AIM

Global Solder Solutions

SMT Troubleshooting Guide

AIM's Practical Solutions

WWW.AIMSOLDER.COM INFO@AIMSOLDER.COM

Excellence is more than a word…
it's our passion

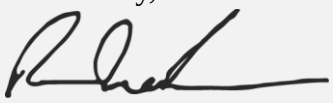
With roots in the world of metal stretching back over 75 years, AIM has evolved from humble beginnings into an international leader in the development, manufacture and application of electronics assembly materials.

Our mission is to offer the most innovative and reliable product solutions available to the electronics industry. At the same time, AIM is keenly focused on the need for customer support at every stage of the professional relationship. In fact, we consider our commitment to providing top-notch technical service to be just as important as our goal of producing market-leading materials.

The key to being a market leader in any industry today rests on the ability to provide customers with unmatched quality, consistency and value throughout the entire process, with products and services delivered locally, and yet still inspired by a global, environmentally-focused view.

We believe that our focus on creating excellence every step of the way is what sets AIM apart, and allows us to meet those challenges. We thank you for the confidence and trust that you have placed in us, and we look forward to continue working closely with you to help you achieve your goals.

Yours truly,



Rick Black
President, AIM

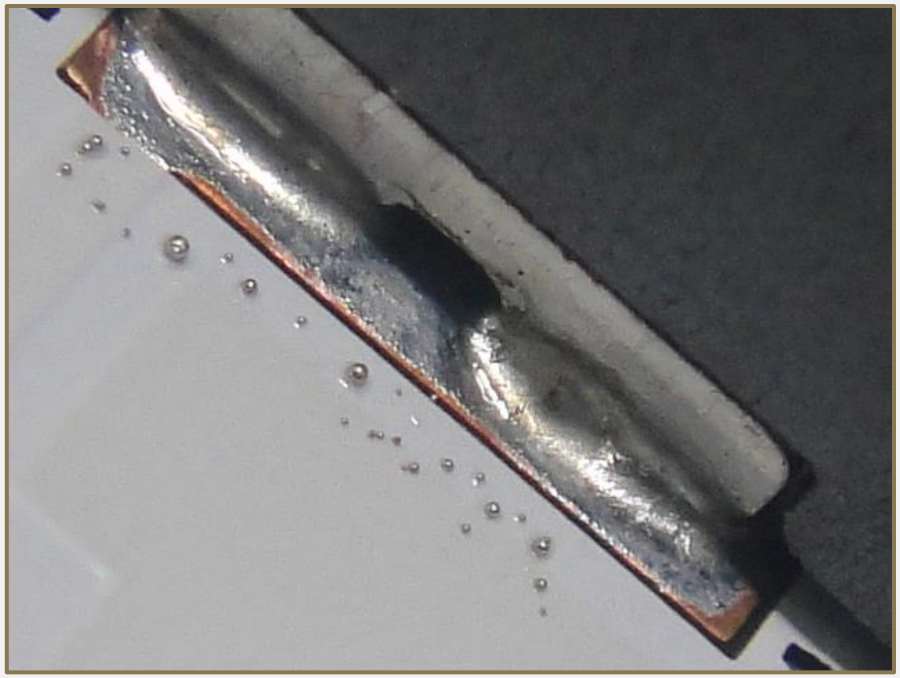
Types of SMT Soldering Defects

Types of Defects

- Solder Balling
- Solder Beading
- Bridging
- Opens
- Voiding
- Tombstoning
- Unmelted Paste
- Disturbed Joints
- Excessive Fillet
- Non-Wetting
- Dewetting

Solder Balling

- Recognized by:
  - Numerous tiny solder balls trapped along the peripheral edge of the flux residue after reflow
  - Balls stuck around fine pitch lands and solder mask



Solder plus Support

Solder Balling

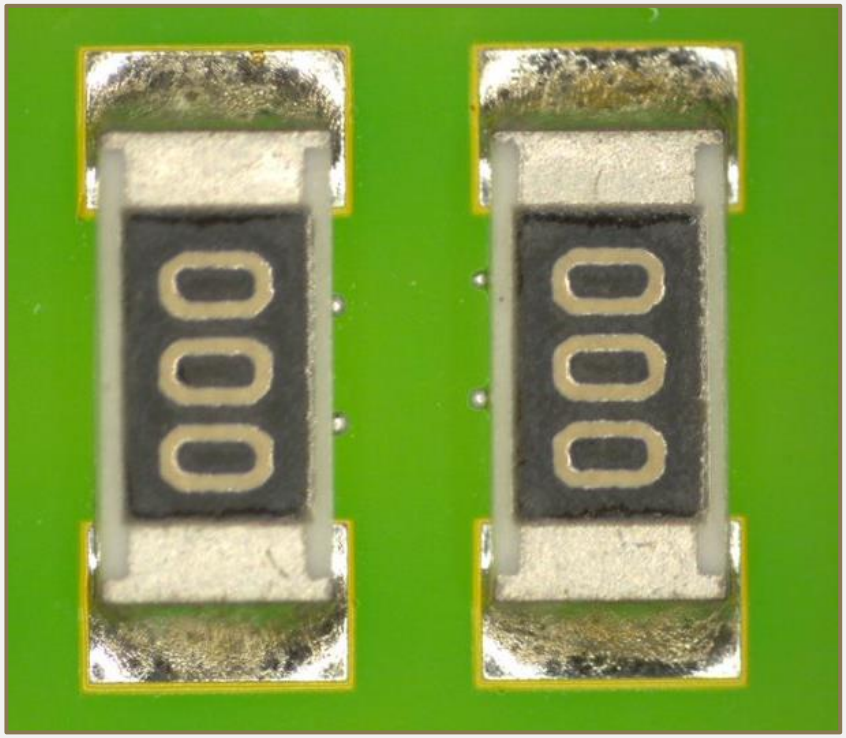
- Possible Causes:
  - Oxidized paste – will be identified by grey halos in appearance around the solder balls or joints
  - Paste has been on PCB too long before reflow
  - Reflow profile too slow on ramp up
- Remedy:
  - Run fresh paste from a different lot under same conditions and see if solder balls go away
  - Check ramp rate in the first 90 seconds of the profile
  - Run recommended profile and see if the problem persists

Solder Spattering

- Possible Cause:
  - Reflow profile too rapid during ramp up. Too rapid of a ramp up to reflow temperature will not allow the volatiles in the paste to be driven off before paste becomes molten.
  - The combination of volatiles and molten solder will result in solder spatter
- Remedy:
  - Run a slower ramp up profile to give the volatiles time to vaporize

Solder Beading

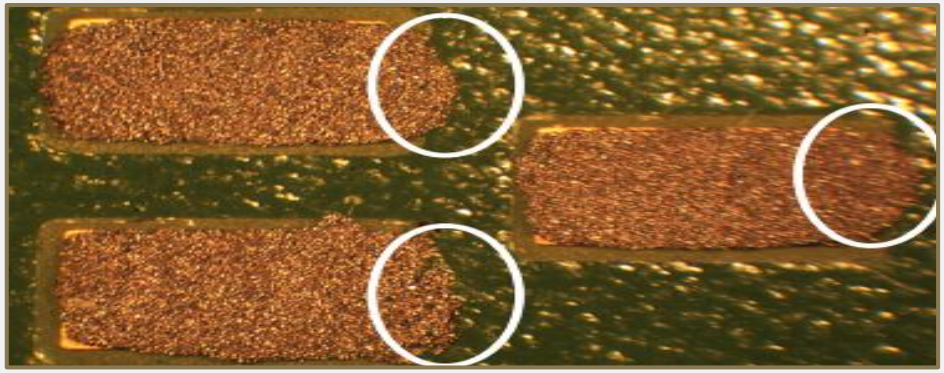
- Recognized by:
  - Single large solder balls that are next to a component or component pad
  - Typically found by discrete components



Solder plus Support

Solder Beading

- Possible Causes:
  - Reflow profile ramp up too slow causing capillary action to draw the unreflowed paste away from the pad.
  - Too low squeegee pressure leaving too much paste behind
  - Excessive amount of solder paste printed on component pads
  - Paste bleeding under the stencil



Solder plus Support

Solder Beading

- Remedies:
  - Use a rapid ramp up of 1.5 to 2.5°C/sec
  - Increase squeegee pressure
  - Reduce the amount of solder paste printed on the pad
  - Check for excess squeegee pressure or improper gasketing

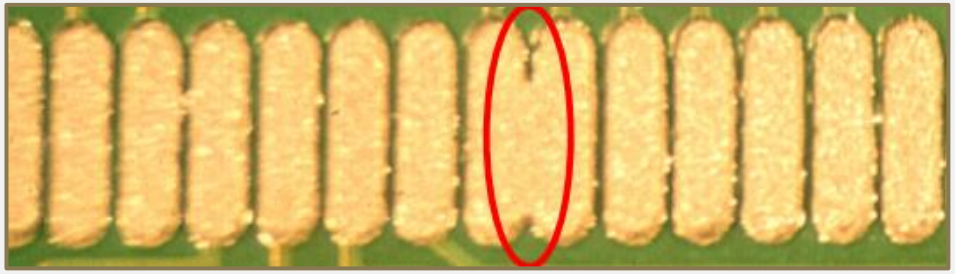
Bridging

- Recognized by solder running from one component contact to another. This could result in a short circuit

Solder plus Support

Bridging

- Possible Causes:
  - Cold slump - due to paste being the incorrect viscosity or too rapid squeegee speed resulting in over shearing of solder paste and degrading its thickeners
- Remedy:
  - Check viscosity of the paste. If correct, lower print speed and run fresh paste to see if problem persists
  - Might be humidity related. Use of fresh paste is recommended



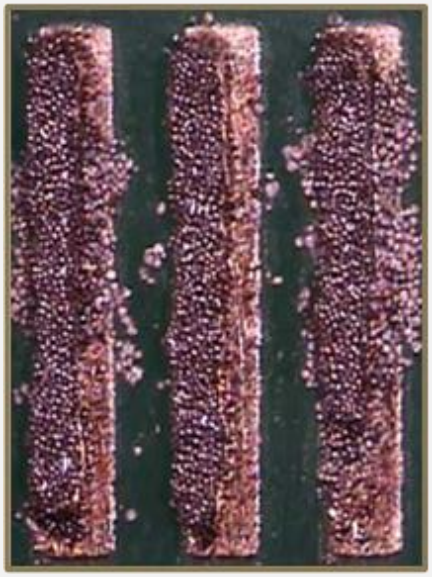
Solder plus Support

Bridging

- Possible Causes:
  - Hot slump - occurs when the paste deposits settle or spread out during the ramp up cycle of the reflow profile and spread to an adjacent pad
- Remedy:
  - Check overall profile length
  - Check ramp up rate. Try shortening ramp up and see if bridging disappears
  - Change the profile type

Bridging

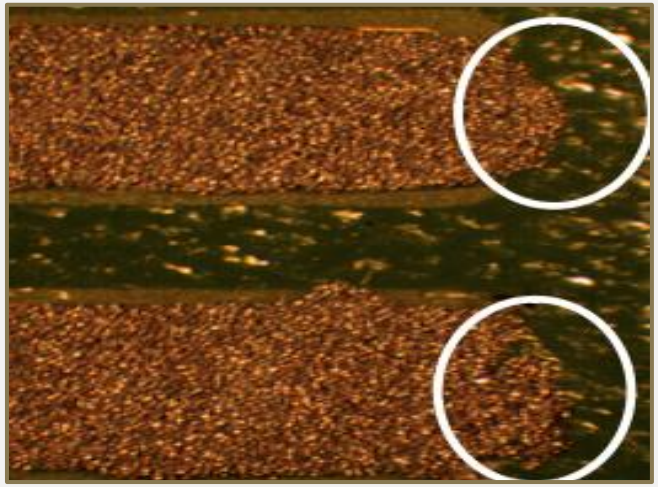
- Possible Causes:
  - Excess squeegee pressure causing paste bleed out
- Remedy:
  - Reduce squeegee pressure causing bleed out



Solder plus Support

Bridging

- Possible Causes:
  - Excess of solder paste being deposited on the pads between the bridge. Upon reflow, the excess solder may flow over to adjacent pad
- Remedy:
  - Reduce aperture dimensions or stencil thickness



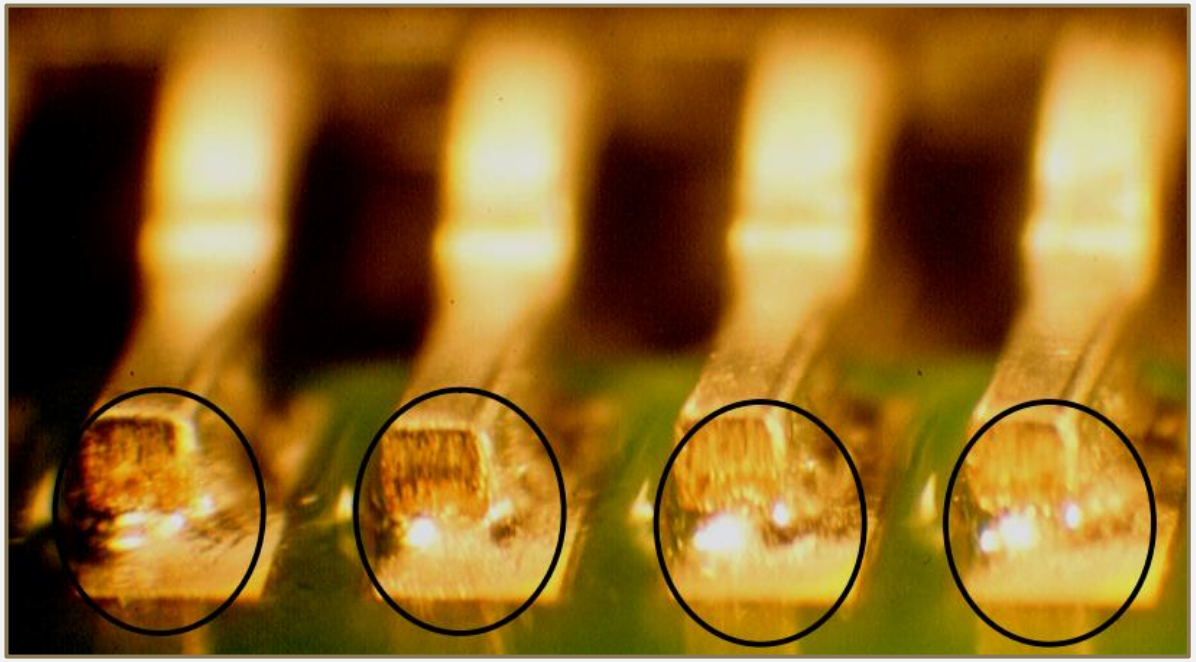
Solder plus Support

Bridging

- Possible Causes:
  - Excessive part placement pressure, insufficient board support or incorrect alignment
- Remedy:
  - Check and correct placement settings

Opens or Insufficient Fillet

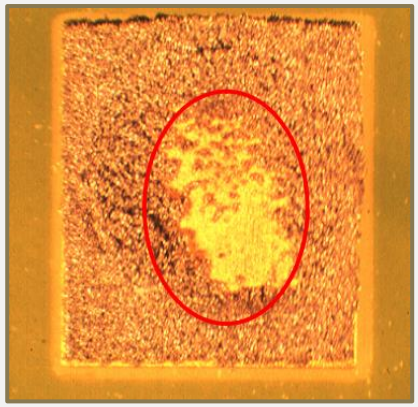
- Recognized by insufficient solder to make a complete bond between the lead and the pad



Solder plus Support

Opens or Insufficient Fillet

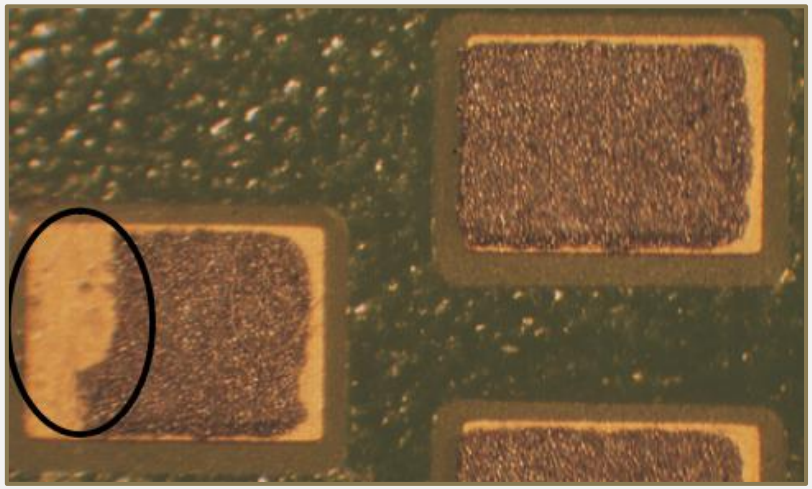
- Possible Cause:
  - Scooping - due to excessive squeegee pressure. The excess pressure forces the edge of the squeegee to dip into the aperture, scooping out paste intended to be deposited
- Remedy:
  - Decrease pressure
  - Use harder durometer polypropylene squeegee
  - Use metal squeegee



