Section 2:
Printed Circuit Board Fabrication & Solderability

Good Boards = Results

Board fabrication is one aspect of the electronics production industry that SMT assembly engineers often know little about. It cannot be stessed enough that poorly fabricated boards WILL cause solderability problems during assembly.

This section covers PCB production basic steps as it effects the assembly of populated circuit boards. In addition, this section will describe how to apply this knowledge to prevent board solderability problems BEFORE the boards ever get to the assembly line.

Soldered boards exiting a reflow oven.

Pad Oxidation and Board Storage

Circuit board fabricators prefer to produce large numbers of boards in a single run. This reduces setup time and offers the benefits of economy of scale when ordering laminate, copper foil, prepreg, and other raw materials of printed circuit boards. These purchasing and manufacturing issues cause them to offer lower per board costs to assemblers if the assembler buys in large lots.

However, even in the era of just-in-time manufacturing, many companies keep boards on the shelf too long. If the assembler goes for the low per board cost and does not populate and solder the boards in a reasonably short time, oxidation buildup on pads, even if they have been plated, can cause solderability problems. Often reworking boards that exhibit poor solderability is more expensive than the savings gained by buying larger lots of cheaper PCBs.

Therefore, it is important for the processing engineer in a surface mount assembly line to work with purchasing and management to determine the safe shelf life of bare boards. Solderability tests can determine the safe time limit. Ensuring a good raw material of boards is one of the most important parts of high yield surface mount assembly. What makes board oxidation particularly critical is that it dooms the process before solder even touches the PCB.

PCB Manufacturing

Substrate Manufacturing

Substrate is manufactured from thin sheets of a dielectric material bonded to a sheet of electrically conductive material. Figure 2-1 illustrates the first three steps of circuit board fabrication.

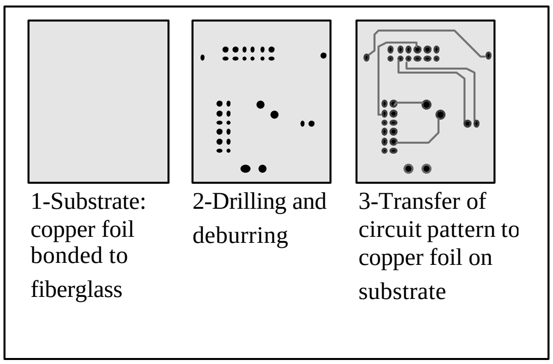


Figure 2-1. The first three steps in circuit board fabrication.

FR4 is the most common substrate used in printed circuit boards. Epoxy resin is used to bond fiberglass to copper foil in the creation of FR4. A fire retardant is added so the substrate can be safely soldered in later processes.

Some other substrates are:

- polyimide/fiberglass, which can sustain higher temperatures and is much harder than FR4
- FR2-fire retardant coated phenolic/paper, which is cheap and used mainly in low cost consumer electronics
- Flex circuits, usually polyimide, in some cases polyester, used in automotive and other applications where space and weight are at a premium. New technology allows adhesiveless material for even thinner flex circuits.

Problems that occur during substrate production usually have little impact on solderability during SMT assembly.

The Subtractive Process

After the substrate has been made, drilling machines bore holes of different diameters in the exact locations on the board. These holes are called vias. These vias are where circuits electrical connections are created between different layers of the board.

Next, an image of the circuit pattern is transferred to the copper foil on the surface of the board with either a UV photoresist film or an ink screening process. In the UV process, another step is required to remove the resist material from areas where the circuit will be. Incomplete removal of this resist material can cause solderability problems later. Resist residue on the copper will not allow solder to bond to the pad.

Then 0.0000150 - 0.000020 inches of copper is chemically deposited (called electroless plating) in the drilled holes. This plating provides a base on which more copper can be electrically plated. *(See Figure 2-2.)*

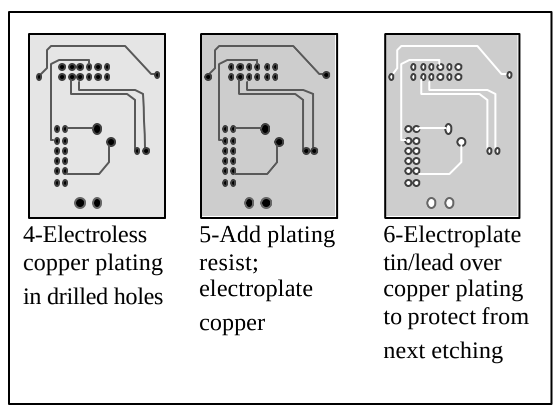
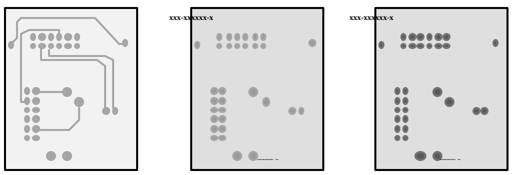


Figure 2-2. The next three steps in producing a PCB.

