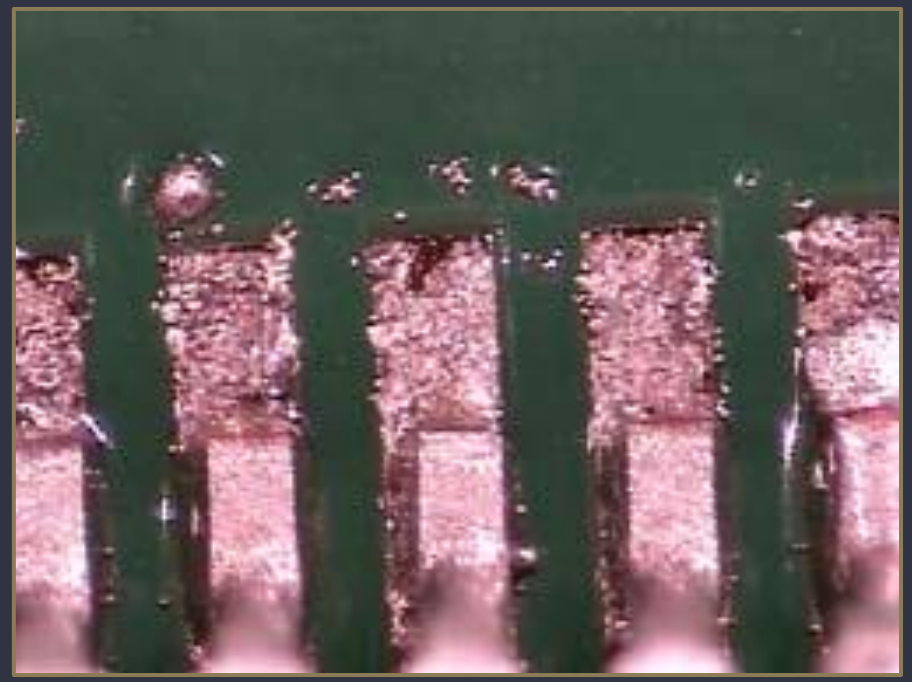
SMT TROUBLE SHOOTING GUIDE

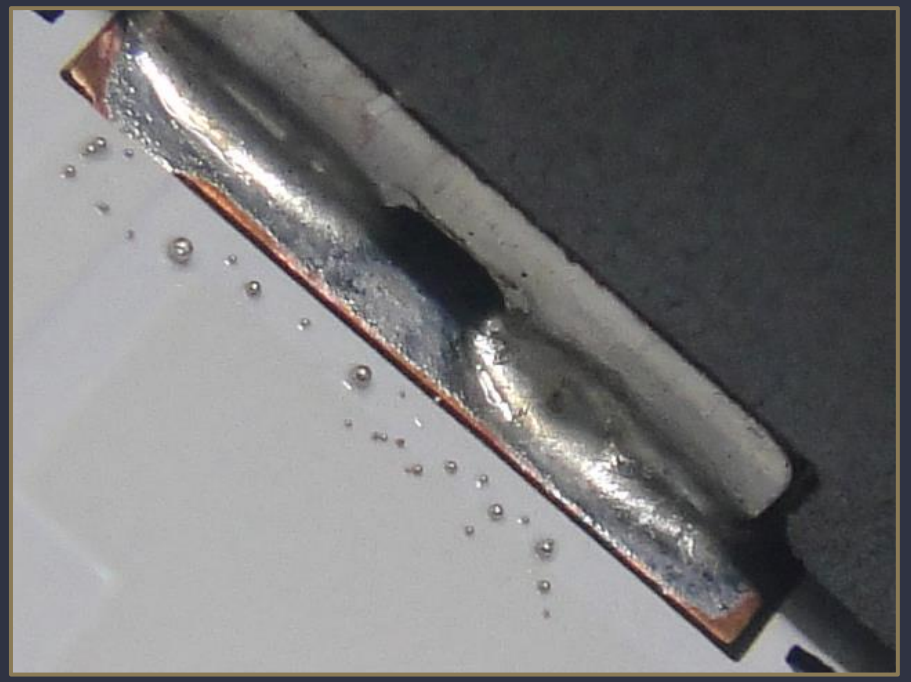
SMT TROUBLE SHOOTING OUTLINE

- Solder balling/spattering
- Solder beading
- Bridging
- Non-wet opens (NWO)/Head in pillow (HiP)
- Insufficient fillet/solder
- Voiding
- Tombstoning
- Cold solder joints
- Disturbed joints
- Excessive fillet/too much solder
- Nonwetting/dewetting
- Grainy joints/graping

SOLDER BALLING/SPATTERING

Definition: Solder balling or spattering refers to the formation of tiny solder balls adjacent to solder joints or scattered across the PCB surface after the reflow process. These unintended solder balls can potentially cause short circuits if they bridge adjacent conductive areas.





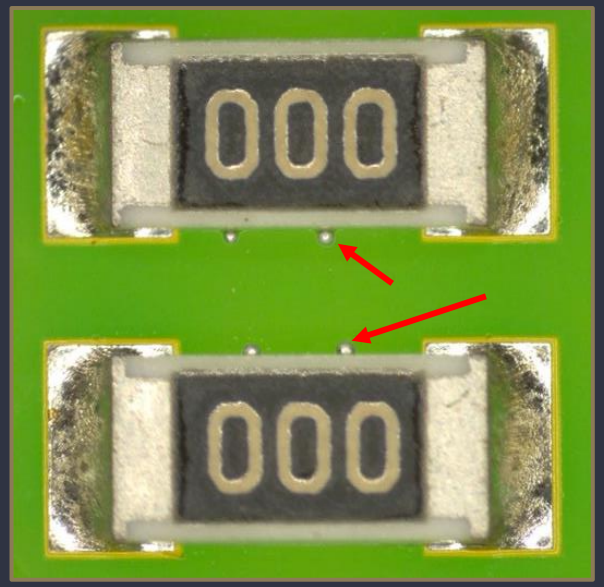
How to Recognize: Solder balling can be visually identified by the presence of small, round solder beads on or around the soldered areas of the PCB.

