Bridging at Reflow, What is the Cause and Can it be Eliminated?

Robert Dervaes, V.P. Technology, FCT Assembly

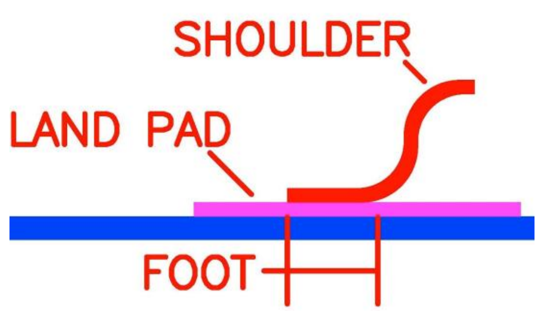
Surface mount technology (SMT) started in the 1960s and became more common in the 1980s. It is the dominant technology in use today. Through-hole technology is still in use, and will be for the foreseeable future, but the drive towards miniaturization of components is only possible with SMT and surface mount devices (SMD). While smaller SMD sizes allow for smaller, and more powerful, assemblies, they present many more challenges during the SMT assembly process.

The SMT assembly process has a multitude of variables that have a direct impact on process yields. Typically this is referred to as "the process window." A wider process window can effectively handle more variation throughout the SMT assembly process while still maintaining high assembly yields. Smaller SMD sizes narrow the process window and require tighter process control in order to maintain those same high assembly yields. The topic of this article will focus on one area of the SMT assembly process that frequently causes defects and subsequent rework to fix the problem – bridging on gull wing devices during reflow.

Everyone in the electronics assembly industry has had to perform assembly rework at one time or another. However, rework costs money, time, and solder joint reliability. All of these have a direct impact on quality whether it is functional failure of the assembly, cost overruns, or missed shipments. Eliminating all rework is the goal, but is that really possible? In the case of bridging on gull wing devices at reflow, the answer is typically "yes."

When moving through the reflow oven, the ideal scenario is to have all of the PCB land pads and component leads at the same temperature when entering liquidus. This will promote more uniform wetting of the molten solder between all of the surfaces. However, component substitutions and incorrect PCB land patterns can disrupt uniform wetting of the solder by creating temperature variations between the PCB land pads and component leads. Molten solder seeks out the highest temperature and temperature differences will change how, and where, the molten solder wets.

In the case of fine pitch, gull wing components, a foot (see image below) that is substantially (< 75%) shorter than the PCB land pad creates a sizeable difference in their thermal masses. The shorter foot has a lower thermal mass, and typically more surface area exposed to the heat, than does the PCB land pad. As a result, the gull wing lead will heat faster, and stay at a higher temperature, throughout reflow. Since molten solder moves towards higher temperatures, the bulk of the molten solder will wet to the foot, and up to the shoulder, of the gull wing lead. With a stencil aperture size based on the PCB land pad and a short enough foot (foot < 75% of the land pad length), there is too much solder volume at reflow for this component.



The gull wing lead has a limit as to how much solder volume will wet to the surface. Any excess flows off of the PCB land pad and bridges to adjacent pads, when looking at fine pitch components. The solution is to reduce the volume of solder paste deposited at print.
In extreme cases, the solder paste volume reduction is more than 50%. In the majority of cases, the volume reduction is between 25% and 40%. In all cases, the amount of solder paste volume reduction is dependent on a size comparison of the foot and the PCB land pad, as well as the solder paste stencil thickness.

It is important to review the gull wing component specification to determine where the gull wing foot touches the PCB land pad. Ideally the foot should be centered on the PCB land pad. However, component substitutions frequently mean the foot is no longer centered. The solder paste brick should be centered about the foot to maintain symmetric wetting of the solder. This is especially important when using lead-free solder as the wetting is not as strong as that of leaded solder.

The relationship between the foot and land pad sizes is highly repeatable and solutions can be applied to any leaded or lead-free assembly containing HASL, ENIG, immersion Ag, or immersion Sn surface finishes. When using lead-free solder and an organic surface protectant (OSP) finish, however, caution must be exercised. The reduction in wetting ability of lead-free solders, coupled with aggressive reductions in solder paste volume, do create scenarios where the lead-free solder does not wet the entire PCB land pad surface. OSP has a limited shelf life and will eventually break down, allowing for oxidation of the underlying copper. It is vital that solder wet the entire surface when using OSP. There are instances where this solution cannot be applied to PCBs with an OSP surface finish.

Bridging at reflow with gull wing components is very common, but can be completely eliminated by a size comparison of the gull wing foot and PCB land pad and then reducing the printed solder paste volume accordingly. I have successfully applied this custom stencil solution to many assemblies over the years. The increased reliability and cost and time savings is definitely worth it.

Visit www.fctassembly.com for more information!