DVD-35C

Solder Paste Printing Defect Analysis and Prevention

Below is a copy of the narration for DVD-35C. The contents for this script were developed by a review group of industry experts and were based on the best available knowledge at the time of development. The narration may be helpful for translation and technical reference.

Section 1

Surface Mount Technology is now the most widely used technology for printed circuit board design and manufacturing. Surface mount offers several economic, quality and performance advantages that have established it as the dominant technology. These include board size and weight reduction, component density and cost, higher speed assembly, and faster operating speeds. The materials, manufacturing equipment and technology methods have matured to the point where repeatable and stable surface mount assembly processes can be implemented and maintained.

The first step of the surface mount process is to apply solder paste to the specific areas of the circuit board where the surface mount components will be attached. Solder paste printing is commonly referred to as stencil, or screen printing. The solder paste is pressed through openings in the stencil or screen onto the lands of the circuit board. This is done by the use of squeegees.

Once solder paste has been applied to the lands, the board is moved to one or more pick and place machines or other automated component placement devices.

The final surface mount operation is reflow soldering. Reflow soldering is a process of joining metallic surfaces by melting the solder in the solder paste as well as any soldering coatings. The heat causes the solder to melt, and since the solder may have been melted before, the term reflow describes the action taking place. Solder paste printing provides the majority of the opportunities for defects in surface mount assembly because every component land is an opportunity for a defect. In many surface mount operations, the opportunity for a defect can be defined as one for each component lead that's soldered plus an additional one for the component itself.

For example, an 84 pin PLCC has 85 opportunities for a defect -- one for each of the 84 solder joints and one for the component itself. The performance of the printing operation has a significant impact on the quality level of the entire surface mount process, as well as on the quality and reliability of the final product that is delivered to the customer.

The availability of sophisticated automated printing equipment, advancements in solder paste, stencil fabrication and improved circuit board fabrication processes can drive the defect rates of the printing process to three to five parts per million levels.

Another factor in the lower defect rates has been the implementation of statistical studies and designed experiments in characterizing and optimizing the assembly process. At the same time that automated stencil printing equipment, and other process and material improvements have become available, the introduction of ultra-fine pitch components has increased the difficulty of successfully printing solder paste.

Surface mount manufacturing operations are now being asked to produce product with 16 and 12 mil pitch components in volume and at the same quality levels as products containing non fine pitch, or components with leads more widely spaced. To successfully maintain the required quality level, the cause for the defects that do occur must be understood so that corrective action can be initiated as soon as possible.

This videotape will define the specific defects associated with solder paste printing, discuss the possible causes and offer some recommended solutions. The first step is to define what is a "good" solder joint. This can be done with laboratory studies and/or using an established corporate/industry standard such as the IPC.

For example, J standard 001, the standard for J-lead solder joints states: "Joints formed to leads having a J shape at the joint site must meet the following dimensional and fillet requirements. For more information on the details of all surface mount solder joints, refer to the IPC videotape series on Surface Mount Solder Joint Evaluation.

The definition of a good solder joint will determine the volume of solder paste that must be deposited on a particular component land. This required solder volume then determines the stencil's thickness and the aperture size for each specific land. The relationship between the stencil thickness and aperture size is called the aspect ratio.

Other factors that influence the final solder joint are the viscosity of the solder paste, the circuit board finish, component leads, the component land geometry and the performance of the entire screen printing process.

In surface mount technology, the solder paste printing process contains the greatest number of variables. This includes the solder paste, stencils, circuit boards, vision systems, squeegees and environmental controls, among others.

Therefore, the printing process must be optimized using statistical studies and designed experiments. Process development based on trial and error, or "guess work" can result in recurring defects. Use of designed experiments is advised. Once the necessary process development work has been completed, the target process parameters can be determined and implemented.

In production, it is important to have a statistical process control program in place to warn the operator of "out of control" process parameters before defective solder joints are produced. The process can then be stopped and corrective action implemented. This type of program will keep defects at an absolute minimum.

Even with the best equipment, materials and process development, defects will occur. When a defect does occur, it is important to understand the cause so that corrective action can be implemented as soon as possible.