Solder plus Support

Opens or Insufficient Fillet

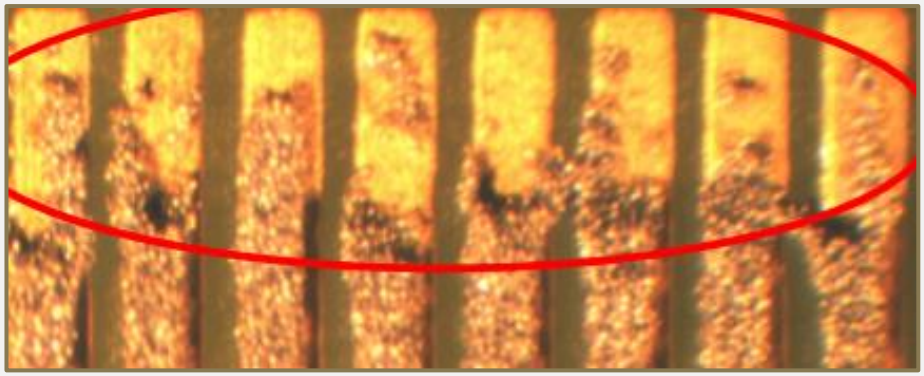
- Possible Causes:
  - A foreign body, dried up paste in stencil aperture, or possible over print of solder mask on the effect pad
- Remedy:
  - Regular examination and cleaning of stencil
  - If over print of solder mask, customer needs to resolve situation with PCB supplier



Solder plus Support

Opens or Insufficient Fillet

- Possible Causes:
  - Squeegee speed too fast to fill the apertures
- Remedy:
  - Slow down squeegee speed and see if problem is corrected



Solder plus Support

Opens or Insufficient Fillet

- Possible Causes:
  - Solder paste wicking up leads as a result of uneven or excess heating
- Remedy:
  - Reduce reflow peak temperature
  - Increase bottom-side heat

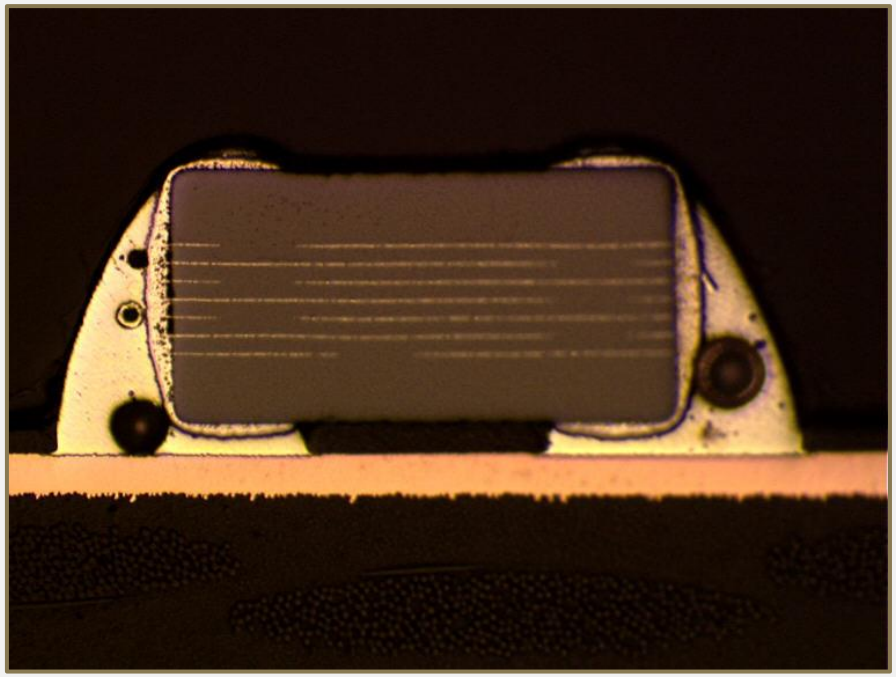
Solder plus Support

Opens or Insufficient Fillet

- Possible Causes:
  - Solder paste viscosity is too high not allowing paste to roll on the stencil and/or fill the apertures
  - Separation speed too fast
  - Separation distance too small
  - Poor stencil cut
- Remedy:
  - Check viscosity and metal content to see if manufacturer's specifications are met
  - Check other settings one by one until solution is found

Voiding

- Recognized by the appearance of many tiny or a few large “bubbles” in the joint. These may be air or flux entrapment
- Commonly found through X-ray or cross section inspection