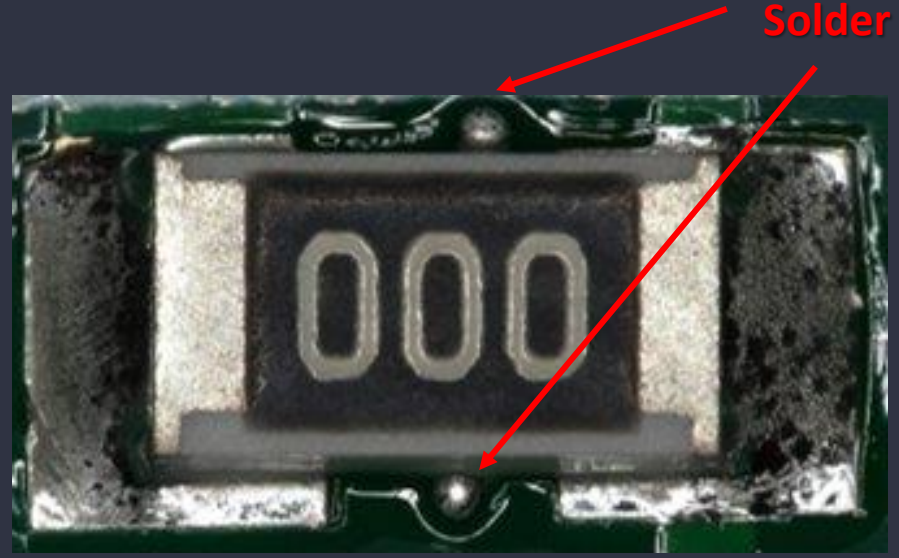
SOLDER BALLING/SPATTERING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Rapid flux volatilization	If the flux within the solder paste volatilizes too quickly, it can spatter solder particles.	Adjust the reflow profile, allowing for a more controlled flux volatilization during the soak stage.
Oxidized solder paste	Oxidized solder particles may not coalesce well with the rest of the paste, leading to isolated solder balls.	Use fresh solder paste and ensure proper storage to minimize oxidation. Make sure paste is not on PCB for too long before reflow. Consider using a mild cleaning agent if oxidation is suspected.
Aggressive reflow profile	An overly aggressive or inappropriate reflow profile can cause solder spattering.	Optimize the reflow profile to ensure a smooth and controlled melting and solidification of the solder.

SOLDER BEADING

Definition: Solder beading refers to the formation of small solder beads or balls typically located adjacent to chip components, such as resistors or capacitors, after the reflow process. Unlike random solder balling, solder beading is usually positioned close to the terminations of passive components.





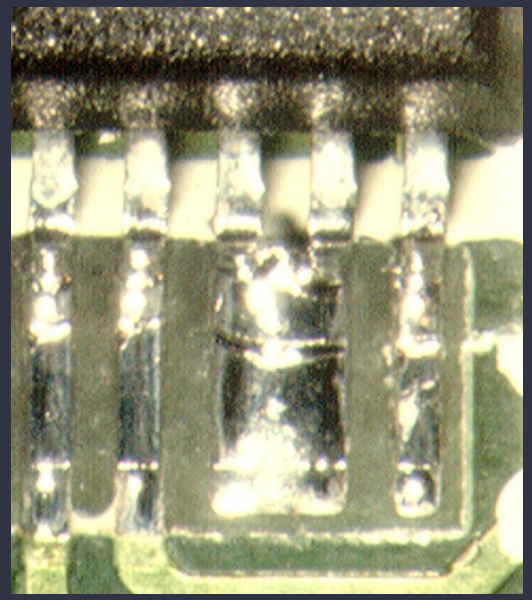
How to Recognize: Solder beading can be visually identified by spotting tiny solder balls positioned near the ends of chip components. These beads are typically larger and more defined than the more randomly dispersed solder balls caused by balling or spattering.

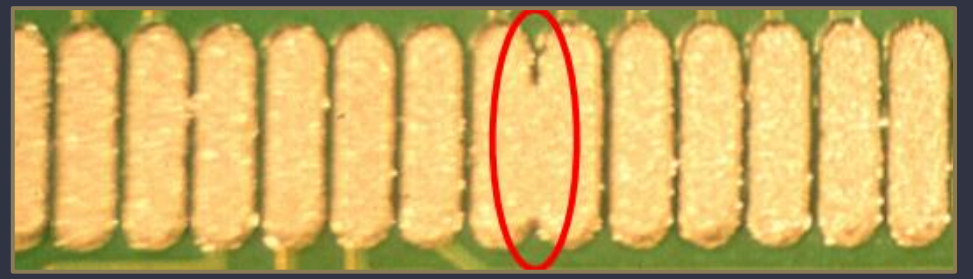
SOLDER BEADING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Excessive solder paste	Depositing too much solder paste can lead to the formation of solder beads, especially near chip components.	Start by increasing squeegee pressure to see if this resolves the issue. If not, adjust the stencil design to ensure appropriate solder paste deposition and ensure accurate printer calibration.
Reflow ramp-up too slow	A slow reflow profile ramp up can cause capillary action to draw the unreflowed paste away from the pad, leading to solder beading.	Adjust the reflow profile, using a ramp up rate of 1.5 to 2.5°C per second.
Oxidation or contamination	Contaminated or oxidized pads or solder paste can hinder coalescence, causing the solder to bead up.	Store solder paste under recommended conditions and ensure PCBs are clean and free from oxidation.

BRIDGING

Definition: A defect in which an unintended solder connection forms between adjacent pads or component leads. This can potentially create short circuits, affecting the functionality of the circuit.





How to Recognize: Bridging can be visually identified by observing solder connections between two adjacent pads or leads where there shouldn't be any. These connections can appear as thin solder filaments or broader solder masses spanning between pads.

