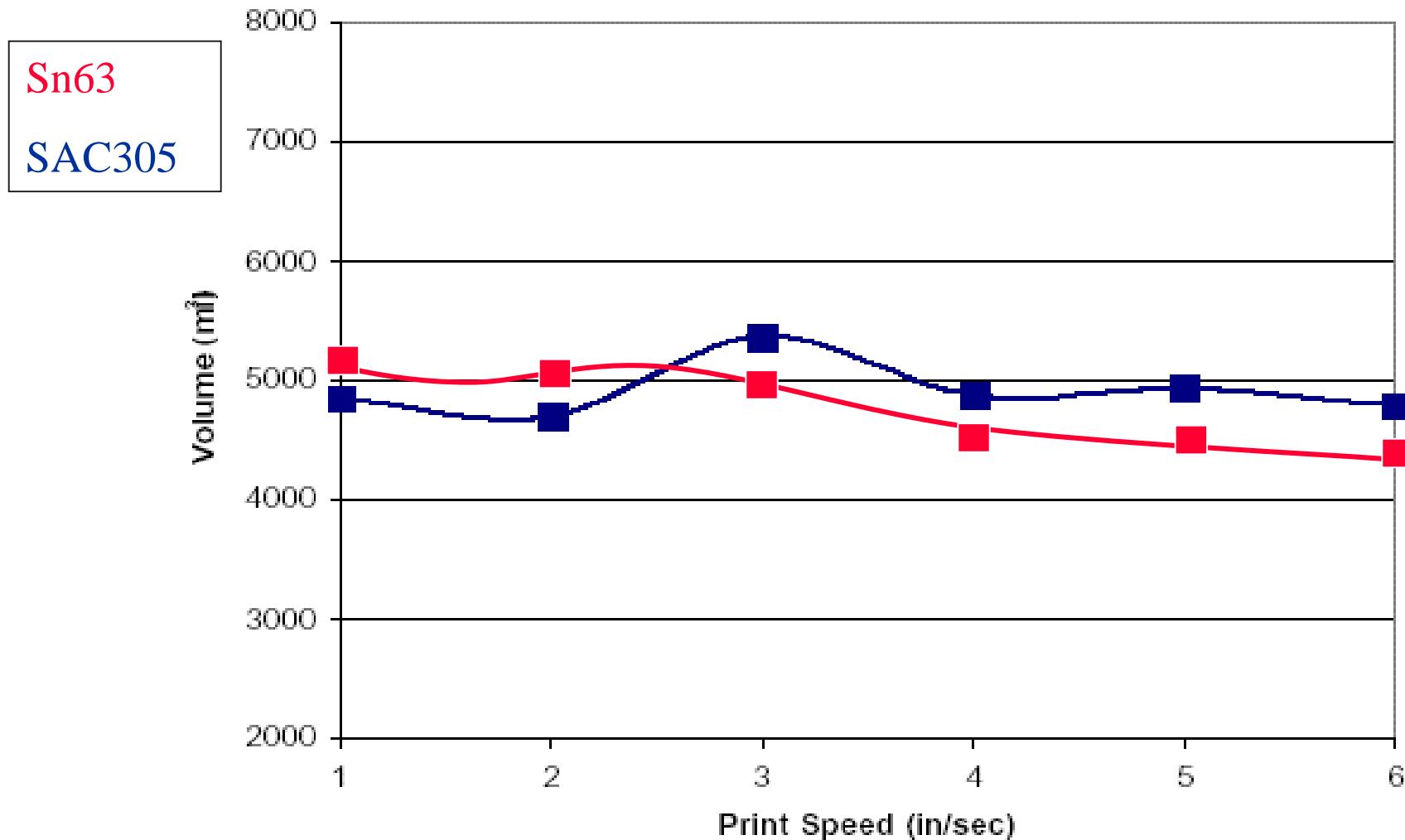


30 boards were printed and a specific pad was chosen to determine the print height consistency between boards.



