DVD-34C
Solder Paste Printing

Below is a copy of the narration for DVD-34C. 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

The art of screen printing originated in China nearly two thousand years ago. Over the centuries it has adapted to new requirements and applications -- most recently in the electronics industry.

Today, with the growth of surface mount technology, the printing of solder paste is a critical operation that provides continuous challenges to equipment development, material manufacturing and operator skills.

In surface mount technology, components are attached to the surface of a printed circuit board, rather than having their leads inserted through mounting holes. Solder paste printing is the first step in the surface mount assembly process.

Solder paste is applied to the specific areas of the circuit board where surface mount components will be attached. These metallized land areas on the circuit board are called pads, or lands.

Solder paste printing utilizes a stencil. The solder paste is pressed into openings in the stencil onto the lands of the circuit board. The stencil apertures, or openings are designed to optimize the amount of solder paste that is deposited on the land patterns.

There are many different types of printers. They all fall into two basic categories -- automatic and semi-automatic. Fully automatic printers are able to run with minimal operator intervention once they've been initialized. Semi-automatic machines require full operator interaction during production. This includes loading and aligning circuit boards, setting parameters, applying solder paste, inspecting the printed boards, unloading the boards, and periodically cleaning the stencil.

Both types of machines require the operator to replenish consumables such as solder paste, wiper paper and solvent. We'll come back and describe the operation of each of these types of machines later.

At this point, let's discuss the key elements of solder paste printing in some detail. These elements include the circuit board, the stencil, the solder paste and the squeegee.

Printed circuit boards may vary in material, shape, size, thickness and number of land sites depending on the specific application. A circuit board usually consists of fiberglass cloth, epoxy resin and various metals. The different types of surface finishes are usually bare copper, with or without a protective coating; solder; flash gold; or other metal combinations. The board will most likely also have a solder mask in it.

The cost of a bare board can range from $1.00 to more than $10,000. It is very important to handle the boards by the edges since contaminants from your fingers can cause solder defects.

The quality of the surface finish is critical to the solder paste printing process. The stencil must contact and seal around the land for optimum printing. The larger the gap is between the stencil and the land -- as the squeegee travels across the stencil -- the more paste that will be deposited on the board, and the more chance of solder bridges which can cause short circuits.

Now, let's take an in-depth look at the stencil. The stencil controls the outside dimensions of the paste to be applied to the lands. A stencil is a thin sheet of metal with cutouts where the paste is to be applied. The thickness of the stencil is a critical factor in controlling the amount of paste deposited onto the lands. The thickness may range from four to twelve thousandths of an inch and is based upon the solder volume required for a reliable joint.

Stencils may be "stepped down" for fine pitch devices. This is where sections of the stencil are etched thinner so less solder paste comes through for those devices. The stencil is glued to a mesh material and stretched onto a metal frame to a specific tension. Stencils are made of brass, stainless steel, molybdenum or nickel. The openings can be cut with a laser, etched with various chemicals or electroformed.

The next key element we'll examine is solder paste. Solder paste is composed of spherical particles of tin/lead alloy, flux and alcohols. When solder paste is melted, it serves as an electrical and mechanical connection in the electronic assembly. The paste should be mixed to a uniform consistency before it is deposited on the stencil. If not properly stirred, the viscosity, or spreadability of the paste can be inconsistent. This will result in uneven coverage across the lands.

Solder paste contains lead which is hazardous to your health. It is important to avoid ingestion or contact to your eyes and skin. Always wash hands before eating and smoking, and always wear protective gloves.

The paste container label should be checked for the specific alloy, mesh size, product description of flux and expiration date or manufacture date. Solder paste has a shelf life and it should not be used after the expiration date. This is because the activity of the fluxes may have deteriorated and the solder will not form a reliable solder joint due to oxidation of the solder balls or separation of the flux. Also, different types of paste or flux should not be combined because the resulting mixture may not reflow properly.

If the paste has been stored in a refrigerator, the container should be allowed to reach room temperature before opening since the paste needs to be smooth. This usually takes three to four hours. It is important to log the time and date the paste was removed from the refrigerator for future reference.

