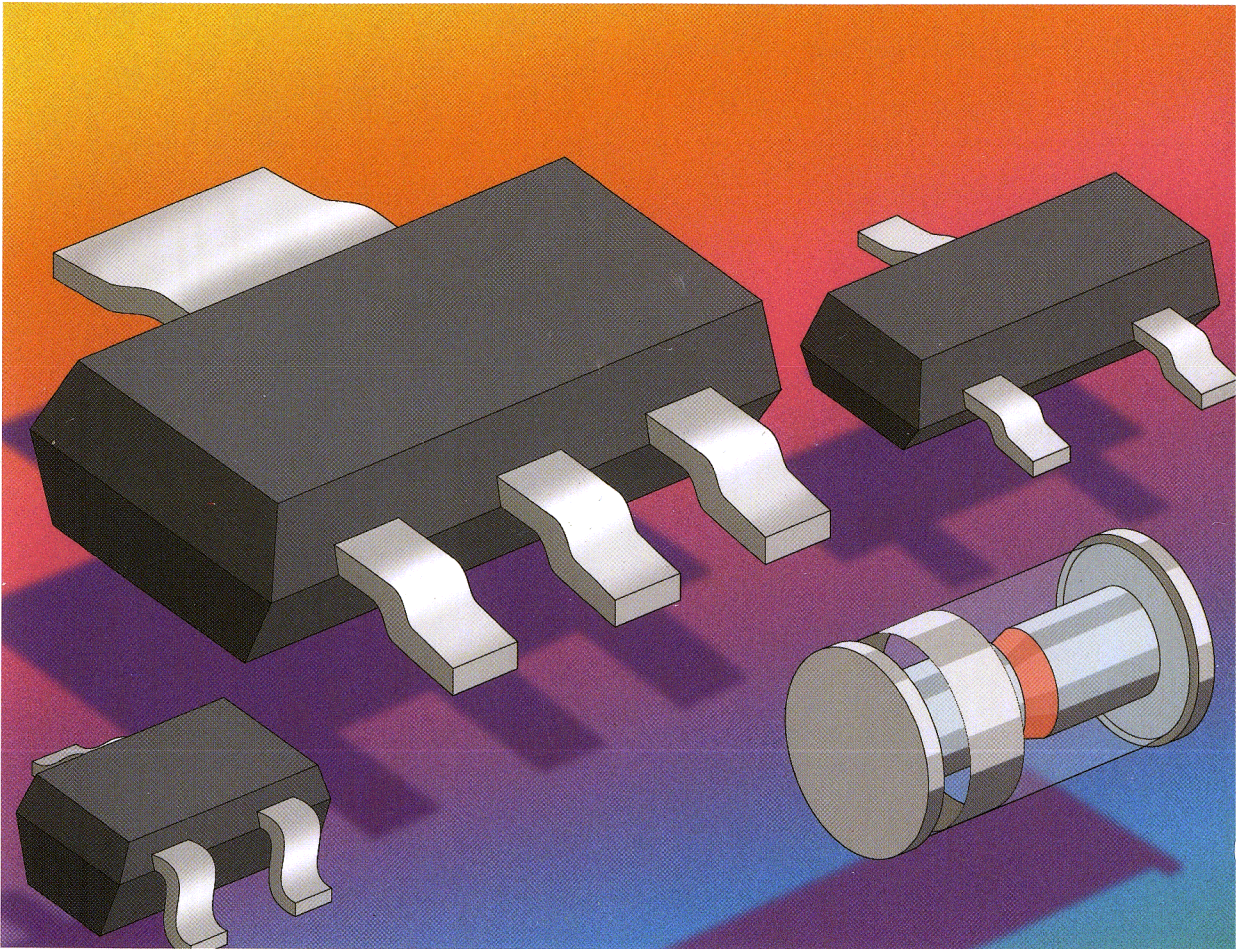


SMD Footprint Design and Soldering Guidelines



1996

DATA HANDBOOK SC18
(Provisional)

QUALITY ASSURED

Our quality system focuses on the continuing high quality of our components and the best possible service for our customers. We have a three-sided quality strategy: we apply a system of total quality control and assurance; we operate customer-oriented dynamic improvement programmes; and we promote a partnering relationship with our customers and suppliers.

PRODUCT SAFETY

In striving for state-of-the-art perfection, we continuously improve components and processes with respect to environmental demands. Our components offer no hazard to the environment in normal use when operated or stored within the limits specified in the data sheet.

Some components unavoidably contain substances that, if exposed by accident or misuse, are potentially hazardous to health. Users of these components are informed of the danger by warning notices in the data sheets supporting the components. Where necessary the warning notices also indicate safety precautions to be taken and disposal instructions to be followed. Obviously users of these components, in general the set-making industry, assume responsibility towards the consumer with respect to safety matters and environmental demands.

All used or obsolete components should be disposed of according to the regulations applying at the disposal location. Depending on the location, electronic components are considered to be 'chemical', 'special' or sometimes 'industrial' waste. Disposal as domestic waste is usually not permitted.

SMD Footprint Design and Soldering Guidelines

Since the introduction of Surface Mount Devices (SMDs), component design and manufacturing techniques have changed almost beyond recognition. Smaller pitch, minimum footprint area and reduced component volume all contribute to a more compact circuit assembly. As a consequence, when designing printed circuit boards (PCBs), the dimensions of the footprints are perhaps more crucial than ever before.

One of the first steps in this design process is to consider which soldering method, either wave or reflow, will be used during production. This determines not only the solder footprint dimensions, but also the minimum spacing between components, the available area underneath the SMD where tracks may be laid, and possibly the required component orientation during soldering.

Although reflow soldering is recommended for SMDs, many manufacturers use, and will continue to use for some time to come, a mixture of surface-mount and through-hole components and so may prefer the wave soldering method, or even a combination of the two.

To help with your circuit board design, this publication gives an overview of both reflow and wave soldering methods, and is followed by the dimensional outline drawings and recommended footprints for our SMD discrete semiconductor packages.

REFLOW SOLDERING PROCESSES

Process steps

There are three basic process steps for single-sided PCB reflow soldering, these are:

- applying solder paste to the PCB
- component placement
- reflow soldering.

Applying solder paste to the PCB

Solder paste can be applied to the PCB's solder lands by one of either three methods: dispensing, screen or stencil printing.

Dispensing is flexible but is slow, and only suitable for pitches of 0.65 mm and above.

With screen printing, a fine-mesh screen is placed over the PCB and the solder paste is forced through the mesh onto the solder lands of the PCB. However, because of mesh aperture limitations (emulsion resolution), this method is only suitable for solder paste deposits of 300 µm and wider.

Stencil printing is similar to screen printing, except that a metal stencil is used instead of a fine-mesh screen. The stencil is usually made of stainless steel or bronze and should be 150-200 µm thick. A squeegee is passed across the stencil to force solder paste through the apertures in the stencil and onto the solder lands on

the PCB (see Fig.1). It does not suffer from the same limitations as the other two printing methods and so is the preferred method currently available.

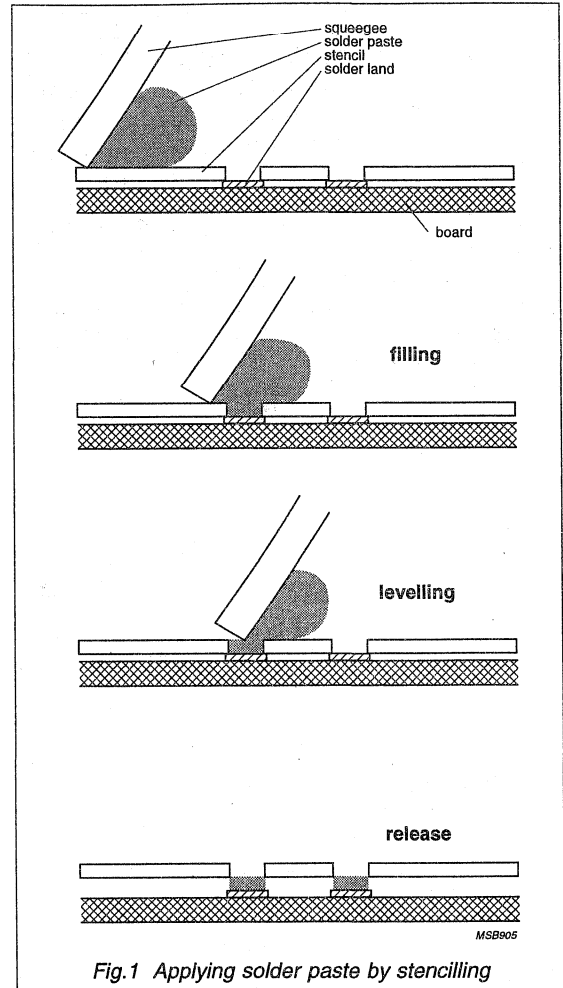


Fig.1 Applying solder paste by stencilling

The reflow soldering environment

It's recommended that for solder paste printing, the equipment is located in a controlled environment maintained at a temperature of 23 ± 2 °C, and a relative humidity between 45% and 75%.

Stencil printing

The printing process must be able to apply the solder paste deposits to the PCB:

- in the correct amounts
- at the correct position on the lands
- with an acceptable height and shape.

SMD Footprint Design and Soldering Guidelines

The amount of solder paste used must be sufficient to give reliable soldered joints. This amount is controlled by the stencil thickness, aperture dimensions, process settings, and the volume of paste pressed through the apertures by the squeegee.

The downward force of the squeegee is counteracted by the hydrodynamic pressure of the paste, and so the machine should be set to ensure that the stencil is just "cleaned" by the squeegee.

Suitable aperture dimensions depend on the stencil thickness. The solder paste deposits must have a flat part on the top (Fig.2, examples 4 and 5), which can be achieved by correct process settings. The footprints given in this book were designed for these correct deposit types. Stencil apertures that are too small result in irregular dots on the lands (Fig.2, examples 1 to 3). If the apertures are too large, solder paste can be scooped out, particularly if a rubber squeegee is used (Fig.2, example 6).