BRIDGING: CAUSES AND SOLUTIONS

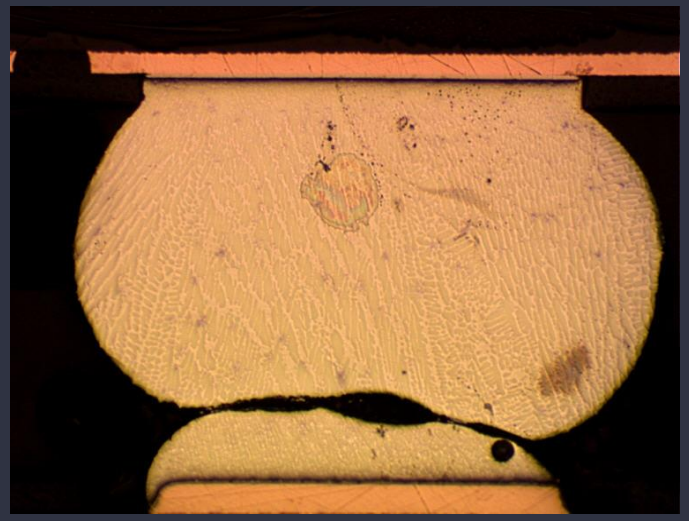
Problem	Cause	Solution
Cold slump	Paste slumps due to incorrect viscosity or as a result of shear forces from the squeegee.	Check viscosity of the paste. If correct, lower the print speed to see if the problem persists. If slumping is humidity related, try using fresh paste.
Excessive solder paste deposition	Too much solder paste is deposited on the pads, leading to an overflow that can connect adjacent pads during reflow.	Ensure proper squeegee pressure during the printing process. Adjust the stencil design, reduce aperture size or thickness if needed.
Reflow profile issues	An inappropriate reflow profile can cause paste deposits to settle or spread out during the ramp up cycle, and spread out to the adjacent pad.	Optimize the reflow profile to ensure complete and uniform melting of the solder paste. Consider shortening the ramp up time or changing the profile type.

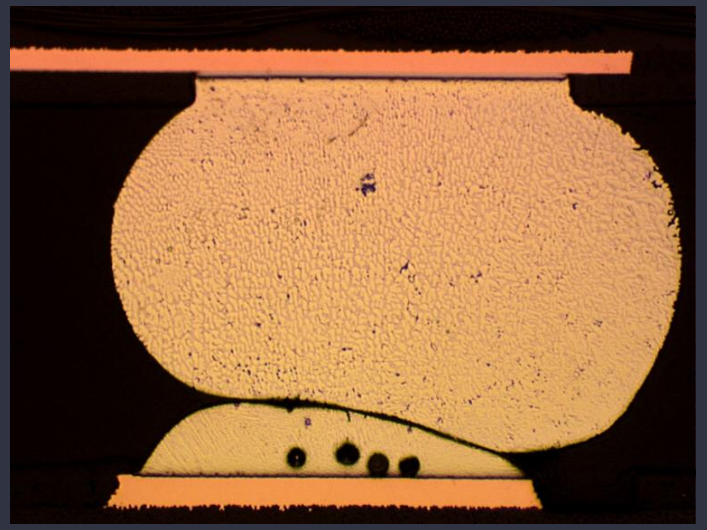
BRIDGING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Inconsistent or uneven forces	Excessive part placement pressure, insufficient board support, or incorrect alignment may push paste into places it doesn’t belong.	Check and adjust placement settings as needed.
Squeegee issues during printing	Incorrect squeegee angle, speed, or pressure during solder paste printing can lead to uneven paste deposition.	Adjust squeegee settings for optimal solder paste deposition.
PCB design issues	Pads that are too close together or improper pad design can make bridging more likely.	Review and adjust PCB pad layouts, considering component specifications and soldering requirements.

NON WET OPENS (NWO)/HEAD IN PILLOW (HIP)

Definition: NWO occur when solder does not wet to one or both connection pads, resulting in an open circuit. HiP are a type of NWO that arise when the solder ball on a BGA or CSP component appears to rest on the solder paste without merging with after reflow. Other names for NWO defects include "foot in mud" and "ball in socket".





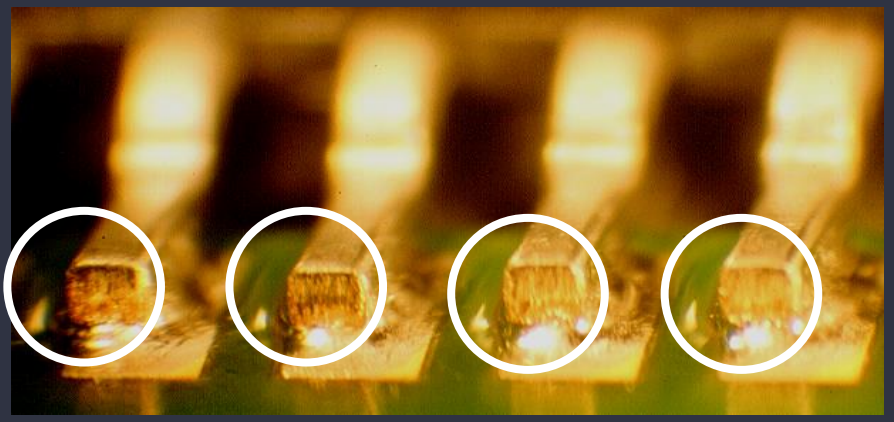
How to Recognize: In general, NWO can be identified visually by observing a clear gap or separation between the solder and one (or both) of the connection pads. It's best recognized using X-ray inspection, where the solder ball and solder paste appear distinct, lacking the expected coalescence. The defect often manifests as an intermittent electrical connection.

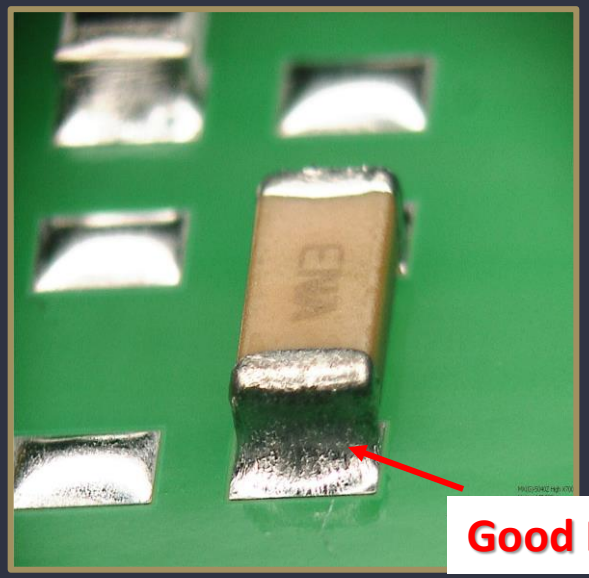
NWO/HIP: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Reflow profile issues	An inappropriate reflow profile can cause the solder paste to reflow before the solder ball has a chance to merge with it, or result in inadequate heat, preventing proper wetting.	Review/optimize the profile to match the solder paste and component specifications. Consider reducing reflow peak temperature or increasing bottom-side heat.
Oxidation of solder ball, paste, pads, or leads	Oxidation prevents solder from wetting properly, leading to NWO/HiP defects.	Ensure components and PCBs are clean and free from oxidation. Consider using a mild cleaning agent .
Insufficient solder paste volume	If there isn't enough solder paste deposited, the solder ball might not make proper contact, leading to HiP.	Adjust the stencil design, print speed, and squeegee pressure settings to ensure adequate solder paste deposition.
Insufficient flux activity	Flux promotes solder wetting by removing oxides from surfaces. If there's insufficient flux activity, wetting can be impeded.	Use solder paste with appropriate flux activity. Consider the application of additional flux if solder paste flux is not sufficient.