Another consideration of solder paste is that the quality begins to degrade rapidly on the stencil. The alcohols or volatiles evaporate. This will change the viscosity of the solder paste, and result in poor printing. The rate of decay also varies with different pastes. Other environmental factors that can affect the paste include temperature, humidity and air flow across the stencil or around the printer.

The rate, or speed of printing can also affect the print quality. It may be necessary to knead the paste if the printer has been idle for a period of time. Solder paste can either be spread onto the stencil manually with a spatula, or dispensed automatically from a cartridge.

The last element to consider in the solder paste printing process is the squeegee. Squeegee blades are used to press the solder paste through the stencil openings. Squeegees are made of polyurethane, polypropylene, stainless steel or composite material.

The polyurethane squeegees are color coded to signify the durometer or hardness of the material. The lower the number, the softer the material. There is no industry standard for color codes as yet.

It is advisable to check the polyurethane squeegee technical specification to confirm the hardness of the blade material. If a blade is too soft it won't properly level the top of paste across the stencil opening as the soft squeegee will expand into the stencil opening removing the solder paste. This effect is called "scooping."

Now let's take a look at what's involved in the mechanical set-up of the printer. This set-up applies to both automatic and semi-automatic printers.

To set up the printer for production runs you will need the correct circuit boards, stencil, solder paste, squeegees and board supports. These will be specified in the documentation for the specific board to be run.

The stencil may need to be installed on the Universal Adapter using the hardware provided. Make sure that the image is centered and the shortest total pattern is from front to back. The Universal Adapter is then mounted into the printer and clamped in place. Some stencils may be installed without an adapter.

The squeegee length should extend about 1/2 inch beyond the stencil pattern on both sides. The blades should be leveled in the squeegee holder using a flat surface such as a light table or a granite block. When the squeegee is level in the holder, it's ready to mount onto the print head.

Tooling pins are based on the size of the circuit board locating holes. If required, they are installed in tooling pin holders and are mounted in the worktable or tooling pin assembly. The mounting is such that the circuit board is now centered and the pattern corresponds to the previously mounted stencil.

If the printer is equipped with a snugger system, mechanical adjustment of the snugger pins must be accomplished. A snugger system provides additional hold down for the circuit board.
Also, sufficient board supports should be installed in specific areas to restrict any up and down movement of the circuit board during the printing process.

Since one pneumatic snugger pressure won't work for every board because of different board thicknesses, a final adjustment of air pressure is done to secure the board without deforming it.

Section 2

Now, let's take a detailed look at the stencil printing process for an automatic printer. The first step in the process is to enable power to the printer. During an automatic printer start-up sequence, a reset is performed which moves all the machine components to a safe park position prior to performing operations.

Once the machine has finished its initialization, the operator uses the software to select a previously programmed board profile for printing. The board profile has been tested to provide optimum values for the many options and variables of the automatic printer. These include how fast the squeegee will travel, how much force the programmable squeegee applies during the print stroke, when and how much solder paste will be applied and how often the underside of the stencil will be cleaned.

If the board profile hasn't been entered into the system, the values of each parameter will be specified in your documentation. You can now enter these values into the computer. The correct selection of the variables is critical to the quality of printing solder paste on the board. Studies have estimated that 64% of the defects in a finished surface mount assembly are directly attributable to the solder paste printing process.

When the proper computer file is selected, the printer automatically positions the stencil to suit the values in the board parameters file.

The operator checks the software to find information needed to finish preparing the machine for printing. This includes length, hardness and attack angle of the squeegees to be used, and the pressure of the solder paste dispenser.

Once the operator has installed the squeegees, completed all adjustments and loaded the proper stencil, the machine is ready to begin a print cycle.

Automatic machines are designed to be placed in production lines, utilizing transportation tracks to carry the boards from one machine to the next. An optical sensor on the input side of the machine senses when a board is placed on the transportation tracks. If a SMEMA interface is being used, the transportation tracks automatically send the board to the printer. SMEMA establishes a standard for the electrical and mechanical interfacing of various equipment.

The board can also be manually placed on the tracks by the operator who then presses a cycle start button to initiate the print cycle.

Once the board has reached the approximate location of its printing position, a second optical sensor stops the transport tracks. The workholder then raises to place the board on its vacuum chambers and board supports.

