SURFACE MOUNT TECHNOLOGY
ABSTRACT

Surface mount technology is the method of attaching both leaded and non-leaded electrical components to the surface of a conductive pattern that does not utilize leads in feed through holes. It is an innovative trend in the field of Printed Circuit. Printed circuit board designing especially those requiring high on board density. This seminar explains how the PCB circuit can be designed step by step.

It is done in a SMT also provides improved shock and vibration resistance due to the lower mass of components. The smaller lead lengths of surface mount components reduce parasitic losses and provide more effective decoupling.
# INDEX

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Page No</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>CHAPTER 1: INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>CHAPTER 2: SURFACE MOUNT TECHNOLOGY</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>2.1. Types of Surface Mount Technology</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>2.2. Fine Pitch Devices</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>2.3. Surface Mount Design</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>2.4. Solder Paste Application</td>
<td>11</td>
</tr>
<tr>
<td>3</td>
<td>CHAPTER 3: ADVANTAGES &amp; DISADVANTAGES</td>
<td>16</td>
</tr>
<tr>
<td>4</td>
<td>CHAPTER 4: CONCLUSION</td>
<td>17</td>
</tr>
<tr>
<td></td>
<td>REFERENCES</td>
<td>18</td>
</tr>
<tr>
<td></td>
<td>APPENDIX I</td>
<td></td>
</tr>
<tr>
<td></td>
<td>APPENDIX II</td>
<td></td>
</tr>
</tbody>
</table>
1 Introduction

Traditional through-hole Dual In-Line Package assemblies reached their limits in terms of improvements in cost, weight, volume, and reliability at approximately 68L. SMT allows production of more reliable assemblies with higher I/O, increased board density, and reduced weight, volume, and cost. The weight of printed board assemblies (PBAs) using SMT is reduced because surface mount components (SMCs) can weigh up to 10 times less than their conventional counterparts and occupy about one-half to one-third the space on the printed board (PB) surface. SMT also provides improved shock and vibration resistance due to the lower mass of components. The smaller lead lengths of surface mount components reduce parasitic losses and provide more effective decoupling.
2.1. Types Of Surface Mount Technology
SMT replaces DIPs with surface mount components. The assembly is soldered by reflow and/or wave soldering processes depending on the mix of surface mount and through-hole mount components. When attached to PBs, both active and passive SMCs form three major types of SMT assemblies, commonly referred to as Type I, Type II, and Type III (see Figure 1).

Figure 1. Surface Mount Technology Board Types
Type I is a full SMT board with parts on one or both sides of the board. Type II is probably the most common type of SMT board. It has a combination of through-hole components and SMT components. Often, surface mount chip components are located on the secondary side of the Printed Board (PB). Active SMCs and DIPs are then found on the primary side. Multiple soldering processes are required. Type III assemblies are similar to Type II. They also use passive chip SMCs on the secondary side, but on the primary side only DIPs are used.

**Figure 2. Typical Process Flow for Total Surface Mount (Type I SMT)**

The process sequence for Type I SMT is shown in Figure 7-3. For a single sided type I, solder paste is printed onto the board and components are placed. The assembly is reflow soldered and cleaned (if needed). For double-sided Type I, the board is turned over, and the process sequence just described is repeated.
Type II & III assemblies go through the process sequence of Type I SMT followed by the sequence for Type III. In general practice, only passive chip components and low pin count gull wing components are exposed to solder wave immersion.

2.2. Fine Pitch Devices

The need for high lead-count packages in semiconductor technology has increased with the advent of application-specific integrated circuit (ASIC) devices and increased functionality of microprocessors. As package lead count increases, devices will become larger and larger. To ensure that the area occupied by packages remains within the limits of manufacturing equipment, lead pitches have been reduced. This, coupled with the drive toward higher functional density
at the board level for enhanced performance and miniaturization, has fostered the introduction of many devices in fine-pitch surface mount packages.

A fine-pitch package can be broadly defined as any package with a lead pitch finer than the 1.27mm pitch of standard surface mount packages like PLCCs and SOPs. Most common lead pitches are .65mm and .5mm. There are even some now available in 0.4mm pitch. Devices with these fine pitches and leads on all four sides are called Quad Flat Packs, (QFPs). The assembly processes most dramatically affected by the fine-pitch package are paste printing and component placement. Fine pitch printing requires high quality solder paste and unique stencil aperture designs. Placement of any surface mount package with 25 mils or less of lead pitch must be made with the assistance of a vision system for accurate alignment.

Placement vision systems typically consist of two cameras. The top camera system scans the surface of the board and locates fiducial targets that are designed into the artwork of the board. The placement system then offsets the coordinates in the computer for any variation in true board location. The bottom camera system, located under the placement head, views the component leads. Since the leads of fine-pitch components are too fragile to support mechanical centering of the device, the vision system automatically offsets for variations in the X, Y, and theta dimensions. This system also inspects for lead integrity problems, such as bent or missing leads.

2.3. Surface Mount Design

2.3.1 Design for Manufacturability

Design for manufacturability is gaining more recognition as it becomes clear that cost reduction of printed wiring assemblies cannot be controlled by manufacturing engineers alone. Design for manufacturability—which includes considerations of land pattern, placement, soldering, cleaning, repair, and test—is essentially a yield issue. Thus, companies planning surface mount products face a challenge in creating manufacturability designs.

Of all the issues in design for manufacturability, land pattern design and interpackage spacing are the most important. Interpackage spacing controls cost-effectiveness of placement, soldering, testing, inspection, and repair. A minimum interpackage spacing is required to satisfy all these manufacturing requirements,
and the more spacing that is provided, the better. With the vast variety of components available today, it would be difficult to list or draw the space requirements for every component combination. In general, most component spacing ranges from 0.040 in. to 0.060 in. The space is typically measured from pad to pad, lead to lead, or body to body, whichever is closest. Smaller spacing (0.040 in) is generally used for low or thin profile parts and small chip components. Taller parts such as PLCCs are usually spaced at 0.060 in. The placement capability of each individual piece of equipment will partially dictate minimum requirements. However, often the ability to rework or repair individual leads, or entire parts, will have a stronger influence on the minimum spacing. Allowing enough space for rework nozzles or soldering irons can save considerable cost by allowing repair of a few bad solder joints versus scrapping the entire board. Thus, each user must set spacing requirements based on the equipment set used. Similar types of components should be aligned in the same orientation for ease of component placement, inspection, and soldering.

2.3.2 Land Pattern Design

The surface mount land patterns, also called footprints or pads, define the sites where components are to be soldered to the PC board. The design of land patterns is very critical, because it determines solder joint strength and thus the reliability of solder joints, and also impacts solder defects, leanability, testability, and repair or rework. In other words, the very producibility or success of SMT is dependent upon the land pattern design. The lack of standardization of surface mount packages has compounded the problem of standardizing the land pattern.

To simplify the land pattern design guidelines, surface mount components are divided into four different categories:
1. 0.050" Pitch J-leaded Devices
2. 0.050" Pitch Gullwing Leaded Devices
3. Sub 0.050" Pitch Gullwing Leaded Devices
4. Chip Components

0.050" Pitch J-Leaded Devices

The following dimensions will be needed:
• Nominal pitch (without tolerance)
• Maximum lead span (use tolerance)
• If several vendors’ parts are proposed for the same pattern, be sure to consider them all when extracting the above dimensions.

An overview of the land pattern design method is:

1. Set the OD (outside distance) using the max lead span.
   • Set the OD equal to the max lead span, plus 0.030", rounded UP to the nearest 0.010".

2. Derive the ID, using the standard pad for this pitch.
   • Subtracting two standard pad lengths from the OD established in (1). The standard pad for this pitch is:
     Pitch = 0.050” Pad Size = 0.025” x 0.075”

3. Set the stencil aperture size.
   • In CAD, make the stencil aperture the same as the metal pad. The stencil vendor will modify the solder paste artwork if necessary, with input from the Manufacturing or Process Engineer.

0.050” Pitch Gullwing Leaded Devices

The following dimensions will be needed:
• Nominal pitch (without tolerance)
• Maximum toe-to-toe lead span (use tolerance)
• Minimum heel-to-heel lead span (if not specified directly, can be calculated by subtracting twice the max foot length from the min toe-to-toe lead span) The following spec is desirable:
• Minimum body width

If several vendors’ parts are proposed for the same pattern, be sure to consider them all when extracting the above dimensions.

An overview of the land pattern design method is:

1. Set the OD (outside distance) using the max lead span.
   • Set the OD equal to the max toe-to-toe lead span, plus 0.020", rounded UP to the nearest 0.010”.

2. Derive the ID, using the standard pad for this pitch.
   • Subtract two standard pad lengths from the OD established in (1). The standard pad for this pitch is:
     Pitch = 0.050” Pad Size = 0.025” x 0.075”

3. Check the ID for adequate fillet.
   • The ID should be no greater than the min heel-to-heel minus 0.030". This allows for a 0.015" fillet on each side. If it passes this test, then this ID is the final ID. If it fails this test, go on to (4).
4. If adequate fillet is not achieved, decrease the ID.
• If the ID fails the test in (3), then determine which of the following is the greater:
  Min heel-to-heel minus 0.030"
  Min body width minus 0.010"
• Set the final ID to whichever is greater rounded DOWN to the next 0.010".
• Calculate the pad length as (OD-ID)/2. Use the standard pad WIDTH from the table in (2).

5. Set the stencil aperture size
• In CAD, make the stencil aperture the same as the metal pad. The stencil vendor will modify the solder paste artwork if necessary, with input from the Manufacturing or Process Engineer.

**Sub 0.050" Pitch Gullwing Leaded Devices**

On rectangular four sided parts, the steps below must be used twice, since different dimensions are required for each axis. For these parts, the same pad and stencil sizes are used on all four sides.

The following dimensions will be needed:
• Nominal Pitch (without tolerance)
• Maximum toe-to-toe lead span (use tolerance)’
• Minimum heel-to-heel lead span (if not specified directly, can be calculated by subtracting twice the max foot length from the min toe-to-toe lead span)

The following spec is desirable:
• Minimum body width

If several vendors’ parts are proposed for the same pattern, be sure to consider them all when extracting the above dimensions.

An overview of the land pattern design method is:
1. Set the ID (inside distance).
• Determine which of the following is greater:
  Min heel-to-heel minus 0.030"
  Min body width (if available) minus 0.010"
• Set the ID to whichever is greater, rounded DOWN to the next 0.010"

2. Derive the OD (outside distance), using the standard pad for this pitch.
• Add two standard pad lengths to the ID established in (1). The standard pad for each pitch is:

<table>
<thead>
<tr>
<th>Pitch Pad Size</th>
<th>1.0mm (approx .0394&quot;)</th>
<th>0.8mm (approx .0315&quot;)</th>
<th>0.65mm (approx .0256&quot;)</th>
<th>0.025&quot;</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.025x0.075</td>
<td>0.018x0.070</td>
<td>0.015x0.070</td>
<td>0.015x0.070</td>
</tr>
</tbody>
</table>

---
3. Check the OD for adequate pad extension
   • The OD should be greater than or equal to the max toe-to-toe plus 0.050". This allows for a 0.025" pad extension on each side. If it passes this test, then this OD is the final OD. Use the standard pad size from the table and skip to (5). If it fails this test, go on to (4).

4. If adequate pad extension is not achieved, increase the OD
   • If the OD fails the test in (3), then set the OD equal to the max toe-to-toe plus 0.050", rounded up to the next 0.010".
   • Calculate the pad length as (OD-ID)/2. Use the standard pad WIDTH from the table in (2).

5. Set the stencil aperture size
   • In CAD, make the stencil aperture the same as the metal pad. The stencil vendor will modify the solder paste artwork if necessary, with input from the Manufacturing Process Engineer.