INSUFFICIENT FILLS, FILLETS, OR SOLDER

Definition: This defect is characterized by a lack of sufficient solder in the joint, leading to incomplete or weak connections. Insufficient fillet means that the solder does not fully form the expected shape, resulting in a potentially weaker joint.





How to Recognize: Insufficient solder can be visually identified by observing that the solder joint appears incomplete or not fully formed. The solder might not have completely covered the pad, or the fillet might appear shallow or non-existent. The joint may also lack the typical concave shape at the component lead-to-pad interface.

INSUFFICIENT FILLET/SOLDER: CAUSES AND SOLUTIONS

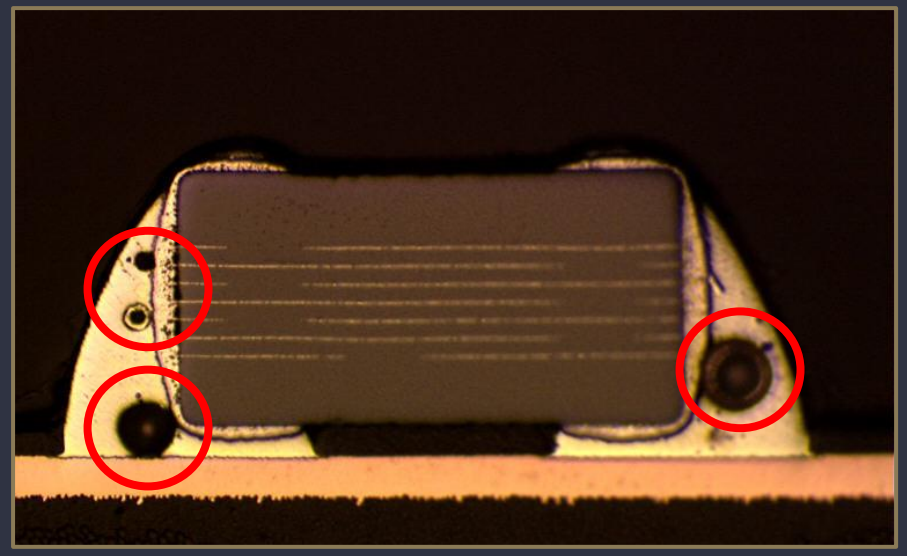
Problem	Cause	Solution
Scooping	Excessive squeegee pressure forces the edge of the squeegee to dip into the aperture, scooping out paste intended to be deposited.	Decrease squeegee pressure.
Inadequate solder paste deposition	If not enough solder paste is deposited on the pad, it can lead to insufficient solder after reflow. This may be caused by poor stencil design, or improper squeegee pressure or speed. It can also result if the separation speed is too fast, not allowing paste to release from the stencil.	Adjust the stencil design and aperture size and ensure proper squeegee pressure and speed during the printing process, as well as separation distance and speed.

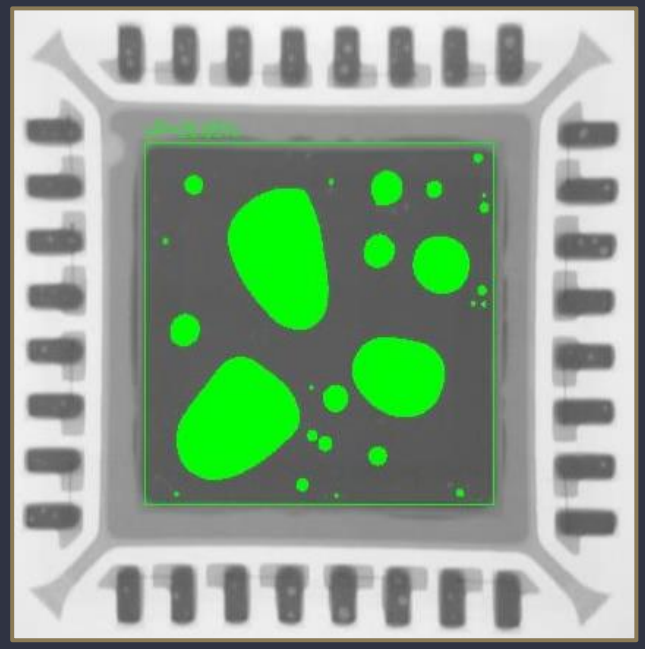
INSUFFICIENT FILLET/SOLDER: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Stencil misalignment	If the stencil is misaligned relative to the PCB, it can lead to uneven or insufficient solder paste deposition.	Ensure machine calibration for accurate stencil alignment and regularly inspect and clean the stencil.
Component placement issues	Misplaced components can drag solder away from the intended location, leading to insufficient solder in the joint.	Ensure accurate and precise component placement using calibrated machinery.
Improper viscosity of paste	If solder paste viscosity is too high, it won't properly roll on the stencil and/or fill the apertures	Check viscosity and metal content to see if these meets the manufacturer's specifications.

VOIDING

Definition: Voiding refers to the formation of air pockets or voids within a solder joint, especially common in Ball Grid Array (BGA) and Leadless Components. These voids can impact the mechanical strength, thermal performance, and electrical integrity of the solder joint.





