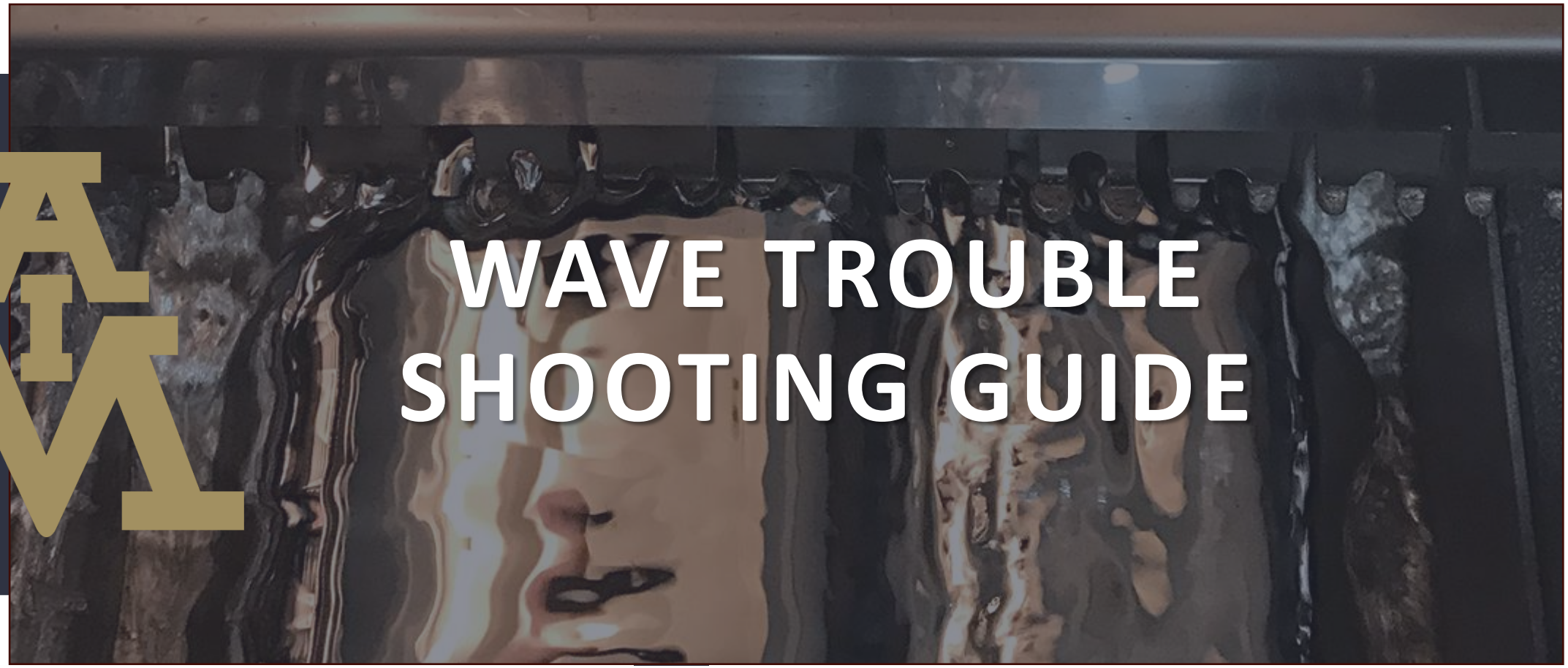


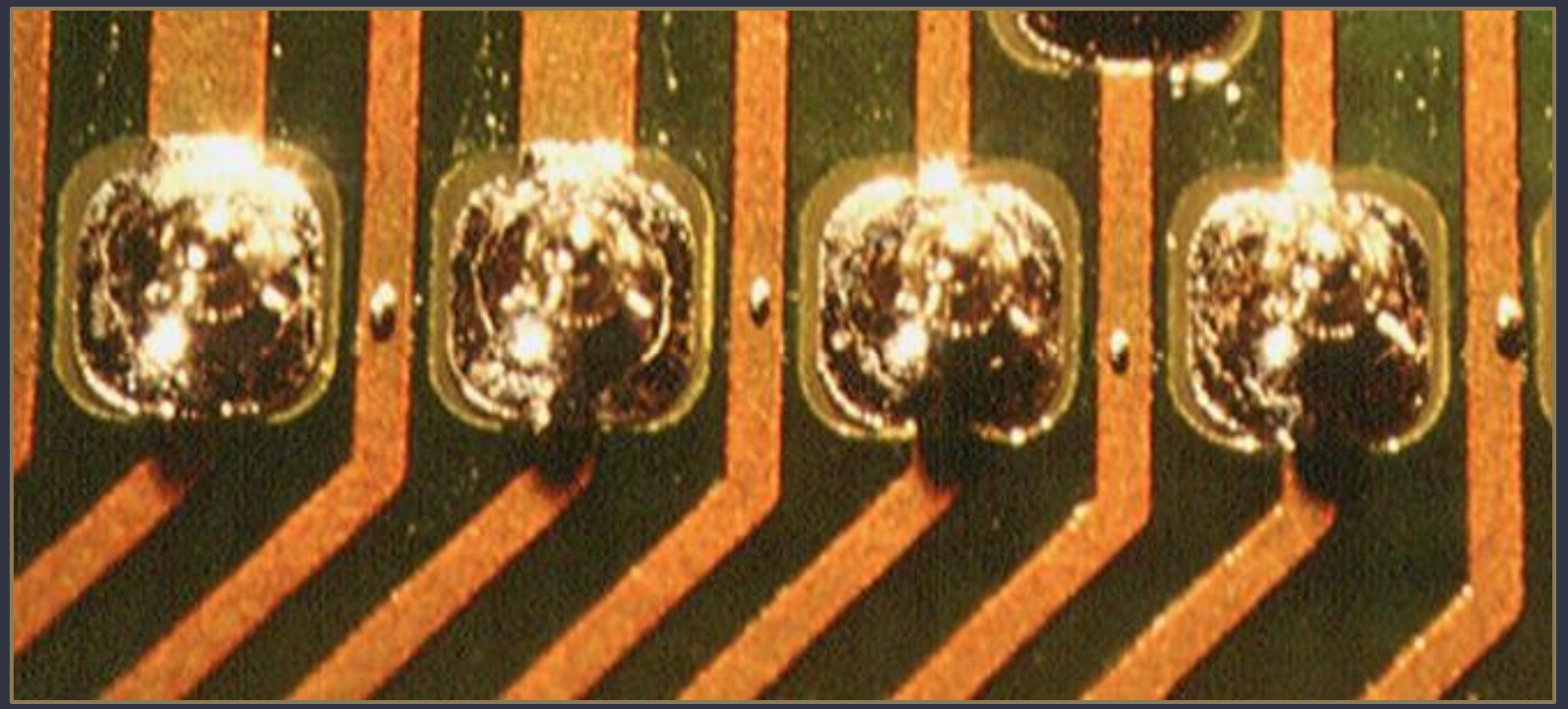
WAVE TROUBLE SHOOTING GUIDE

WAVE TROUBLE SHOOTING OUTLINE

- Solder balls
- Bridging
- Pin holes
- Webbing
- White haze
- Icycling
- Cold/disturbed solder joints
- Excess solder
- Nonwetting/dewetting
- Dull/grainy joints

SOLDER BALLS