It should be remembered that the entire surface mount process including screen printing, automated component placement, manual component placement, reflow soldering and cleaning can all contribute to defects.

On occasion, a defect that is attributed to the solder paste printing process may be caused by either the component being placed incorrectly, the product being handled improperly, and an incorrect reflow oven profile, among others. IPC has an SMT Process Guideline and Checklist for all surface mount processes. Each section contains a list of problems often observed during a specific part of the surface mount assembly process. For example, section 5.0 deals exclusively with solder paste application, providing detailed checklists for potential problems and corrective actions.

During the remainder of this program we'll look at some common solder paste print defects, their possible causes, and suggested remedies. For more details, refer to section 5 of the IPC document.

Section 2

The common solder paste printing defects involve solder paste coverage, bridging, alignment, solder paste volume and the shape of the deposit. Maintenance of stencil cleanliness and the paste properties at the time of stencilling are critical for good paste deposits.

Before we continue, let's review some of the important considerations of solder paste. Solder paste consists of solder particles in flux with additives to control viscosity. Solder paste ages and changes with temperature. Never mix old and new pastes since the fluxes and thinners will evaporate at different rates. Finally, the viscosity of solder paste changes with both the amount of kneading, or working and the speed of the squeegee.

The first thing to examine when the printed substrate exits the screen printer is coverage. Lack of coverage means that lands are mostly or totally devoid of solder paste. If more than one percent of the lands lack coverage, especially in one area, then there may be a solder paste printing or set-up problem. If less than one percent of the lands lack coverage, **and** these lands are at the edges of devices that are perpendicular to the squeegee, it may also indicate a printing problem.

If the solder paste has been sitting for about ten minutes, you should increase the amount of paste on the stencil. The specific time is a function of the paste being used. Check to see that the paste is rolling during printing if it has been sitting.

This is an example of a good paste roll, compared to what the roll looks like when the paste is dried out. If the paste has been sitting for more than about ten minutes, then the paste should be kneaded.

If less than one percent of the lands lack coverage, **and** these lands are at the center of the devices, then there are probably clogged apertures on the stencil. The stencil can be cleaned by either manually wiping it with a lint-free cloth and an appropriate solvent -- depending on the type of paste, or by activating the stencil wiper on automatic printers which wipes and vacuums the underside of the stencil.

The next printing defect we'll discuss is bridging. Bridging occurs when there is solder paste connecting one land to another. This can cause electrical short circuits in the finished product. Bridging is typically caused by poor gasketing between the stencil and the board due to substrate quality or slumping of solder paste. Also, good contact between the stencil and board at the time of the deposit can aid in preventing bridging. If bridging occurs, here are some solutions depending on the symptom. If bridges are present on less than five percent of the lands, you should wipe the stencil bottom clean.

The stencil is also wiped for the remaining bridging conditions, along with following the specific remedies. If bridges are present on more than five percent of the lands **and** lands are parallel to the squeegee blade, decrease the squeegee downstop and go to off contact printing. The "snap-off" distance in off contact printing allows the stencil to release from the board before the squeegee raises.

If bridges are present on more than five percent of the lands **and** the lands are both parallel and perpendicular to the squeegee blade **and** no previous action has been taken, try the following remedies.

First, try decreasing the squeegee pressure. If this doesn't work, try increasing the squeegee pressure while decreasing the squeegee speed. If decreasing squeegee speed doesn't solve the problem, increase the squeegee speed and decrease the downstop. The next thing to try is increasing the downstop and decreasing the snap-off distance.

If nothing has worked so far and the condition persists, increase the snap-off distance and check the circuit board for quality of hot air leveling, surface finish and legends. Then try reducing the stencil aperture size.

The next type of defect we'll be examining is misalignment. Misalignment occurs when the stencil apertures don't perfectly align to the position of the land patterns on the circuit board.
In most printers, fiducials are used to align the stencil to the board. Good use of fiducials can help improve process yields, particularly with high lead count fine pitch components.

If an alignment error is detected when using an automatic stencil printer **and** the fine pitch mode is disabled, try enabling the fine pitch mode for two print cycles. The fine pitch mode verifies the position of the stencil and the board each time prior to printing.

If an alignment error is detected **and** the fine pitch mode is enabled, you will need to add X, Y or Theta offsets as required with magnitude equal and opposite to that of the misalignment.