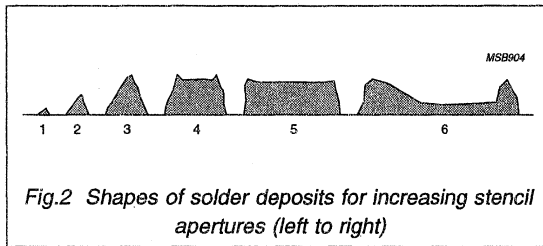


Fig.2 Shapes of solder deposits for increasing stencil apertures (left to right)

Ideally, the deposited solder paste should sit entirely on the solder land. The tolerated misplacement of solder paste with respect to the solder land is determined by the most critical component (SOT363/SC70-6). The solder paste deposit must be deposited within 100 μm with respect to the solder land.

Furthermore, the tackiness (tack strength) of the solder paste must be sufficient to hold surface-mount devices on the PCB during assembly and during transport to the reflow oven. Tack strength depends on factors such as paste composition, drying conditions, placement pressure, dwell time and contact area. As a general rule, component placement should be within four hours after the paste printing process.

Squeegee

The squeegee can be either metal or rubber. A metal squeegee gives better overall results and so is recommended, however with step stencils, a rubber squeegee has to be used. The footprints given at the end of this book were designed for application by both types of squeegee.

Stencil apertures

Stencil apertures can be made by either:

- etching
- laser cutting
- electroforming.

Of the three methods, etching is less accurate as the deviation in aperture dimensions with respect to the target is relatively large (target is +50 μm at squeegee side and 0 μm at PCB side).

Laser-cut and electroformed stencils have smaller deviations in dimensions and are therefore more suitable for small and fine-pitch components (Fig.3)

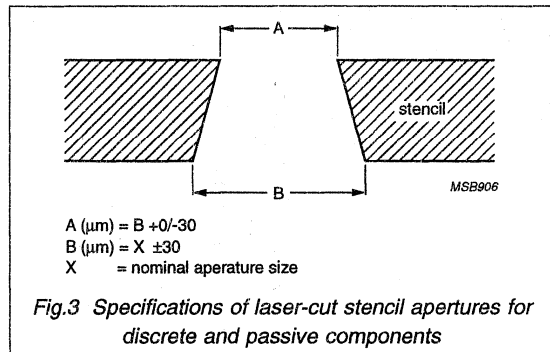


Fig.3 Specifications of laser-cut stencil apertures for discrete and passive components

A useful method of controlling the stencil printing process during production is by monitoring the weight of solder paste on the board which may vary between 80% and 110% of the theoretical amount according to the target (designed) apertures. Smearing and clogging of a small aperture cannot be detected with this method.

Solder paste

Reflow soldering uses a paste consisting of small nodules of solder and a flux with binder, solvents and additives to control rheological properties. The flux in the solder paste can be rosin mildly activated or rosin activated.

The requirements of the solder paste are:

- good rolling behaviour
- no slump during heat-up
- low viscosity during printing
- high viscosity after printing
- sufficient tackiness to hold the components
- removal of oxides during reflow soldering.

Suitable solder paste types have the following compositions:

- Sn62Pb36Ag2
- Sn63Pb37
- Sn60Pb40.

SMD Footprint Design and Soldering Guidelines

Component placement

The position of the component with respect to the solder lands is an important factor in the final result of the assembly process. A misaligned component can lead to unreliable joints, open circuits and/or bridges between leads.

The placement accuracy is defined as the maximum permissible deviation of the component outline or component leads, with respect to the actual position of the solder land pattern belonging to that component or component leads on the circuit board (Fig.4).

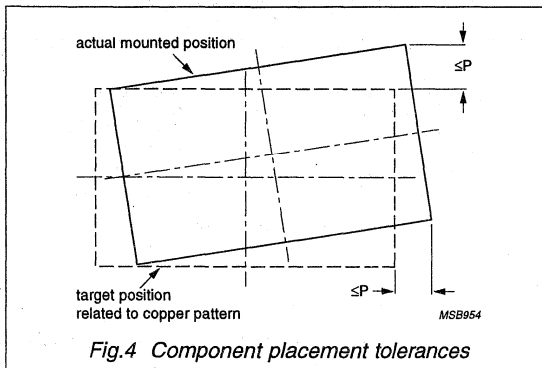


Fig.4 Component placement tolerances

A maximum placement deviation (P) of 0.25 mm is used in these guidelines, which relates to the accuracy of a low-end placement machine. A higher placement accuracy is required for components with a fine pitch. This is given in the footprint description for the components concerned.

Besides the position in x- and y-directions, the z-position with respect to the solder paste, which is determined by the placement force, is also important. If the placement force is too high, solder paste will be squeezed out and solder balls or bridges will be formed. If the force is too low, physical contact will be insufficient, leads will not be soldered properly and the component may shift.

Reflow soldering methods

There are several methods available to provide the heat to reflow the solder paste, such as convection, hot belt, hot gas, vapour phase and resistance soldering. The preferred method is, however, convection reflow.

Convection reflow

With this method, the PCBs passes through an oven where it is preheated, reflow soldered and cooled (see Fig.5). If the heating rate of the board and components are similar, however, preheating is not necessary.

During the reflow soldering process, all parts of the board must be subjected to an accurate temperature/time profile. Figure 5 shows a suitable profile framework for single-sided reflow soldering and the first side of double-sided print boards. It's important to note that this profile is for discrete semiconductor packages. The actual framework for the entire PCB could be smaller than the one shown, as other components on the board may have different process requirements.

Reflow soldering can be done in either air or a nitrogen atmosphere. If soldering in air, the temperature (T_p) must not exceed 240 °C on the first side of a double-sided print board with organic coated solder lands. This is because peak temperatures greater than 240 °C reduce the solderability of the lands on the second side to be soldered. This peak temperature can rise to 280 °C when soldering the second side with organic coated solder lands in air.

If soldering in a nitrogen atmosphere, a peak temperature of 280 °C is allowed for double-sided print boards or single-sided reflow soldering. Soldering in a nitrogen atmosphere results in smoother joint meniscus, smaller contact angles, and better wetting of the copper solder lands.

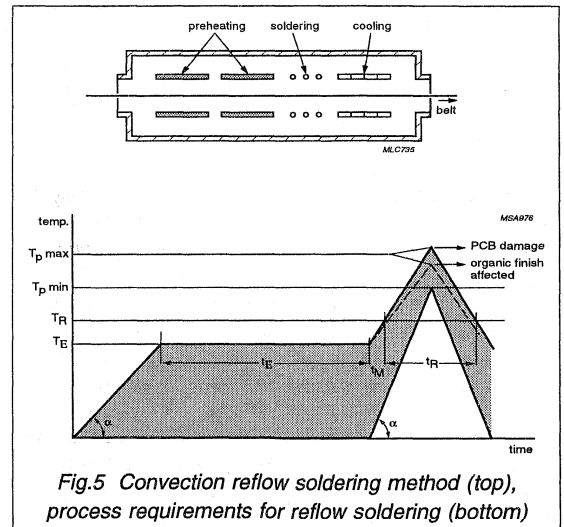


Fig.5 Convection reflow soldering method (top), process requirements for reflow soldering (bottom)

where:

$\alpha \leq 10 \text{ }^\circ\text{C/s}$	$t_E \leq 1 \text{ min.}$, if possible (else $\leq 5 \text{ min.}$)
$T_E \leq 160 \text{ }^\circ\text{C}$	
$T_R = 180 \text{ }^\circ\text{C}$	$T_{p \text{ min}} = 205 \text{ }^\circ\text{C}$
$t_R \leq 70 \text{ s}$	$T_{p \text{ max}} = 240 \text{ }^\circ\text{C}$ for soldering the first side of a double-sided board with organic finish.
$t_M = 2 - 30 \text{ s}$	$T_{p \text{ max}} = 280 \text{ }^\circ\text{C}$ for all other cases

SMD Footprint Design and Soldering Guidelines

The profile can be achieved by correct combinations of conveyor speed and heater temperature. To check whether the profile is within specification, the coldest and hottest spots on the board have to be located.

To do this, you should dispense solder paste deposits regularly over the surface of a test board and on the component leads. Set the oven to a moderate temperature with maximum conveyor velocity and pass the test board through. If too many solder paste dots melt, lower the oven's temperature. Continue passing test boards through the oven, while lowering the speed of the belt in small steps.

The deposit that melts first indicates the warmest location, the one that melts last indicates the coldest location. Paste dots not reflowed after two runs must be replaced by fresh dots. Thermocouples have to be mounted at the coldest and warmest location and temperature profiles measured.

DOUBLE-WAVE SOLDERING PROCESSES

Process steps

There are four basic process steps for double-wave soldering, these are:

- applying adhesive
- component placement
- curing adhesive
- double-wave soldering.

Applying adhesive

To hold SMDs on the board during wave soldering, it's necessary to bond the component to the PCB with one or more adhesive dots. This is done either by dispensing, stencilling or pin transfer. Dispensing is currently the most popular technique. It is flexible and allows a controlled amount of adhesive to be applied at each position. Stencil printing and pin transfer are less flexible and are mainly used for mass production.

The component-specific requirements for an adhesive dot are:

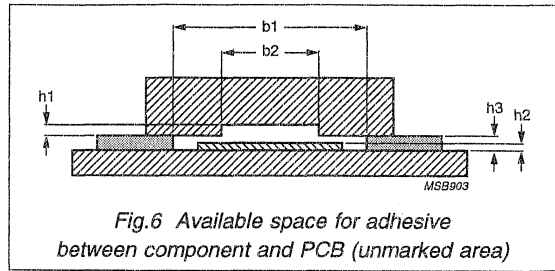
- shape (volume) of the adhesive dot
- number of dots per component
- position of the dots.

Volume of adhesive

There must be enough adhesive to keep components in their correct positions while being transported to the curing oven. This means that the deposited adhesive

must be higher than the gap between the component and the board surface. Nevertheless, there should not be too much deposit as it may smear onto the solder lands, where it can affect their solderability.

The gap between a component and printed board depends on the geometry of the board and component (Fig.6).



where:

- h1 = component stand-off height
- h2 = solder resist (and track) height on PCB
- h3 = copper height on PCB
- b1 = gap between solder lands on the PCB
- b2 = gap between metallisations of the component

Table 1 gives guidelines for volumes of adhesive dots per package. The spreading in volumes should be within $\pm 15\%$.