Definition: Solder balls are tiny, round solder deposits adjacent to solder joints or scattered across the PCB surface after the reflow process. They can potentially cause short circuits if they bridge adjacent conductive areas.



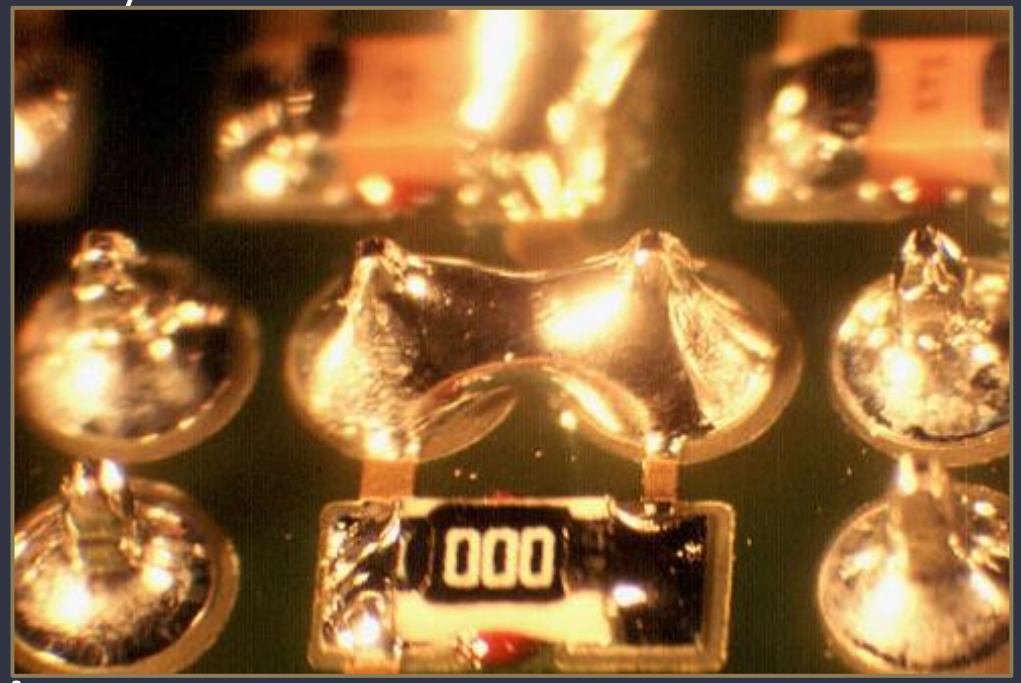
How to Recognize: Solder balls can be visually identified by the presence of small, spherical shapes of solder dispersed over the surface of the PCB.