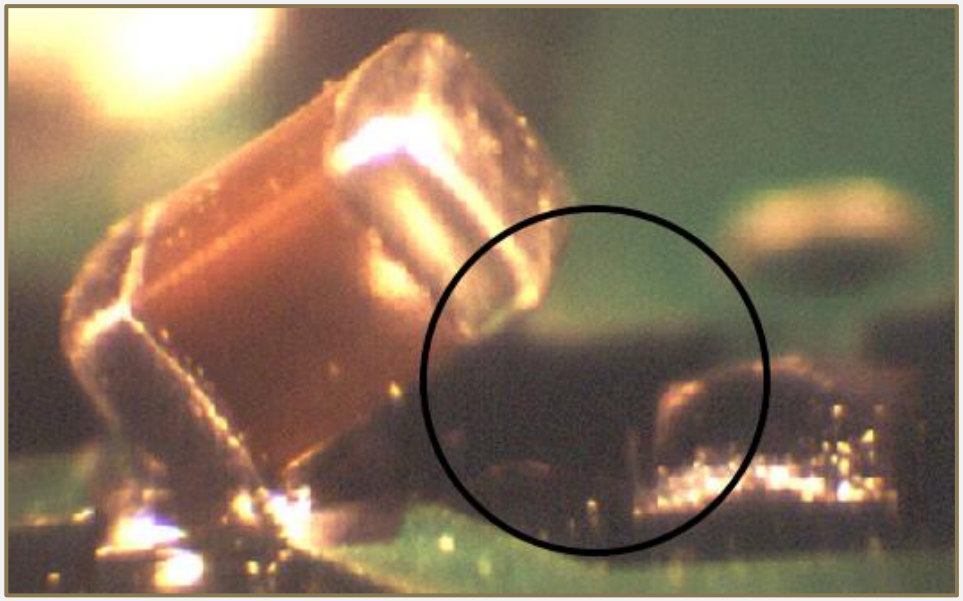
Solder plus Support

Voiding

- Possible Causes:
  - Peak temperature is too low
  - Too short of time at peak temperature
  - Soak too long
  - Soak too hot
  - Excessive ramp rate
- Remedy:
  - Inspect reflow profile and adjust accordingly

Tombstoning

- Recognized by chip type components standing on end after reflow
- Root cause is unequal soldering forces being exerted on each end of component.
  - Common in vapor phase reflow



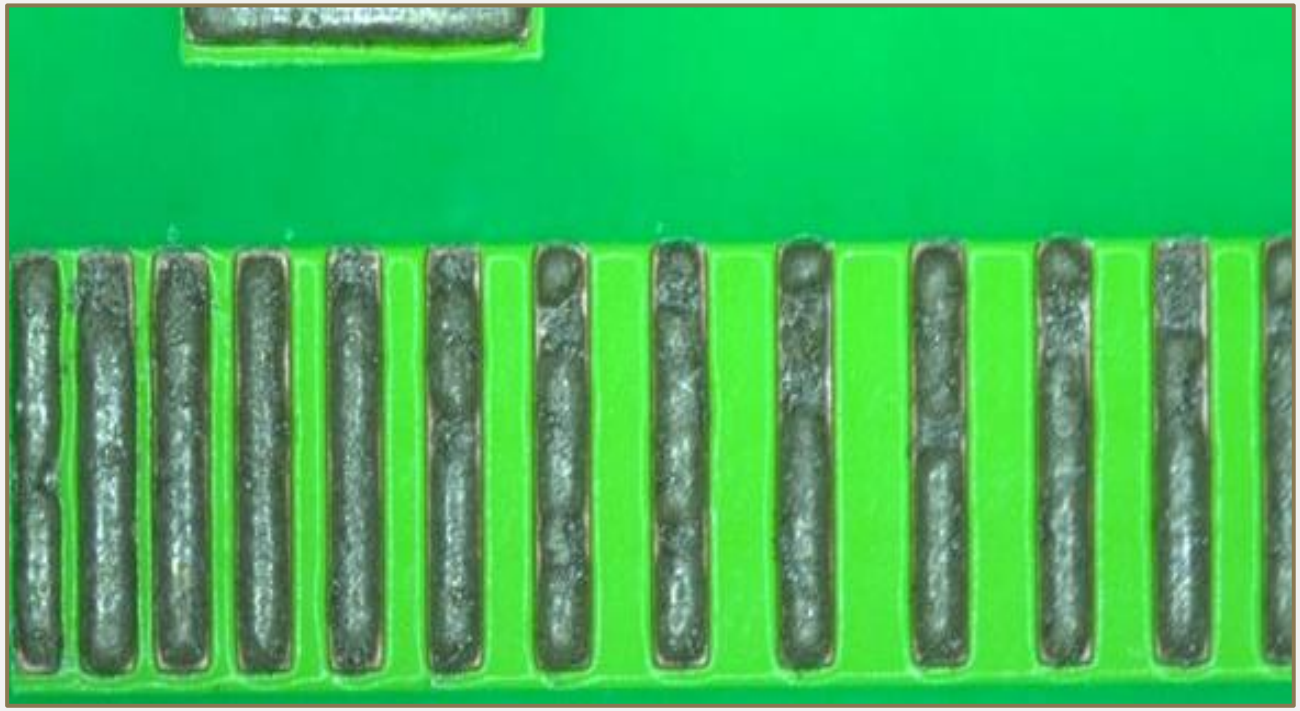
Solder plus Support

Tombstoning

- Possible Causes:
  - Unequal heat sink, such as ground plane that draws heat from one end of the component
  - Solderability of the components may be marginal
  - Unequal placement on the pads prior to reflow
    - Will result in an imbalance of wetting forces
- Remedy:
  - Increase duration of soak cycle to ensure all parts of the PCB are at uniform temperature
  - Test components for solderability
  - Alter placement position to ensure equal placement on pads