**Chip Components**

The following dimensions will be needed:
   • Maximum overall component length.
   • Minimum termination-to-termination gap (if not specified directly, can be calculated by subtracting twice the max termination thickness from the minimum overall component length).
   • Maximum component height.
   • Maximum termination height (may be the same as component height).
   • Nominal termination width.

May be the component terminal width, such as 0.050" on an 0805 Component. On components where the termination is narrower than the body (such as molded tantalum capacitors), use the nominal width of the termination alone. If a nominal is not stated, split the difference between the minimum and maximum width If several vendors’ parts are proposed for the same pattern, be sure to consider them all when extracting the above dimensions.

An overview of the land pattern design method is:
1. Set the OD using the max component length.
   • For components that can be wave or reflow soldered (most components), set the OD, using:
     \[ OD = \text{Max component length} + 2 \times (\text{max termination height, or } 0.040", \text{ whichever is LESS}) + 0.010" \] [for placement tolerance]. Rounded UP to the next 0.010". This leaves plenty of room for wave soldering as well as reflow soldering.
• For components that will be reflow soldered only (such as those taller than 0.090"), set the OD, using:
OD = Max component length + 1 * (max termination height, or 0.040", whichever is GREATER) + 0.010" [for placement tolerance] Rounded UP to the next 0.010.
2. Set the ID, using the min termination-to-termination gap
• ID = Minimum termination-to-termination gap. Rounded DOWN to the next 0.010.
For parts smaller than 0805, the rounding down to the next 0.010" in the above step may result in a gap that is too small. The formula has not yet been modified to consider these small parts.
3. Determine pad length from OD and ID
• Pad length = (OD-ID)/2
4. Set pad width, using nominal termination width
• If the component has a full width termination, set the pad width equal to the nominal device width, rounded to the nearest 0.005". For example, on 0805, use 0.050"; on 1210, use 0.100".
• If the component has a termination width smaller than the component width, set the pad width equal to the nominal termination width.
5. Set the stencil aperture size
• In CAD, make the stencil aperture the same as the metal pad. The stencil vendor will modify the solder paste artwork if necessary, with input from the Manufacturing or Process Engineer

2.3.3 Design for Testability

In SMT boards, designing for testability requires that test nodes be accessible to automated test equipment (ATE). This requirement naturally has an impact on board real estate. In addition, the requirement impacts cost, which is dependent upon defects. A lower number of test nodes can be tolerated when defect rates are low, but higher defect occurrence demands adequate diagnostic capability by allowing ATE access to all test nodes. Most companies use bed-of-nails in-circuit testing for conventional assemblies. Use of SMCs does not impact testability if rules for testability of assemblies are strictly observed. These rules require that (1) 0.050-in. and 0.100-in. test probes are used, (2) solder joints are not probed, and (3) through-hole vias or test pads are used to allow electrical access to each test node during in-circuit testing. If possible, this electrical access should be provided both at top and bottom, with the bottom access being necessary. The main drawback of providing all the required test pads is that the
real estate savings offered by SMT are somewhat compromised. To retain these savings requires development of some form of self-test or reliance upon functional tests only. However, self-test requires considerable development effort and implementation time, and functional tests lack the diagnostic capability of in-circuit tests. Designing for manufacturability, test, and repair are very important for yield improvement and thus cost reduction.

2.4. Solder Paste Application

2.4.1 Solder Paste Printing

Solder paste plays an important part in reflow soldering (Type I and Type II SMT). The paste acts as an adhesive before reflow and even may help align skewed parts during soldering. It contains flux, solvent, suspending agent, and solder of the desired composition. Characteristics such as viscosity, dispensing, printing, flux activity, flow, ease of cleaning, and spread are key considerations in selecting a particular paste. Susceptibility of the paste to solder ball formation and wetting characteristics are also important selection criteria. In most cases, solder paste is applied on the solder pads before component placement by stenciling.

