
DVD-33C

Introduction to Surface Mount Assembly

Below is a copy of the narration for DVD-33C. 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

NARRATOR

This video will introduce you to the surface mount assembly method for connecting electronic components to printed circuit boards.

NEWSREEL VOICE

Since the birth of the transistor there has been a continuous evolution in electronic component technology -- and package style.
The wires or leads that extend out of the component package have consistently increased in number - and changed in size, shape and orientation.

Regardless of the style or size of the leads, the mounting and interconnecting of the components onto a circuit board has typically been performed by inserting the component leads into holes in the circuit board - and then applying solder - to create the interconnection.

The soldering operation could be performed manually with a soldering iron – or by passing the entire assembly over a wave of molten solder.

During the 1970s, a different style of component began to emerge. Some of these components still had leads extending out from the package -- but the leads were designed to be attached to the surface of the board, rather than inserted through the mounting holes. Other versions of these new component styles had metallized terminations instead of the leads – again intended for interconnection to the surface of the board.

These surface mounted components offered several advantages over through hole components. First, since the leads or terminations didn't require mounting holes, the opposite side of the board was now freed up for other uses -- such as additional conductors or components. The move away

from through hole components provided the potential for significant size and weight reduction - often as much as fifty percent for a typical multilayer board.
But even with surface mounted components there was still a need for some type of holes to make the interconnections between the individual layers of circuitry. A much smaller hole called a "via" was used to perform this function.

It is this combination of smaller components, use of both sides of the circuit board and the via holes for interconnection that creates this significant increase in circuit density.

There are other benefits of surface mounting that are less visible but equally important. For electronic circuits that are designed to operate at high signal speeds or high frequencies, the components can now be located closer together. The less distance the electronic signals have to travel from one component to the other, the faster the signals will arrive -- with less overall degradation to the "quality" of the signal.

This ability to move the signals between components more quickly is a major breakthrough. Now the circuit boards can be less of a limitation on the overall processing speed of the circuit -- and even faster operating components can now be used.

But like everything else, there are new tradeoffs and problems that must be overcome before surface mounting can replace conventional through hole technology. One of the most significant differences is the configuration of the solder joints. Since surface mounted components do not have the mechanical advantage of a mounting hole, a clinched or bent lead, and both sides of the board to secure the connection, there can be a major difference in the overall strength of the component attachment.

Component packaging is another big difference between surface mount components and their through-hole counterparts. Passive chip components such as resistors and capacitors have plated component ends for terminations rather than wires. This allows them to be much smaller and placed closer together on the board than through-hole components. It also allows them to be placed on the bottom side of a through-hole board that has active components on the other side.

Active surface mount components such as integrated circuits, or ICs can come in different package styles and lead configurations.
Inside the package, the IC - or chip - is bonded to the surface mount package using wires smaller than the diameter of a human hair.

Plastic encapsulated surface mount packages utilize two different types of external leads to attach to the board -- gull wings and J-leads. There are also ceramic leadless packages.

The gull wing lead resembles the shape of a seagull's wing in that it bends down and out. Packages utilizing gull wings include quad flat packs, or QFPs which house microprocessors that function as the brains of a computer; small outline integrated circuits, or SOICs which have two sides; and small outline transistors, or SOTs that contain three gull wing leads. Gull wings are often quite fragile and need to be handled very carefully to prevent damage. The SOT leads are generally stiff.

J-leads resemble the letter J and are bent underneath the package. Packages utilizing J-leads include plastic leaded chip carriers, or PLCCs and small outline J-leads, or SOJs.

The leadless packages are called Leadless Ceramic Chip Carriers, or LCCCs. LCCCs have groove shaped terminations rather than leads. These terminations are known as castellations. The castellations provide shorter signal paths which allow higher operating frequencies.

It is important that you be able to identify the different types of surface mount components. Some surface mount components utilize fine pitch technology, meaning the spacing from the center of one lead to the next is less than 50 thousandths of an inch, or 50 mils. The current standard is 25 mils. In contrast, through-hole lead centers are spaced at 100 mils. A challenge in surface mount assembly is to accurately place a fine pitch package on a circuit board and make good solder connections.