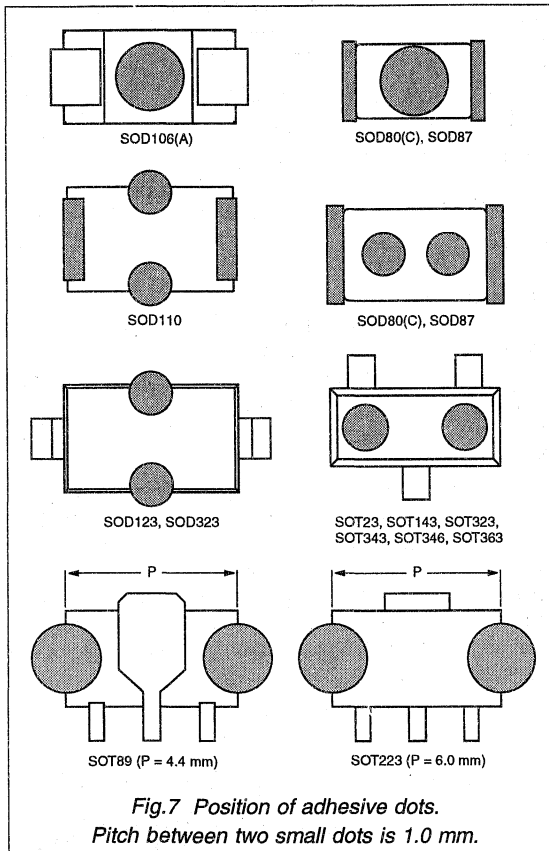
TABLE 1 Guidelines for volumes of adhesive dots		
component	number of dots	volume per dot (mm ³)
SOT89	2	0.3
SOD106(A)	1	0.65
SOD80(C), SOD87	1	0.5
	2	0.08
SOD123	2	0.07
SOD110, SOD323	2	0.065
SOT323, SOT343, SOT353, SOT363	2	0.045
SOT23, SOT143, SOT346	2	0.06
SOT223	2	0.70

SMD Footprint Design and Soldering Guidelines

Number, position and volume of dots per component

Figure 7 shows the recommended positions and numbers of adhesive dots for a variety of packages. SOD106(A), SOT89 and SOT223 packages require much larger adhesive dots compared with those for other components. SOD80(C) and SOD87 packages can have one large adhesive dot (recommended) or two smaller adhesive dots.

Note: For optimum power dissipation, the SOT89 requires a good thermal contact (i.e. good solder joint) between the package and the solder land. During wave-soldering, however, flux may not always reach the total soldering area beneath the component body, which in turn can lead to an incomplete solder joint. If the SOT89 is double-wave soldered, therefore, power derating must be applied.



Nozzle outlet diameter

Depending on adhesive type and component size, the nozzle outlet diameter of the dispenser can vary between 0.6 and 0.7 mm for the larger dots, and between 0.3 and 0.5 mm for the smaller dots.

As the rheology of the adhesive is temperature dependent, the temperature in the nozzle must be carefully controlled before dispensing. The required temperature depends on the adhesive type, but is usually between 26 °C and 32 °C to maintain the adhesive's rheology within specification during dispensing. Thermally curing epoxy adhesives are normally used.

Adhesives

Beside the nozzle diameters, different adhesive types are also used for different component sizes. Small components can be secured during assembly and wave soldering with a thin (low green strength) adhesive, which can be dispensed at high speeds. For larger components (such as QFP and SO packages), a higher green strength adhesive is required.

Component placement

Positioning components on the PCB is similar in practice to that of reflow soldering.

To prevent component shift and smearing of the adhesive, board support is important while placing components. This is particularly important when placing the SOD106(A) package.

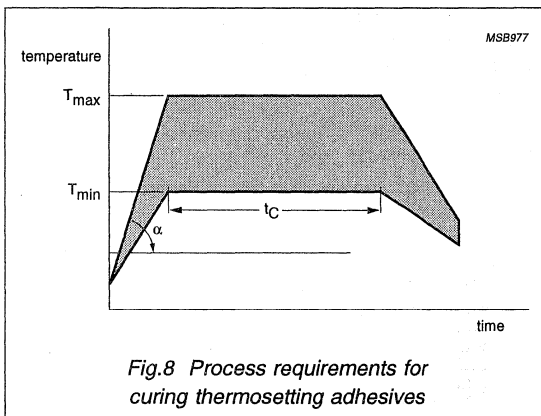
SMD Footprint Design and Soldering Guidelines

Curing the adhesive

To provide sufficient bonding strength between component and board, the adhesive must be properly cured. Figure 8 gives general process requirements for curing most thermosetting epoxy adhesives with latent hardeners. The temperature profile of **all** adhesive dots on the PCB must be within this framework. It's important to note that this profile is for discrete semiconductor packages. The actual framework for the entire PCB could be smaller than the one shown, as other components on the board may have different process requirements.

To check whether the profile is within specification, the temperature of coldest and hottest spots must be measured. The coldest spot is usually under the largest package: the hottest spot is usually under the smallest package.

The adhesive can be cured either by infrared or hot-air convection.



where:

- $T_{max} \leq 160 \text{ }^\circ\text{C}$
- $T_{min} \geq 110 \text{ }^\circ\text{C}$
- $t_c \geq 3 \text{ minutes}$
- $\alpha \leq 100 \text{ }^\circ\text{C/min}$ (some adhesives allow higher heating rates)

If $T_{min} > 125 \text{ }^\circ\text{C}$, t_c may be $< 3 \text{ min.}$, depending on adhesive specification.

Bonding strength

The bonding strength of glued components on the board can be checked by measuring the torque force. For small components the requirements are given in Table 2. No values are specified for larger packages.

component	minimum bonding strength	target bonding strength
SOD323, SOD110, SOT323, SOT363, SOT343, SOT353	110 cNcm	250 cNcm
SOD80(C), SOD87	200 cNcm	350 cNcm
SOT23, SOT346, SOT143	150 cNcm	250 cNcm

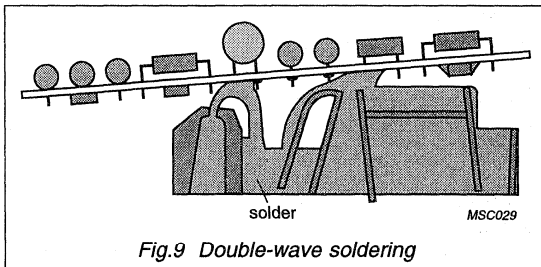
Wave soldering

After applying adhesive, placing the component on the PCB and curing, the PCB can be wave soldered.

The wave soldering process is basically built up from three sub-processes. These are:

- the fluxing process
- the preheating process
- the (double) wave soldering process.

Although listed here as sub-process they are in practice combined in one machine. All are served by one transport mechanism, which guides the PCBs at an incline through the soldering machine. It's important to note that the PCB must be loaded into the machine so that the SMDs on the board come into direct contact with the solder wave (see Fig.9).



SMD Footprint Design and Soldering Guidelines

In principle, two different systems of PCB transports are available for wave soldering:

- **Carrier transport**
PCBs are mounted on a soldering carrier, which moves through the soldering machine, taking it from one sub-process to the next. The advantage of carrier mounting is that the board is fixed and warpage during soldering is reduced.
- **Carrierless transport**
PCBs are guided through the soldering machine by a chain with grips. This method is more convenient for mass production.

The fluxing process

Fluxing is necessary to promote wetting both of the PCB and the mounted components. This ensures a good and even solder joint.

During the fluxing process, the solder side of the PCB (including the components) are covered with a thin layer of solder flux, which can be applied to the PCB either by spraying or as a foam. Although several types of solder flux are available for this purpose, they can be categorized into three main groups:

- non-activated flux (e.g. rosin-based fluxes)
- mildly activated flux (e.g. rosin-based or synthetic fluxes)
- highly activated flux (e.g. water-soluble fluxes)

The choice for a particular flux type depends mainly on the products to be soldered.

Although there is always some flux residue left on the PCB after soldering, it's not always necessary to wash the boards to remove it. Whether to clean the board can depend on:

- The type of flux used (highly activated fluxes are corrosive and so should always be removed).
- The required appearance of the board after soldering.
- Customer requirements.

The preheating process

After the flux is applied, the PCB needs to be preheated. This serves several purposes: it evaporates the flux solvents, it accelerates the activity of the flux and it heats the PCB and components to reduce thermal shock.

The required preheat temperature depends on the type of flux used. For example, the more common low-residue fluxes require a preheat temperature of 120 °C (measured on the wave solder side of the PCB).

The (double) wave soldering process

The PCB first passes over a highly intensive (jet) solder wave with a carefully controlled constant height. This ensures good contact with the PCB, the edges of SMDs and the leads of components near to high non-wetted bodies. The greater the board's immersion depth into this first wave, the fewer joints will be missed.

If the PCB is carrier mounted, the first wave's height, and thus the board's immersion depth, can be greater. Carrierless soldering is more convenient for mass production, but the height of the wave must be lower to avoid solder overflowing to the top side of the board. The height of the jet wave is given in Table 3 along with an indication of soldering process window. This information is based on a 1.6 mm thick PCBs.

The second, smoother laminar solder wave completes formation of the solder fillet, giving an optimal soldered connection between component and PCB. It also reduces the possibility of solder bridging by taking up excessive solder.

To reduce lead/tin oxides and possibly other solder imperfection forming during soldering, the complete wave configuration can be encapsulated by an inert atmosphere such as nitrogen.

TABLE 3
Process ranges for carrierless and carrier double wave soldering

	carrierless	carrier
Preheat temperature of board at wave solder side (°C)	120 ± 10	
Heating rate preheating (°C/s)	$\Delta T/\Delta t \leq 3$	
First (jet) wave wave height with respect to bottom side of board (mm)	1.6 +0.5/-0	3.0 +0.5/-0
Second (laminar) wave height with respect to underside of the board (mm) - relative stream velocity with respect to the board	Double sided overflow 0.8 +0.5/-0 0	
Solder temperature (°C)	250 ± 3	
Contact times (s) - first (jet) wave - second (laminar) wave	0.5 +0.5/-0 2.0 ± 0.2 (plain holes) 2.5 ± 0.2 (plated holes)	
PCB transport angle (°)	7 ± 0.5	
Solder alloys	Sn60Pb40 Sn60Pb38Bi2	

SMD Footprint Design and Soldering Guidelines

ASSESSMENT OF SOLDERED JOINT QUALITY

The quality of a soldered joint is assessed by inspecting the shape and appearance of the joint. This inspection is normally done with either a low-powered magnifier or microscope, however where ultra-high reliability is required, video, X-ray or laser inspection equipment may be considered.