SOLDER BALLS: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Insufficient preheat	Inadequete preheating can prevent the flux from fully activating and cleaning the PCB, leading to incomplete solder wetting and solder balls.	Ensure preheat temperature and duration are sufficient to fully activate flux; verify PCB's surface temperature is approprioate before entering the solder wave.
High humidity	High humidity can affect the soldering process by causing condensation on the PCB, leading to solder ball formation.	Maintain a controlled environment with low humidity in the soldering area. Use dehumidifiers and monitor humidity levels regularly.
Plated through hole conditions creating pin holes	Defects in plated through holes, such as voids or pin holes, can trap flux and gases. During soldering the trapped flux and fases expand, causing solder balls.	Inspect PTHs for defects before soldering.

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 leads where there shouldn't be any. These connections can appear as thin solder filaments or broader solder masses spanning between leads.

BRIDGING: CAUSES AND SOLUTIONS

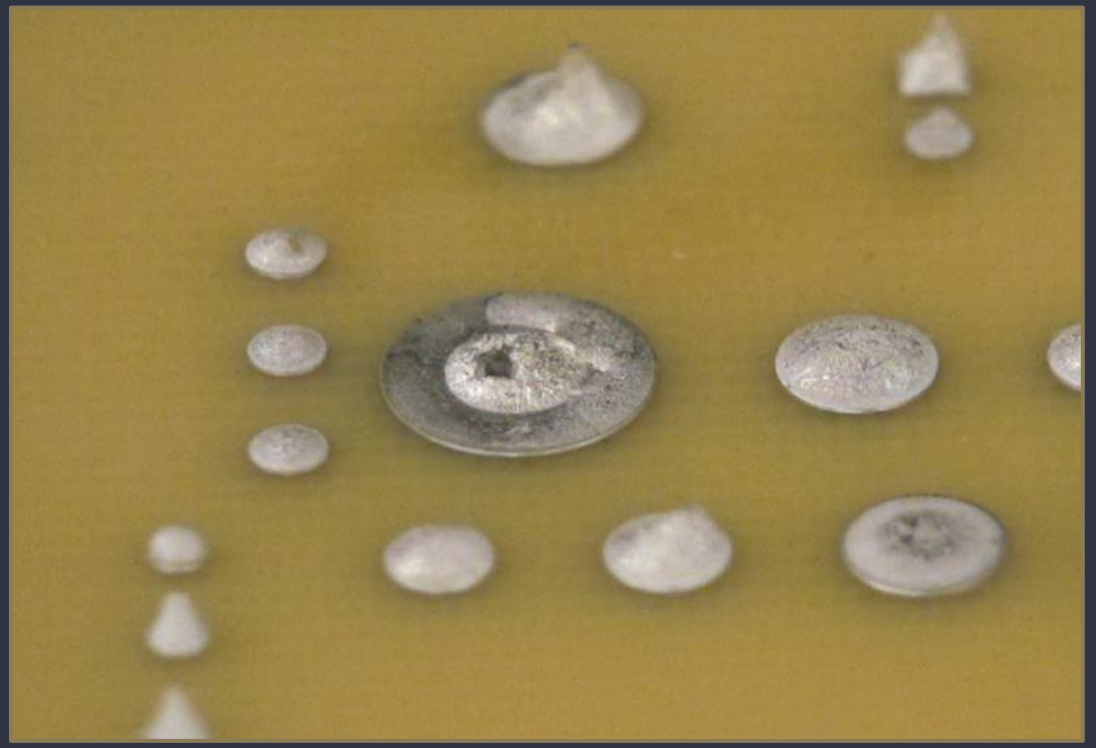
Problem	Cause	Solution
Component leads bent or too closely spaced	When component leads are bent or too closely spaced, it increases the likelihood of solder bridging between them during the soldering process.	Ensure proper handling and placement of components to maintain correct lead spacing. Use a lead-forming tool to straighten bent leads before soldering.
Excess solder	Excess solder can create bridges between adjacent leads as the solder tends to spread and connect unintended areas.	Adjust the solder wave parameters to reduce the amount of solder applied. Consider using solder masks to control the flow of solder and prevent bridging.
Inadequate flux	If insufficient flux is applied or if it is not effective, it may not promote proper drainage of solder, leading to bridging.	Ensure adequate and even application of flux to the PCB. Verify that the flux is compatible with the soldering process and check that the fluxer is functioning correctly.

BRIDGING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Immersed too deep in wave	Immersing the board too deep in the solder wave can cause excessive solder to accumulate, leading to bridging.	Adjust the immersion depth of the PCB in the solder wave to the optimal level.
Picking up dross in the wave	Leads can pick up dross (oxidized solder) from the solder wave, which can contribute to bridging.	Regularly clean and maintain the solder pot to minimize dross formation.
Contaminated solder	Contaminated solder can lead to poor solder flow and increased likelihood of bridging.	Use high-purity solder and ensure proper storage to prevent contamination. Regularly test and replace solder in the solder pot to maintain its quality.
Poor component solderability	Components with poor solderability can cause incomplete or uneven solder joints, leading to bridging.	Conduct solderability testing and use appropriate flux to enhance wetting. Store components in a controlled environment to prevent oxidation and degradation.