Section 2

At this point, let's take a look at the surface mount assembly process. There are three basic parts to this operation -- solder paste application, placement of components and reflow soldering.

The first step of the process is to apply solder paste to the specific areas of the circuit board where the surface mount components will be attached. These metallized areas are called lands. Solder paste application is a printing process that utilizes a screen or stencil. The solder paste is pressed through openings in the screen or stencil to the lands on the circuit board.

The stencil openings should match the precise locations of these land patterns. The stencil is usually made out of nickel plated metal or stainless steel. The land patterns in the stencil are often cut with a laser. Other stencils are more flexible and their patterns are created using a chemical etching process. Screens function in the same manner as stencils but will have an open wire mesh through which the solder paste is pressed. The type of stencil or screen used is based on the accuracy of the printer.

Stencils may be "stepped down" for fine pitch parts. Sections of the stencil are etched thinner where the fine pitch patterns are located so less solder will come through during the printing process. The stencil fits into a frame which attaches to the screen printer. It must be accurately adjusted to be in precise alignment with the circuit board.

Solder paste is now pressed through the stencil onto the board - with a squeegee. The squeegee may be made of hard rubber or stainless steel. There are a number of variables in the solder paste application process. It's important that these variables be understood so that accurate prints are made.

Now let's take a closer look at the solder paste application process. Your company will have documentation that specifies the board to be assembled; the vacuum plate, or board fixture which holds the board during printing; the stencil; solder paste; the depth of the paste and other process parameters; as well as the computer program for the screen printer. Also included will be a list of the electronic components and their quantities.

If you happen to have a computer controlled system, the first step is to load the correct computer program. Otherwise, use the normal mechanical type set-up. The computer program allows the operator to interact with the screen printer and to control the machine operation.

Next, the stencil is loaded. The correct orientation of the stencil will be specified in the documentation. At this point, the rails on the conveyor are set. The rails hold the board on the conveyor during the printing operation and are set up for the size of the board. The vacuum plate, or board fixture, sits underneath the stencil. The purpose of the vacuum plate is to support the board during the printing operation. If there are parts on the underside of the board, spacers or standoffs are placed in specified locations on the vacuum plate so that these components will be protected.

Using the computer controls, the operator now cycles the conveyor so the board now moves into the printer. The machine should be set for the board thickness so that the edges of the board meet the edges of the rails.

At this point, the vacuum plate moves the board in position underneath the stencil and aligns the board with the stencil using fiducials on the board and the stencil. The distance between the board and the stencil should be set at this time per your company's standard operating procedure. This separation is called the "snap-off" distance. Some paste application is done on contact with no snap-off.

After the solder paste has been deposited, the stencil lifts away or snaps off immediately behind the squeegee and returns to its original snap-off position. If there was no space between the stencil and the board, the stencil would smear the solder paste between the lands which would cause bridging, or short circuits.

Next, the pressure of the squeegee should be set. This is done by putting paper underneath and adjusting the squeegee so there is even pressure across the paper. On some screen printers, this adjustment is done automatically.

The solder paste, which consists of little balls of solder held together by flux and other non-metallic materials, is now stirred for about thirty seconds. There are different types of solder pastes that are appropriate for specific kinds of screen printers and stencils. Your documentation will identify the solder paste to be used. Stirring the solder paste will bring it to its proper viscosity.

The paste is then spread on the stencil with a spatula or applied from a cartridge across the stencil in front of the squeegee. The thickness of the paste is partially dependent on the size of the 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.

At this point, the tilt, or downward pressure of the squeegee is set. It is important to begin this adjustment with too little pressure rather that too much. Too much squeegee pressure can damage the stencil. You will know the squeegee is adjusted properly when there isn't a film of solder paste left on the stencil.

If there's too little pressure or the paste is not correct, the squeegee will leave a film of paste on the stencil. This can also cause solder peaks, or spikes on the board. If there's slightly too much pressure, not enough solder will be deposited on the board. Too high of a squeegee pressure may also cause "scooping" if using a soft squeegee blade.