How to Recognize: Voids are typically not visible on the surface of the solder joint. They are most commonly detected using X-ray inspection, where they appear as dark spots or areas within the X-ray image of a solder joint.

VOIDING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Trapped volatiles in solder paste	Volatile components in the solder paste, such as solvents, can outgas during reflow and become trapped, forming voids.	Adjust the reflow profile to allow volatiles to escape more effectively or select solder pastes formulated to reduce voiding. Consider modifying stencil design to aid flux volatilization (ex. I/O overprint). Choose a solder paste designed to minimize voiding.
Reflow profile issues	An inappropriate reflow profile can lead to incomplete outgassing or improper solder coalescence, leading to voids. Voids may form if peak temperature or time above liquidus are too low, if the soak is too long or too hot, or if the ramp rate is excessive.	Optimize the reflow profile, ensuring a proper soak period and peak temperature to allow for complete outgassing.

VOIDING: CAUSES AND SOLUTIONS

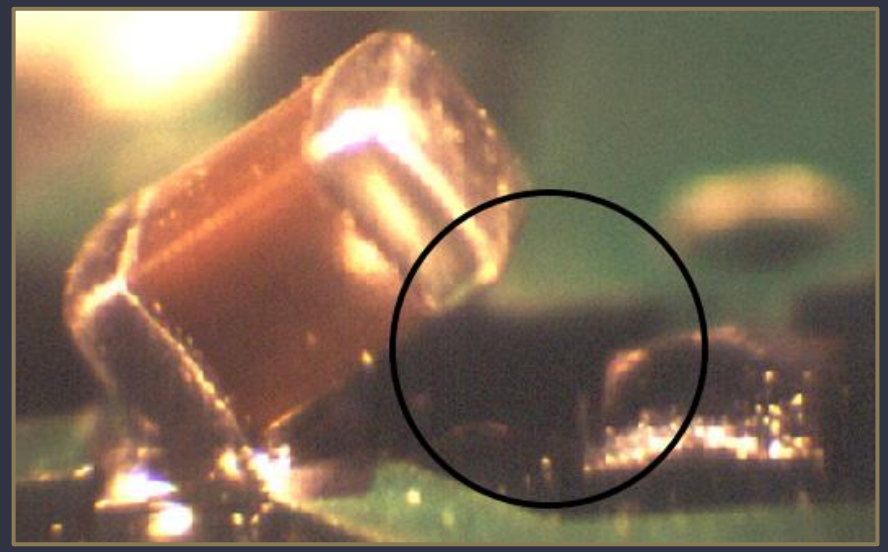
Problem	Cause	Solution
BTC voiding	The BTC design may have internal voids or gaps that release gas during reflow. Flux with high viscosity or poor wetting properties can also trap gas. Incorrect stencil design or applying too much solder paste can further exacerbate void entrapment.	Choose BTC components with minimal internal voids or gas-trapping features. Select flux with low viscosity and strong wetting properties. Adjust stencil design for proper paste release, thickness, and consistent aperture sizes. Apply the correct amount of solder paste to prevent excess.
BGA voiding	Uneven placement of solder balls or solder paste underneath the BGA package. BGA component warping during reflow can create voids, as can uneven placement of solder palls or paste, or inconsistent/oversized solder balls.	Ensure even placement and uniform solder paste under the BGA. Minimize component warpage through proper handling and reflow profile. Use precise solder ball size and distribution.

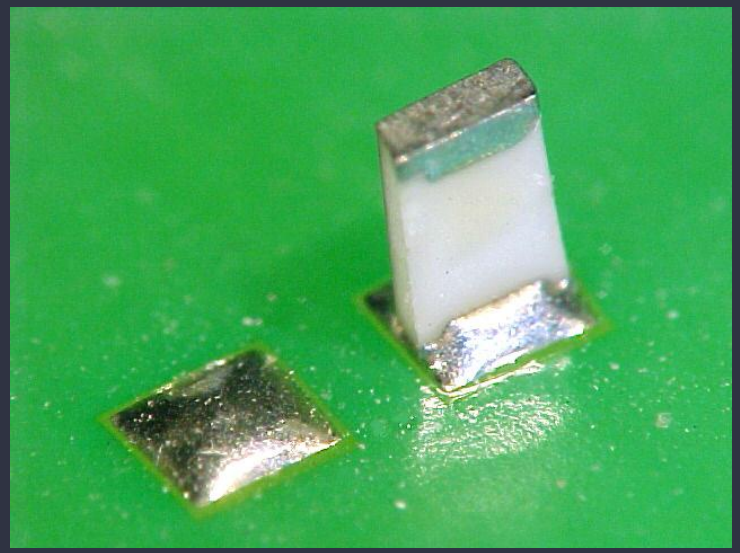
VOIDING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
QFN voiding	Inadequate solder mask openings or coverage can trap gas. Inconsistent or excessive solder paste application or QFN component warping during reflow can also lead to voids.	Ensure proper solder mask design with adequate openings. Precisely control the amount of solder paste applied. Adjust reflow profile to prevent warpage, or increase solder paste volume as needed.
Chip component voiding	Inconsistent or excessive solder paste application, incorrect placement or alignment of chip components, or chip component warping during reflow can lead to voiding.	Precisely control solder paste volume for chip components, ensure accurate placement and alignment, and adjust reflow profile to minimize warpage.

TOMBSTONING

Definition: A defect in which a two-terminal component (like a resistor or capacitor) stands vertically on one end after reflow soldering. This results in one end of the component being soldered correctly while the other end lifts off the pad, resembling a tombstone.





