1 Component Surface Mount Technology (SMT)

Intel® Manufacturing Enabling Guide

May 2010

Disclaimers

Contents

Figures

Tables

1.1 Introduction

Traditional through-hole Dual In-Line Package (DIP) assemblies reached their limits in terms of improvements in cost, weight, volume, and reliability at approximately 68L. Surface Mount Technology (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 (PBA's) using SMT is reduced because Surface Mount Components (SMC's) 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.

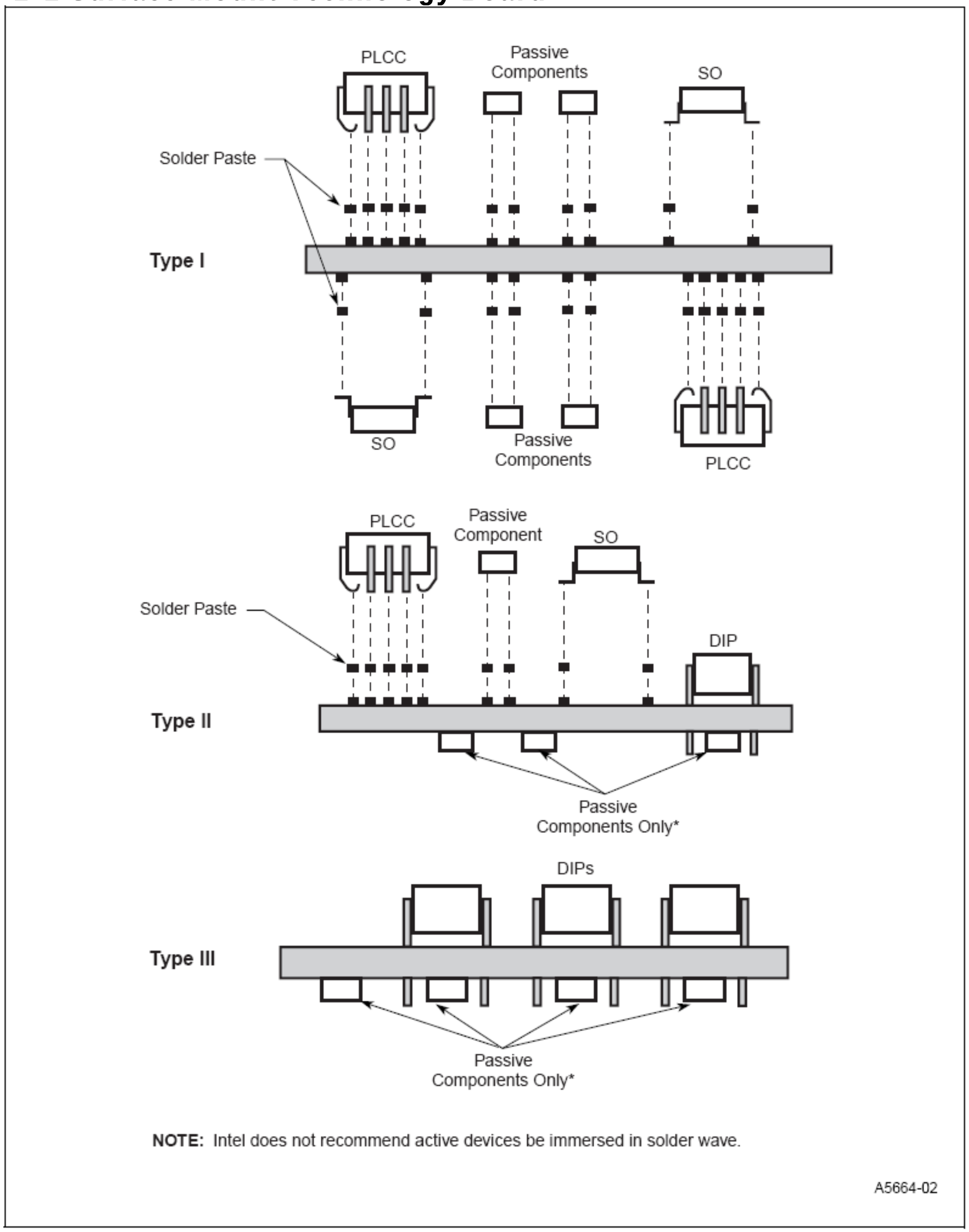
The smaller size of SMC's and the option of mounting them on either or both sides of the PB can reduce board real estate by four times. A cost savings of 30% or better can also be realized through a reduction in material and labor costs associated with automated assembly.

