Problem Description:

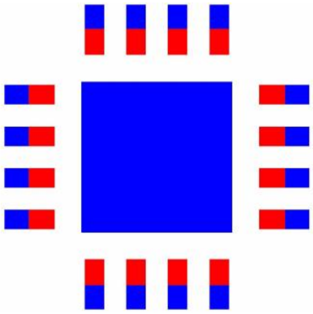
SMT components QFN 16-pin and QFN 28-pin are having insufficient solder volume at reflow. 0402 components are experiencing tombstone and solder ball problems at reflow. The assembly is also experiencing cold solder joints and poor wetting at reflow.

Top
QFN 16-pin
QFN 28-pin
0402 components

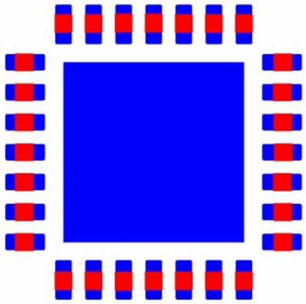
Bottom
0402 components

Root Cause:

QFN



Allegro QFN 16-Pin
(red: termination, blue: SMT pad)



Microchip QFN 28-Pin
(red: termination, blue: SMT pad)

For leadless components, the terminal should cover 90%-100% of the SMT pad on the printed circuit board. On this assembly, the Allegro terminals cover 50% (terminal length is 0.0157" and SMT pad length is 0.0315") of the SMT pad and the Microchip terminals cover 47% (terminal length is 0.0157" and SMT pad length is 0.0335") of the SMT pad. The root cause of the insufficient solder volume is a result of too much surface area for the solder to wet to with enough left over to form an acceptable solder joint.

On leadless components, solder will uniformly wet to the surfaces of the terminal and SMT pad. When the terminal is 90%-100% of the SMT pad, solder is shared between the two surfaces. When the SMT pad is longer than the terminal, it creates surface area that is not shared between the two surfaces. Solder will be pulled from the joint during reflow to cover the additional surface area and the result is insufficient solder volume at the terminal.

There are many assemblies where SMT pads are lengthened on leadless components to aid in the inspection process or provide SMT pad area for rework. Unfortunately, lengthening the SMT pad creates rework. In some cases, the component specifications provide recommended land pad designs that are incorrect, based on thermal mass and the amount of surface area that has to be covered by solder.

Solder Balls

The assumption is the solder balls are located around the 2-pin chip components. The solder paste array files that were sent for this assembly did not indicate any home plate apertures were used on the stencil. If this is the case, the root cause of the solder ball problems could be the lack of home plate apertures and the no-clean solder paste. However, the current reflow profile also shows a very steep temperature ramp in the soak zones of the oven. Ramping more than 2°C/sec, and more than 3°C/sec, can cause very rapid outgassing which can easily "spit" solder away from the SMT pads and onto the soldermask. The steep temperature ramp can also exhaust the activators in the flux too quickly. There are simply not enough activators left in the later stages of the oven to thoroughly clean the SMT pads, component leads, and solder powder. Solder powder with oxidation still left on the solder particles prevents coalescing of the solder, leading to poor wetting and solder ball problems.

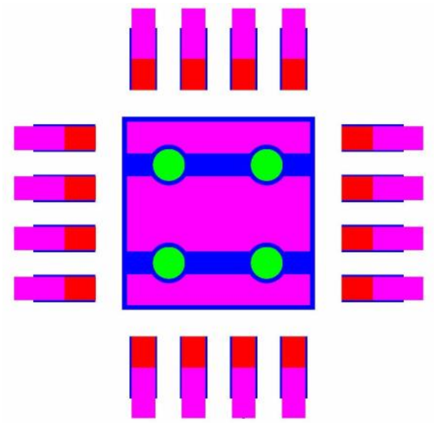
Cold solder and Poor Wetting

The steep temperature ramp can cause cold solder joints and poor wetting by exhausting the activators too quickly. The current profile length is also approximately six minutes. This is beyond the recommended 3.5-4.5 minute profile length. The longer the profile length, the more time available for oxidation to build up after the flux has removed it. This build-up of oxidation can cause joints to look like "cold solder." It can also cause "graping" where solder balls are trapped in the solder joint.