PIN HOLES

Definition: Pin holes are small voids or holes that appear in solder joints, particularly in plated through holes (PTHs). They can compromise the mechanical strength and reliability of the solder joint, leading to potential failures in the PCB.



How to Recognize: Pin holes can be identified visually by observing small holes or eruptions in the solder fillet.

PIN HOLES: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Moisture of plating solution in PCB laminate	Moisture or residual plating solution trapped in the PCB laminate can outgas during soldering, creating pin holes.	Properly bake PCBs before soldering to remove moisture and plating solutions. Store PCBs in a dry environment to prevent moisture absorption.
Inadequate preheat	Insufficient preheating may leave flux solvents in the PTH, which can vaporize during soldering and create pin holes.	Adjust the preheat temperature and duration to ensure complete evaporation of flux solvents. Regularly verify and maintain the preheat profile to achieve consistent results.
Flux has absorbed water	Flux that has absorbed water can lead to steam generation during soldering, resulting in pin holes.	Use fresh flux and ensure it is stored in a controlled environment to prevent water absorption. Replace flux regularly and verify that it is within its shelf life and stored according to manufacturer recommendations.

PIN HOLES: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Physical blockage due to foreign body in hole	Foreign bodies or contaminants in the PTH can block solder flow and create voids or pin holes.	Inspect and clean PCBs to remove any foreign bodies or contaminants before soldering. Implement rigorous cleaning and inspection processes to ensure PTHs are clear of blockages.
Top of plated through hole prematurely solidifying	The top of the PTH can solidify prematurely if the soldering process is not well-controlled, leading to incomplete solder fill and pin holes.	Optimize solder wave parameters, including temperature and contact time, to ensure proper solder flow and fill in the PTHs. Adjust the conveyor speed and solder wave height to achieve consistent soldering without premature solidification.

WEBBING

Definition: Webbing is a defect characterized by thin strands or filaments of solder bridging between component leads, pads, or other conductive areas. This can cause short circuits and impact the functionality and reliability of the PCB.

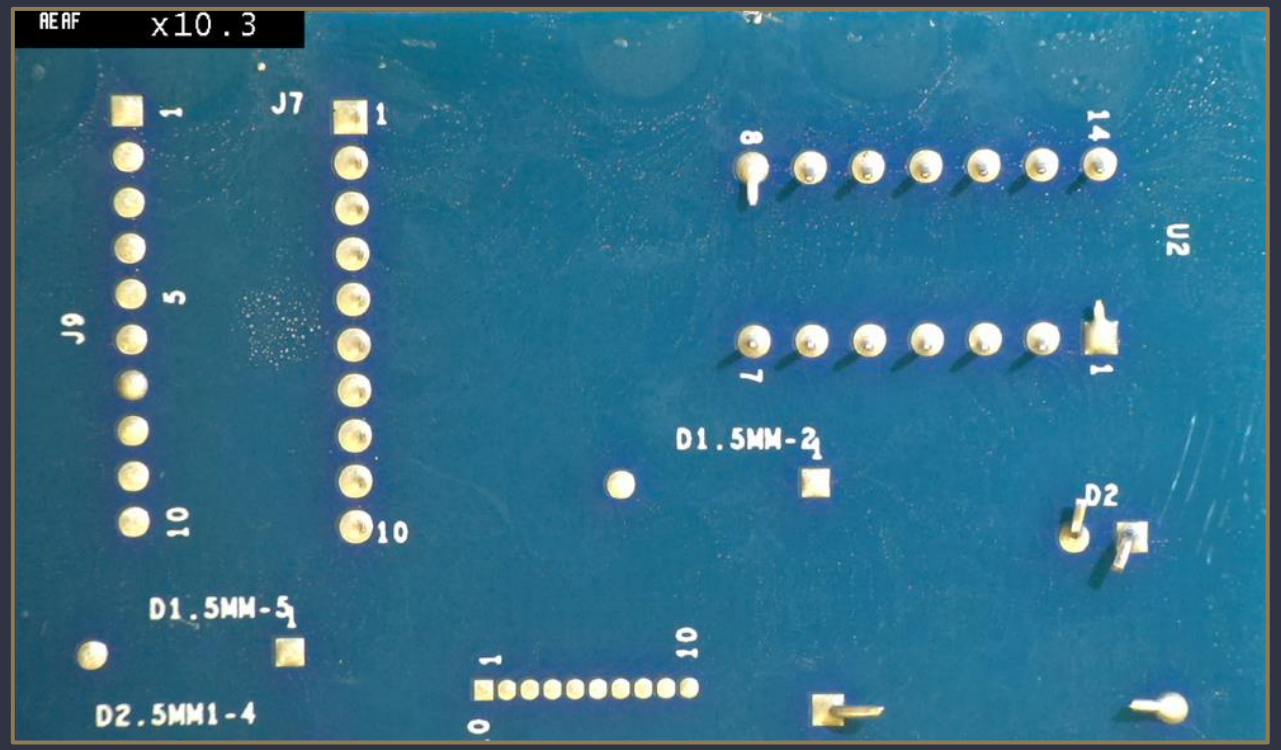
How to Recognize: Webbing can be visually identified by observing a spider web like extension of solder across the nonconductive portion of the PCB.