1.2 Types of Surface Mount Technology

SMT replaces DIP's 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 PB's, both active and passive SMC's form three major types of SMT assemblies, commonly referred to as Type 1, Type II, and Type III (see Figure 1-1).

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 SMC's and DIP's 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 SMC's on the secondary side, but on the primary side only DIP's are used.

Figure 1-1 Surface Mount Technology Board



The process sequence for Type III SMT is shown in Figure 1-2. Leaded components are inserted, usually by automatic equipment. The assembly is turned over, and adhesive is applied. Next, passive SMC's are placed by a "pick-and-place" robot, the adhesive is cured, the assembly is turned over, and the wave-soldering process is used to solder both leaded and passive SMC's in a single operation. Finally, the assembly is cleaned (if needed), inspected, repaired if necessary, and tested. For this type of board, the surface mount components used are chip components and small pin count gull wing components.

The process sequence for Type I SMT is shown in Figure 1-3. 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 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.

Figure 1-2 Typical Process Flow of Underside Attachment (Type III SMT)

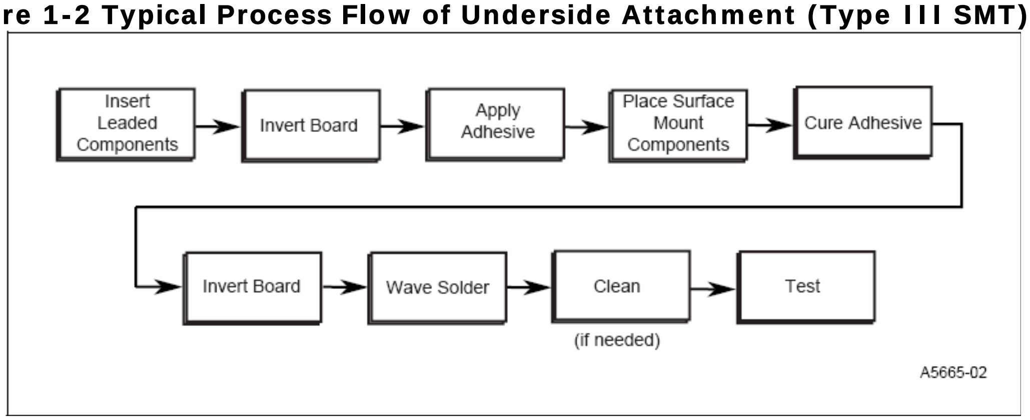
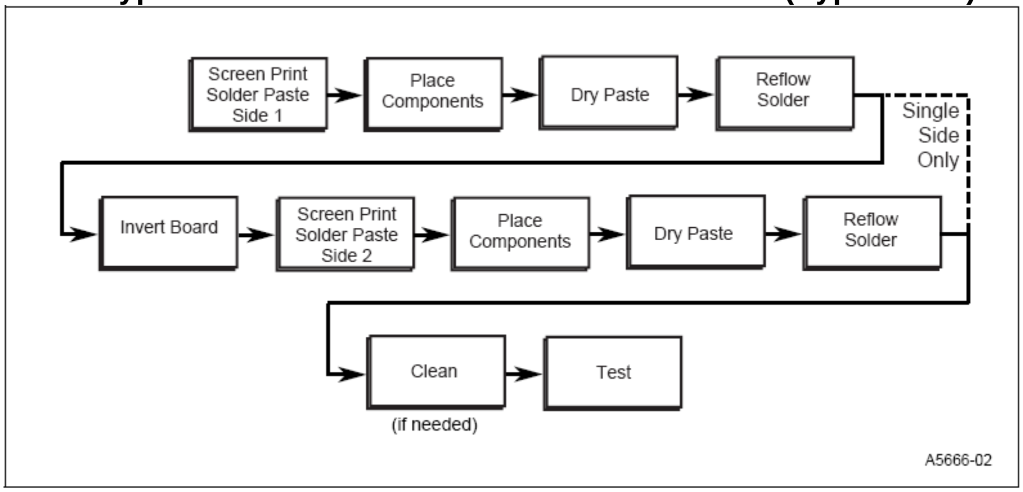


Figure 1-3 Typical Process Flow for Total Surface Mount (Type I SMT)



1.3 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 Plastic Leaded Chip Carriers (PLCC's) and Small Outline Packages (SOP's). 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 (QFP's).

