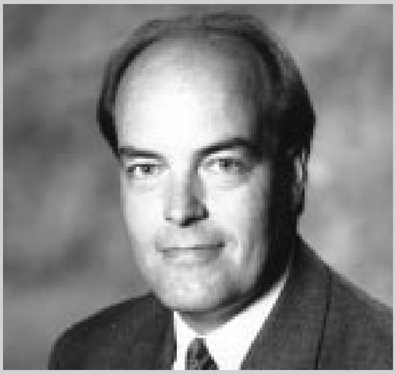
Process Solutions



Bob Willis

A PCB defects guide

During manufacture, packaging, distribution, and assembly there are always opportunities for defects to occur. In order to reduce the cost of scrap, you need not only to minimize the number of scraped boards but find the defective PCBs as early in the process as possible.

Upon examining defective boards scraped by manufacturers or assemblers, it is surprising to find how many boards are rejected on purely cosmetic grounds. Knowing the time and effort that has gone into each circuit's manufacture, I find this level of rejects quite disappointing.

To avoid unnecessary rejection, it is imperative that you know something about the real defects that occur during fabrication and assembly of PCBs. The fabrications stage defects include weak knee, plating voids, resist misalignment, tooling hole damage, de-lamination, dendrite formation, legend contamination, knee cracks, and inner layer separation.

Fabrication defects

The week knee is encountered during assembly of tin-/lead-coated PCBs or during testing for solderability by the circuit fabricator. During application of tin/lead by solder leveling, the edge of the hole may have a coating thickness of <1µm. If this occurs, then solderability life becomes short. When you are soldering, the solder may not rise on the plate, through the hole, and cover the knee. You can remedy this problem by correctly setting the solder leveling system and providing a tin/lead thickness of 2µm to 5µm.

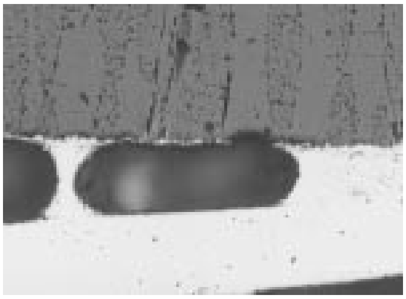


***Figure 1**: During soldering in the assembly stage, plating voids cause outgassing resulting in pin/blow holes in the solder joints.*

Plating voids (**figure 1**) can be caused by a number of process stages, but occur most commonly during electroless copper deposition. If you see poor coverage prior to copper plating, you can expect voids to be present. These defects are often difficult to see without destructive testing because they do not have much effect on electrical measurements.

The voids may be caused by poor drilling leaving a rough surface, poor agitation allowing bubbles to form in the hole, or by process contamination.

Solder-mask overlap or misalignment is often caused by poor design where too little room is left around pads. As a guide, keep all resist apertures 0.1mm to 015mm larger than the pad when you are using a photo-imaged resist. If you screen-print solder resist, you can use the 0.3-to-0.36mm range. I do not recommend a 1:1 resist to pad design.

Do not use tooling holes for multiple stages during fabrication—dedicate them for a specific process. If you reuse tooling holes for assembly after they have been used during fabrication, you may cause considerable damage to occur and affect their accuracy.

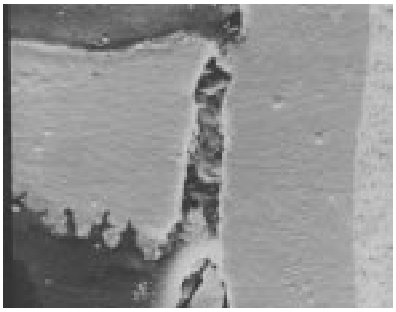


***Figure 2**: Inner-layer separation occurs when de-smearing, a cleaning process, is not properly conducted.*

You may encounter dendrites when testing circuits during humidity and voltage exposure. Dendrites are copper growths forming between contacts when you apply a voltage to the circuit. They form during electrolysis because of ionic contamination on the surface of the board. The contamination may come from the plating solution or reflow or leveling fluxes that are not properly cleaned off the surface of the circuit.

Legend ink contamination is another common problem. It is usually caused by poor circuit design rules. Ink printing is achieved by screening and is an inaccurate process particularly on large panels. During printing, the screen that defines the image stretches; hence, the inaccuracy. Design engineers often do not leave enough room around features, and the tolerance level drops.

Defects from thermal stress

You may have often seen de-lamination during a soldering process—solder leveling, wave soldering, or reflow. Its cause is moisture in the board expanding and pushing apart the layers of the board.

Figure 3*: The component has lifted because of the thick solder mask; during reflow it acted like a fulcrum lifting the part just above the pad surface.*

Cracking on the knee of the copper through-hole plating is uncommon today, but you should know about it. During soldering or the application of solder by leveling, the laminate significantly expands. The expansion takes place in the Z axis and, if the copper is not ductile, a crack occurs. The problem is related to what you add to the copper plating vat.

