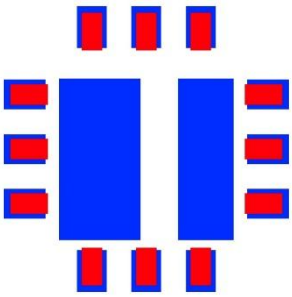
FCT Assembly Solves Bridging Issues at Reflow

Tuesday, March 27, 2012

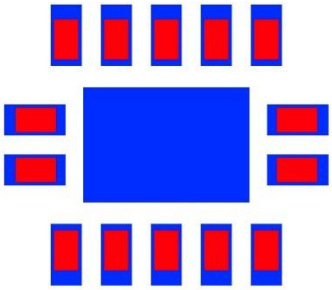
Root Cause

As the electronics assembly industry evolves, printed circuit board (PCB) features and surface mount technology (SMT) components continue to get smaller and smaller. This miniaturization shrinks the process window at print, placement, and reflow, increasing the opportunities for defects.

Just recently, Bob Dervaes, V.P. of Technology, performed a post reflow analysis on an assembly having bridging problems at reflow. The MLP 12-pin and SOT762 14-pin components, below, were bridging from termination-to-termination and from termination to the center GND/thermal pad, requiring post reflow rework.



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



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

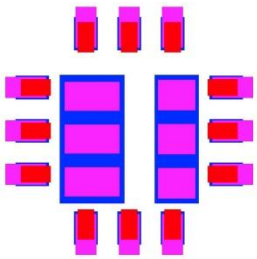
For the bridging from termination-to-GND, the smallest clearance between the land pads for the terminations and the center GND pad is 0.006" for the MLP and 0.007" for the SOT762. This clearance is extremely small and can easily be bridged as the solder paste heats up and spreads during reflow.

The bridging from termination-to-termination is a result of stencil apertures that are too wide. The assembly has 1 oz copper, prior to through-hole plating, on the outer layers. There will be a size reduction, at the top of the PCB land pad, due to the copper etching process. For 1 oz copper, the reduction will be between 0.002" and 0.003". If the stencil aperture width is not reduced accordingly, solder paste can squeeze out between the stencil and PCB and create bridging on fine pitch devices.

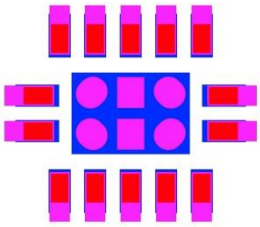
Recommendation

To eliminate the bridging, the clearance between the solder paste bricks on the termination land pads and center pad needs to be increased. The stencil aperture width also needs to be decreased to compensate for the land pad size reduction, due to etching. However, solder volume is extremely critical on small components and large reductions in the width can cause insufficient solder volume problems at reflow. The stencil aperture length needs to have a corresponding increase to ensure sufficient solder volume. For leadless components, length should always be increased on the outer edge of the PCB land pad. Increasing the length to the inside increases the possibility of flux migrating along the component body, causing bridging.

FCT Assembly has developed proven formulas that compare the termination and PCB land pad sizes to calculate the required stencil volume. We recommend the following stencil aperture sizes:



- MLP 12-Pin: 0.0295” x 0.015” (40% volume reduction for thermal pad.
    - Stencil apertures shifted 0.002” to the outside to increase the clearance between the solder paste bricks and the center GND pads
    - Clearance between the solder paste brick and center pad is a minimum of 0.0125”



- SOT762 14-Pin: 0.030” x 0.009” (40% volume reduction for thermal pad.
    - Stencil apertures shifted 0.0027” to the outside to increase the clearance between the solder paste bricks and the center GND pads.
    - Clearance between the solder paste brick and center pad is A minimum of 0.0093”

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

About FCT Assembly Root Cause Analysis

FCT Assembly offers Complimentary Root Cause Analysis services to customers that encounter assembly problems anywhere in the SMT assembly process. Our technical support team has been working in the electronics assembly industry for decades and can address yield problems at print, placement, and reflow. The majority of yield problems can be resolved with just the PCB files, component specifications, and reflow profiles, if needed. For yield problems that require physical printing, placement, and reflow, we have a state-of-the-art SMT assembly test lab where defects can be duplicated to determine the optimum solution. Our goal is always to eliminate post reflow rework.

Visit www.fctassembly.com for more information!