Then, a 0.0010 - 0.0020 inch layer of electroplated copper is added to the chemically deposited layer. Tin/lead plating---not solder, because the metals are still discrete---covers the copper to protect it from oxidation and the subsequent etching solution step.

Now it is necessary to etch off the plating resist which again exposes the copper foil on the board surface that remains from the very first step of the process. Then the tin/lead can be etched off the circuit traces and pads.



7-Etch off resist material; then etch tin/lead from circuits & pads

8-Apply solder mask to protect board, and legend ink with part no. & other information

9-Apply solder or organic coating to pads (HASL, fuser, or chemical bath)

Figure 2-3. The finishing steps PCB fabrication.

The Protective Process

When the electroplated tin/lead (Figure 2-2) has been removed, the board is electrically complete. Solder mask is applied to seal out contaminants.

Applying Solder Mask; Hot Air Surface Leveling (HASL) or Organic Coating

Solder mask can be dry film or liquid. Both are photoimageable and require a very clean environment. After mask is applied over the entire board surface, UV-blocking artwork is placed over the board. The artwork blocks UV radiation from those places where solder mask must be removed so that the pads can be exposed for component placement during assembly.

UV radiation is then applied to the board. The mask that is protected by the UV-blocking artwork remains soft. The mask not protected by the artwork is exposed to UV, which begins polymerization of the mask. Polymerization is the linking of the polymer molecules so the mask becomes hard enough to protect the board surface from scratches and contaminants.

The mask that was protected by UV radiation is still soft, and is removed by high pressure water spray in a machine called a developer. After this, the board is heat cured to harden the mask to its final form.

Then the pads must be coated for a last time to prevent oxidation. For about a decade this usually was done with a hot air leveler, which dips the board into a solder pot for 3-10 seconds.

As the board comes out of the solder, air knives blast it on both sides, leveling the solder on the pads.

This, however, can cause problems during assembly:

- Bad air knives leave an uneven pad surface so components cannot lay flat.
- The solder in the hot air leveler is contaminated by a tiny amount of copper from each board. The solder left on the pads thus becomes less solderable.
- Too much time in the solder pot creates a thick layer of intermetallic alloy, which weakens joints made in assembly.

To overcome these issues, some board fabricators are applying an organic compound that protects the pads, but without the thermal and mechanical stress that happens during hot air leveling.

Solderability Testing

PCB fabrication can be a source of assembly problems, particularly with regards to oxidation and misapplication of solder mask. Engineers and machine operators who understand fabrication learn to make careful inspection of the raw board as the first step in troubleshooting SMT solderability problems.

Several solderability test methods are commonly practiced. Regardless of the test method you use, the process engineer should evaluate the test's level of complexity, the time required and cost compared to the negative impact of the solderability problem. Here are some quick and easy test approaches that have proven useful for basic troubleshooting.

A Simple and Fast Test for Determining Pad Solderability on Bare Boards

If solderability is in doubt, a simple test is to stencil solder paste on a bare board and then reflow it without components. If the joints *look* good, but there is still doubt, have a testing lab do an X-ray to determine copper content of the solder or other potential causes of poor solderability. If the pads exhibit proper adhesion, then check the electronic components themselves. The following tests measure solderability of both component leads.

The Dip and Look Test for Component Leads

The dip and look test is similar in its ease and simplicity to the pad test. It requires a few pieces of equipment and some work space, but can be done quickly without interrupting production.

If you have established that solderability of the pads is good, take a board that has been through the screen printer, but has no components on it, to use for this test. Heat the board on a hot plate, or by whatever means are available that can be controlled.

With a thermocouple attached to one of the leads, use a tweezer to place the surface mount component on its pad. Some kapton tape placed over the component to hold it in place helps secure the part.

Heat the assembly through the thermal cycle *(See Section 3 – Reflow Profiling)* appropriate for the solder paste, using the thermocouple as a guide. Let the assembly cool, remove flux residue if necessary, and then inspect the lead for wetting.

Notes:

Wetting Balance: A More Involved Component Lead Test

Wetting balance is good test because its accuracy relies on instruments rather than the judgment of the person doing the test. The drawback is that those instruments are often not readily available and they require training for proper use. In addition, this test requires a comparison against a known specification established on a component with proven solderable leads.

To perform the wetting balance test, attach a lead with a clamp to the arm of a force sensor. A solder pot with molten solder is raised so that the lead goes at least 1/8” (3mm) into the solder.

Leave the lead in the solder for 5 seconds. Remove the lead. The force pushing against the lead should be reduced as the flux activates and cleans the surface of the lead if the flux is working and the lead is not too corroded for it to work. This reduction of force will be recorded on the force sensor, and is compared with force results from the lead on the good solderability component.

Notes: