**AM I AT RISK TO USE THIS PCB**

When you consult in electronic manufacturing, perhaps the most common question you receive is, “am I at risk if I use these PCBs”? There is only one truly definitive way to answer the question and that is with an examination from a qualified laboratory. However, production pressures, laboratory lead times and cost usually require some “less than definitive” answer. Even with access to a laboratory and control of the lead time, I sometimes find it necessary to get some type of an answer “on the fly”. This usually occurs at a customer location with the product running. To address this need for less than perfect guidance, I have developed a series of simple tests that can be performed at the floor level with equipment generally available to the process engineer. The following is one of my top ten!

**Poor Wetting and De-wetting of the solder**

Poor wetting of the attachment pad is usually manifest in an irregular attachment to the solder pad and obvious areas where no solder adhered. I also find this with pads where the solder does not “wet” the pad to edges. In the solder process, the liquid solder must melt and dissolve the surface finish (no matter the type) in order for the tin to base metal intermetallic to form. In the case of ENIG (Electroless Nickel Immersion Gold), the base metal will be nickel after the (very thin) gold layer is dissolved into the solder. There is no visual examination of the ENIG surface that will give an indication of the solderability of the nickel. The following test works with all finishes, but it is particularly suited to ENIG

- Remove an array or single board depending upon how the PCB is normally processed.
  - Do no preparation (such as baking) to the array.
- Perform the normal paste print process.
  - Do not place any parts.
- Hold (delay) the array for the normal hold time and placement time of this board.
- Run through the fusion oven with the correct profile.
- Perform any post cleaning that would normally occur.
- If SMT both sides, run the array through the second side process.

Once complete, examine the attachment locations under a microscope at 10-20 X. The fused paste should have a uniform and rounded appearance on each pad. The past will attempt to form a parabolic shape in each dimension of the pad. If the pad is square (not common), the shape will be a uniform parabola. With rectangular pads, the shape will be a curve in both axis. If this is the result, the PCB is perfectly solderable. As you see degradation from this perfect shape, you will see corresponding soldering issues that require touch up in your process.

Look very closely at the second side of the SMT process. Every surface finish suffers some degradation when exposed to the Paste-Fuse-Clean process. Therefore, the second side is always the least solderable. This side is also where poor surface finishes will have the most impact on your solder connection. It is also the side that usually has the most difficult parts to solder!
One other benefit of this solderability test (and the reason for no preparation of the PCB) is that if you have moisture in the PCB or poor curing of the innerlayers, the PCB will show signs of delamination or meazeling.

A variant of this test is also useful. The joint strength of ENIG is significantly lower than HASL (Hot Air Solder Level). The strength of the ENIG joint is entirely dependent upon the condition of the underlying nickel barrier metal. This condition can be a series of things from passivation, cracks, or even captured contaminants all of which are easily covered by the immersion gold. I call this a “Push/Pull Pin Test”.

Bend a small diameter solid wire (28-30 AWG) into an “L” shape.

Push this small “L” into the (unfused) solder paste of several SMT pads on the PCB.

Do not make any attempt to “help” the solder connection by moving the wire around on the pad.

Complete the fusing and cleaning process.

Do the same thing on the second SMT operation (if there is one).

When the process is complete, grasp the exposed end of the small wire and pull at a 45 degree angle to the PCB until you pull the wire and pad from the PCB. Do this for all wires you installed. Examine the failure mode of the solder connection. If the wire pulled the pad from the PCB, then the ability of the solder to “wet” the pad is sufficient. This is correct even if the solder joint is not as “pristine” as a perfect joint. The reason is that the failure mechanism is the attachment of the copper pad to the “butter coat” epoxy of the PCB. The joint strength does not need to be any better than the attachment of the pad to the PCB.

All of my “Top Ten” are as follows in no particular order because on any given day, one of them will be #1! Future articles will address some of these.

• Poor Wetting and De-wetting of the solder
• Weak Solder joint strength
• Paste Print Coverage
• Insufficient hole fill of through-hole components
• Poor wetting of “topside” pads
• Solder Mask liftage
• De-lamination or Meazeling
• Contamination
• Warpage
• Low Tg (Glass Transition Temperature)

There is no “cliff” where the process goes from good to bad, only a slope from no rework to total rework. The guidance that I can give is that as you progress down the slope you will incur more rework. Every shop must make their own decision of when that is excessive. My goal has always been to have no rework. One step in this direction is to not process PCBs that do not pass these tests.

Stanley L Bentley, P.E.