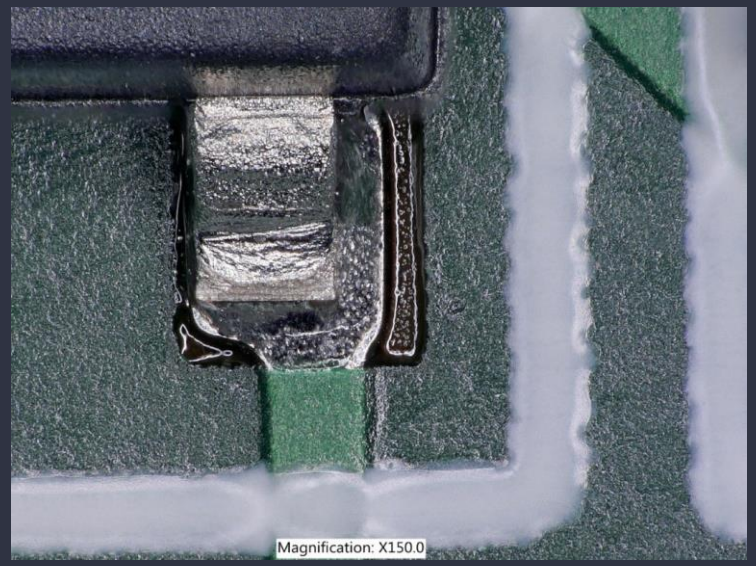
How to Recognize: Tombstoning can be visually identified by observing components such as resistors or capacitors standing vertically instead of lying flat against the PCB. The defect is more prominent with 0402, 0201, or smaller component sizes.

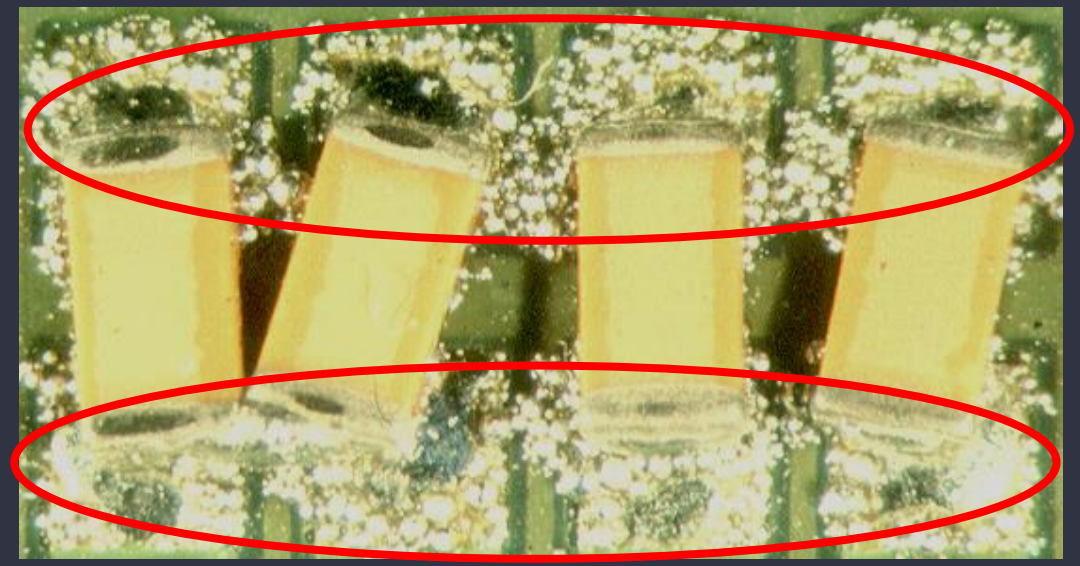
TOMBSTONING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Uneven solder paste wetting or volume	If one side of the component wets (melts and bonds) faster than the other, it can result in unequal forces, causing the component to lift.	Ensure consistent solder paste deposition across pads and optimize the reflow profile for uniform heating. Consider testing components for solderability.
Component misplacement	If a component is placed off-center, it increases the chance of tombstoning.	Ensure accurate and precise component placement using calibrated machinery.
Thermal imbalance during reflow	Different thermal characteristics such as unequal heat sink, or uneven heating during reflow can cause one end of the component to reflow before the other.	Optimize the reflow oven profile and ensure uniform heating across the PCB. Increase duration of soak cycle to ensure all parts of the PCB are at uniform temperature.

COLD SOLDER JOINTS

Definition: A situation where portions of the solder paste do not properly reflow, leaving residues of unreflowed solder particles and flux. This results in a joint that may be porous, weak, and lacking the typical smooth appearance of a properly reflowed joint.





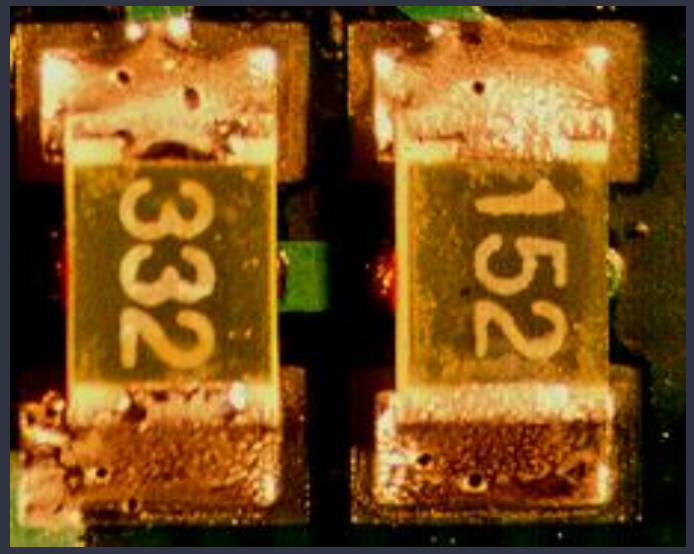
How to Recognize: The defect can be visually identified by observing a solder joint that appears rough, granular, or matte instead of the typical smooth and shiny appearance. Under magnification or X-ray, unreflowed solder particles might be visible within the joint.

COLD SOLDER JOINTS: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Inadequate reflow profile	If the reflow oven does not achieve the required peak temperature or doesn't maintain it long enough, the solder paste might not fully reflow.	Review and adjust the reflow profile, ensuring it matches the solder paste's specifications.
Expired or degraded solder paste	Solder paste that's past its shelf life or has been improperly stored may not reflow consistently. Oxidized solder particles within the paste may not melt and coalesce properly during reflow.	Always use fresh solder paste and adhere to recommended storage conditions to maintain its efficacy. Consider using a mild cleaning agent if oxidation is suspected.
Reflow in inappropriate atmosphere	Some solder pastes – including ultra fine pastes – may require specific atmospheres, like nitrogen, for optimal reflow.	Ensure the reflow atmosphere is compatible with the solder paste being used. Consider reflowing in a nitrogen-rich environment if recommended.

