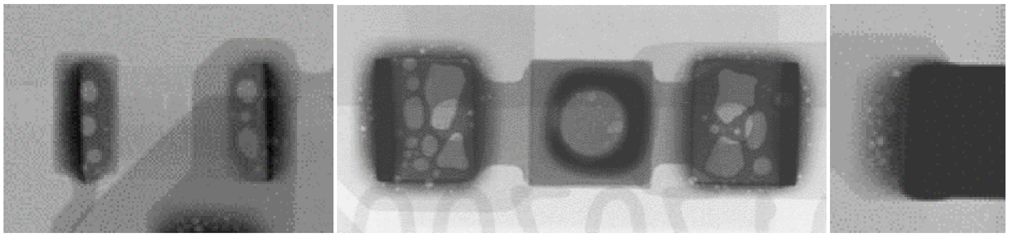
Voiding under Non-BGA's?

As requirements increase for high reliability assemblies, voiding under regular SMT joints continues to gain notice. There is a large amount of data available on acceptable voids levels in BGA's, however very little data is available for void levels under other SMT components. Due to lack of supporting data, the safest method is still to perform die shear testing to determine if the joint strength or integrity is being sacrificed.

When taking a look at the below pictures, 2 types of voids can be seen:

- Large voids underneath the component body
- Small voids in the fillet area around the component



Large voids: For the large voids underneath the component, this is most often caused by outgasing of the solder paste flux. In this case, flux did not have time to escape from underneath the component body. This is quite common and is nearly impossible to fully eliminate without the use of reflowing in vacuum.

However, this flux entrapment can be improved my modifying the reflow profile. The solder paste is composed of roughly 50% flux, and 50% powder by volume (11.5% flux by weight). Of the flux, roughly 50% is solvents that must outgas. Therefore nearly 25% of the solder paste by volume needs to outgas during the reflow profile. By adding a soak zone to the profile, more of this outgasing is allowed to occur prior to reflow. A majority of the volatiles are then removed before the reflow portion of the profile. Increasing the time above liquidus to ~70 seconds can also help allow additional time for any remaining solvents/outgasing to burst out of the molten solder.

Increasing the peak temperature should also be considered, however each application can give different results. By increasing the peak temperature to 245-250C, the solder will become more fluid and thus allow voids to more easily flow through and escape. However at the same time, at temperatures of 250C, more organics may begin to decompose that otherwise would have remained solid at temperatures of 235-240C. Therefore movements in the peak temperature should be trialed to find an optimum for the specific assembly.

Another common cause of voiding can be outgasing of the board through micro-vias embedded in the pads. Outgasing from trapped moisture in the board will flow along the glass fibers in the FR4 and escape upwards at micro-via locations. If these micro-vias are underneath a pad, they will cause voiding and profile modifications often will not be enough. Pre-baking and storing of boards in a Nitrogen environment can help with this root cause of voids.

Small voids in perimeter: The smaller voids in the fillet and underneath are often called "champagne voids" on Immersion Silver substrates. With other plated finishes, the same type of void can occur. During the plating process, if the plating is not performed correctly, contaminants from the plating bath can be plated in with the metal. These contaminants then outgas during the reflow causing small 25-50 micron voids along the surface of the substrate. These are difficult to remove, but profile modifications can help.

Conclusion: Complete elimination of all voids under joints is very difficult without the use of vacuum reflowing. However through profile optimization, a reduction of the voiding should be possible. Unfortunately the void reduction will need to be balanced with solderability and other defects, so the optimized profile for one assembly, may not work on the next. There are no industry specifications as to how much voiding is considered good or bad under passive components. Testing the shear strength of the components can determine if they are in fact reliable. The majority of the joint strength comes from the heal fillets in a joint. However void levels underneath should be monitored as well.

Please call your Kester technical support expert at 1-800-2KESTER (1-800-253-7837) for more information.