Improving SMT Assembly Yields

Eliminating Insufficient Solder Volume with Leadless SMDs

*Robert Dervaes, V.P. Technology, FCT Assembly*

Regardless of where one is in the world, the desire to pack "more power" into smaller devices is a common thread. Advancements in technology and miniaturization go together like peanut butter and jelly. A cellular telephone from the 1990s had considerably more computing power than what NASA had available to launch the Apollo missions in the 1960s and 1970s. The NASA computers took up rooms and rooms of space. The 1990s cellular telephone would fit into a small bag. A modern smartphone is orders of magnitude more powerful, than either, and fits into one's pocket.

The majority of electronics today utilize, to some extent, surface mount technology (SMT) components. Without SMT, the reduction of component sizes, and product sizes, would be extremely difficult. However, reducing the component sizes increases the complexity of putting them onto printed circuit boards (PCB). The smaller the size, the more difficult it can be to print solder paste onto the PCB, precisely place the component, and send it through the reflow oven with no problems.

There are a myriad of SMT defects (bridging, solder balls, voids, etc.) that one has to try and prevent. In some cases there is only one assembly build and one has to try to get it "right" the first, and only, time. With the more widespread use of leadless packages, insufficient solder volume is one SMT defect that is occurring more frequently. This SMT defect is costly, but can be repaired at rework. Is this a defect one has to live with or is it possible to eliminate this altogether?

The short answer is "yes." The longer version involves a comparison of the leadless package termination size and that of the PCB land pad. The required ratio of the two is very consistent and adjustments to the solder paste stencil can prevent insufficient solder volume problems at reflow.

For sufficient solder paste volume, the optimum, based on SMT assembly, PCB land pad length is ~110% of the leadless termination length (see figure 1). However, inspection and rework of leadless components is extremely difficult when almost 100% of the land pad is underneath the component. For this reason, the majority of leadless land pad designs will lengthen the PCB land pad.

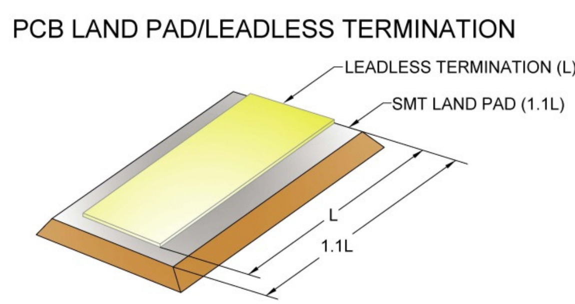


FIGURE 1

While in the reflow oven, the leadless package design tends to obstruct more of the convection air flow and IR, compared to a gull wing style component. Unless the PCB has extremely heavy copper weights, the leadless termination and PCB land pad temperatures will therefore increase fairly uniformly, and be close together at liquidus. This will produce a uniform wetting of the solder across the surfaces. Increasing the land pad length beyond 110% of the termination length increases the surface area that the solder has to cover. This will limit the formation of acceptable solder fillets, if the solder paste volume is not increased at print.

The volume increase is based on the size difference of the leadless termination and PCB land pad and is applied to the stencil. Referring to figure 2, the volume increase is calculated as follows:

Volume increase (%) = $50 * \left(1 - \left(\frac{L}{P}\right)\right)$

PCB LAND PAD/LEADLESS TERMINATION STENCIL DESIGN

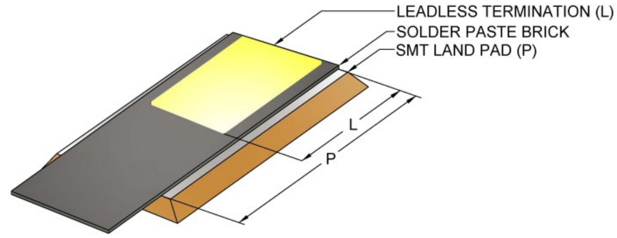


FIGURE 2

The additional solder paste volume should always be printed to the “toe” side for leadless components (extending the solder paste brick further underneath the leadless package should be avoided, due to the bridging potential) and increasing the stencil aperture width should also be avoided. Extending solder paste beyond the SMT land pad, up to 0.040”, is not a problem with leaded or lead-free solders (both coalesce and pull back onto the SMT land pad extremely well). However, it is extremely rare that an overprint gets even close to 0.040” with leadless components (an overprint to this extent is typically reserved for paste-in-hole applications). The majority of the time the extension is somewhere between 0.005” and 0.010”.

In addition to increasing the stencil aperture size, the stencil foil thickness is also very important. For the majority of leadless components, a 0.005” foil thickness is required. If the foil thickness has to be reduced to accommodate other SMT components, the stencil aperture volume for the leadless components should be increased accordingly. Solder volumes are critical and it does not take much of a volume reduction to start causing yield problems.

Leadless components are here to stay. While that may cause many to lose sleep, eliminating rework, due to insufficient solder volume, is possible. A comparison of the leadless termination and PCB land pad will determine whether or not the rework department should report to work. Catching this scenario at stencil design can fix the problem before it occurs and make one’s job a little easier. The cost is minimal. The effect on yields is priceless.