WEBBING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Improper curing of the laminate or solder mask	If the laminate or solder mask is not properly cured, it can lead to poor adhesion and contamination, resulting in webbing.	Baking the PCB can sometimes correct an improperly cured mask or laminate condition.
Inadequate flux	Insufficient flux application can result in poor solder wetting and increased likelihood of webbing.	Apply an adequate and even amount of flux across the PCB. Verify that the fluxer is functioning properly and adjust the flux volume as needed. Consider substituting a more viscous flux or increasing quantity.
Dross in solder wave	Dross (oxidized solder) in the solder wave can create turbulence and contaminants that contribute to webbing.	Regularly skim and remove dross from the solder pot to maintain clean solder.

WHITE HAZE

Definition: White haze is a whitish residue or film that remains on the surface of the PCB after wave soldering. This residue can affect the appearance and potentially the reliability of the PCB by interfering with further processing or causing corrosion over time.



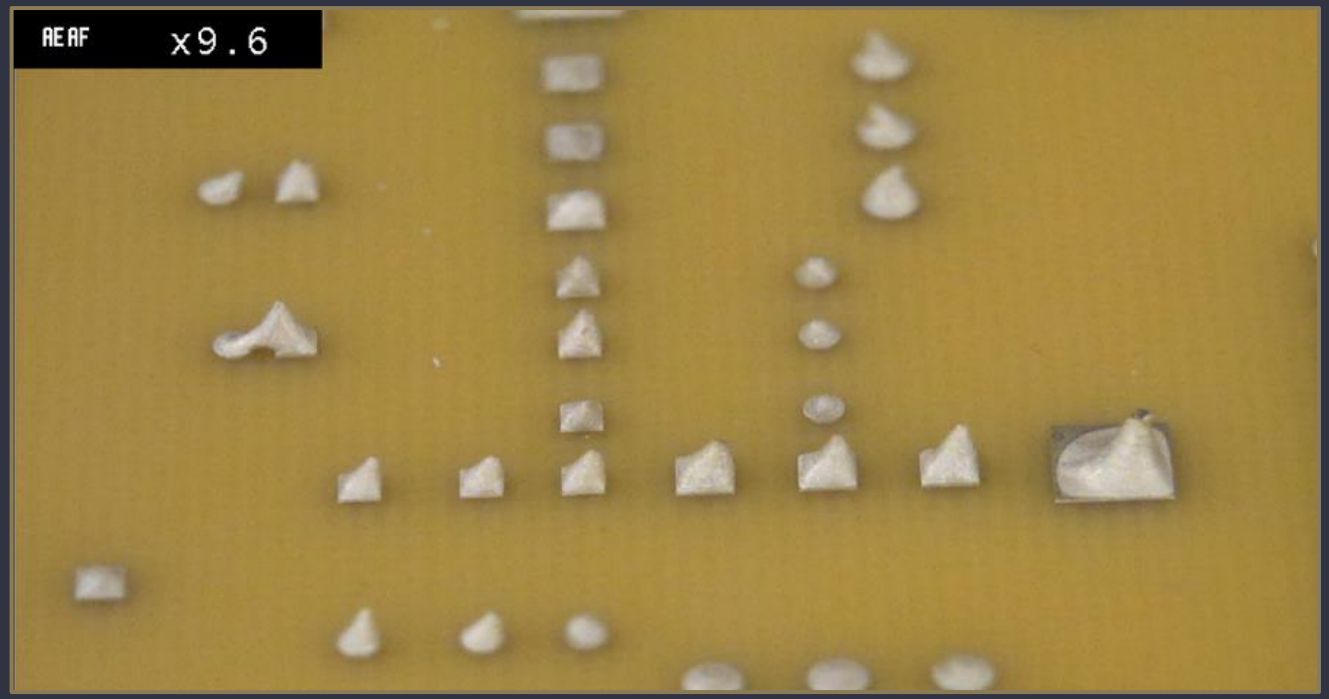
How to Recognize: White haze is recognized as a hazy, white residue on the nonconductive portion of the PCB that cannot be removed by washing.