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.

Other manufacturing issues for assembling fine-pitch components on PC boards include:

1. Printing various amounts of solder paste on the 25-mil and 50-mil lands. One stencil thickness will usually suffice. But stencils may be stepped down to a thinner amount for fine pitch aperture areas to keep volumes lower to prevent bridging.
2. Cleaning adequately under and around package leads.
3. Baking of the packages to remove moisture. Thin QFP's are susceptible to a problem known as popcorning where moisture in the plastic can literally explode when heating in reflow or rework and crack the plastic package.
4. Handling of the packages without damaging fragile leads.

These challenges are by no means insurmountable. Many equipment choices have already found solutions to these issues.

1.4 Surface Mount Design

1.4.1 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 manufacturable 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" to 0.060". The space is typically measured from pad to pad, lead to lead, or body to body, whichever is closest. Smaller spacing (0.040") is generally used for low or thin profile parts and small chip components. Taller parts such as PLCC's are usually spaced at 0.060". 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.

The spacing between the pads of conventional and surface mount components may be as large as 0.100", so that auto-insertion equipment may be used for conventional components. Clear spaces of at least 0.050” should be allowed around all edges of the PC boards if the boards are tested off the connector, or 0.100”, if vacuum seal is used for testing, such as bed-of-nails.

Another manufacturing consideration is the alignment of components on the PC board. Similar types of components should be aligned in the same orientation for ease of component placement, inspection, and soldering.

Via holes are used to connect SMC lands to conductor layers. They may also be used as test targets for bed-of-nails probes and/or rework ports. Via holes may be covered with solder mask material if they are not required for node testing or rework. Such vias are called tented or capped vias.

Via holes may be placed under surface mount components. However, in Type II and Type III SMT (mix-and-match surface mount), via holes under SMC’s should be minimized or tented with solder mask to prevent trapping of flux under the packages during wave soldering. For effective cleaning, via holes should only be located beneath SMC’s in Type I SMT assemblies (full surface mount) that are not wave soldered.

1.4.2 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, cleanability, 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. There are a variety of package types offered by the industry, and the variations in a given package type can be numerous. For example, for the SOP’s, there are not only two lead types (gull-wing and J-lead), but there are multiple body types such as narrow, wide and thin. In addition, the tolerance on components varies significantly, adding to the manufacturing problems for SMC users.

In this section, general guidelines are presented for land pattern designs that accommodate reasonable tolerances in component packages, process, and equipment used in manufacturing. These guidelines are based on manufacturability and environmental testing of different land pattern designs for reliability.

To simplify the land pattern design guidelines, surface mount components are divided into five different categories:

1. 0.050" Pitch J-Leaded Devices
2. 0.050" Pitch Gull-wing Leaded Devices
3. Sub 0.050" Pitch Gull-wing Leaded Devices
4. Chip Components
5. BGA Components

Again, with the large variety of SMT parts available today, listing every pad size would create a very long list. So, instead of providing specific pad sizes, the general formulas for the land pattern designs are given for each category. There are several different approaches to dimensioning pads. In addition to the guidelines below, IPC also publishes its own set of guidelines. Each customer should study several options and decide which is best for their application.

1.4.2.1 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.

Comments:
The outer (heel) fillet is the important one for J-Leaded devices.

1.4.2.2 0.050” Pitch Gull-wing 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.

1. An overview of the land pattern design method is:
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.

Comments:
The inner (heel) fillet is the important one for Gull-wing devices. Toe fillets are not required, as they add little strength, and often don't form anyway, due to lead trimming after plating (end of toe may be bare copper or alloy 42).