When all the settings are correct, the stencil should be cleaned by wiping with a lint free cloth with an appropriate solvent or water depending on the type of paste. An air hose is also used to blow out any solder left in the openings. During production, the frequency of cleaning is dependent on the board and solder paste being used. If the stencil is not regularly cleaned, the holes will clog and solder paste won't go through, or the excess paste will be deposited where not desired. This can create voids, open circuits, or bridges.

At this point, a test board is run through the printer. The fiducials, or alignment marks on the stencil should be checked against the fiducials on the board. Assuming they are both in alignment, the solder paste application is performed.

Now the board is visually checked for crisp prints. The paste deposit is then checked for correct thickness. This test system can check paste on each corner of the board for uniform thickness, or in other critical locations as desired.

If the solder paste application was done correctly, all the boards in the run can now be printed. Let's watch this operation for a moment so you can understand the solder paste application process. (Pause)

Some boards may also require through-hole assembly along with surface mount components attached to the bottom side of the board. In this instance, going through a wave soldering operation may unsolder the components on the underside of the board. To prevent this, a drop of adhesive is placed between the lands so that the components are glued to the board and do not depend on the solder joint to hold them on. This is done after solder paste application and prior to component placement.

The adhesive is applied from a glue tube using air pressure. A computer controls the air pressure, the nozzle used, the amount of glue dispensed and the precise locations the drops of glue are applied.

Section 3

Once solder paste has been applied to specific areas of the circuit board as defined by the stencil, the board is moved to one or more pick and place machines for component placement. These machines may have single or multiple placement heads that can be fitted with different size nozzles for picking up and placing components of varying sizes.

Smaller components are on tape reels that are connected to metal feeders. A computer program contains the part size, shape, thickness and number of leads which are verified before the component is placed on the circuit board. This is done by taking a picture of the component and comparing it to the description stored in the computer.

Larger component packages such as QFPs and PLCCs are stacked in waffle trays. These parts are sub-robot fed to main robots that place these components. Other large components may be fed from tape reels or from component tubes.

Now, let's take a closer look at what's involved in placing these components. In the same manner as solder paste application, the pick and place machines will have specific programs that need to be loaded based upon the particular board to be run. The documentation will explain exactly how to load the programs, set the rails for the boards and how to load the components.

Each pick and place machine runs off a different program. There are internal sensors that control when one machine is done and another board is ready to place additional parts. As we mentioned, the majority of components are on tape reels of various sizes. It is important to load the tape reels on the correct size metal feeders.

The component variables have to do with the size of the component; the width, or distance between components on the reel; and the pitch, or angle of the feeder. It is important to pay close attention when changing reels because many components have identical appearances and similar part numbers. These part numbers must be checked carefully against those specified on the documentation.

The component feeders are powered either by air pressure, mechanical action, or electricity. The program controls the operation of the feeders in terms of how many times a part is picked and placed. It is also important to verify that the proper nozzles are on the placement heads.

Once the pick and place machines are properly set up and the correct tape reels are connected to the feeders, the component placement operation can begin. Machines with multiple heads can place components on two boards at a time. Also, several machines can be connected in-line so that components can be placed faster.

As the nozzle places the component, an infrared beam activates a camera that takes a picture of the component to be placed. That picture is compared to the part description stored in the computer. The computer verifies that the leads are not splayed or bent, and that the part is being placed in the correct location on the board.

If the component is the wrong size, or the leads are defective, the nozzle will drop the component on the conveyor before placing it and the machine will automatically stop. At that point, check to see that the correct parts are loaded on the right feeder.

Other problems can include the wrong size feeder for the size of the component reel, a malfunction in the feeder itself, or the component being picked up incorrectly because the wrong size nozzle is on the placement head. This is why it's important to double check every aspect of the pick and place operation so that expensive errors and machine down time don't occur.