WHITE HAZE: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Improper curing of the laminate or solder mask	Inadequate curing of the laminate or solder mask can lead to incomplete polymerization, resulting in residues that form a white haze during soldering.	Baking the PCB can sometimes correct an improperly cured mask or laminate condition.
Incomplete flux activation	Insufficient preheating or incorrect flux can prevent full activation, leaving residues that manifest as a white haze.	Adjust the preheat temperature and duration to ensure complete activation of the flux. Verify that the flux used is appropriate for the soldering process and that it has not absorbed moisture or degraded.
Excessive humidity	High humidity can lead to moisture absorption in the flux, causing white haze due to incomplete evaporation of solvents during soldering.	Maintain a controlled environment with low humidity. Use dehumidifiers and monitor the humidity levels to keep them within acceptable ranges.

ICYCLING

Definition: Icicling refers to the formation of sharp, elongated projections of solder on the joints or leads after the wave soldering process. These solder spikes can cause short circuits, damage components, and create aesthetic and functional defects.



How to Recognize: Icycling can be visually identified by the presence of conical or flag shaped extensions of the solder fillet.

ICYCLING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Inadequate flux	Insufficient flux application can lead to poor wetting and drainage, resulting in the formation of icicles as the solder solidifies.	Ensure adequate and uniform flux application across the PCB. Verify that the fluxer is functioning properly and adjust the flux volume as necessary to promote effective solder drainage.
Pot temperature too low	A solder pot temperature that is too low can prevent proper solder flow and increase viscosity, leading to icicling as the solder does not drain quickly enough.	Increase the solder pot temperature to the recommended level for the specific solder alloy. Regularly calibrate and monitor the solder pot to maintain consistent temperatures.
Soldering surface unusually heat absorbent	A soldering surface that absorbs heat excessively can cause the solder to solidify too quickly, resulting in icicling.	Ensure that the preheat stage is adequate to bring the PCB to a uniform temperature, reducing heat absorption during soldering.

ICYCLING: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Lead picking up dross in the wave	Leads that pick up dross (oxidized solder) from the solder wave can create irregularities, leading to icicling.	Regularly skim and remove dross from the solder pot to maintain clean solder. Implement a dross management system to minimize the formation of dross and ensure smooth solder flow.
Wrong plated through hole to wire ratio	An incorrect ratio of plated through hole (PTH) size to wire diameter can cause insufficient solder fill and irregular solder flow, leading to icicling.	Design PCBs with the correct PTH to wire diameter ratio, ensuring that the holes are appropriately sized for the components being used. Follow design guidelines and industry standards for optimal PTH dimensions.

COLD/DISTURBED SOLDER JOINTS

Definition: These occur when the solder joint is not properly formed due to movement during solidification or insufficient heat. They are characterized by a dull, grainy, or uneven appearance and can result in weak electrical and mechanical connections.

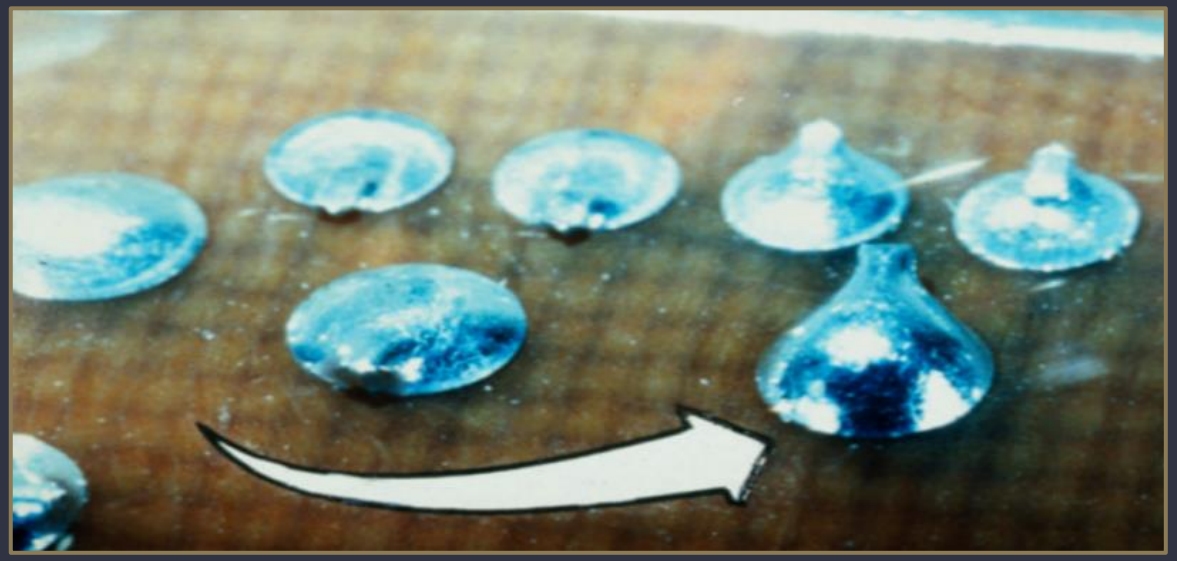
How to Recognize: The defect can be visually identified by observing rough and dull finish on the fillets in conjunction with unacceptable mechanical strength of the joint.

