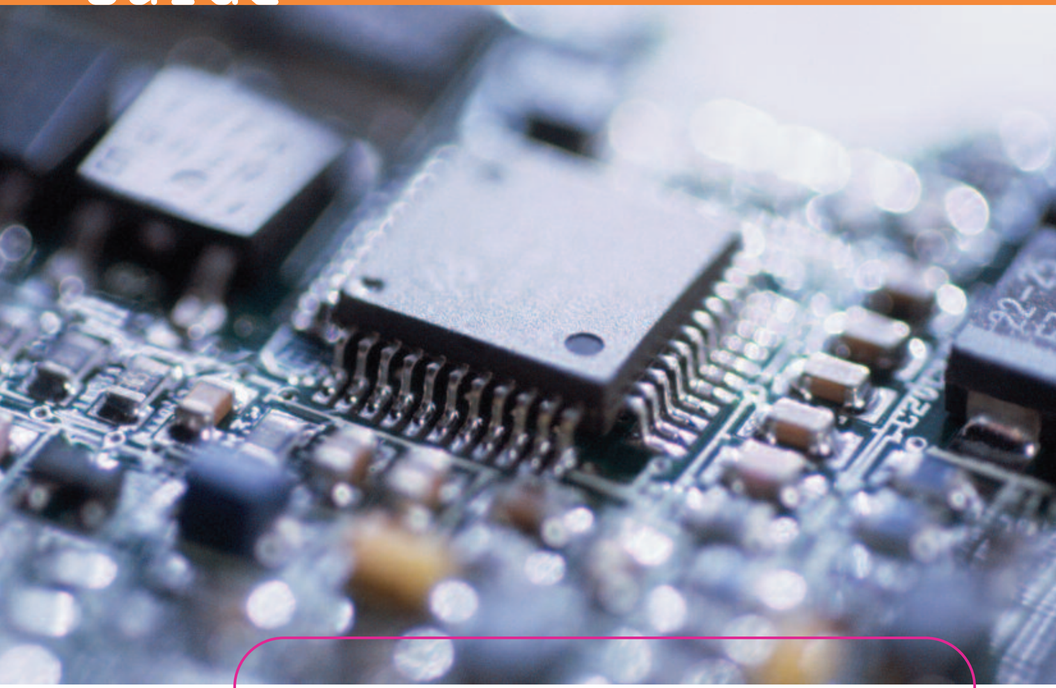
SMT Troubleshooting Guide



Easy-to-use advice for common SMT assembly issues.

Cookson Electronics

www.alpha.cooksonelectronics.com

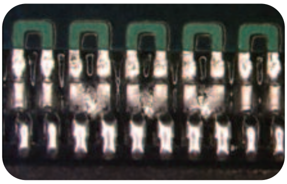
With this easy-to-use Troubleshooting Guide, you can learn to troubleshoot common SMT issues. After using it a few times, it will become an essential companion for you and anyone in your company responsible for operating an SMT line.

This Guide offers troubleshooting advice for common SMT assembly issues by process defect. If your issue is not resolved after following the steps to help identify the possible root cause and solution, please contact your Cookson Electronics representative who will be able to provide you with further assistance.

Table of Contents

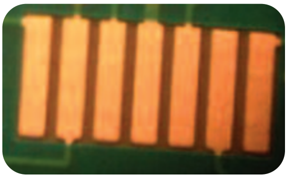
Bridging

Definition: Solder connecting, in most cases, misconnecting two or more adjacent pads that come into contact to form a conductive path.