DISTURBED JOINTS

Definition: A disturbed solder joint is one where the solder has been disrupted or moved during its solidification process. The joint may not have the desired mechanical and electrical properties.



How to Recognize: Disturbed joints can be visually identified by their irregular or deformed appearance. Instead of the typical smooth and shiny surface, a disturbed joint might appear grainy, matte, or even contain visible solder waves or ripples. The joint might lack the desired concave fillet shape, appearing flattened or stretched.

DISTURBED JOINTS: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Movement during solidification	If the PCB or components are moved or disturbed while the solder is still in its molten state, it can result in a disturbed joint.	Ensure there's no movement or vibration of the assembly until the solder has fully solidified post-reflow.
Rapid cooling rate	An overly aggressive cooling rate post-reflow can cause the solder to solidify irregularly.	Adjust the reflow profile, particularly the cooling rate, to ensure a controlled and gradual solidification of the solder.
Inadequate reflow profile	An inappropriate reflow profile can cause uneven heating and cooling, leading to disturbed joints.	Review and adjust the reflow profile to ensure uniform heating and controlled cooling.

EXCESSIVE FILLET/TOO MUCH SOLDER

Definition: A situation where more solder than necessary is present in a solder joint. The solder joint might appear bulky, and in extreme cases, it could bridge with adjacent joints or components. Excessive solder can result in mechanical stress on components, leading to potential reliability issues.

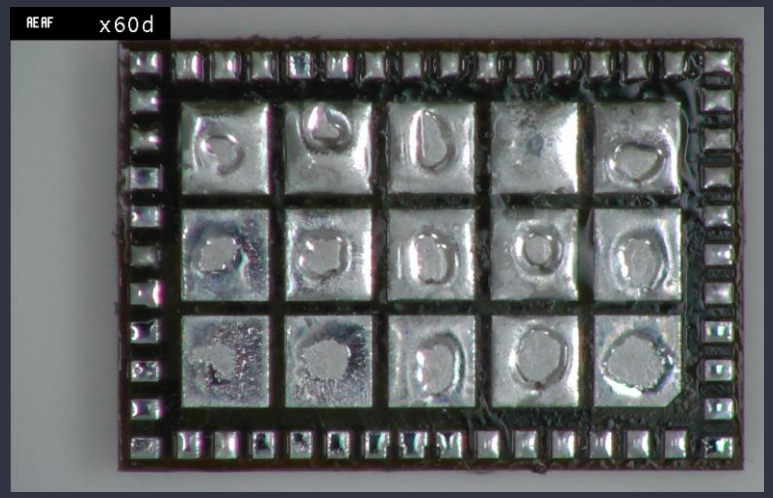
How to Recognize: The defect can be visually identified by observing a solder joint that appears larger and bulkier than typical. The fillet might extend significantly up the component lead or even touch the component body. In severe cases, the excessive solder might form bridges or shorts with neighboring components or traces.

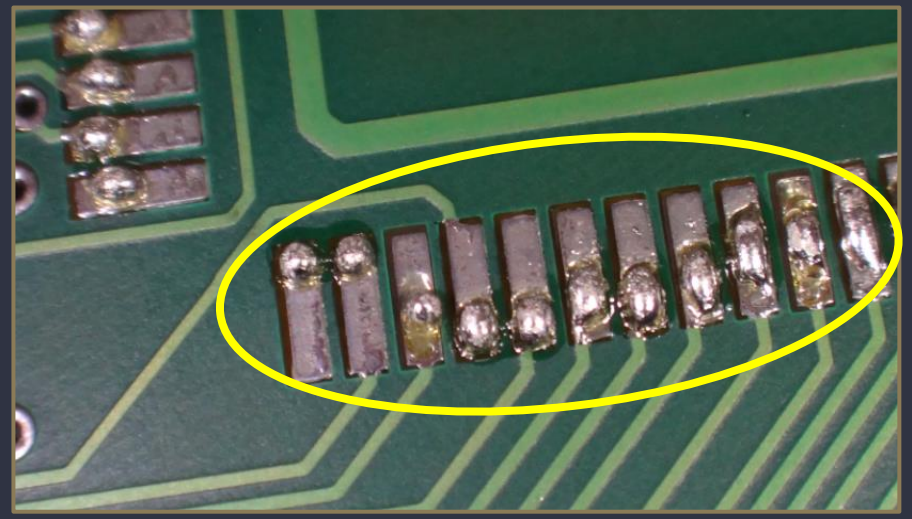
EXCESSIVE FILLET/SOLDER: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Excessive solder paste deposition	Depositing too much solder paste can lead to excessive solder after reflow.	Adjust the stencil design, thickness, and aperture size. Also, ensure proper squeegee pressure and speed during the printing process.
Stencil misalignment or wear	A worn-out stencil or one that's misaligned can deposit more solder paste than intended.	Regularly inspect the stencil for wear and ensure machine calibration for accurate alignment. Replace worn stencils as necessary.
Bleed out	Poor stencil design, excess squeegee pressure, improper gasketing, or viscosity that is too high can result in paste bleeding beyond the intended print area before reflow.	Inspect stencil and adjust squeegee pressure to see if this resolves the issue. Check the viscosity of the paste to make sure it is within specification. Reduce squeegee speed in case excessive shear forces are causing the bleed out.

NONWETTING/DEWETTING

Definition: Nonwetting refers to the failure of molten solder to form a cohesive and continuous bond with the metal surface it's intended to wet. Dewetting occurs when the solder initially wets the surface but then recedes, leaving areas of the base material exposed and often resulting in an irregular or rough solder surface.





How to Recognize: Both nonwetting and dewetting can be visually identified. Nonwetting will show areas where the solder has not bonded to the surface at all. Dewetting will show areas where solder initially bonded but then pulled back, leading to exposed base material and a rough, irregular solder finish.

NONWETTING/DEWETTING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Misregistered solder mask	If the solder mask is not printed correctly, this can create areas that do not wet.	The customer should resolve this issue with the PCB manufacturer.
Copper migration or exposed copper	If components or boards are old, they may have suffered copper migration. Scratches on pads can also result in exposed copper.	Ensure components and boards are unexpired or newer. Check boards for scratches or defects and make sure there are no abrasive contaminants.
Oxidation of surfaces	Oxidation of the PCB pads, component leads, or solder itself can prevent proper wetting.	Ensure that components and PCBs are clean and free from oxidation. Store solder paste and other materials under recommended conditions. Consider using a mild cleaning agent if oxidation is suspected.