Both sides of the PCB should be carefully examined: there should be no misaligned, missing or damaged components, soldered joints should be clean and have a similar appearance, there should be no solder bridging or residue, and the PCB should be assessed for general cleanliness.

Unlike leaded component joints where the lead also provides added mechanical strength, the SMD relies on the quality of the soldering for both electrical and mechanical integrity. It is therefore necessary that the inspector is trained to make a visual assessment with regard to long-term reliability.

Criteria used to assess the quality of an SMD solder joint include:

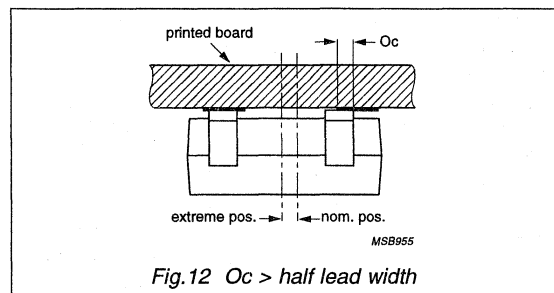
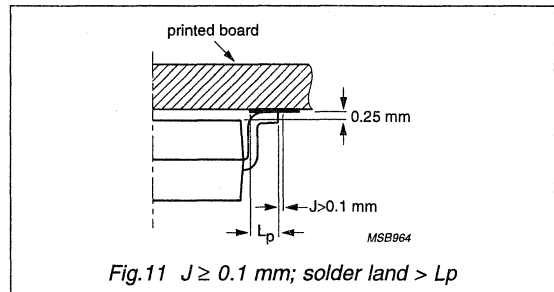
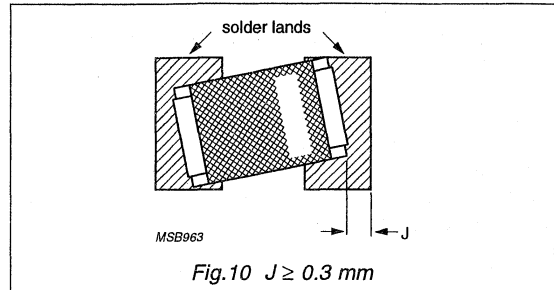
- correct position of the component on the solder lands
- good wetting of the surfaces
- correct amount of solder
- a sound, smooth joint surface.

Positioning - if a lead projects over the solder land too far an unreliable joint is obtained. Figures 10 to 12 show the maximum shift allowed for various components. The dimensions of these solder lands guarantee that, in the statistically extreme situation, a reliable soldered joint can be made.

Good wetting - this produces an even flow of solder over the surface land and component lead, and thinning towards the edges of the joint. The metallic interaction that takes place during soldering should give a smooth, unbroken, adherent layer of solder on the joint.

Correct amount of solder - a good soldered joint should have neither too much nor too little solder: there should be enough solder to ensure electrical and mechanical integrity, but not so much that it causes solder bridging.

Sound, smooth joint surface - the surface of the solder should be smooth and continuous. Small irregularities on the solder surface are acceptable, but cracks are unacceptable.



SMD Footprint Design and Soldering Guidelines

FOOTPRINT DEFINITIONS

A typical SMD footprint, is composed of:

- solder lands (conductive pattern)
- solder resist pattern
- occupied area of the component
- solder paste pattern (for reflow soldering only)
- area underneath the SMD available for tracks
- component orientation during wave soldering.

Solder lands (conductive pattern)

The dimensions of the solder lands given in these guidelines are the actual dimensions of the conductive pattern on the printed board (Fig.13). These dimensions are more crucial for fine-pitch components.

Note: The solder land dimensions are designed to give optimum soldering results. They do not take into account the copper area for optimum power dissipation. If an extra area is required to improve power dissipation, it should be coated with solder resist. This is especially important for power packages such as SOD106(A), SOT89 and SOT223.

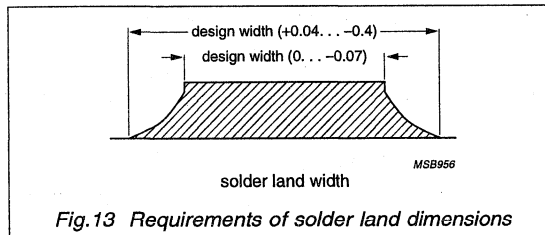


Fig.13 Requirements of solder land dimensions

Solder resist pattern

The solder resist on the circuit board prevents short circuits during soldering, increases the insulation resistance between adjacent circuit details and stops solder flowing away from solder lands during reflow soldering.

In contrast to the tracks, which must be entirely covered, solder lands must be free of solder resist. Because of this, the cut-outs in the solder resist pattern should be at least 0.15 mm or 0.3 mm larger than the relevant solder lands (for a photo-defined and screen printed solder resist pattern respectively). The solder resist cut-outs given with the footprints in these guidelines are sketched and their dimensions can be calculated by using the above rule. Consult your printed board supplier for agreement with these solder resist cut-outs.

Occupied area of the component

A minimum spacing between components is necessary to avoid component placement problems, short circuits during wave or reflow soldering and dry solder joints during wave soldering caused by non-wettable component bodies. These problems can be avoided by placing the components so the occupied areas do not overlap (Fig.14).

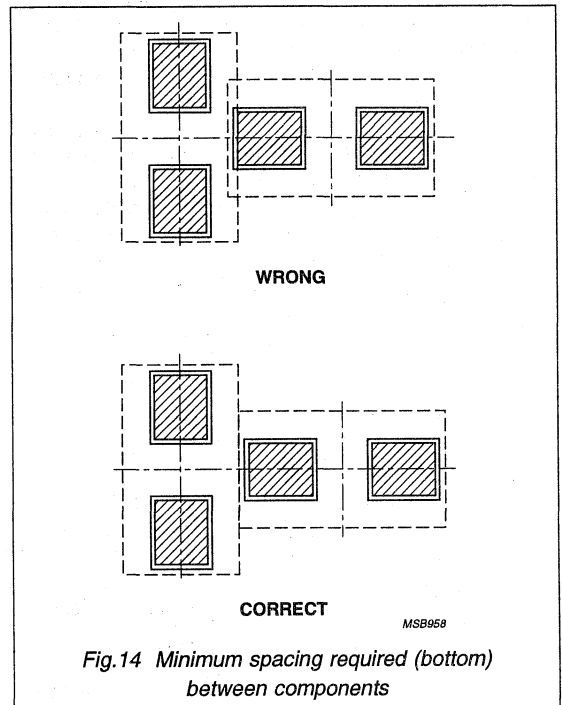


Fig.14 Minimum spacing required (bottom) between components

Solder paste pattern

It's important to use a solder paste printer which is optical aligned with the PCB's copper pattern for the reflow footprints presented here. This is because, for these footprints, the solder paste deposit must be within a 0.1 mm tolerance with respect to the copper pattern.

To ensure the right amount of solder for each solder joint, the stencil apertures must be equal to the solder paste areas given by the footprints.

SMD Footprint Design and Soldering Guidelines

Area available for tracks (conductive pattern)

Tracks underneath leadless SMDs must be covered with solder resist. However, as solder resist can sometimes be thin or have pin holes at the edges of tracks (especially when applied by screen printing), an additional clearance for tracks with respect to the actual metallisation position of the mounted component should be taken into account (Fig.15).

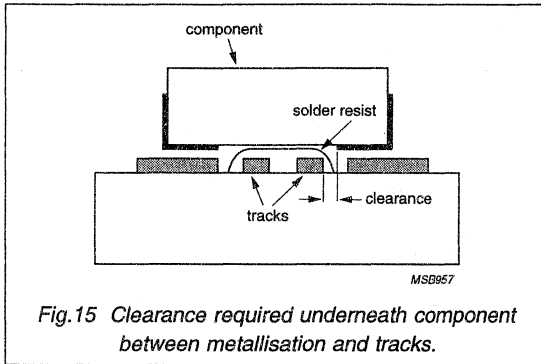


Fig.15 Clearance required underneath component between metallisation and tracks.

For components that need the additional clearance, the footprints on the following pages give the maximum space for tracks not connected to the solder lands

(clearance ≥ 0.1 mm), for low-voltage applications. The number of tracks in this space is determined by the specified line resolution of the printed board.

Component orientation during wave soldering

Where applicable, footprints for wave soldering are given with the transport direction of the PCB. This is given as either a "preferred transport direction during soldering" or "transport direction during soldering".

Components with small terminals and non-wettable bodies, have a smaller risk of dry joints, especially when using carrierless soldering as the components are placed according to the "preferred orientation".

Components have no orientation preference for reflow soldering.

RECOMMENDED FOOTPRINTS

The recommended footprints for our discrete semiconductor packages are given on the following pages along with their dimensional outline drawing.

In addition to its standard footprints, SOD110 has a reflow footprint for high thermal cycling load applications (see Fig. 16). This footprint has larger solder lands to give a fatter solder fillet and, therefore, longer solder fatigue life. A SOD110 mounted on this footprint has a lower self-aligning ability, and has a higher risk of displacement when soldered in a nitrogen atmosphere.