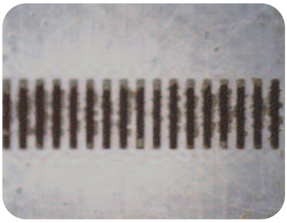

Possible Causes: PCB

Description	Recommendations
SMD pads will contribute to coplanarity issue resulting in poor gasketing during printer setup.	Highly recommended to remove solder mask between adjacent pads especially for fine-pitch components



Possible Causes: Stencil

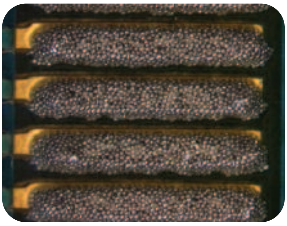
Description	Recommendations
Dirty stencil with paste underneath will contaminate the bare board on the next print, attributing a potential bridge.	• Verify zero print gap set up. • Ensure minimum print pressure. • Increase wipe frequency. • Use different cleaning chemicals.
Stencil tension	Ensure stencil tension is tight. Poor stencil tension will make it impossible to have a good setup for consistent print definition.
Aperture Design	For fine pitch component, it is highly recommended to have the opening slightly smaller than landing pad size to improve stencil to PCB gasketing.

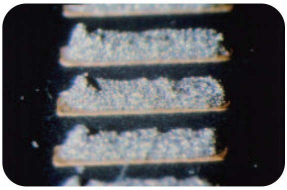


Bridging

Possible Causes: Screen Printer

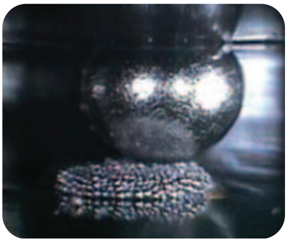
Description	Recommendations
Poor gasketing – paste oozes out beneath stencil during printing, increasing chance of wet solder paste bridges	• Zero print gap between stencil and PCB • Check paste smear underneath stencil. • Check sufficient stencil tension.
Misaligned print will challenge the paste to pull back to pads during molten stage, increasing the potential for bridging.	Ensure print accuracy and consistency for both print strokes.
Smearing and bridging phenomenon on the next printed board after stencil cleaning operation	• Verify stencil is dry after cleaning and before next print. • Standard cleaning mode is wet/vacuum/dry.
Poor print definition with dog ears especially on fine-pitch components	• Check board support. • Adjust separation speed to achieve minimum dog ears. *NB: Different paste chemistry requires different separation speed to minimize dog ears.*
Dented squeegee blades could result in uneven print pressure.	Check squeegee blades condition.

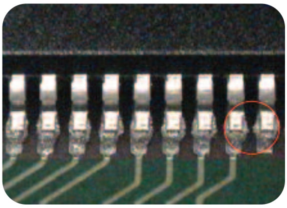




Possible Causes: Component Placement

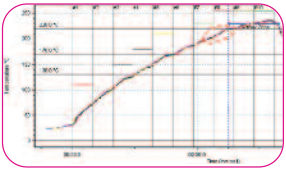
Description	Recommendations
Placement inaccuracy will narrow the gap between pads, increasing the chance of bridging.	• Verify component placement pressure. • Use X-ray to verify BGA placement. • Use microscope for QFPs.
Excessive component placement pressure will squeeze paste out of pads.	• Verify actual component height against data entered in the machine • Component placement height should be ±1/3 of paste height.





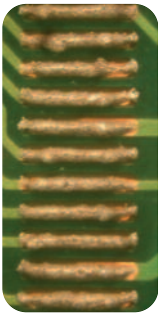
Possible Causes: Reflow Profile

Description	Recommendations
Extended soak will input more heat to the paste and result in paste hot slump phenomenon.	Adopt a straight ramp to spike profile, without soak zone if possible.

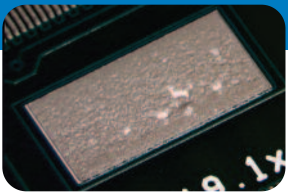


Possible Causes: Solder Paste

Description	Recommendations
Dry paste phenomenon – irregular print shape and inconsistent print volume	• Paste expiry • Operating temperature within supplier's recommendations. Check temperature inside printer. Normal requirement around 25°C, 50%RH • Do not mix using new and old paste.
Paste oozes out of pads, may form connection with adjacent pads.	• Operating temperature within supplier's recommendations • Verify with another batch of paste to confirm problem is batch-related. • Perform cold and hot slump test result using IPC-TM-650 Method 2.4.35.



