Process Solutions

Bob Willis

Detecting PCB outgassing before circuit assembly

The subject of outgassing, as it relates to printed circuit assemblies during the flow soldering operation, has been an area of debate for many years. Until recently, the soldering equipment was blamed for pin holes and blowholes. However, any soldering system would have to be operated well outside effective soldering limits, and with poor quality control for outgassing to occur.

A research program of the Soldering Science and Technology Club, involving the National Physical Laboratory (NPL), various manufacturers, and trade groups, has established the cause of outgassing. Blowholes and pin holes are caused by the escape of gas through the plated copper hole during the soldering operation. The gas is water vapor escaping through the molten solder. Voids are formed when the gas continues to escape during the solidification of the solder joint. Outgassing may result in pin holes, blowholes, or sunken joints.

Outgassing can be eliminated if the copper plate is uniform in thickness and impervious to the escaping gas. The quality of drilling, the deposition of the electroless copper, and the thickness of electroplated copper, all play an important role in inhibiting outgassing. The drilling may cause problems due to the irregular surface that the copper will be required to cover and the need to bridge glass strands protruding into the through-hole plating.

Eliminating outgassing

You can eliminate outgassing by using the correct specifications for the PCB. Most national specifications quote a minimum thickness of copper plating in the through-hole connection. The specification states a minimum of 20-µm thickness but allows voids to be present also in the plating. Most PCB makers, of course, far exceed that quality.

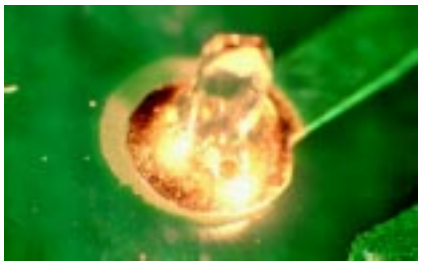

***Figure 1**: Water trapped inside the glass epoxy gets super heated during the soldering process. The escaping vapor may cause blowholes in solder joints.*

Tools needed to test for outgassing

1. Sample PCBs for evaluation.
2. Canada Bolson oil or suitable alternative, which is optically clear for visual inspection and can be easily removed after test. Such oils are generally offered by optical microscope suppliers.
3. A suitable hypodermic syringe for applying oil in each hole, and blotting paper for removing excess oil.
4. A microscope with top and underside lighting. You may use any other suitable optical aid that provides between 5 to 25x magnification, and a light box.
5. A soldering iron with suitable temperature control.

You should not request boards to be manufactured according to the national specification or to a specification close to it. Some companies accept boards to the national specifications, and end up with products that outgas during soldering. Passing the responsibility for soldering standards to second-party manufacturers will not solve your problem. The assembler will be required to rework the solder joint to improve the cosmetic appearance. And touching up solder joints has never been shown to improve the quality of the joint; in fact, repeated rework can cause damage.

You can reduce outgassing by baking the PCBs prior to the soldering process in order to remove moisture. It is essential to use the boards right after baking because all glass epoxy is hydroscopic and takes on moisture during storage. You must bake the PCBs at the correct temperature and with the correct time duration.

Non-destructive evaluation

You can conduct a simple, non-destructive test on PCBs with plated through holes at goods receipt, during production or on final assemblies to check for outgassing. Select a sample board or part of a board for examination. Inject each hole for examination with optically clear oil using a hypodermic syringe. This

prevents excess oil from being applied to the board. For effective examination, it is necessary for the oil to form a concave meniscus on the surface of the hole—the concave form allows an optical view of the complete, plated through hole. You can easily form a concave meniscus and remove excess oil using a blotting paper. If any air trapment is present, apply more oil until you get a clear view of the internal surface.

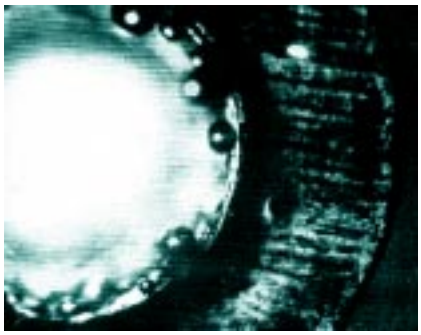

Figure 2*: This testing method provides the opportunity to record the results on both video or film for future reference/discussion.*

Next, mount the sample board over a light source to allow illumination of the hole. Require an optical viewing aid to examine the hole during test. A magnification of 5 will allow you to view bubble formation. For a more detailed examination of the through hole use a magnification of 25 times.

In the next stage reflow the solder in the plated through holes. This also locally heats the surrounding board area. The easiest method is to apply a fine-tipped soldering iron to the pad area on the board or to a track connecting to the pad area. The tip temperature may vary but 260°C is normally satisfactory. Examine the hole as you apply the soldering iron. Seconds after the complete reflow of the tin/lead plating in the through hole, you will see bubbles emanating from thin or porous areas. Outgassing is seen as a constant stream of bubbles which indicates pin holes, cracks, voids, or thin plating.

Generally, outgassing continues for a considerable time—in most cases it continues until the heat source is removed. This may last one to two minutes, and the heat may cause a discoloration of the board material. However, your assessment can generally be made within 30 seconds of the application of heat. After testing you may clean the board with a suitable solvent to remove the oil.

This test allows fast and effective examination of the surface of the copper or tin/lead plating. It may be used on through holes with non-tin/lead surfaces as well. In case of organic coatings, any bubbling due to the coatings will cease within a few seconds. AEE

Bob Willis *is an independent process consultant. Further process advice and information is available on his Internet home page at* ***http:///www.bobwillis.co.uk****. You may also e-mail him at* ***eps@bobwillis.co.uk*** *or fax* ***+44-1245-496123****.*