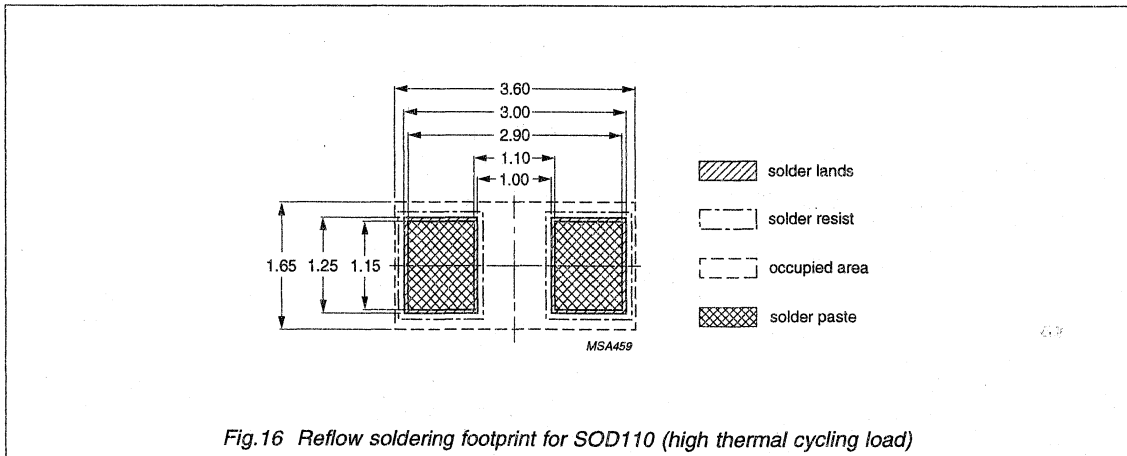
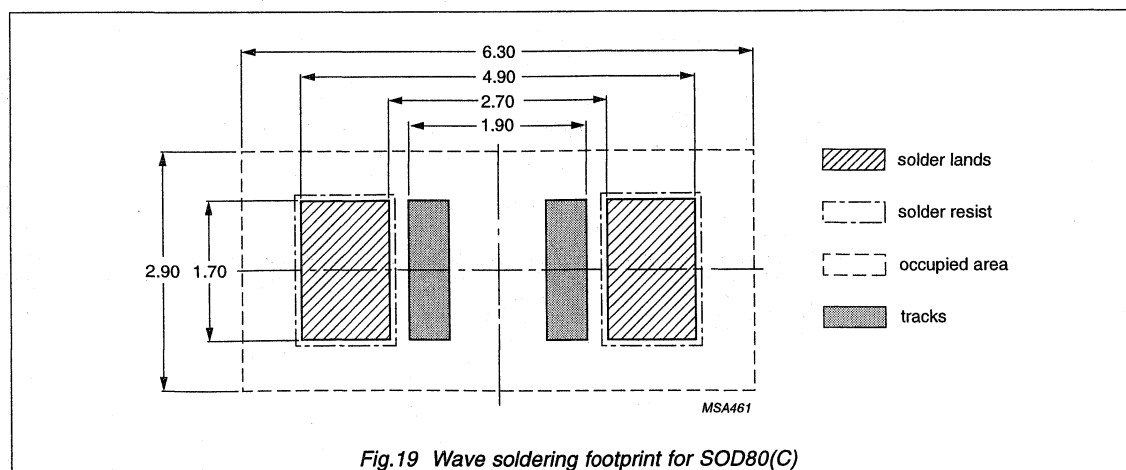
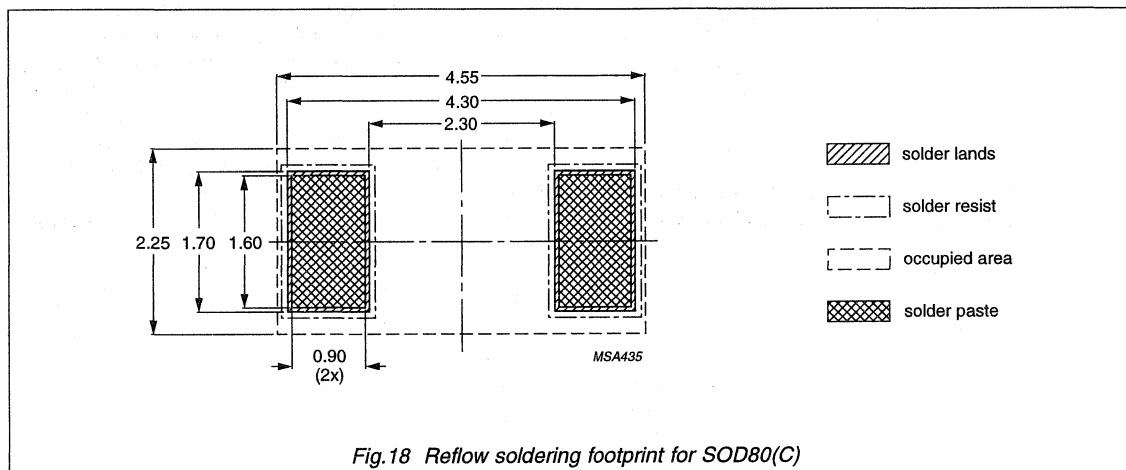
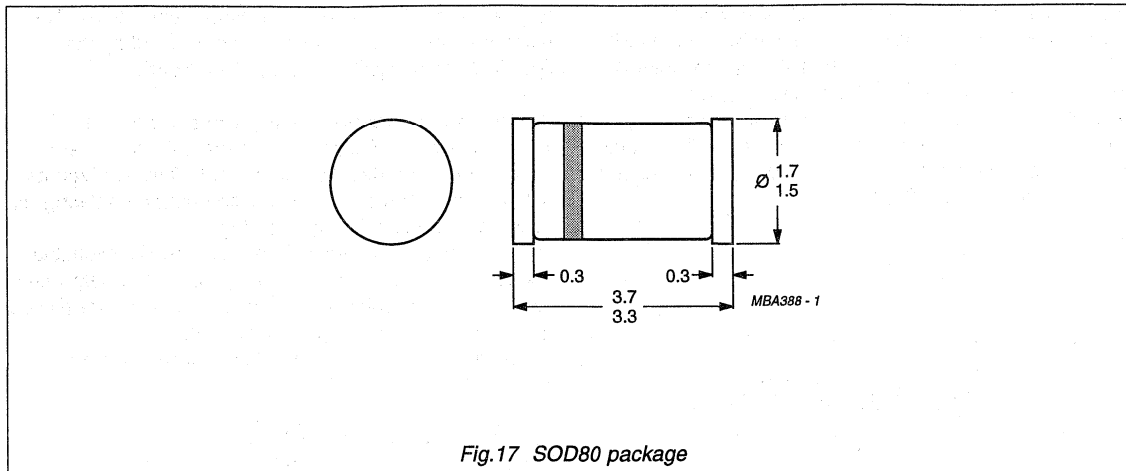


Fig.16 Reflow soldering footprint for SOD110 (high thermal cycling load)

SMD Footprint Design and Soldering Guidelines



SMD Footprint Design and Soldering Guidelines

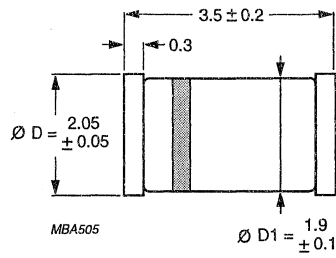


Fig.20 SOD87 package

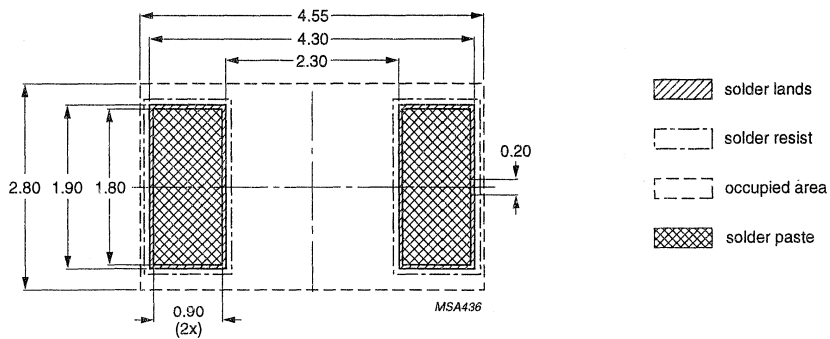


Fig.21 Reflow soldering footprint for SOD87

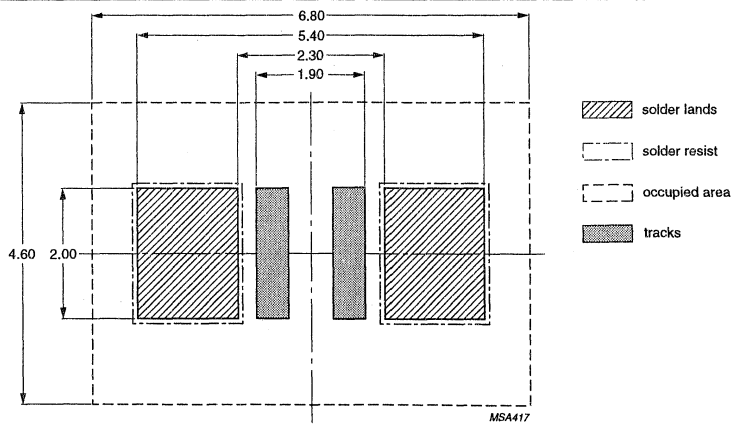
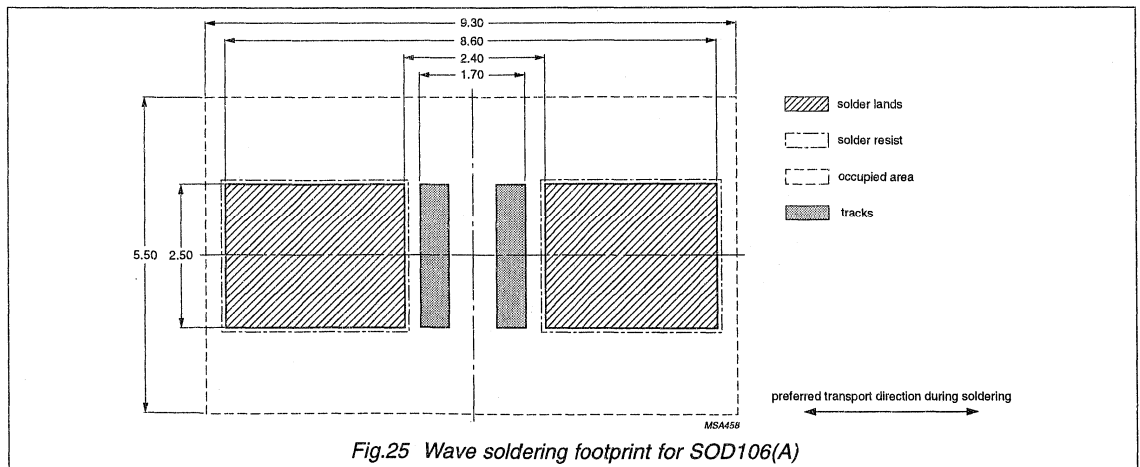
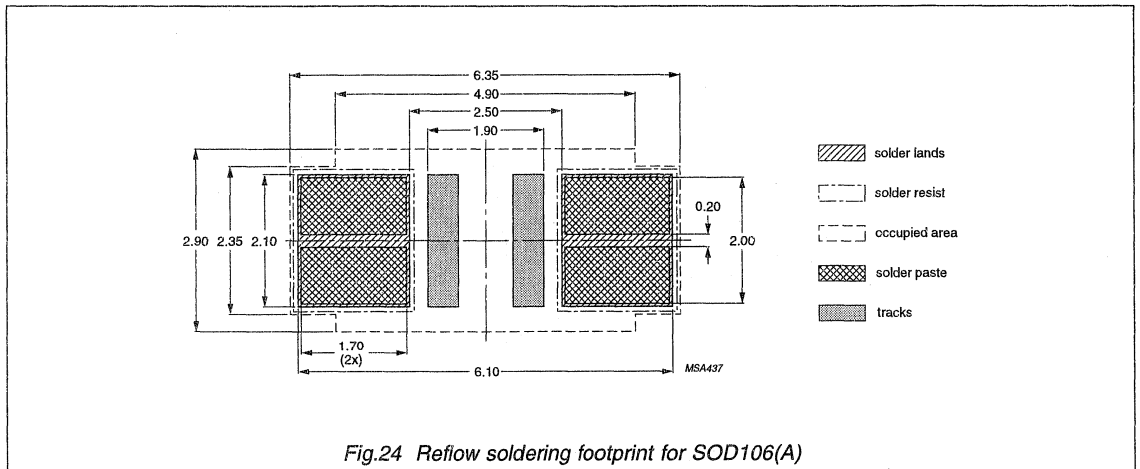
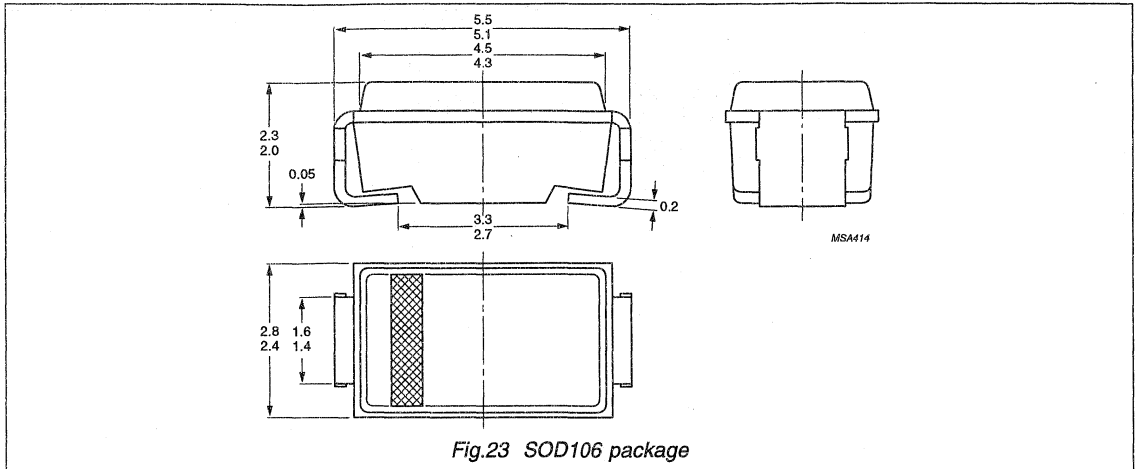
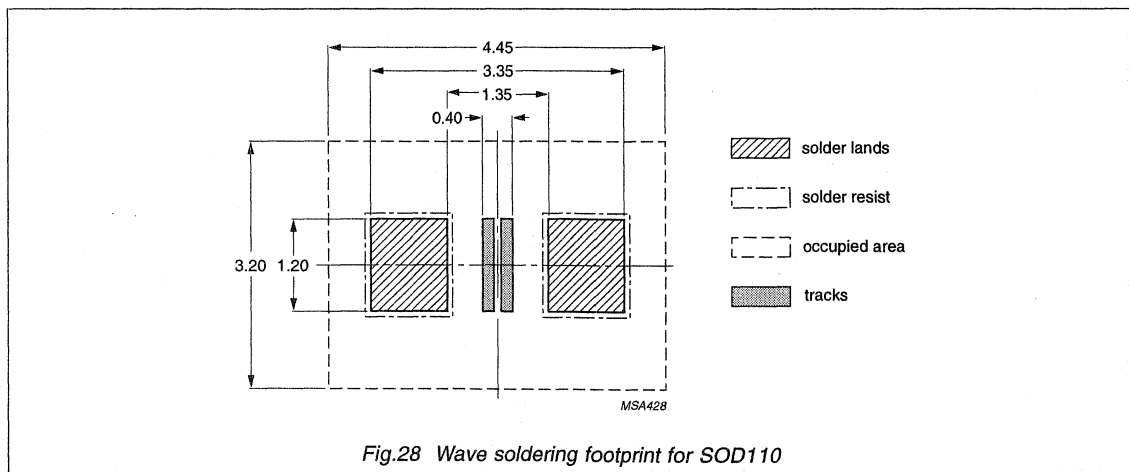
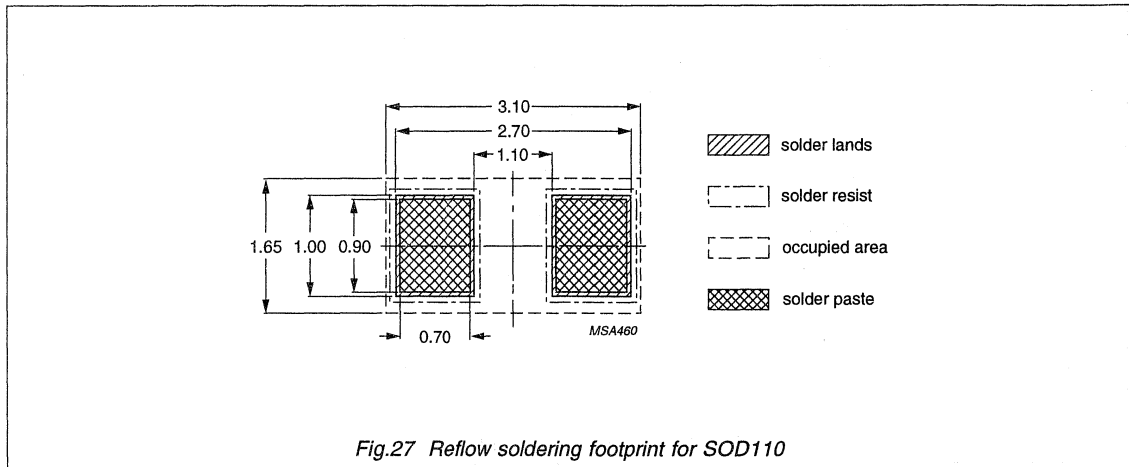
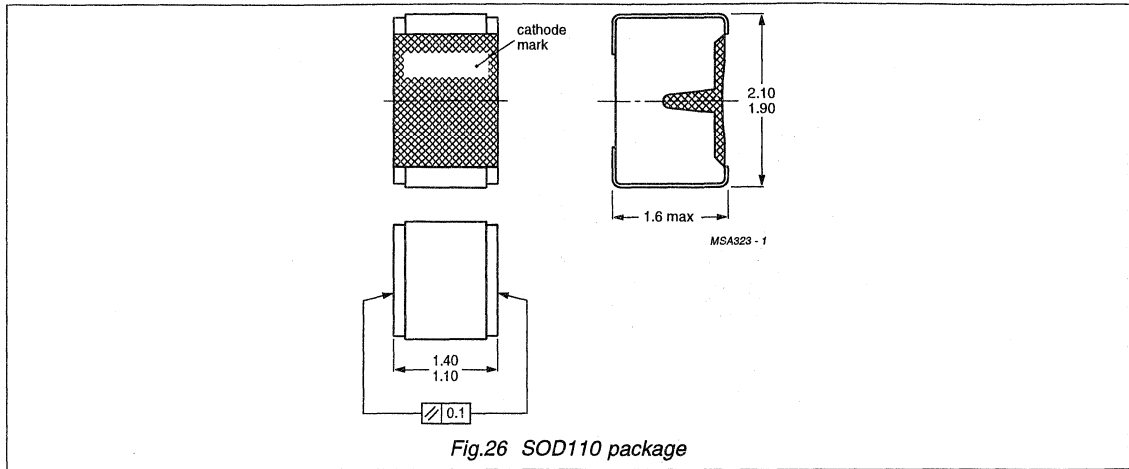


Fig.22 Wave soldering footprint for SOD87

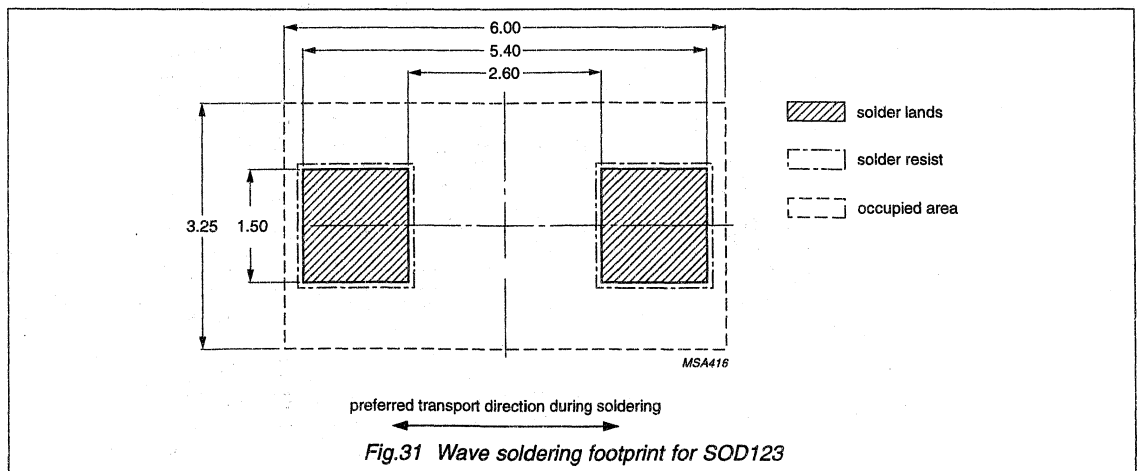
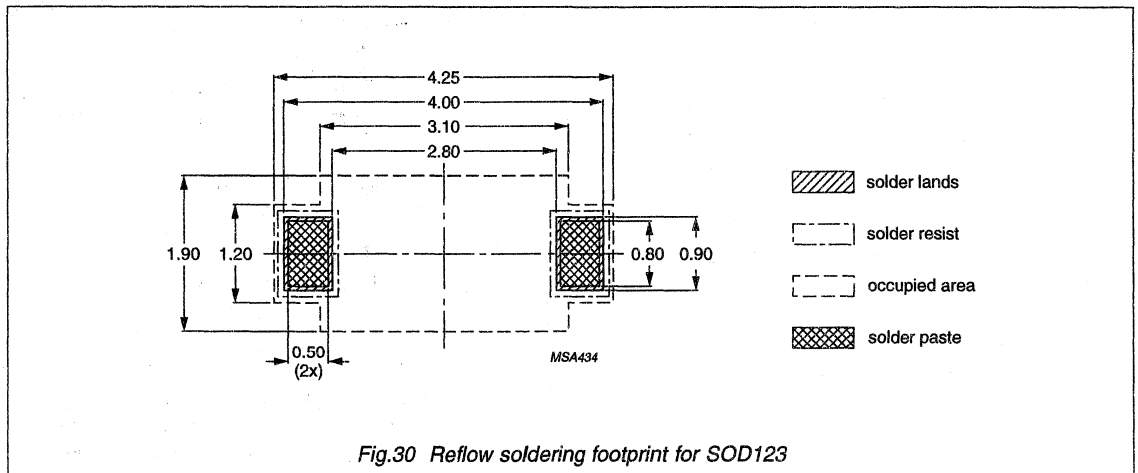
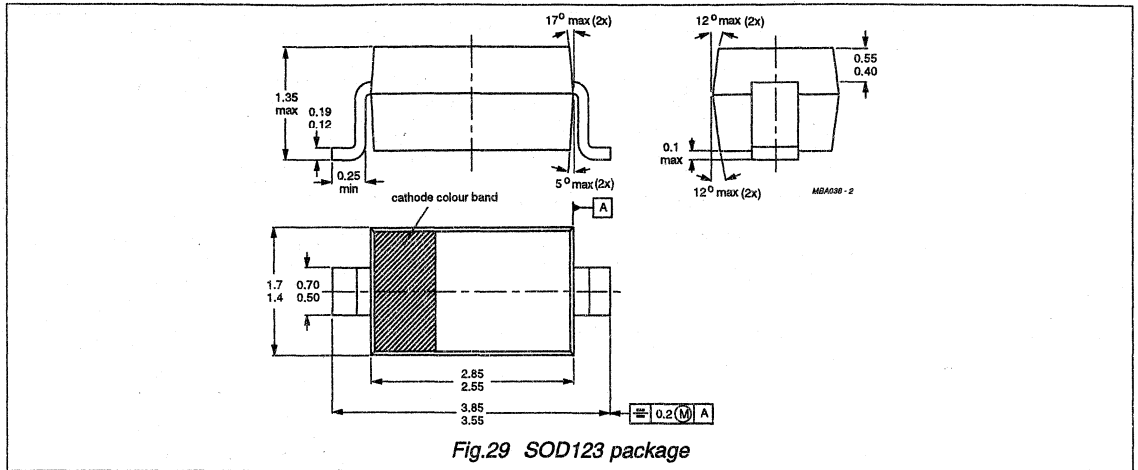
SMD Footprint Design and Soldering Guidelines



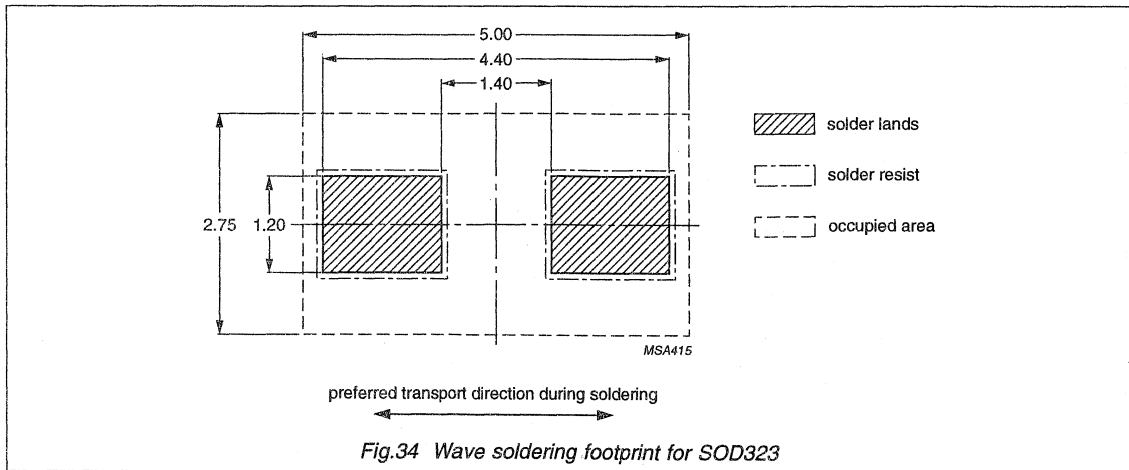
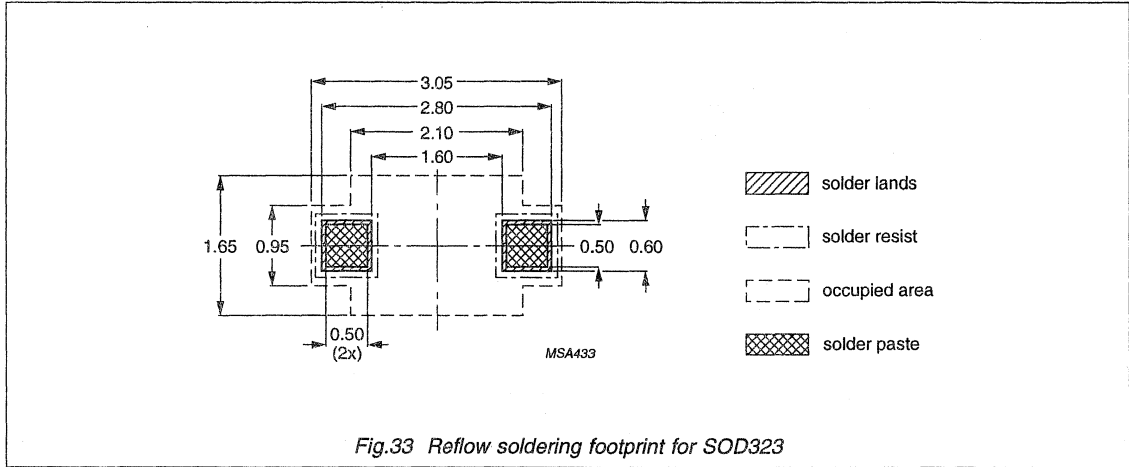
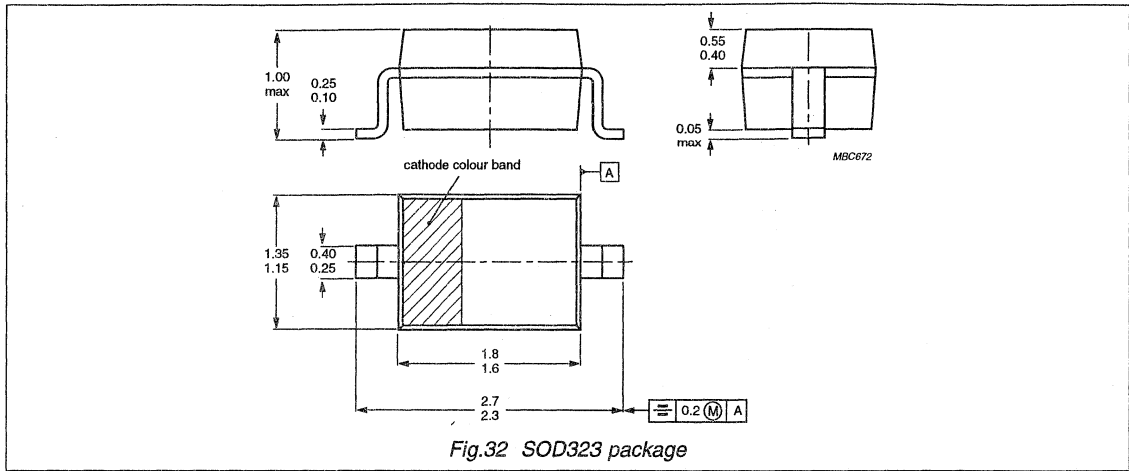
SMD Footprint Design and Soldering Guidelines