Another defect that arises due to thermal stress is inner-layer separation in multilayer boards (**figure 2**). This happens whenever the adhesion of the copper through-hole plating is not sound. Make sure that the through-hole is properly cleaned prior to metallization to avoid this problem.

Assembly defects

Since a significant portion of assembly work is now a surface mounting process, we will focus on this aspect. The defects list includes crystallized and cracked joints, joint failure, voids, component lift, solderability problems, and solder beading and wicking.

You may often come across crystallized joints on leadless ceramic chip carrier (LCCC) packages. It is a surface effect on the solder joint and may be caused by the joint remaining in the liquid phase for too long with no flux on the surface. It does not effect the joint quality and is a cosmetic defect.

Cracked joints result from flexure of the board assembly. They may also be caused by quality control staff probing the joints during inspection after reflow or, in the case of subcontracted assemblies, at goods receipt on the customer's site.

It is surprising just how much force can be developed with an inspection probe. It is well in excess of the 800-to-1,000g, which is seen during joint pull-off measurement.

Joint failures may occur during assembly even if the soldering operation is successful. Failure may constitute the separation of the joint from the pad. This can be avoided only by proper cleaning prior to the plating operation in the fabrication stage.

You may also see voids in surface-mount joints, which are caused by either volatile gas not escaping or non-metallic material not being displaced before the solder solidifies. You will often find voids under leads, the heel of gull wing leads, and in BGA terminations. Excessive development of voids reduces joint pull strength but does not necessarily reduce the reliability of the joint.

Although paste suppliers provide basic profiles, it is still necessary that you make adjustments to these profiles to avoid voiding, improve visual appearance of the joint surface, and reduce the amount of flux left after reflow.

Component movement and lifting (**figure 3**) are related to the soldering process and design of the board. If one termination reflows and wets before the other, surface movement is generated—rotational or vertical. Different solderabilities of the two terminations also cause this problem.

The use of vapor-phase reflow is more likely to cause movement than convection because of the fluid movement on the board surface. The incidence of component lift is increasing due to the increased reduction in the size of passive parts.

Poor pad wetting is seen often with nickel/gold and copper PCBs. They are relatively new coatings compared with tin/lead and not many know enough about them. Poor solderability of the copper pads may be caused by aging, poor handling, or excessive reflow temperature. If none of these have caused the defect, then the solderability was poor when the board was supplied.

Poor solderability of leads causes poor wetting. The solder paste during reflow may wet the pads but may be unable to flow up the leads. In this case, it is a component problem which needs to be discussed with the supplier. As a minimum requirement, there should be 5µm of tin/lead coating on the pins to provide a long solderable life.

Of beads and balls

Solder balls forming at the side of components and not at the joint surface are referred to as solder beads (**figure 4**). This is simply to avoid the confusion between the many other solder-ball phenomena.

As the board passes through the reflow oven and into the reflow zone, the paste turns into a liquid. All the solder balls coalesce together to form a solder joint. The solder beads form during assembly when solder paste gets under the body of the component. As the solder balls coalesce under the part, the increase in the size of the solder lifts the part to allow the liquid to escape. The component lowers back onto the board,

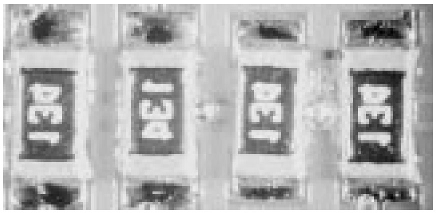


***Figure 4**: Solder beads often result from paste slumping. Remember to check this possibility when tracking down the cause of beads.*

leaving the solder bead.

To determine when the paste enters under the parts you need to check paste printing quality. Remove parts prior to reflow and check for paste. When the board has exited check for paste under parts, pass a fully loaded board through reflow, changing the final zone temperature to prevent paste reflow. By finding out when the paste gets under the parts, you will be able to eliminate the problem.

If you have this problem with LCCCs, it may be due to placement. When the component is placed onto the solder paste, it is forced under the body of the device. The LCCC termination and ceramic body are relatively flat and, therefore, some paste displacement must occur.

Another solder problem is wicking, which occurs when one of the board surfaces has poor solderability. This causes the solder to flow to one surface in preference to the other. Wicking may also occur if there is a significant difference between the reflow temperatures of the pad and pins. Correct temperature profiling will eliminate this problem. AEE

__Bob Willis__ is an independent process consultant. Please e-mail your comments and questions to __eps@bobwillis.co.uk__, or fax __44-01245-496123__. His home page is at __www.bobwillis.co.uk.__