Stencils are etched stainless steel or brass sheets. Stencils are essentially the industry standard for applying solder paste. Screens with emulsion masks can be used but stencils provide more crisp and accurate print deposits. The types of solder paste available fall under three main categories: Rosin Mildly Activated (RMA), water-soluble Organic Acid (OA), and no-clean. Each of these has advantages and disadvantages as listed in the Table 1, and choosing one over the others depends on the application and the product type.
Table 1. COMPARISON OF SOLDER PASTES

<table>
<thead>
<tr>
<th>Type</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>RMA</td>
<td>Stable chemistry, Good properties</td>
<td>Needs chemical solvent or Saphonication for Cleaning</td>
</tr>
<tr>
<td>OA</td>
<td>Cleaned using pure water, Good cleanability</td>
<td>Humidity sensitive, Water leaches into Waste stream</td>
</tr>
<tr>
<td>NO CLEAN</td>
<td>No cleaning process</td>
<td>May leave some Invisible residue behind</td>
</tr>
</tbody>
</table>

2.5. Surface Mount Components and Their Placement

2.5.1 Component Packaging

Most active components are available in surface mount. However, connectors and sockets are still through-hole, often for strength considerations, which will keep us in mix-and-match format for some time to come. Surface mount components are available in various shipping media. The most common is tape and reel. It requires fewer machine reloads allowing more machines run time. Trays are also used, generally for large packages such as QFPs. The EIA specification RS-481A has standardized reel specifications for passive components and active components.

2.5.2 Component Placement

Requirements for accuracy make it necessary to use auto-placement machines for placing surface mount components on the PB. The type of parts to be placed and their volume dictate selection of the appropriate auto-placement machine. There are different types of auto-placement machines available on the market today: (A) in-line, (B) simultaneous, (C) sequential, and (D) sequential/simultaneous. In-line placement equipment employs a series of fixed-position placement stations. Each station places its respective component as the PB moves down the line. These machines can be very fast by ganging several in sequence. Simultaneous placement equipment places an entire array of components onto the
PC board at the same time. Sequential placement equipment typically utilizes a software-controlled X-Y moving table system. Components are individually placed on the PC board in succession. These are currently the most common high-speed machines used in the industry. Sequential/simultaneous placement equipment features a software-controlled X-Y moving table system. Components are individually placed on the PC board from multiple heads in succession. Special features such as vision capability, adhesive application, component testing, board handling, and capability for further expansion may be of interest for many applications. Vision capability is especially helpful in accurate placement of fine-pitch packages. Machine reliability, accuracy of placement, and easy maintenance are important to all users.

2.6. Soldering

Like the selection of auto-placement machines, the type of soldering process required depends upon the type of components to be soldered and whether surface mount and through-hole parts will be combined. For example, if all components are surface mount types, the reflow method will be used. However, for a combination of through-hole and surface mount components, reflow soldering for surface mount components followed by wave soldering for through-hole mount components is optimum.

2.6.1 Infrared/Convective Reflow Soldering

There are basically two types of infrared reflow processes: focused (radiant) and non-focused (convective). Focused IR, also known as Lamp IR, uses quartz lamps that produce radiant energy to heat the product. In non-focused or diffused IR, the heat energy is transferred from heaters by convection. A gradual heating of the assembly is necessary to drive off volatiles from the solder paste. This is accomplished by various top and bottom heating zones that are independently controlled. After an appropriate time in preheat, the assembly is raised to the reflow temperature for soldering and then cooled. The most widely accepted reflow is now "forced convection" reflow. It is considered more suitable for SMT packages and has become the industry standard. The advantage of forced convection reflow is better heat transfer from hot air that is constantly being replenished in large volume thus supplying more consistent heating. For wave soldering components, must be spaced sufficiently far apart avoid bridging or shadowing (inability of solder to penetrate properly into small spaces). This is less important for reflow soldering but sufficient space must be allowed to enable rework should it be required.
Soldering Precautions

The soldering process can create a thermal stress on any Semiconductor component. The melting temperature of solder is higher than the maximum rated temperature of the device. The amount of time the device is heated to a high temperature should be minimized to assure device reliability. Therefore, the following precautions should always be observed in order to minimize the thermal stress to which the devices are subjected.

1. Always preheat the device.
2. The delta temperature between the preheat and soldering should always be less than 100°C. Failure to preheat the device can result in excessive thermal stress which can damage the device.
3. The maximum temperature gradient should be less than 5°C per second when changing from preheating to soldering.
4. The peak temperature in the soldering process should be at least 30°C higher than the melting point of the solder chosen.
5. After soldering is complete, forced cooling will increase the temperature gradient and may result in latent failure due to mechanical stress.
6. During cooling, mechanical stress or shock should be avoided.

WAVE soldering

It is done using a wave soldering machine which uses the ultraviolet energy. It is important to avoid the possibility of thermal shock during soldering and carefully controlled preheat is therefore required. The rate of preheat should not exceed 4°C/second and a target figure 2°C/second is recommended. Although an 80°C to 120°C temperature differential is preferred, recent developments allow a temperature differential between the component surface and the soldering temperature of 150°C (Maximum) for capacitors of 1210 size and below with a maximum thickness of 1.25mm. The user is cautioned that the risk of thermal shock increases as chip size or temperature differential increases. Mildly activated rosin fluxes are preferred. The minimum amount of solder to give a good joint should be used. Excessive solder can lead to damage from the stresses caused by the difference in coefficients of expansion between solder, chip and substrate. AVX terminations are suitable for all wave and reflow soldering systems. If hand soldering cannot be avoided, the preferred technique is the utilization of hot air tools.
Cleaning

In general, cleaning of SMT assemblies is harder than that of conventional assemblies because of smaller gaps between surface mount components and the PB surface. The smaller gap can entrap flux, which can cause corrosion, which leads to reliability problems. Thus, the cleaning process depends upon the spacing between component leads, spacing between component and substrate, the source of flux residue, type of flux, and the soldering process. RMA cleaning requires chemicals and has waste affects to deal with. OA cleaning uses water that must flush down the drain. However in this chemistry, lead is often found in the wastewater and creates an environmental concern. No clean is generally becoming the preferred solder process since it eliminates cleaning all together. This eliminates the environmental issues and saves in capital costs. One of the key issues in SMT has been to determine the cleanliness of SMT assemblies. The Omega meter is a common tool originally used for DIP boards these boards check for ionic contaminates left on the PB by measuring the electrical resistance between adjacent traces or circuits.

Repair/Rework

Repair and rework of SMT assemblies is easier than that of conventional components. A number of tools are available for removing components, including hot-air machines for removing active surface mount components. As with any rework tool, a key issue in using hot-air machines is preventing thermal damage to the component or adjacent components. No matter which tool is used, all the controlling desoldering/soldering variables should be studied, including the number of times a component can be removed and replaced, and desoldering temperature and time. It is also helpful to preheat the board assembly to 150°F - 200°F for 15 to 20 minutes before rework to prevent thermal damage such as measling or white spots of the boards, and to avoid pressure on pads during the rework operation. To prevent moisture induced damage, SMT components may require bake-out prior to removal from the board.
3. ADVANTAGES & DISADVANTAGES

3.1 ADVANTAGES

Since the surface mounting component’s weight is ten times less than their conventional counter parts and since they occupies 3 times less than the space required by the ordinary components the size, weight and the volume of the printed board assemblies using this technology is reduced considerably, without any reduction in reliability. Also it allows improved shock and vibration resistance due to lower mass of components.

3.2 DISADVANTAGE

The only disadvantages is about the cost all the operations in this technology is carried out by innovative machines, and since this machines are of high cost, this cannot be implemented by individuals.
4. Conclusions

The major technical considerations for implementing SMT include surface mount land pattern design, PB design for manufacturability, solder paste printing, component placement, reflow soldering, wave soldering, cleaning, and repair/rework. These areas must be studied and thoroughly understood to achieve high quality, reliable surface mount products. The smaller size of SMCs and the option of mounting them on either or both sides of the PB can reduce board real estate by four times. A cost savings of 30% or better can also be realized through a reduction in material and labor costs associated with automated assembly.
References

1) www.linear.com
2) www.naist.org