Section 3

This section will examine defects dealing with solder volume and its relationship to the shape of the printed solder paste. Here is an example of a crisp, uniform print. As you can see, the shapes of these prints are not uniform.

This shape is called dog ear and may relate to excess solder volume. If this is the case, increase the squeegee pressure. The following remedies should be tried if the solder volume is nominal and the shape is mostly Dog Ear. If the snap-off distance is less than ten thousandths of an inch, you should increase snap-off distance to 20 thousandths of an inch, then subsequently by increments of five thousandths of an inch to a maximum of thirty five thousandths of an inch if necessary. If snap-off distance is at maximum, go back to the original setting and increase squeegee speed and pressure.

If squeegee speed and pressure are at maximum, then restore original speed and pressure and check for paste residue in corners or ends of stencil apertures, along with checking the viscosity of the solder paste. If the original snap-off distance is greater than thirty thousandths of an inch, then go to zero snap-off, or contact printing -- and turn on slow release.

If there is contact printing and the release speed is fast, try decreasing the slow snap-off distance by fifteen thousandths of an inch. Advanced stencil fabrication such as laser cutting or additive processes can assist in eliminating the problem.

Temperature and humidity can also be a factor in improper printing. Make sure you follow the recommendations of your solder paste supplier.

If the prints are still unsatisfactory, restore original squeegee speed and pressure and check for paste in the corners or ends of the stencil apertures, and for the viscosity of the solder paste.

This shape is called Pump and relates to having excessive solder volume. When this condition exists you should increase the viscosity of the solder paste and/or decrease squeegee pressure.

This shape is called scoop and relates to an insufficient volume of solder. When this condition exists, try decreasing squeegee pressure and/or increasing squeegee speed. It is also possible that the squeegee blade is too soft. Try a higher durometer squeegee -- or change to a metal blade.

This shape is called slope and also relates to an insufficient solder volume. When this condition exists, try increasing the paste deposit or decreasing the viscosity. You can also decrease squeegee speed and/or increase squeegee pressure.

Another defect symptom relating to shape is the edge definition of the print. If you are doing off-contact printing, remedies include wiping the stencil, increasing snap-off incrementally, not to exceed 35 thousandths of an inch, and going to contact printing.

If you are doing on-contact printing, remedies include wiping the stencil, decreasing snap-off speed and increasing snap-off distance, or increasing snap-off speed and decreasing snap-off distance, and going to off-contact printing.

Section 4

In this final section, we'll examine each of the remedies we've discussed, and summarize some general rules for applying the corrections. When cleaning the stencil bottom, wipe with a lint-free cloth using an approved solvent if necessary and vacuum. Do not used compressed air as this causes airborne lead particles.

For kneading the solder paste, activate the knead cycle, or knead manually with a spatula. If you need to increase the amount of paste on the stencil and the last paste dispense was more than 20 print cycles ago, then simply dispense the paste. If you need to increase the paste deposit and the last paste deposit was less than 20 print cycles ago, then temporarily increase squeegee pressure by 10% until three acceptable boards have been produced.

If you need to change the squeegee pressure, then vary the pressure in increments of ten percent from its current setting. To increase or decrease squeegee speed, then raise or lower the speed in increments of two tenths of an inch per second.

If you are decreasing the downstop and the downstop is greater than or equal to seventy thousandths of an inch, then reduce downstop by ten thousandths of an inch. In decreasing the snap-off, reduce it by ten thousandths of an inch.

This program has examined the various defects related to the printing of solder paste. First, we took a look at surface mount technology -- in terms of process sophistication and evolution as the dominant technology for circuit board design and manufacturing. And we saw how many of the defects in the surface mount operation are solder paste printing defects. Then we discussed common areas of printing defects -- solder paste coverage, bridging, misalignment, the shape of the deposit and the solder paste volume -- and the recommended solutions.

It's important that you learn how to identify these defects, and to understand the ways of modifying the process to maintain quality prints.

To a large extent, the quality of the solder paste printing operation will depend on your attention to detail, and your ability to correctly recognize and analyze defects. And if there's ever anything that puzzles you, be sure to ask questions.