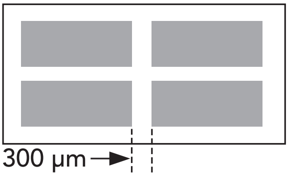
Insufficient Fill



Definition: Amount of solder paste deposited on PWB at printer station is much less than stencil opening design.

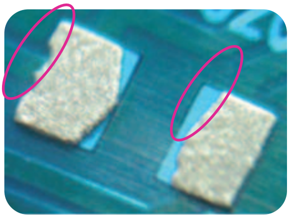
Possible Causes: Stencil

Description	Recommendations
Paste scooping effect especially on large pads	Segment the large opening into smaller apertures.

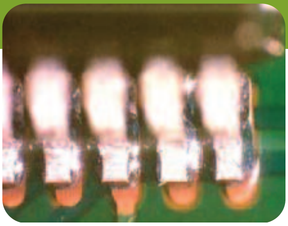


Possible Causes: Screen Printer

Description	Recommendations
Paste does not roll into aperture	• Reduce print speed. • Increase print pressure. • Adopt lower squeegee contact print. • Ensure paste is not expired or dry. • Ensure sufficient board support. • Reduce squeegee pressure.



Insufficient Solder



Definition: Amount of solder paste deposited on PWB at printer station is much less than stencil opening design or, after reflow, insufficient solder to form a fillet at the component leads.

Possible Causes: Stencil

Description	Recommendations
Solder paste adheres on the stencil aperture walls	• Area ratio > 0.66 • Aspect ratio > 1.5 • No burr on stencil aperture edge

Possible Causes: Screen Printer

Description	Recommendations
Print definitions	• Verify print setup • Reduce print speed to provide sufficient time for paste to roll into aperture.

Possible Causes: Reflow Profile

Description	Recommendations
Mismatch in CTE between component and PCB can cause solder wicking effect which may look like insufficient solder on pads.	• Attach thermocouple on component and PCB. • Apply soak profile to minimize delta T before reflow zone. • Set bottom zones to be higher temperature if possible, to keep PCB hotter than component leads.

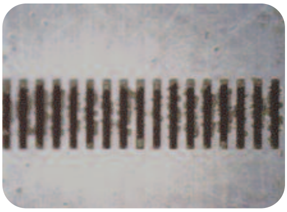
Possible Causes: Solder Paste

Description	Recommendations
Solder paste viscosity	Check paste conditions such as dry paste phenomenon by verifying if paste rolls or skids along print direction.

Random Solder Balls

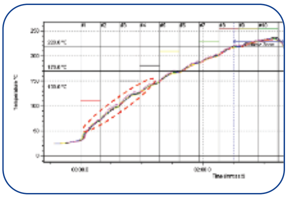
Definition: After reflow, small spherical particles with various diameters are formed away from the main solder pool.

Possible Causes: Stencil



Description	Recommendations
Paste stuck under the stencil will be transferred onto the solder mask of the next PCB.	• Verify zero print gap set up. • Check minimum print pressure used. • Check cleaning efficiency such as wet/dry/vacuum. • Check wipe frequency.

Possible Causes: Reflow Profile



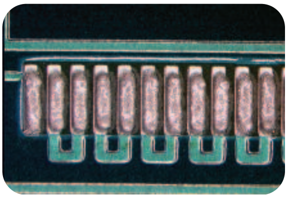
Description	Recommendations
Fast ramp-up rate or preheat rate will not allow sufficient time for the solvent to vaporize off gradually.	Slow preheat rate is recommended, typically < 1.5°C/sec from room temperature to 150°C.

Possible Causes: PCB Moisture

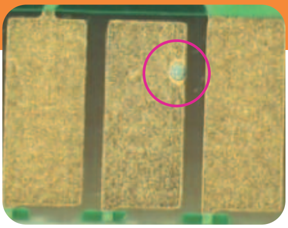
Description	Recommendations
Trapped moisture may result in explosive vaporization.	Especially for lower grade PCBs such as FR2, CEM1, tends to absorb moisture. Bake 120°C for 4 hours if necessary.

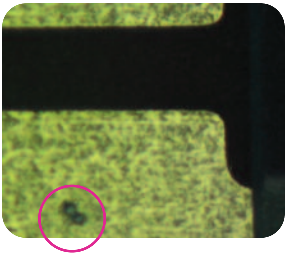
Possible Causes: Solder Paste

Description	Recommendations
Especially for water-soluble solder paste which is hygroscopic, it tends to have limited stencil life because of moisture absorption.	• Minimize exposure time • Printer temperature and humidity to be within recommendation • Try new lot of solder paste to verify paste integrity. • Use coarser powder size if possible as fine powder size has more oxides and tends to slump more readily.



Solder Spattering





Definition: Solder Spatter phenomenon is very similar to solder balling, but the concern is usually about solder deposited onto Au fingers.

Possible Causes: PCB

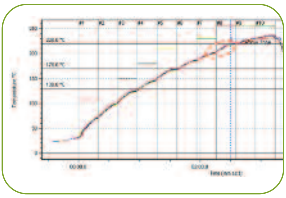
Description	Recommendations
Handling of boards	• Do not mix clean and washed boards. • Open fresh PCBs from package when ready to run. • Ensure working area is cleaned thoroughly and not contaminated with solder paste remains.
Bare boards contamination	Inspect bare PCBs to capture and filter solder found on bare PCB before printing station.

Possible Causes: Screen Printer

Description	Recommendations
Ineffective cleaning of stencil wipe will transfer small particles of solder onto the top surface of the next bare board.	• Ensure wipe frequency is set correctly. • Use effective solvent, preferably SC10. • Use printer machine camera to inspect the effectiveness of stencil cleaning.

Possible Causes: Reflow Profile

Description	Recommendations
Control the flux out-gassing rate to minimize explosive solder scatter on Au pads.	For SAC 305, set slow ramp rate of 0.3-0.4°C/sec from 217-221°C.

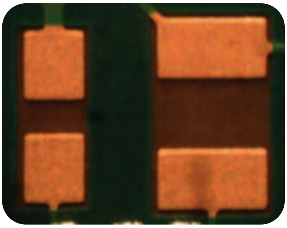


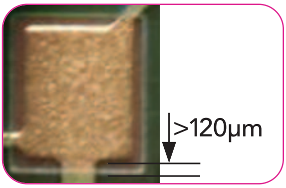
Mid-Chip Solder Balls (MCSB)

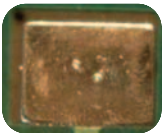
Definition: After reflow, large solder ball(s) is/are located on the side of the chip components, between the terminations and away from the pads.

Possible Causes: PCB

Description	Recommendations
Solder dissociation and does not adhere on solder mask.	• Remove solder mask between pads. • Gap between pad and solder mask is recommended to maintain at least 75µm~100µm, preferably >120µm. • Solder mask may not be centralized around pad.

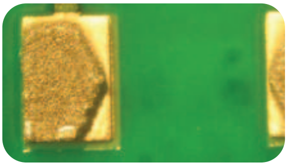


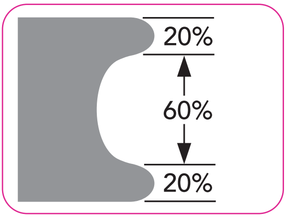




Possible Causes: Stencil Design

Description	Recommendations
Excess paste squeezes underneath component body tends to dissociate with the main body of solder during reflow.	Home plate or U-shape design may help to reduce the amount of paste potentially squeezed under the component body, onto the mask. *NB: Aperture reduction may not be suitable for component size smaller than 0603. Besides, LF alloy has higher surface tension and does not pull back after reflow.*





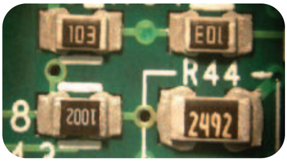
Mid-Chip Solder Balls (MCSB)

Possible Causes: Screen Printer

Description	Recommendations
Paste smearing on solder mask	• Printer set up for zero print gap, verified by paste height consistency without dog ears • Print alignment accuracy

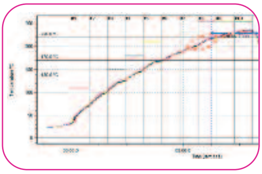
Possible Causes: Component Placement

Description	Recommendations
Excessive placement pressure will squeeze paste on pad	• Verify actual component height against data entered in the machine. • Component placement height should be ±1/3 of paste height.



Possible Causes: Reflow Profile

Description	Recommendations
Extended soak will input more heat to the paste and result in paste slump phenomenon.	Adopt a straight ramp to spike profile, without soak zone if possible.

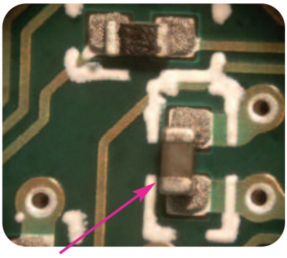


Tombstoning

Definition: A tombstone, sometimes called Manhattan effect, is a chip component that has partially or completely lifted off one end of the surface of the pad.

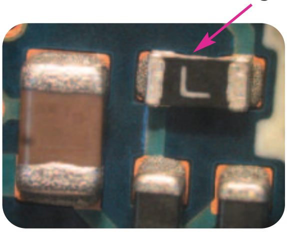
Possible Causes: Pad Design

Description	Recommendations
Component body must cover at least 50% of both pads.	If component terminations are not covering >50% of pads, high tendency to have imbalance wetting force, resulting in tombstoning. Feedback to supplier for alteration if possible.
Unequal pad size especially with ground pad on one side	Unequal size means different solder volume, increasing potential for unequal wetting force. If due to design limitation, use a gradual soak ramp rate just before reaching liquidus point, e.g., SAC305, soak @ 190-220°C for 30-45 sec.



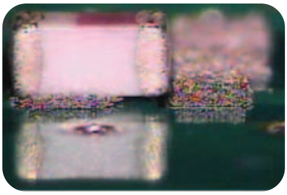
Bad design

Good design



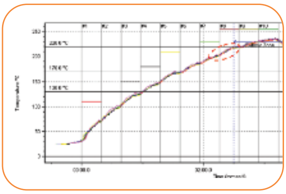
Possible Causes: Placement Accuracy & Pressure

Description	Recommendations
Skew placement will create imbalance wetting force on both pads.	Check other components placement accuracy. Re-teach fiducials if all component shifted, else edit that specific location manually.



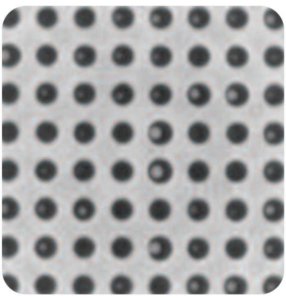
Possible Causes: Reflow Profile

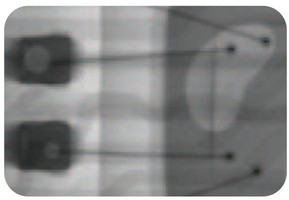
Description	Recommendations
Extend soak zone can aid in balancing the wetting force on both pads before paste reaching to molten state	Focus at 30°C before alloy liquidus temperature, e.g., for SAC305, liquidus @ 220°C, ensure soak at 190~220°C for minimum of 30 seconds



Voiding

Definition: Voids in solder joints are empty spaces within the joint, increasing concern about voiding, especially on BGAs and large pads such as LGAs. Two main contributors of voiding are
(i) outgassing of flux entrapped
(ii) excessive oxidation.


BGA


LGA


Passive Component

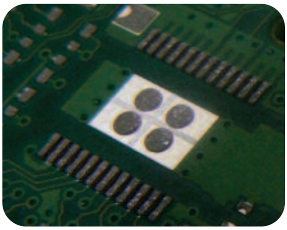
Possible Causes: PCB

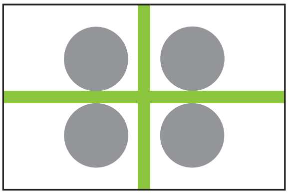
Description	Recommendations
Micro-via holes on pads trapped flux and air pockets	• Typically via holes <6mils will be more difficult to vaporize the flux or air trapped. • Plug the blinded via holes before printing. • Double print helps to pack more solder paste into via holes. • Use finer powder size. • Avoid printing paste on top of via holes, instead aperture opening designed around it.

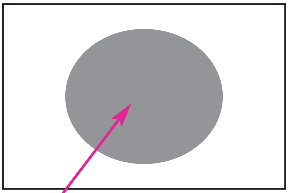
Voiding

Possible Causes: Stencil

Description	Recommendations
For large pads such as LGA, massive solder volume has a lot of flux to vaporize during reflow. Any trapped flux will result in voids.	• Reduced amount of solder deposit • Total solder volume reduction can be as high as 45%. • With solder mask in between, break the large aperture into small openings. • Without solder mask, cut a large round opening in the middle.



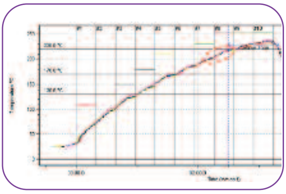




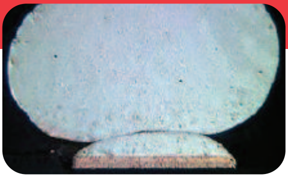
55% opening

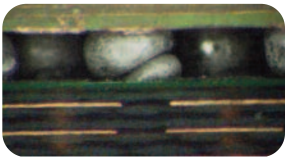
Possible Causes: Reflow Profile

Description	Recommendations
Flux entrapped without sufficient time to outgas	• Establish soak zone from 170~220°C for 60-90 sec. • Also make sure profile set between 130~220°C for 180 sec.
Oxidation rate predominates	• Adopt short profile concept to preserve flux activity, no soak zone. • Use nitrogen if possible.



BGA Head-on-Pillow

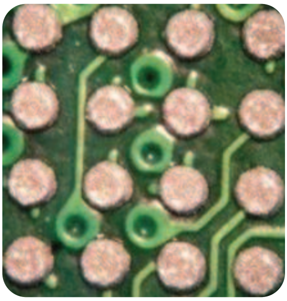




Definition: Head-on-pillow is an assembly defect in which the spheres from a BGA or CSP don't coalesce with the solder paste on the PCB pad. It is important to differentiate head-on-pillow from a defect caused simply by insufficient reflow temperature, which is characterized by distinct solder spheres from the paste that have not been properly melted on the pad and BGA solder sphere. With head-on-pillow the soldering temperature is sufficient to fully melt the solder sphere and paste deposit, but an impediment to the formation of a proper solder joint exists.

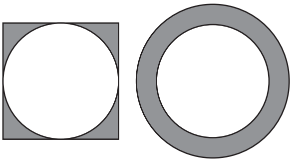
Possible Causes: Screen Printer

Description	Recommendations
Irregular print definition across the pads may hinder some solder bump locations to be in contact with solder paste.	Verify print definition and measure print height consistency



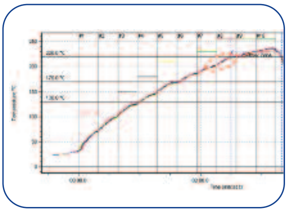
Possible Causes: PCB/Component

Description	Recommendations
Increase paste deposition volume to better compensate for substrate warpage.	Increase print volume by using square aperture vs. round opening, or enlarge overall deposition volume without jeopardizing bridging.
BGA coplanarity issue	Increase solder volume.
Oxidized BGA solder balls	• Use higher activity paste. • Use nitrogen reflow.

