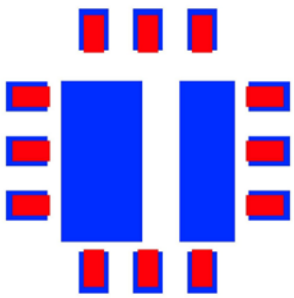
FCT Assembly LEADERS IN TECHNOLOGY	**Fine Line Stencil Root Cause Analysis Report**	**Rev:** A	**Document no:** **RootCause**

Problem Description:

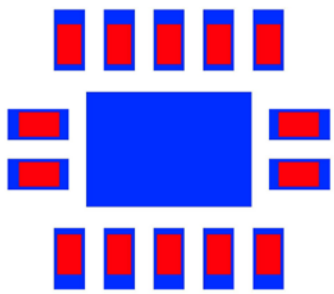
SMT components U3 (12-pin MLP) and U12 (14-pin SOT762) are having bridging problems at reflow

Top
MLP 12-pin: U3
SOT762 14-pin: U12

Root Cause:



MLP 12-Pin
(red: termination, blue: SMT pad)



SOT762 14-Pin
(red: termination, blue: SMT pad)

On these two components, the smallest clearance between the land pads for the terminations and the center GND pad is 0.006” for U3 and 0.007” for U12. Based on this layout, I am expecting the bridging to be between the termination land pad and the center GND pad rather than between the component terminations. The components will settle as the solder paste heats up and the solder paste will spread. Bridging on U3 is most likely on the bottom 3 connections and possibly to the 6 on the sides.

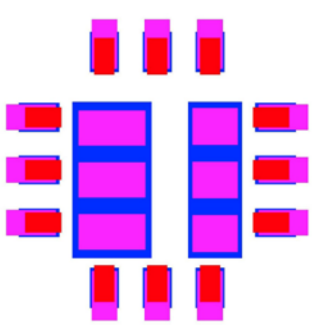
For U12, bridging can be between the 4 side pads and the center GND pad and between terminations. The land pads extend too far beneath the component body. If the solder paste bricked is not shifted outward, there is a lot of solder paste beneath the body. Flux can migrate along the body causing bridging between terminals.

The size of the terminations was also compared to the size of the land pads. For leadless components, the termination should cover 90%-100% of the SMT pad on the printed circuit board. On this assembly, the U3 terminations cover 90% of the length, but the width of the land pad (0.017”) is too wide for the termination width (0.0118”). The land pad width is typically 0.002” wider than the termination. At reflow, solder will be taken away from the joint to fill the additional surface of the land pad. If the stencil aperture volume is not increased, insufficient solder can be a problem. This will be difficult to see since the joint width is covered by the component.

For U12, the terminations only cover 65% (termination length is 0.0157” and SMT pad length is 0.024”) of the SMT pad. The land pad width is good, but the extra length of the land pad will create insufficient solder volume at reflow. The insufficient solder volume, for both U3 and U12, may be hidden at the moment due to the more obvious bridging.

Recommendation:

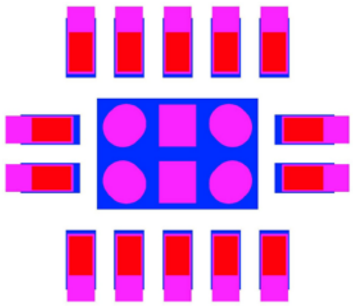
To eliminate the bridging, and potential insufficient solder, problems with the components, the solder paste bricks need to be shifted and increased. Fine Line Stencil has developed proven formulas that calculate the required volume increase to eliminate the insufficient solder volume without causing bridging. The recommended stencil aperture sizes are as follows:



MLP 12-pin: 0.0295” x 0.015” (20% volume increase for terminals and 40% volume reduction for thermal pad)

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

Stencil apertures shifted 0.002” to the outside to increase the clearance between the solder paste bricks and the center GND pads



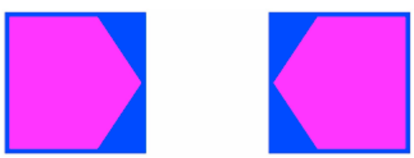
SOT762 14-pin: 0.0273” x 0.011” (19% volume increase for terminals and 40% volume reduction for thermal pad)

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

Stencil apertures shifted 0.0027” to the outside to increase the clearance between the solder paste bricks and the center GND pads

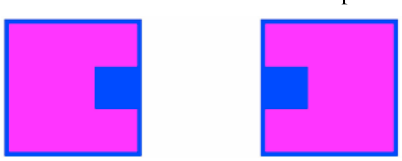
Solder Balls

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.