COLD/DISTURBED SOLDER JOINTS: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Movement while joint is still molten	Any movement of the PCB or components while the solder joint is still in its molten state can disturb the solder, leading to a cold or disturbed joint.	Adjust the conveyor speed to ensure that the board moves steadily and smoothly through the solder wave. Minimize vibrations in the soldering equipment and environment to reduce the risk of movement while the solder is molten.
Inadequate solder wave temperature	If the solder wave temperature is too low, it can prevent proper solder flow and wetting, resulting in cold joints.	Ensure the solder wave temperature is set to the recommended level for the specific solder alloy being used. Regularly calibrate and monitor the soldering equipment to maintain consistent temperatures.

EXCESS 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.



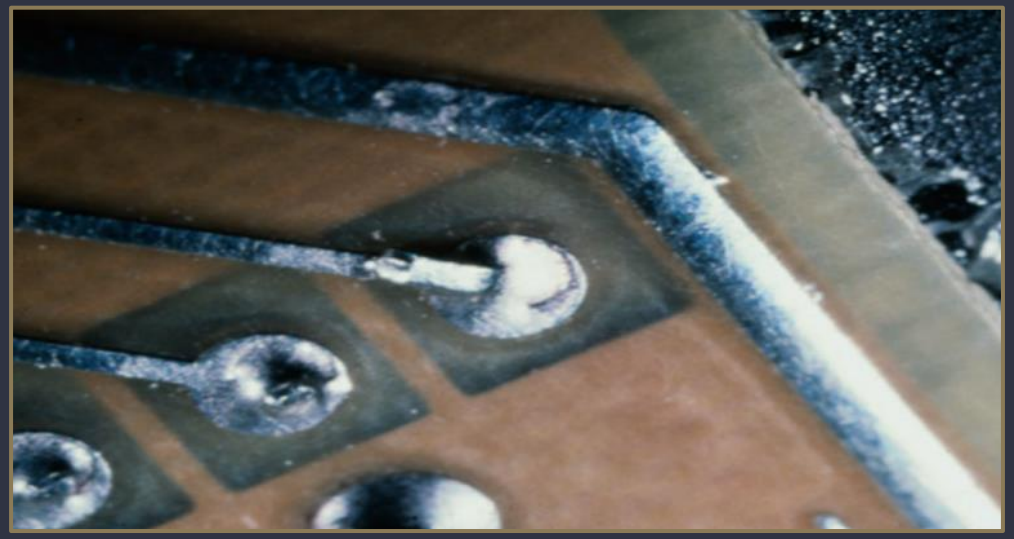
How to Recognize: The defect can be visually identified by observing a solder joint that appears larger and bulkier than typical. The fillet might appear bulbous and you may be unable to see the contours of the lead and land.

EXCESS SOLDER: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Poor drainage of solder	Improper PCB orientation, wave turbulence, or incorrect solder pot height can hinder proper solder drainage.	Ensure correct PCB orientation, adjust solder wave parameters to reduce turbulence, and set the solder pot height to the correct level.
Low temperature of solder or preheat	Low temperatures can prevent proper solder flow and drainage, leading to excess solder on the joints.	Maintain appropriate solder pot temperature and optimize preheat temperature to ensure proper flux activation and PCB heating.
Contamination	Contaminants in the solder can affect its flow characteristics, leading to excess solder deposition.	Use high-purity solder, regularly replace or filter the solder, and clean the solder pot to remove dross and contaminants.
Incorrect wave exit angle or speed	The angle at which the PCB exits the solder wave and the speed of the conveyor can impact solder drainage, leading to excess solder.	Adjust PCB exit angle to promote better solder drainage and set the conveyor speed to ensure smooth, consistent movement through the solder wave.

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.
Contaminated solder	Contaminated solder can introduce impurities that interfere with proper wetting, causing dewetting during the soldering process.	Use high-purity solder and ensure proper storage to prevent contamination. Regularly monitor and test the solder for impurities, and maintain the solder pot by removing dross and contaminants frequently.
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. 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 flux with the appropriate activity level for the application. Ensure proper flux application and regularly clean and inspect flux sprayers and foamers.
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.
Temperature too low	An unsuitable solder pot temperature can prevent proper solder wetting.	Review and optimize the solder pot temperature to ensure it matches the solder and component specifications.
Poor hot air solder levelling during PCB manufacturing	Inadequate hot air solder leveling can leave irregular or insufficient older coating on the PCB, leading to dewetting during wave soldering.	Employ quality control measures to ensure detection and correction of any HASL-related issues before the PCBs proceed to the soldering stage.