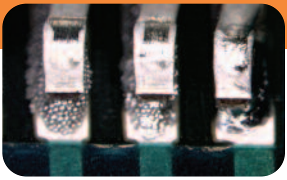


Possible Causes: Reflow

Description	Recommendations
Board warpage especially for double reflow boards or thin PCBs (<1mm thick)	• Critical to minimize time above Tg, (typically 130°C for FR4 boards) with BGAs mounted. Target to maintain < 2 min if possible. • For second reflow cycle, try to adopt lower preheat to reduce warpage occurrence.
Variance in CTE between PCB and BGA	Ensure minimum delta temperature difference between the BGA component and the rest of the components on the board. Apply short soak if necessary.
Paste hot-slump effect will aggravate BGA open joints if there are coplanarity issues.	Minimize time from 150°C to liquidus temperature.
Long soak profile may exhaust the flux capacity before reflow.	If a long soak is mandatory for complex board, use nitrogen cushion the flux capacity in overcoming oxidation rate.



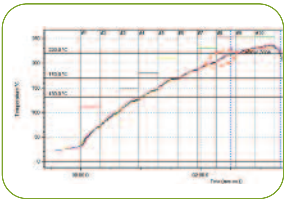
Grainy Joints



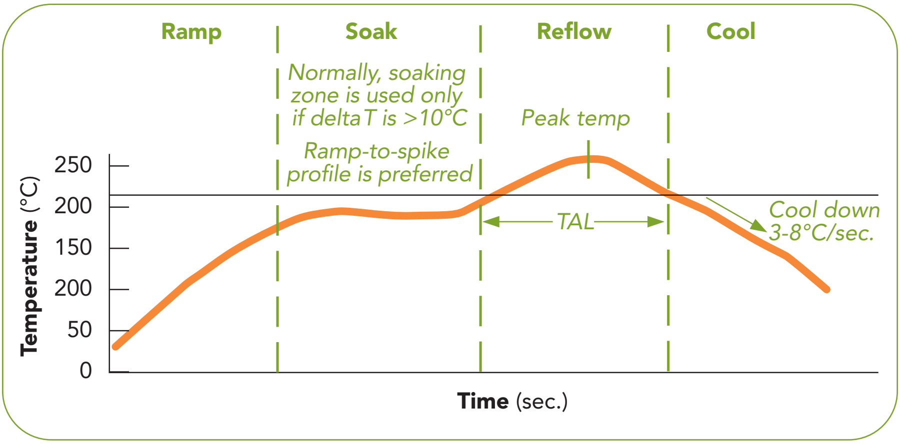
Definition: Sometimes called "Cold Solder," it is recognized by dark, non-reflective, rough surfaces from an alloy that is normally bright and shiny.

Possible Causes: Reflow

Description	Recommendations
Insufficient heat absorbed by the solder	Ensure a TC is properly attached to this component. Verify peak temperature is at least 15°C above alloy liquidus and time above liquidus (TAL) > 45 sec.
Excessive heat imposed	Adopt a ramp-to-spike profile with soak zone to minimize oxidation and flux exhaustion. If soaking is mandatory, use nitrogen reflow whenever possible.
Cooling rate is too slow	Ensure alloy cooling rate from molten solder is 3-8°C/sec. Fast cooling rate will result in fine-grain structure appearance and looks shiny.



Ramp-to-spike profile



www.alpha.cooksonelectronics.com

Global Headquarters
109 Corporate Boulevard
South Plainfield, NJ 07080
USA
Tel: +1-800-367-5460

European Headquarters
Forsyth Road, Sheerwater
Woking, Surrey GU21 5RZ
UK
Tel: +44 (0) 1483 758400

Asia/Pacific
1/F, Block A, 21 Tung Yuen St.
Yau Tong Bay, Kowloon
Hong Kong
Tel: 852-3190-3100

Issued 10/08
SM982

Cookson Electronics

www.alpha.cooksonelectronics.com