1.4.2.3 Sub 0.050" Pitch Gull-wing Leaded Devices

On rectangular 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:

Pitch	Pad Size
1.0mm (approx .0394")	0.025" x 0.075"
0.8mm (approx .0315")	0.018" x 0.070"
0.65mm (approx .0256")	0.015" x 0.070"
0.5mm (approx 0.0197")	0.012" x 0.070"

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 or Process Engineer.

1.4.2.4 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 a 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 = Max component length + 2 * (max termination height, or 0.040", 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".

Warning: 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.

1.4.2.5 Ball Grid Array (BGA) Components

For BGA's, the land pattern of the component substrate (where the ball is attached) and the land pattern of the mounting structure (printed board) should be as similar in diameter as possible. For optimum solder joint reliability, component manufacturers have determined that

the land pattern or pad on the component should be slightly less than the ball diameter. The amount of reduction is based on the original ball size, which is used to determine the average land. In determining the relationship between nominal characteristics, a manufacturing allowance for land size has been determined to be 0.1 mm between the Maximum Material Condition (MMC) and Least Material Condition (LMC). The information shown in Table 1-1 provides data on land patterns and their variation to accommodate six common ball diameters.

Table 1-1 BGA Land Size Approximations

Nominal Ball Diameter (mm)	Reduction	Nominal Land Diameter (mm)	Land Variation (mm)
0.75	25%	0.55	0.60 - 0.50
0.60	25%	0.45	0.50 - 0.40
0.50	20%	0.40	0.45 - 0.35
0.45	20%	0.35	0.40 - 0.30
0.40	20%	0.30	0.35 - 0.25
0.30	20%	0.25	0.25 - 0.20
0.25	20%	0.20	0.20 - 0.17
0.20	20%	0.15	0.15 - 0.12
0.15	20%	0.10	0.10 - 0.08

1.4.3 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 SMC’s does not impact testability if rules for testability of assemblies are strictly observed. These rules require that (1) 0.050” and 0.100” 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 is 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. The following sections address process issues in the manufacturing of surface mount assemblies that play a critical role even when boards are designed for manufacturability.

1.5 Solder Paste Application

1.5.1 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 may even 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. A rubber or metal squeegee blade forces the paste through stencil openings that precisely match the land patterns on the

PB. 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 1-2, and choosing one over the others depends on the application and the product type. Today, No Clean solder pastes are the most common solder pastes used in the electronics manufacturing industry.

Table 1-2 Comparison of Solder Paste Types

Type	Advantages	Disadvantages
RMA	• Stable chemistry • Good properties	• Needs chemical solvent or saponification for cleaning
OA	• Cleaned using pure water • Very good cleanability	• Humidity sensitive, seen as short shelf life, solder ball tendency • Water leaches lead into waste stream
No-Clean	• No cleaning process, equipment or chemicals • Eliminates effluent issues	• May leave some visible residue behind

1.6 Surface Mount Components and Their Placement

1.6.1 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 machine run time. Trays are also used, generally for large packages such as QFP's. The EIA specification RS-481A has standardized reel specifications for passive components and active components.

1.6.2 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. Simultaneous firing of heads is possible.

Many models of auto-placement equipment are available in each of the four categories. Selection criteria should consider such issues as the kind of parts are to be handled, whether they come in tubes, trays, or tape and reel, and whether the machine can accommodate future changes in other shipping media. Selection and evaluation of tapes from various vendors for compatibility with the selected machine is very important. Off-line programming, teach mode, and edit capability, as well as CAD / CAM compatibility may be very desirable, especially if a company has already developed a CAD / CAM database. 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.

1.7 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.

1.7.1 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. While large mass devices on the PB will heat more slowly than low mass devices, the deltas are small allowing all parts to see nearly the same heat cycle.

1.8 Cleaning

Since No Clean solder pastes are the most widely used in the industry today, cleaning of electronic assemblies is not too common today. In general, however, 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 effluents 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. For SMT, the industry also uses Surface Insulation Resistance (SIR) surface mount boards. These boards check for ionic contaminates left on the PB by measuring the electrical resistance between adjacent traces or circuits

1.9 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 to 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. The guidelines outlined in Chapter 5 should be followed.

1.10 Conclusion

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. For more detail on board assembly processing, see Chapter 2.

1.11 Revision Summary

Original publication