NONWETTING/DEWETTING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Inadequate flux activity	Flux facilitates solder wetting by removing oxides. Insufficient or inactive flux can lead to nonwetting.	Use solder paste with appropriate flux activity. If solder paste flux is insufficient, consider applying additional flux.
Contaminated surfaces	Residues, contaminants, or handling can lead to surface contamination, preventing proper wetting.	Ensure all surfaces are clean before soldering. Implement cleaning processes if contamination is suspected.
Inappropriate reflow profile	An unsuitable reflow profile can lead to inadequate heat, preventing proper solder wetting.	Review and optimize the reflow profile to ensure it matches the solder paste and component specifications.
Rapid cooling	For dewetting specifically, rapid cooling can cause the solder to pull back after initial wetting.	Adjust the reflow profile, especially the cooling rate, to ensure a controlled solidification of the solder.

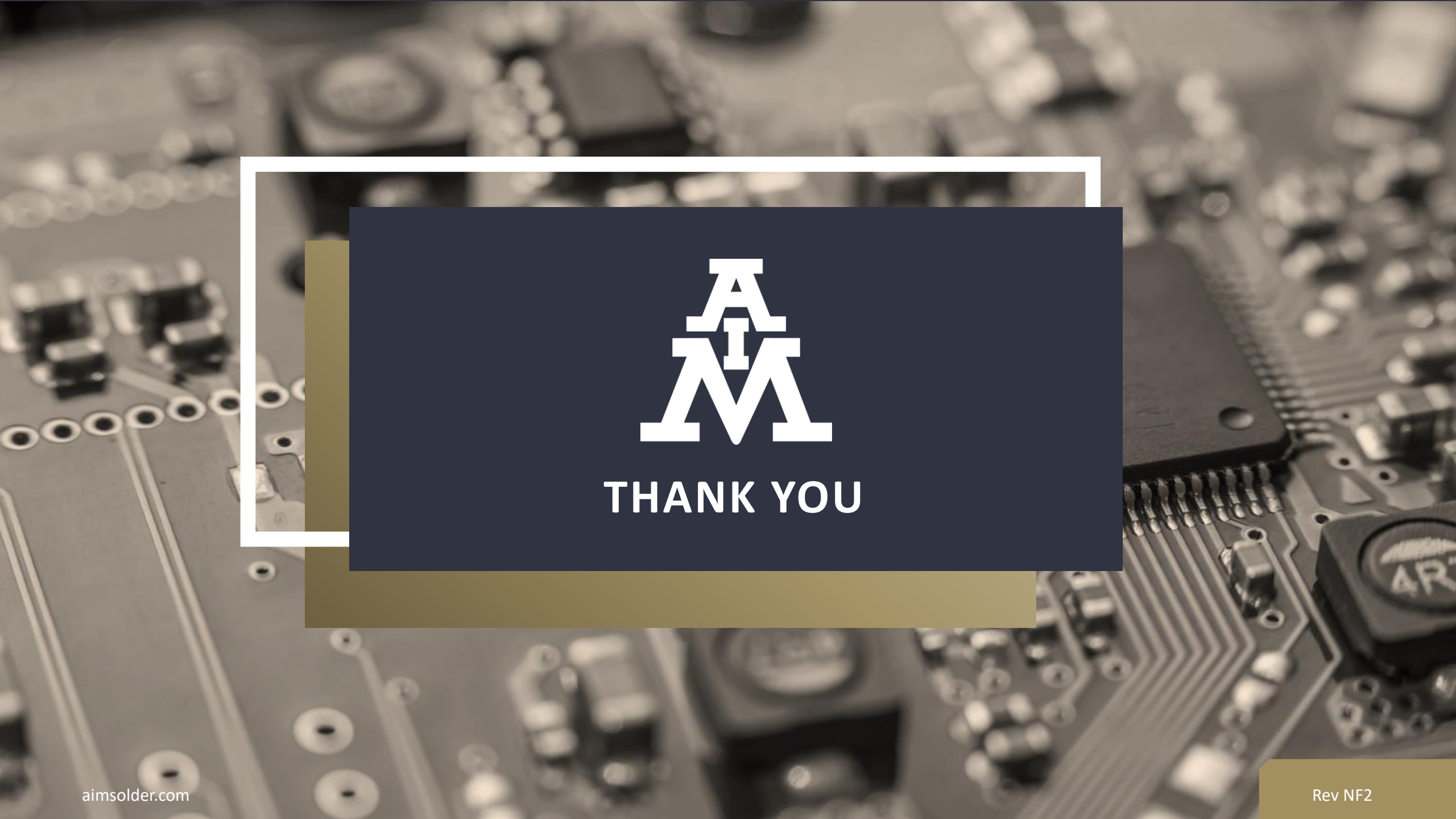
GRAINY JOINTS/GRAPING

Definition: Grainy joints, also known as graping, refer to a solder joint defect where the solder appears rough and grainy, or like a bunch of grapes after reflow.

How to Recognize: Grainy joints can be visually identified by the distinct granular texture on the solder joint surface.

GRAINY JOINTS/GRAPING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Oxidized solder particles	If solder particles within the paste are oxidized, they may not coalesce smoothly, leading to a grainy appearance.	Use fresh solder paste and ensure proper storage conditions to prevent oxidation. Consider a mild cleaning agent if oxidation is suspected.
Aggressive reflow profile	An overly aggressive reflow profile, especially during the ramp-up stage, can cause solder particles to reflow prematurely and individually. If the flux within the solder paste volatilizes too quickly, it can lead to graping.	Optimize the reflow profile to ensure a gradual and uniform melting of solder particles. In particular, adjust the soak and ramp-up stages, to allow for a more gradual flux volatilization.
Reflow in air vs nitrogen	Reflowing in air, especially for some solder paste formulations, can lead to increased oxidation and grainy joints.	Consider reflowing in a nitrogen atmosphere, especially for lead-free solder pastes that may be more susceptible to oxidation.


THANK YOU