Sn63
SAC305

Consistency of Print Volumes During Standard to High-Speed Printing



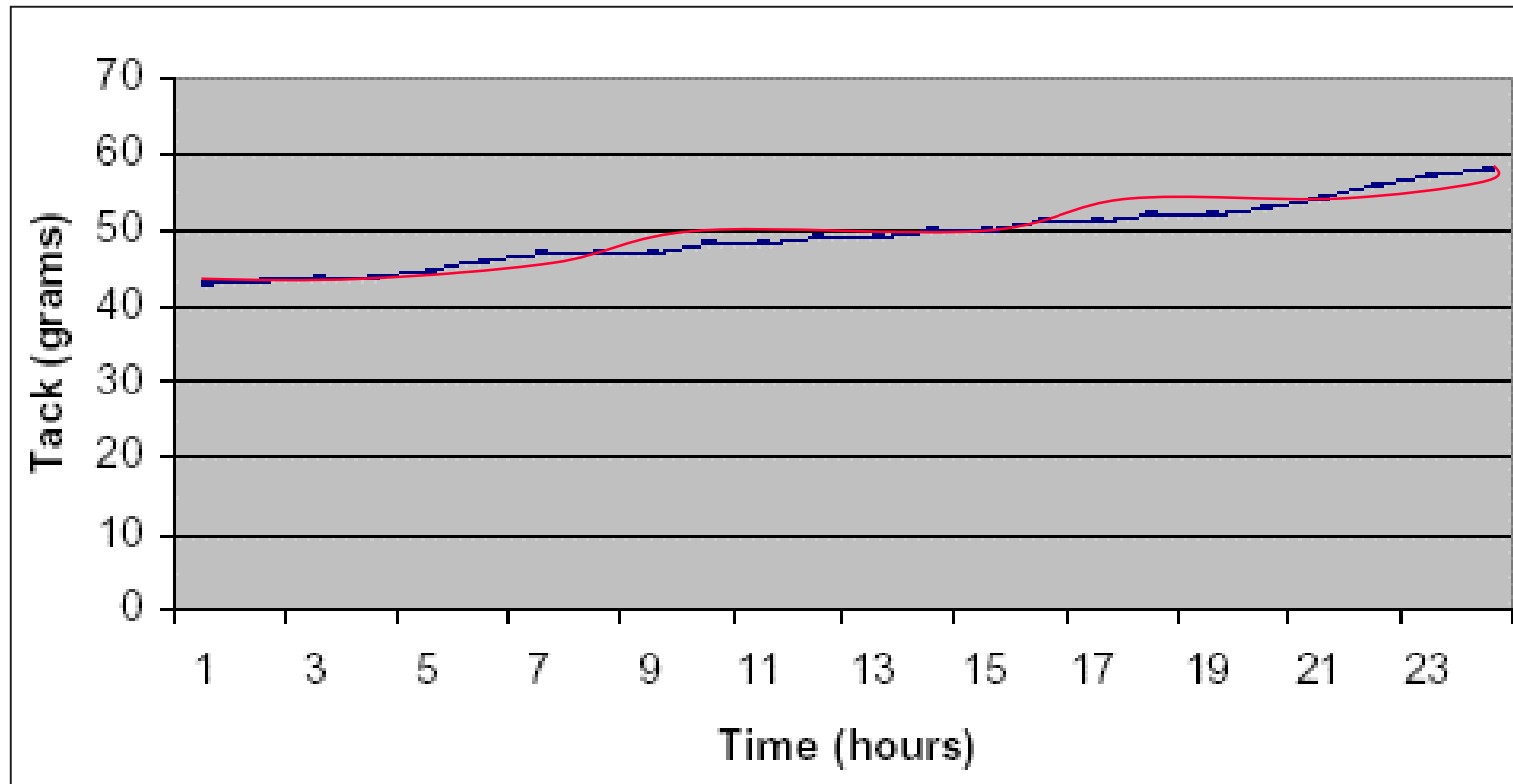
Tack Characteristics



Tack versus Time at Ambient Conditions (72°F (22°C)
at 40% RH) over 24 hours.