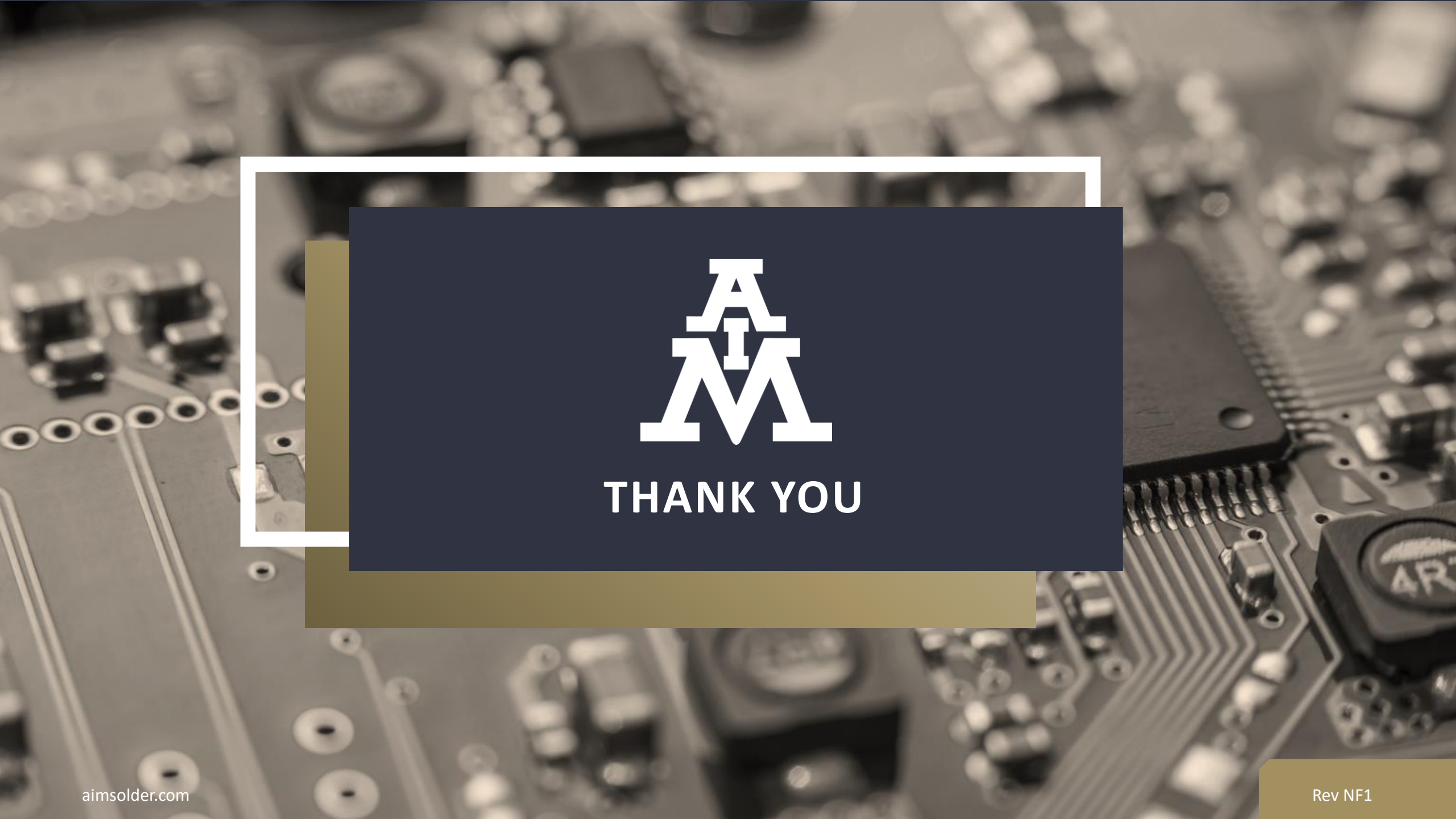
DULL/GRAINY JOINTS

Definition: These are solder joints that lack the shiny, smooth appearance typically expected. They may appear matte or have a rough, uneven surface, indicating potential issues with the soldering process or materials..

How to Recognize: Dull/grainy joints can be visually identified by dark, nonreflective, rough surfaces from an alloy that is normally bright and shiny.

DULL/GRAINY JOINTS: CAUSES AND SOLUTIONS

Problem	Cause	Solution
Alloy naturally has dull finish	Some solder alloys, such as those with higher tin content or lead-free solders, may naturally have a duller finish compared to traditional tin-lead solders.	Verify the expected finish for the specific solder alloy. If the dull finish is inherent to the alloy, confirm that the solder joints meet all other quality and reliability criteria despite the appearance.
Contamination	Contamination of the solder or PCB pads with oxides, residues, or other impurities can prevent proper wetting and lead to dull or grainy joints.	Use high-purity solder and ensure proper storage to prevent contamination. Clean the PCB pads thoroughly before soldering and maintain the solder pot by regularly removing dross and other contaminants.
Improper soldering temperature	Incorrect soldering temperatures, either too low or too high, can affect the solder joint appearance and lead to dull or grainy surfaces.	Ensure solder pot and preheat temperatures are set to correct levels for the specific alloy. Regularly calibrate and monitor the soldering equipment to maintain consistent temperatures.


THANK YOU