The automatic printer contains a set of universal vacuum chambers which are set up to create a worknest for the circuit board. Board supports are strategically placed within the worknest to provide a stable base for the board during the print cycle.

An optional type of workholder, called a dedicated workholder, can be used in place of a universal workholder when customization for certain board types is required. A dedicated workholder can be used to work around areas of a board that present problems to the print cycle. For example, large cutouts or holes in the board can make vacuum hold down ineffective. These workholders also offer the advantage of printing double-sided boards, as well as the fastest possible changeover time from board to board.

Once the board is placed on the workholder, the vacuum system engages to hold the board down in position on the workholder.

If additional holddown is required during the printing process, or if there is a requirement to print the board with no vacuum at all, a snugger system can be used. Snuggers firmly lock the board into position to keep it from shifting while raising it to meet the stencil. They also keep it in place during the squeegee stroke.

When the board is received inside the screen printer, the workholder raises to a specific height for alignment correction. The printer aligns the board position relative to the stencil openings using an electronic vision system.

The camera will search for previously trained board targets, which are called fiducials. Once the fiducials are found, the system performs a series of calculations, and makes the necessary movements required to position the stencil over the board in the best possible location for printing.

Once the board has been positioned, the workholder raises the board up to the stencil for printing. The distance between the board and the stencil is called the snap-off distance.

If no snap-off distance is used, the board raises until it contacts the stencil. If snap-off is required, the workholder stops at a pre-set distance from the stencil.

The squeegee head now moves into position in front of the print area on the stencil, and then lowers to contact the stencil at preset pressure and downstop settings. If the machine is equipped with a single squeegee or dual tilt squeegee head, the squeegee lowers in accordance with the setting assigned by the operator during the machine set-up stage. The squeegee then strokes across the stencil, applying solder paste into the stencil openings onto the lands of the circuit board underneath.

At the end of the print stroke, the squeegee automatically raises from the surface of the stencil. If snap-off distance is used, the stencil is allowed to release from the board before the squeegee raises.

The workholder then lowers, stopping at a height to allow the vision system to perform its two and three dimensional process verifications. During the verification process, lands that have been previously selected as targets are viewed to evaluate the performance of the printing process. During the two dimensional verification process, the lands are viewed for proper coverage of solder paste on the land.

If too little solder paste has been applied, a proper solder connection may not be achieved. If too much solder paste is applied, the paste may slump, or bridge over onto the adjacent lands. Bridging causes electrical short circuits in the finished product.

The three dimensional verification process checks for solder paste deposit height. Deposits under the desired height range are singled out as defects.

Upon completion of the verification process, the results are displayed to the operator who then has the option of accepting or rejecting the board. If the print quality does not fall within process requirements, the operator may remove the board from the remainder of the production process, saving the time and expense of rework.

The workholder then completely lowers, returning the circuit board to the transportation tracks which moves the board to the output side of the machine. While the transport tracks are carrying the board out of the printer, the paste dispenser automatically lays down a new bead of paste.

The paste dispenser cycles at preset intervals to provide a constant bead of paste on the stencil in preparation for the next print stroke. It provides total control over the application of solder paste onto the stencil.

This is an important consideration since too little paste produces lands with incomplete prints, and too much paste allows excess paste to dry which can create clogging in the stencil openings.

To eliminate paste clogging around the stencil openings, a stencil wiper is primed with solvent to prepare the wiper for a pass across the underside of the stencil. The wiper then moves to the underside of the stencil print area and wipes the bottom of the stencil to remove any solder paste deposits.

Regular cleaning of the underside of the stencil prevents the spread of built up paste around the stencil openings to the open areas between the printed lands, decreasing the threat of solder bridging and subsequent electrical shorts. At the completion of the wiper's pass, the wiper moves back to its park location in preparation for the next print cycle.

The last step in the process is when a request for a board from a downstream component placement machine is received by the automatic printer. The transport tracks then resume motion to deliver the board to the tracks for the downstream machine. When the board has left the printer's tracks, an electronic request for the next board to be input is sent and the entire process repeats itself.