Sn63

SAC305



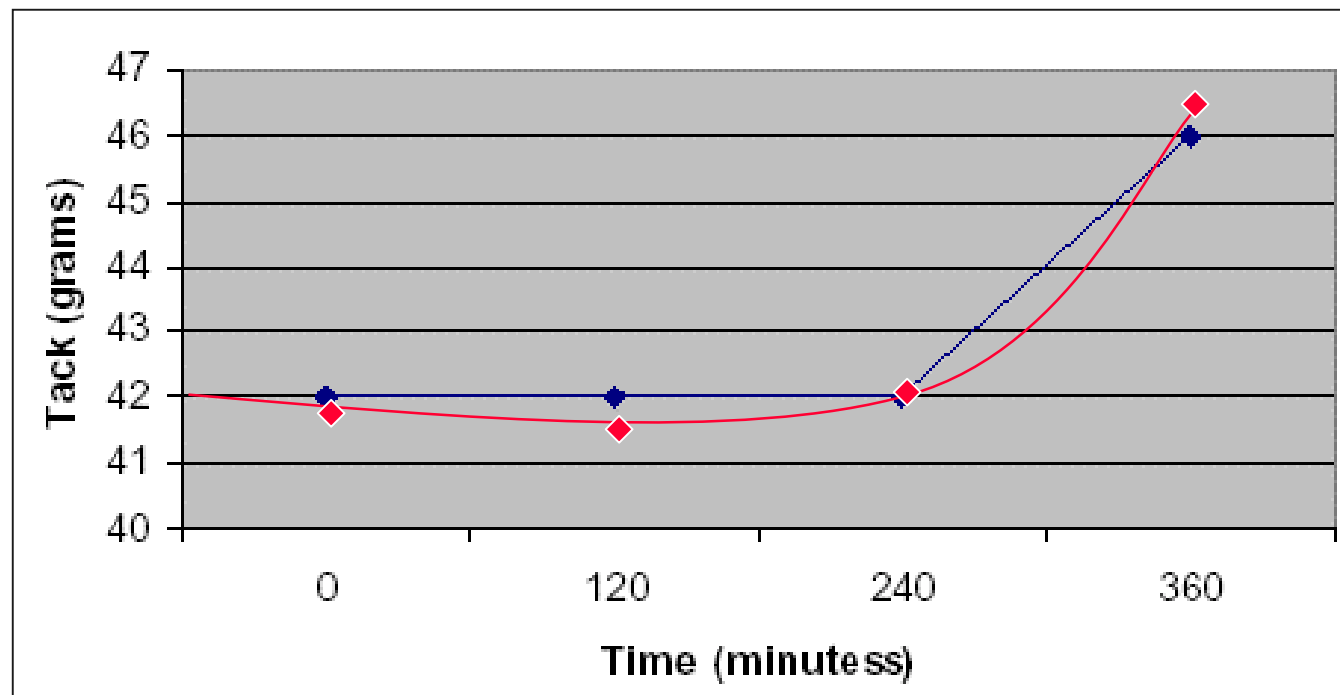


Tack Characteristics

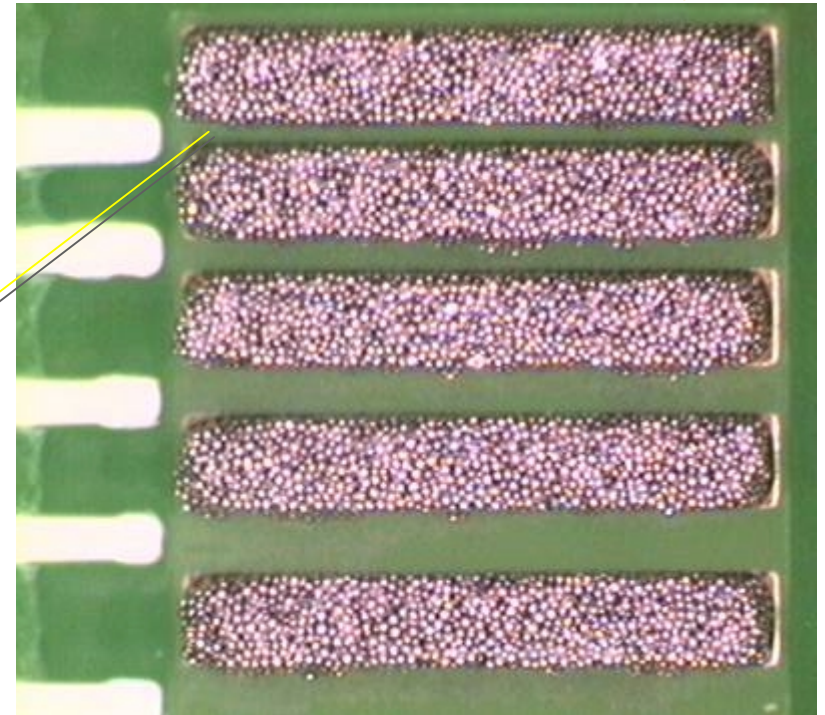
Tack versus Time at Humid Conditions (72°F (22°C) at 75% RH) per IPC-TM-650.