In addition to tape reels, pick and place machines also mount larger components from waffle trays. These trays are loaded in a multi-tray unit. When the last component in a tray is placed on the board, the sub-robot tray remover nozzle removes the empty tray and drops it. Then the sub-robot component nozzle picks up the part from the next tray and transfers it to the robot head for the placement operation. Meanwhile, other large components are picked and placed from tape reels and/or component tubes.

Let's observe the operation of these pick and place machines so you get a complete understanding of how they work. It's important that the parts are accurately placed on the board before the assembly goes into the reflow soldering operation.

If components are not properly centered on the lands, they should be manually aligned with a tweezer at this time. This is very difficult with finer pitch parts and should be avoided. If this happens, engineering should be notified to make program corrections. Also, if a component is missing, it should be placed manually.

The last surface mount operation is reflow soldering. Reflow soldering is a process of joining metallic surfaces by the melting of the solder in the solder paste as well as any solder coatings by the addition of external heat. The heat causes the solder to melt or flow, and since the solder may have been melted before, the term reflow describes the action taking place.

The two most common methods for reflow soldering are infrared and forced convection. Vapor phase is also a method being used, but not as commonly.

In the first method, infrared radiation provides most of the heat to melt the solder. Some heat is also provided by convection. This system can add the correct amount of heat quickly.

In convection systems, air or nitrogen is heated and blown onto the circuit board to reflow the solder. Although most of the heat is provided by convection, a small percentage comes from infrared radiation. This combination provides uniform and quick heating to the surfaces.

Most systems today use a combination of infrared and convection to accomplish the soldering process. About 60 percent of the heat transfer comes from convection through the hot air in the oven. The remaining 40 percent of the heat transfer comes from infrared radiation or heat panels.

The first area inside the machine is a preheat zone which transfers a low amount of heat to allow for a gradual temperature rise. Preheating prevents the board and some of its components from being damaged by receiving all the required heat at one time. This preheat operation causes the flux in the solder to activate which cleans oxides from the lands. It also burns off any moisture and volatiles on the board. If these liquids don't slowly evaporate prior to soldering, they may be turned abruptly to gas at soldering temperatures and explode out through the solder. This will cause blowholes and solder balls.

As the assembly travels through the machine, the next heating zones are set at higher temperatures. These are the zones where reflow of the solder and wetting take place. The amount of heat in the reflow area and the speed of the conveyor will determine how much heat is transferred.

The last phase of the operation is a cooling down process. This is when the solder solidifies. Now the components are mechanically and electrically connected to the board.

Glue that has been applied to the board is cured using the same basic process as reflow soldering except with lower heat.

Following the reflow soldering operation, the boards are visually inspected for proper solder connections. If there are any problems, the board can be touched up at a rework station.

The assemblies then go through a wash and clean operation to clean off undesired contaminants - including any remaining flux residue. These contaminants can attract dust - which could cause short circuits and other electrical failures. If no-clean, low residue flux was used, this operation may not be required.

If surface mount components are to be placed on the secondary side of this board, the entire operation will be repeated to complete the assembly. During the secondary reflow soldering operation the lower heating panels will be set at a lower temperature. This will minimize the amount of time that the first pass solder is in a liquidous state, allowing the surface tension to hold the components onto the board. If through-hole components are to be inserted on the other side of the board, the assembly is transferred to the through-hole area.

In most companies, the final operation for the assemblies is testing. The boards are checked for electrical continuity, meaning opens and shorts, and for proper parts installed correctly. The assemblies are also given a functional electrical test to make sure all the components are working properly.

Next, the assemblies are placed in high temperature ovens and have power applied to them for a specific period of time. When they are removed they are again tested for proper operation. These tests are called "burn-in tests." Burn-in testing is done to verify the operation of the assembly in demanding operating conditions.

The completed boards can now be installed in the products they were designed for.

It's important that you understand all phases of the surface mount assembly process. This will allow you to do the best job possible. When you're not sure about something, always ask a question. A single error can result in costly rework, and can even ruin an entire assembly. Your company's ability to compete depends on each employee doing the best quality job possible.