Section 3

Now let's take a look at the operation of semi-automatic printers. Many of the board profile values that were set by the computer, or entered into the computer on automatic printers will have to be set manually on semi-automatic machines.

These may include the rail sizes on the transport tracks; squeegee pressure, stroke length, stroke speed and downstop; and snap-off distance. In addition, manual alignment is often performed, as well as spreading the paste with a spatula, inspecting the board and cleaning the stencil.

Squeegee pressure is manually set using the pressure adjust knob. This type of adjustment is often done by putting paper underneath and adjusting the squeegee so there is even pressure across the paper.

Squeegee stroke length is manually set by adjustable stops. Stroke length is normally 1/2 inch greater than the pattern on the stencil -- on both front and back ends.

Squeegee stroke speed is manually set by adjusting flow regulators. Speed of the stroke is normally the same from front to back and back to front.

Downstop sets a maximum downward point of squeegee travel on semi-automatic printers. This is set by turning the downstop adjustment knob until the indication matches the setting specified in the documentation.

Snap-off distance, when required, is set manually by a snap-off adjustment knob.

Although many printers align the circuit board to the stencil with a vision system, some semi-automatic printers require manual alignment. This is accomplished by placing a mylar carrier onto supports. The mylar film on the carrier should entirely cover the circuit board to be printed. A printing cycle is now performed onto the mylar film.

Alignment is accomplished by manually adjusting knobs to bring the circuit board lands underneath the mylar into alignment with the solder paste deposits on top of the mylar. After proper alignment is achieved, all adjustments are locked into place and the mylar carrier is removed.

At this point the solder paste can be stirred. The paste is then spread onto the stencil with a spatula. The thickness of the paste is partially dependent on the size board being processed and the properties of the paste. There should be a roll of paste approximately one half to three quarters of an inch in diameter ahead of the squeegee during application.

The printing process may now occur. Let's watch this cycle.
After the board has been printed, it should be visually checked for well-defined prints. It is then taken to a laser measurement system so the paste deposit can be checked for correct thickness. Many of these systems check paste on each corner of the board for uniform thickness, and in other critical locations as desired.

The stencil should be manually cleaned by wiping it with a lint free cloth and an appropriate solvent or water -- depending on the type of paste. The frequency of cleaning will depend on the board and the solder paste being used. If the stencil is not regularly cleaned, the apertures will clog and solder paste won't go through, or the print won't be well defined.

The last topic we'll be discussing comes under the heading of general operator maintenance. The biggest concern in this area is the safety of the operator. When working on equipment care should be taken to wear proper safety equipment such as safety glasses and rubber gloves. This is due to the hazardous nature of solder paste and cleaning solvents.

In terms of materials, operators have the responsibility to remove all unconsumed material when the job is completed. This material should either be restocked or disposed of properly based upon local, state and federal regulations. Cleaning materials, rags, etc. should be disposed of properly as well.

Unused circuit boards should be free of debris. If boards have been improperly printed, they should be cleaned and restocked.

Also, at the completion of the job, all specific tooling must be checked for wear or damage, then cleaned or replaced, and returned to proper storage for future use. Some examples include checking squeegee blade material for edge nicks after the paste has been fully removed; checking the stencil for dents in the pattern area; and checking tooling pins or board supports for accumulated buildup of debris or solder paste.

Finally, all operator accessible areas on the printer should be visually inspected. Any visible malfunction such as frayed wire or worn belts should be reported to the maintenance department.

Cleaning the system should be done in accordance with the operator maintenance manual provided. Also, the printer should be checked for foreign objects, such as circuit boards, tools or debris that may have fallen into the machine during the job run. Then all covers and safety interlocks must be inspected to ensure the safe and efficient processing of the next job.

This program has taken a detailed look at the solder paste printing process, the most critical step in the surface mount assembly sequence. We discussed the key elements of solder paste printing; explained how to set up the machine; described the operation of automatic printers; explained the operation of semi-automatic printers and how to manually set parameters; and discussed the operator maintenance procedures.

It's important that you understand all the elements of solder paste printing. This will allow you to do the best job possible. A single error can result in costly rework, and can slow down production schedules. Your company's ability to compete depends on each employee doing the best possible job.