Sn63

SAC305

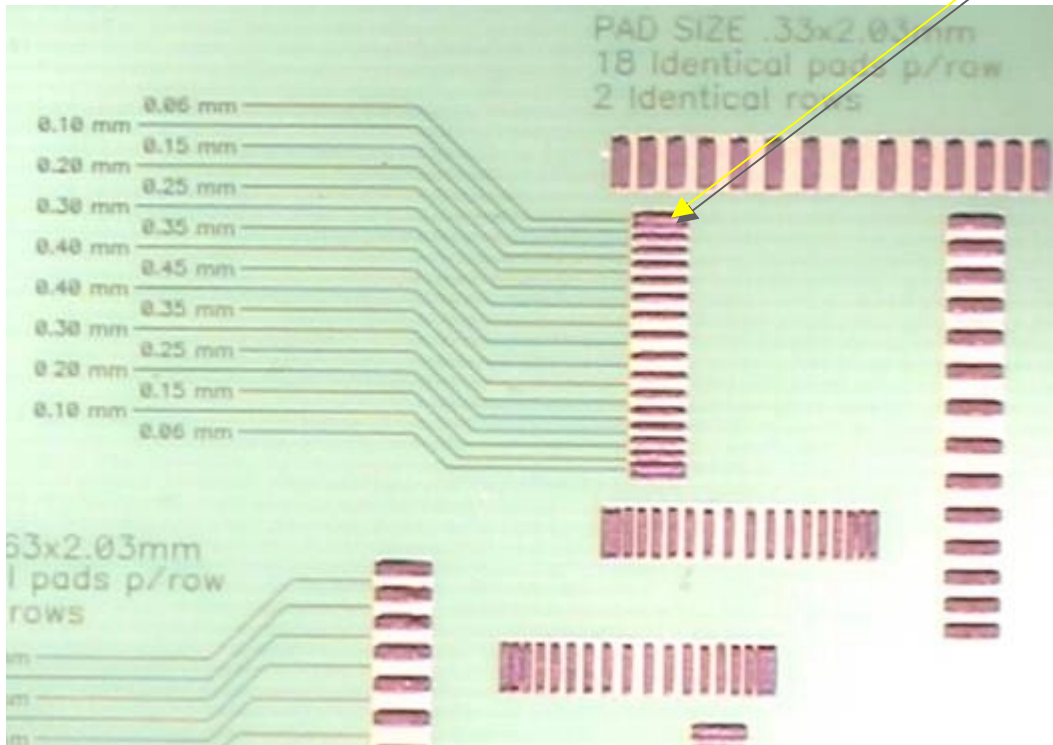


Fine Pitch Printing Without Slumping

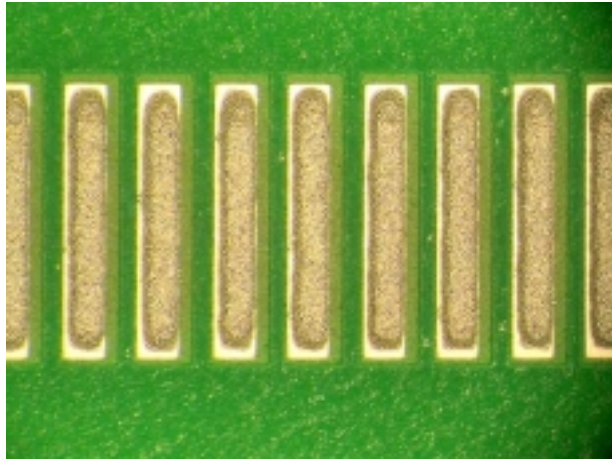


.06mm gap

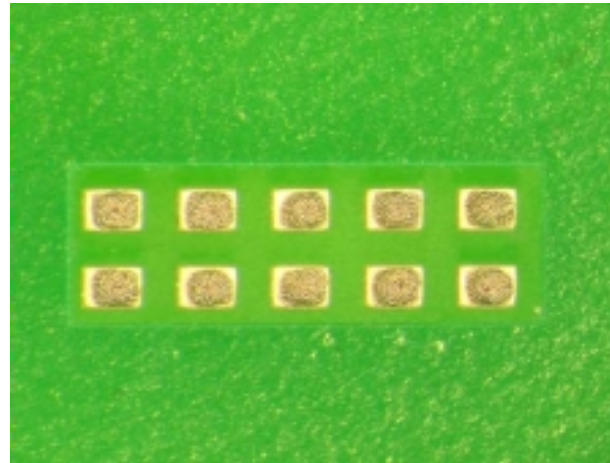
IPC Slump Test Pattern



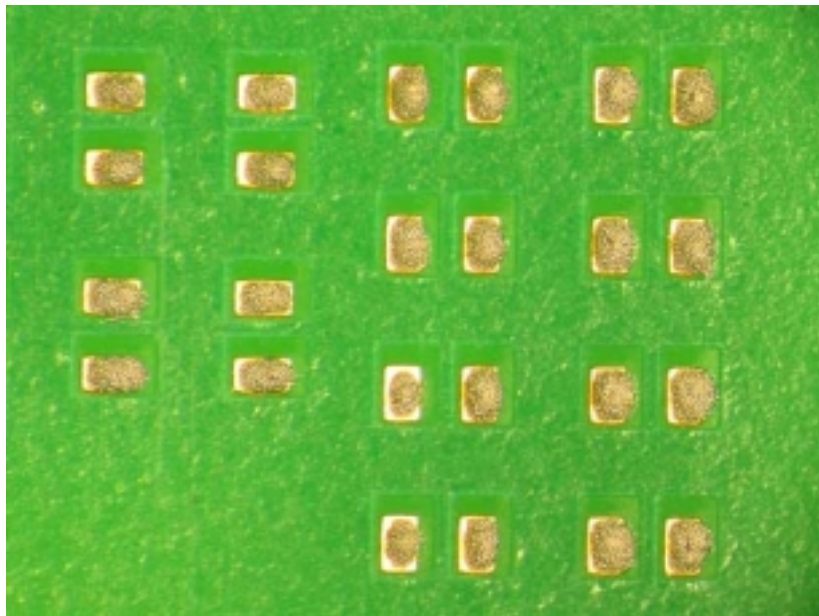
Lead-Free Solder Paste Fine Pitch Printing



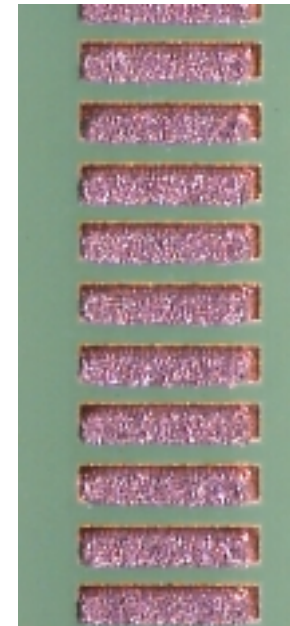
QFP 0.020" pitch



0201 Components



01005 Components



8 mil gap

Conclusion



- High-output, repeatable lead-free SMT printing is taking place at many manufacturers today.
- While printing results significantly depend on the specific solder paste being implemented, it is equally critical to ensure that the appropriate print settings are understood, instituted and verified.
- Issues such as aperture-to-pad ratios, print cycle times, squeegee speed and pressure, separation distance and speed, and snap-off should be examined closely and adjusted in accordance with the solder paste in use and the specific application.