Recommendation:

QFN

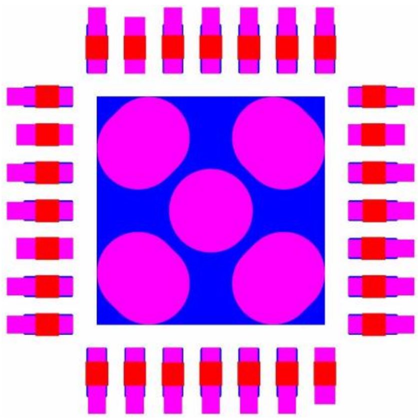
To eliminate the insufficient solder problems with the QFN components, the volume of solder printed must be substantially increased, given the small leadless terminals and large, comparatively, SMT pads. Fine Line Stencil has developed proven formulas that calculate the required volume increase eliminate the insufficient solder volume problem without causing bridging. The recommended stencil aperture size is as follows:



Allegro QFN 16-pin: 0.0118" x 0.0415" (25% volume increase for terminals and 40% volume reduction for thermal pad)

(blue = SMT pad, red = IC foot, magenta = recommended stencil aperture)

To minimize voids in the thermal pad, the four vias are not pasted. Applying solder paste around, but no directly over, the open vias minimizes the amount of flux and solder that gets lost to the via barrels. Voids in thermal pads with open vias are common not due to trapped gases, but insufficient solder volume. Not printing directly over the vias retains more of the solder volume for the joint between the PCB and component thermal pads.



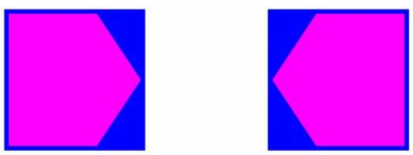
Microchip QFN 28-pin: 0.0118" x 0.0445" (26.5% volume increase for terminals and 40% volume reduction for thermal pad)
(blue = SMT pad, red = IC foot, magenta = recommended stencil aperture)

Some of the stencil apertures on the Microchip QFN are shorter and wider. This is due to the 0.0445" long aperture printing onto open vias. Solder volume will be lost to the open vias. The stencil apertures in several locations were shortened and widened to keep the same volume, but avoid the vias.

There are no open vias in this QFN. The "5-dot" aperture minimizes voids and skewing of the QFN.

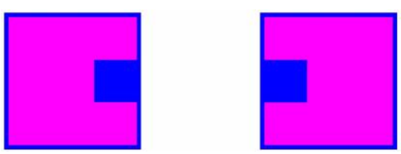
Solder Balls

For no-clean solder paste, a home plate aperture is recommended to prevent solder balls from forming during reflow. The most common home plate design is called a "standard home plate" and is as follows:



From a physics perspective, this design is not optimized to eliminate solder ball problems. Solder balls generate at the midpoint of the chip body. The above design still has solder paste at the midpoint. The surface area of the solder paste also decreases as it moves further underneath the chip body. This can allow tombstone problems to occur since there is not enough adhesion between the solder paste and chip body to keep the body down during the reflow process.

The much more effective home plate aperture is called "U-shape." It is as follows:



Solder paste is removed from the chip midpoint. This is much effective in eliminating solder ball problems. There is also constant, and approximately 25% more, surface area between the solder paste and underneath the chip body. This helps in preventing tombstone problems. Lastly, the solder will wet to fill the area removed at the chip midpoint. As it does, equal-and-opposite wetting forces are produced that help keep the chip self-centered during reflow. Boards with centered chips have a much higher “visual quality” than one where the chips are skewed.

Cold solder and Poor Wetting

To improve the wetting and visual appearance of the solder joints, it is recommended that the oven settings be changed. These changes will also help with solder balls and tombstone problems.

Recommended Oven Settings

Belt speed: 16 in/min
Zone 1: 212°F (100°C)
Zone 2: 266°F (130°C)
Zone 3: 329°F (165°C)
Zone 4: 392°F (200°C)
Zone 5: 455°F (235°C)
Zone 6: 491°F (255°C)