Unmelted Paste

- Recognized by unreflowed paste on the component pads



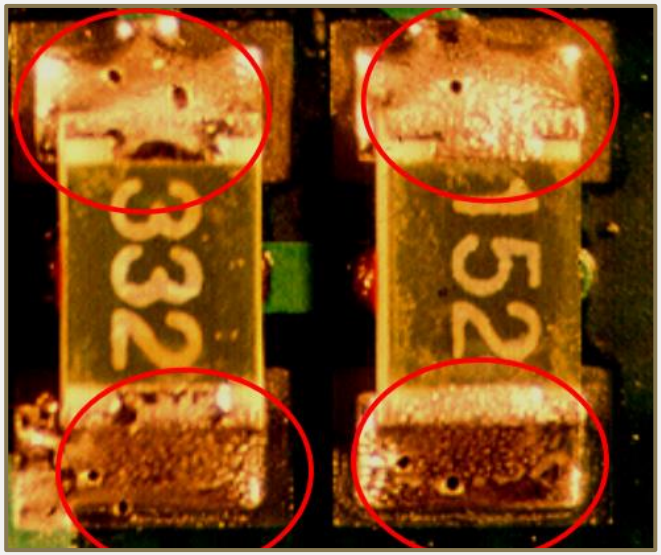
Solder plus Support

Unmelted Paste

- Possible Causes:
  - Too low of a reflow profile peak
  - Too short of time at peak temperature
  - Too long of a soak zone
  - Too hot of a soak zone
  - The wrong alloy/reflow profile is being used
- Remedy:
  - Inspect reflow profile/alloy and adjust accordingly

Disturbed or Cold Joint

- Recognized by dull rough appearance of solder in an alloy that is normally bright and shiny.
- Testing of joint will result in findings of reduced mechanical strength as a result of little or no intermetallic bond



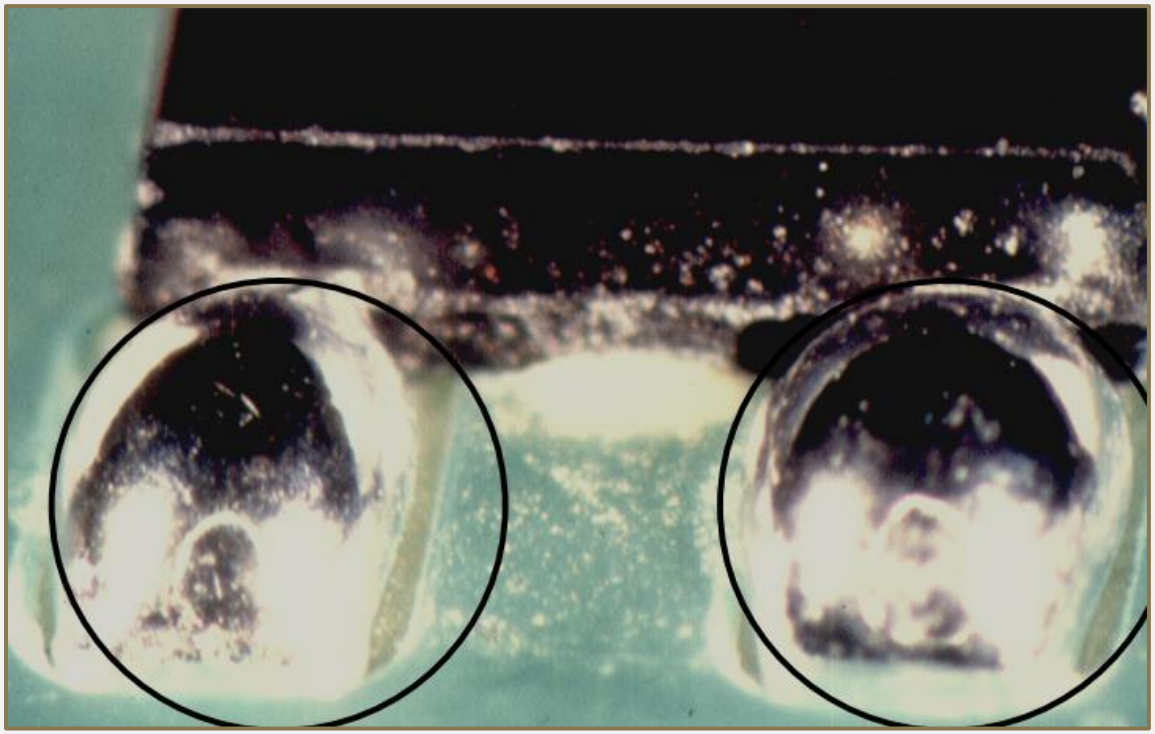
Solder plus Support

Disturbed or Cold Joint

- Possible Causes:
  - A source of vibration that could be transmitted to the PCB after reflow of the solder paste while the solder is still in a liquid state
    - Often is a jerking belt or conveyor in the oven
  - Too long of a cooling cycle
  - Alloys with high percentages of lead are naturally dull
- Remedy:
  - Fix the source of vibration and/or adjust the reflow cool down profile so that the joints are solidified upon exit

Excess Solder on Fillet

- Recognized by bulbous convex appearance of joint where the outlines of the leads are obscured by the quantity of solder on them



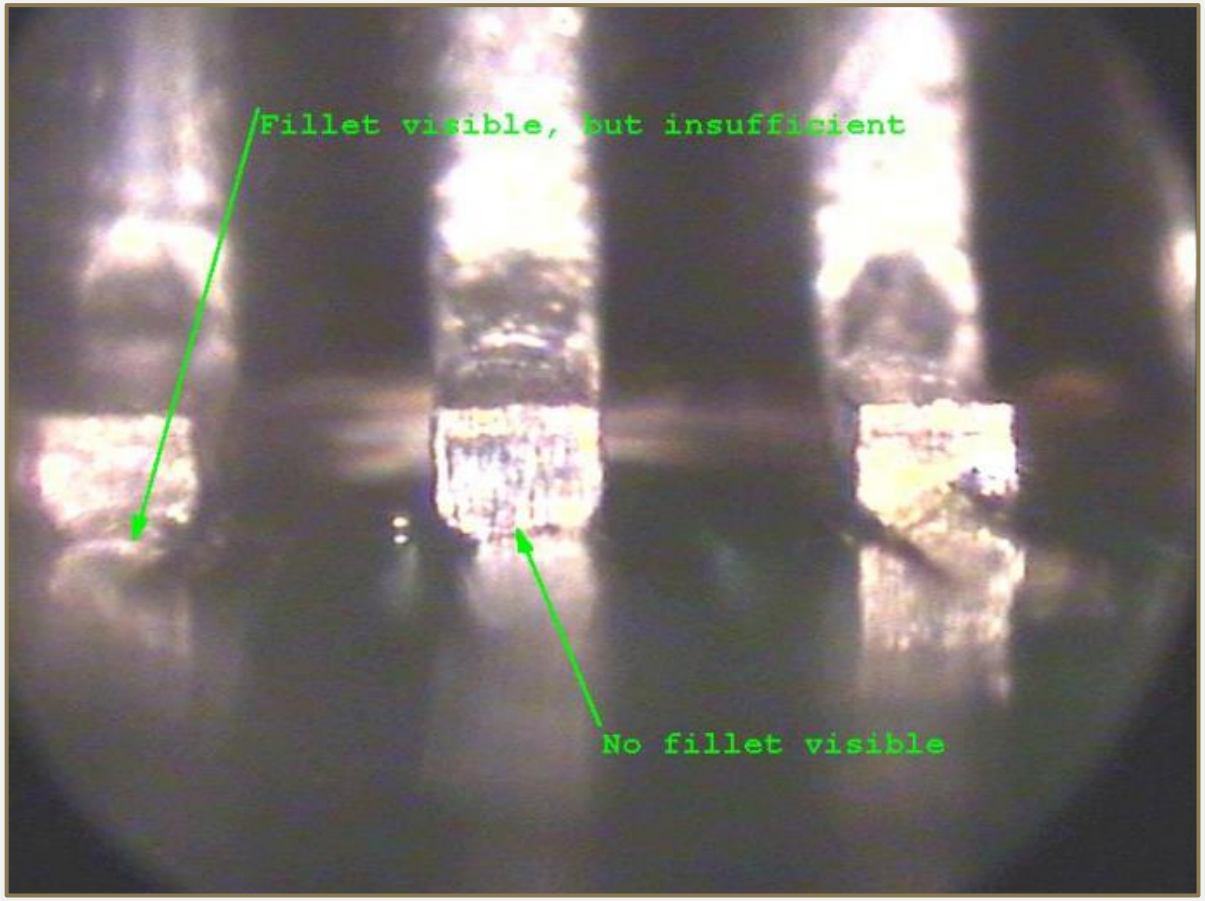
Solder plus Support

Excess Solder on Fillet

- Possible Causes:
  - Conditions which have caused too much paste to be deposited on the pad of the joints with excessive solder on them
    - Stencil thickness
    - Squeegee pressure
    - Snap-Off
    - Bleed out
    - Aperture Size/Design
- Remedy:
  - Reduce the amount of paste printed on the affected areas

Non-Wetting

- Recognized by insufficient solder fillet



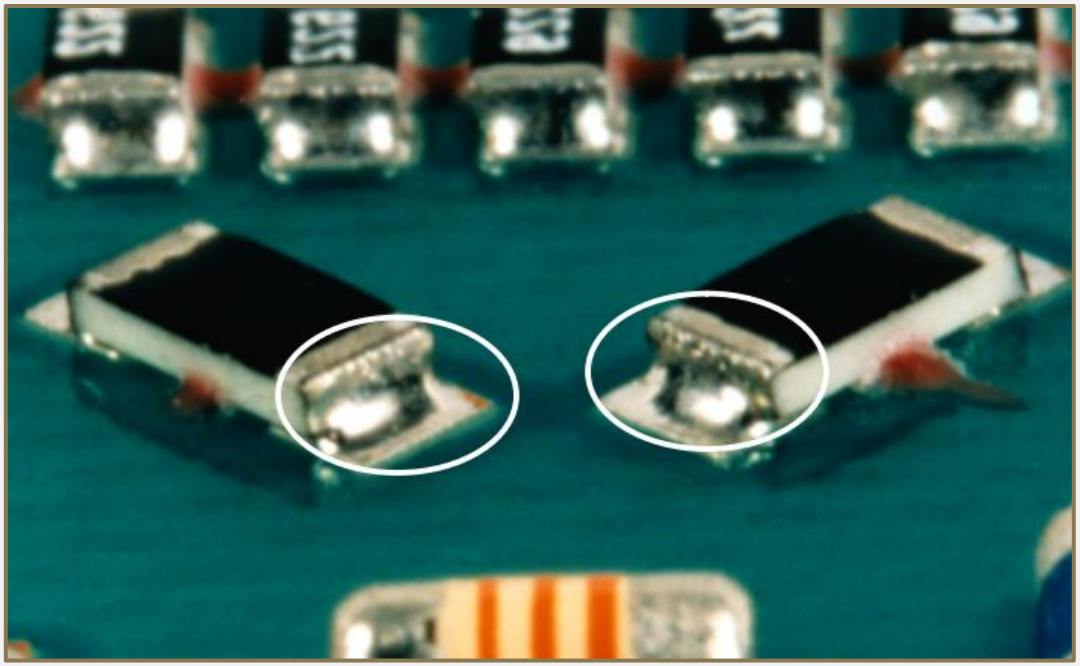
Solder plus Support

Non-Wetting

- Possible Causes:
  - Grease, oil or dirt on the surface to be soldered
  - Bleeding or misregistered soldered mask
  - Excessive, incorrectly placed, or excessive adhesive
  - Surfaces too heavily oxidized for solder paste being used
  - Old boards and/or components being used that have suffered from copper migration
  - Scratched pad resulting in exposed copper
  - Poor activation of paste resulting from paste chemistry or reflow profile
- Remedy:
  - Investigate each possible cause one by one until solderability is restored

Dewetting

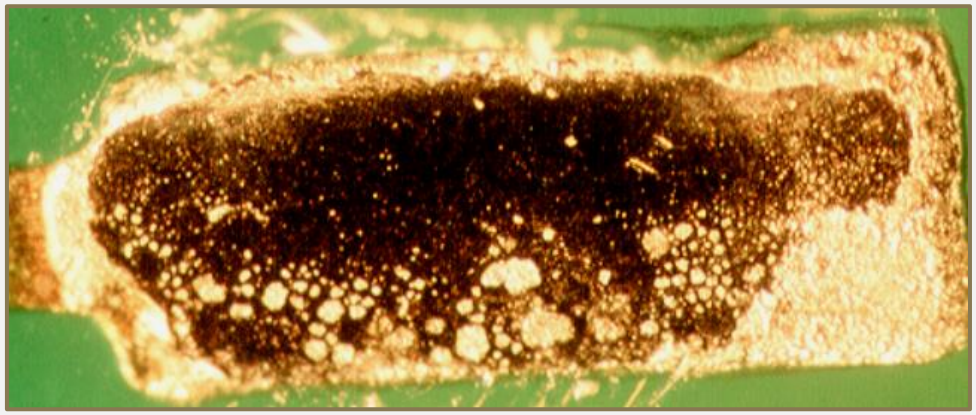
- Recognized by metal wetting initially, then pulling back to form droplets of solder on the surface
- Dewetting will give the surface a tinned appearance



Solder plus Support

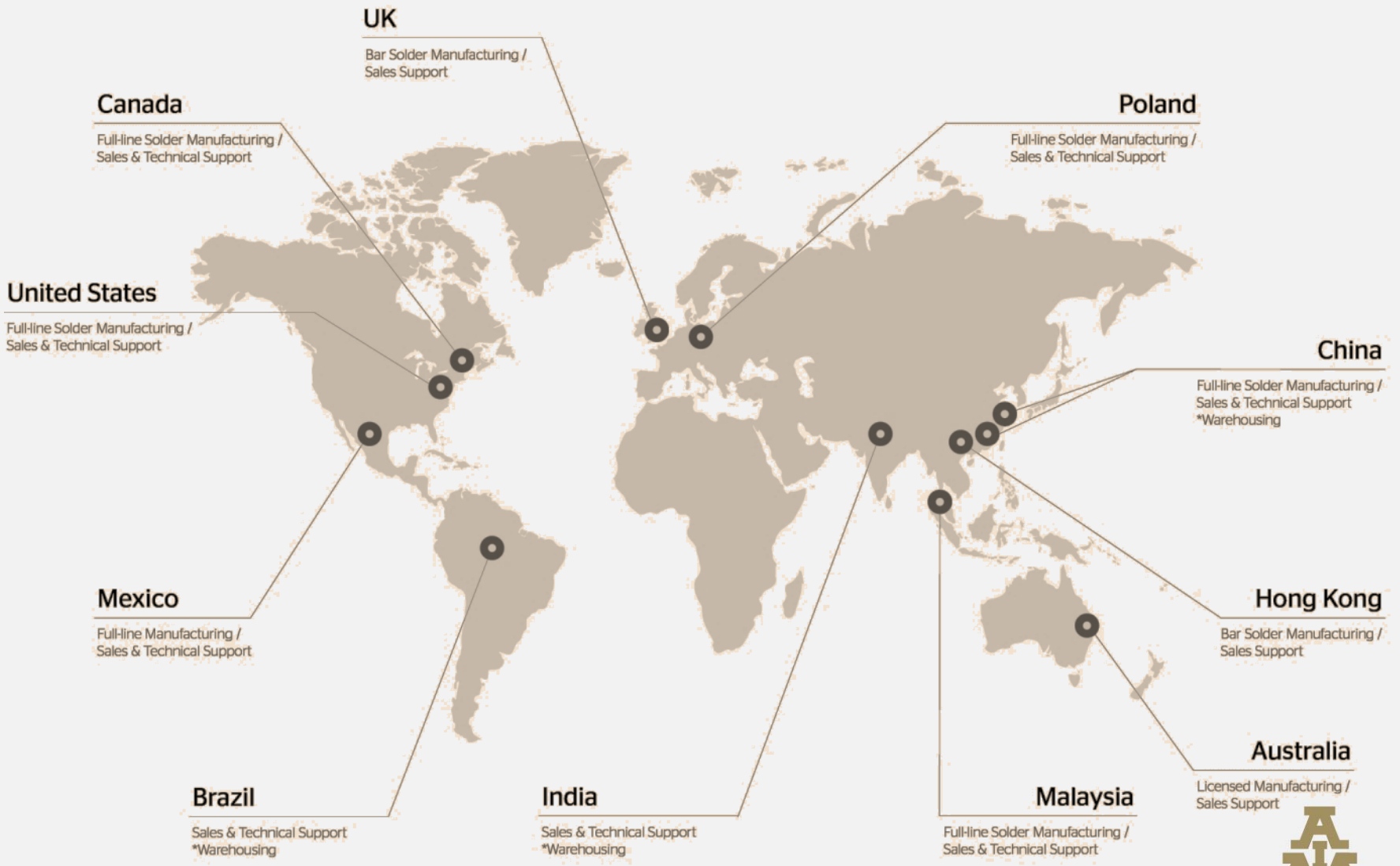
Dewetting

- Possible Causes:
  - Contamination of surface by abrasives
  - Poor plating
  - Poor hot air solder leveling during PCB manufacturing
- Remedy:
  - Investigate each possible cause one by one until solderability is restored



Solder plus Support

AIM Global Locations



Solder plus Support

Thank You

Solder plus Support

www.aimsolder.com info@aimsolder.com