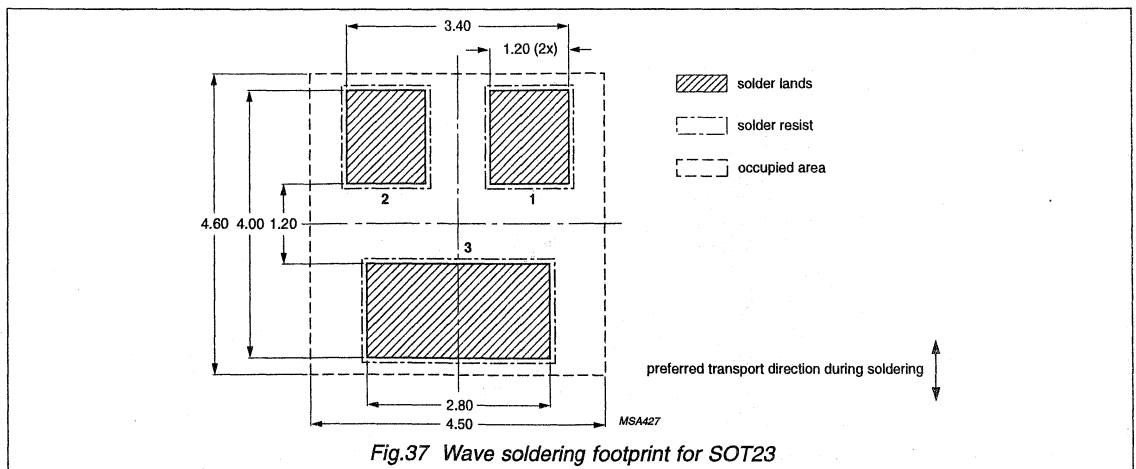
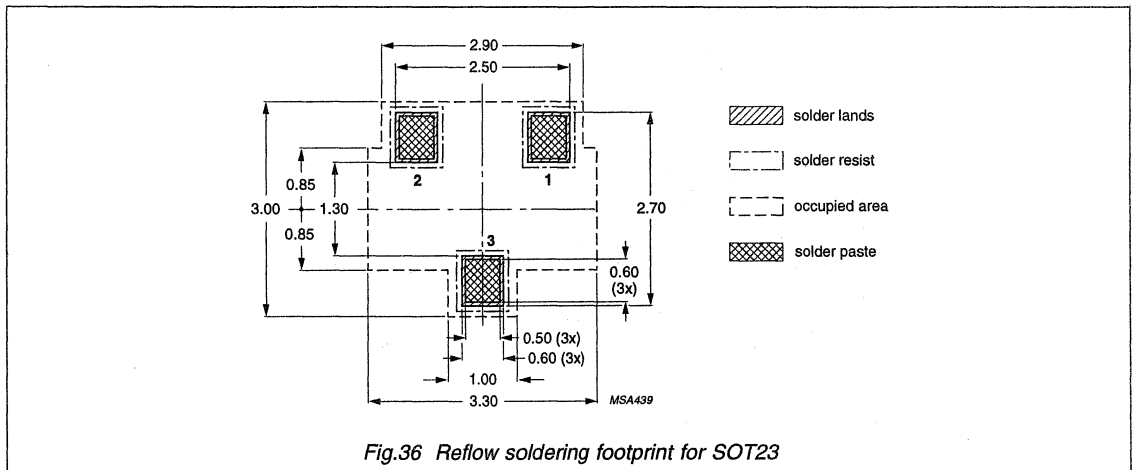
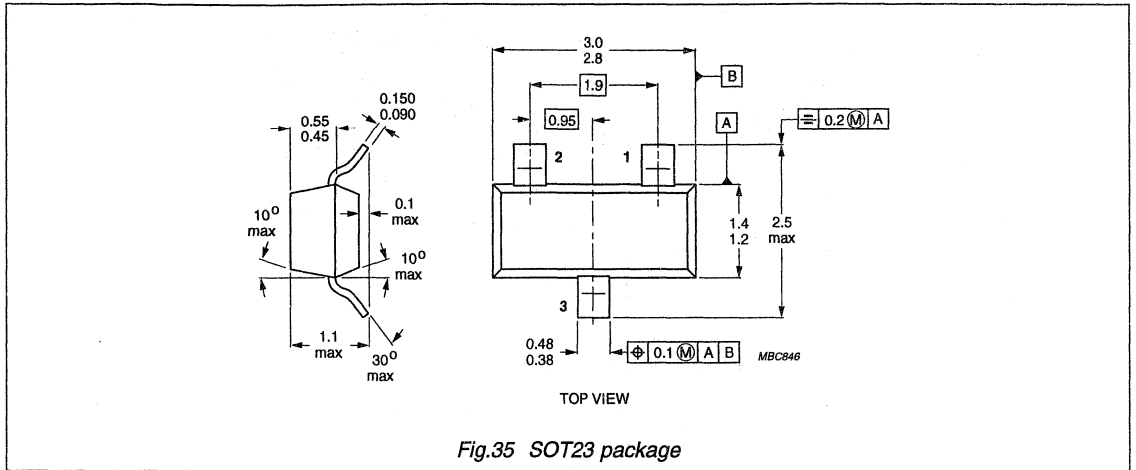
SMD Footprint Design and Soldering Guidelines



SMD Footprint Design and Soldering Guidelines



SMD Footprint Design and Soldering Guidelines



SMD Footprint Design and Soldering Guidelines

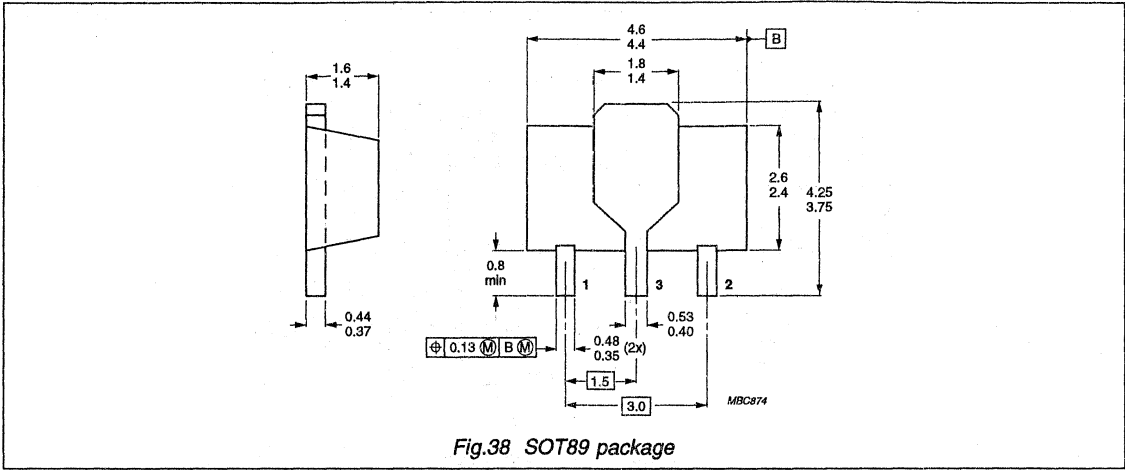


Fig.38 SOT89 package

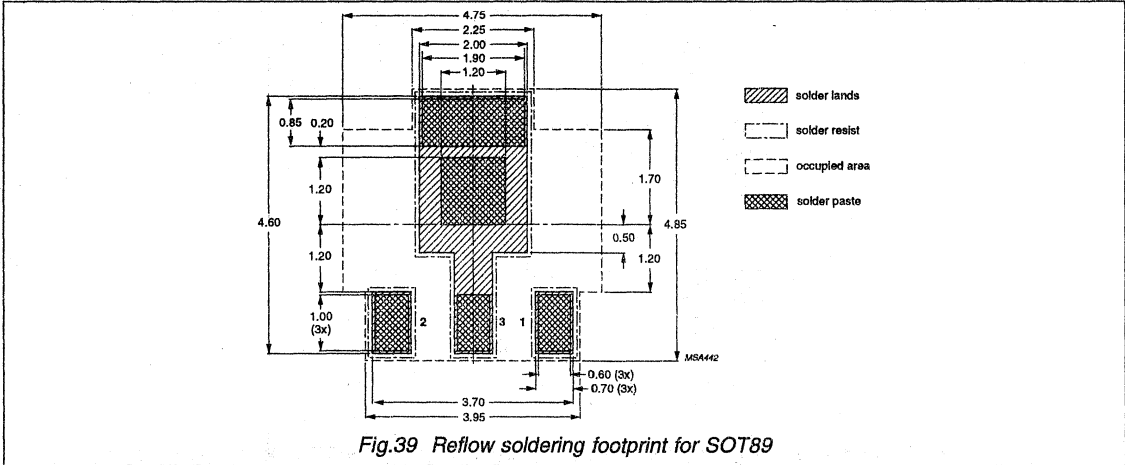


Fig.39 Reflow soldering footprint for SOT89

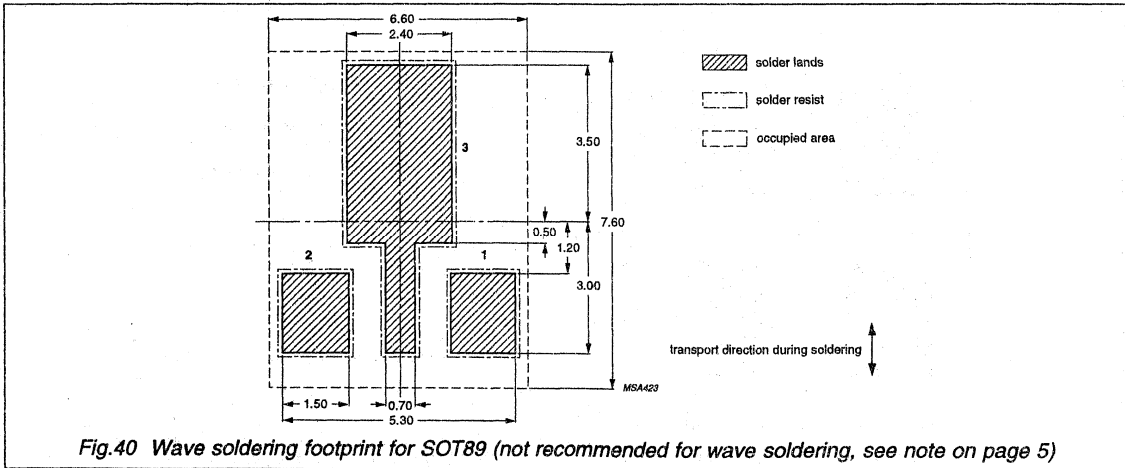
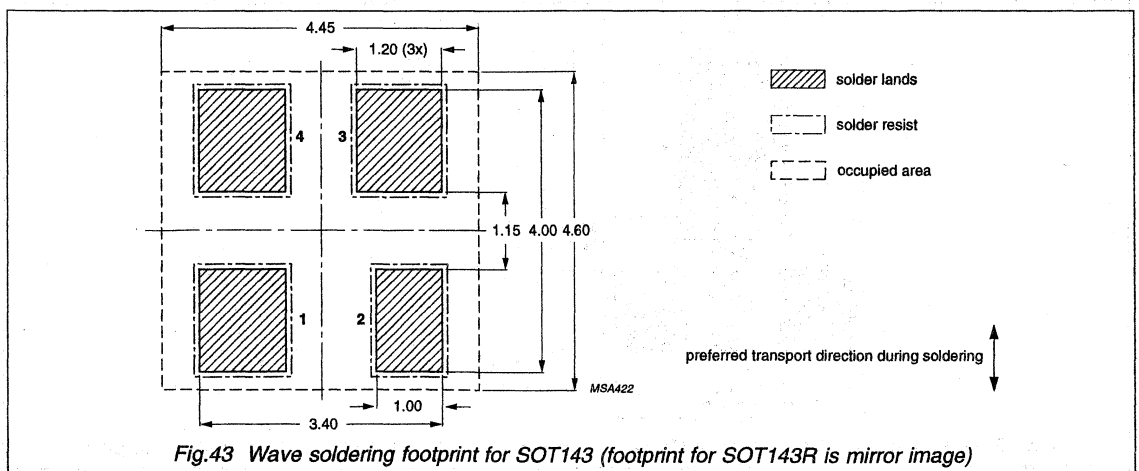
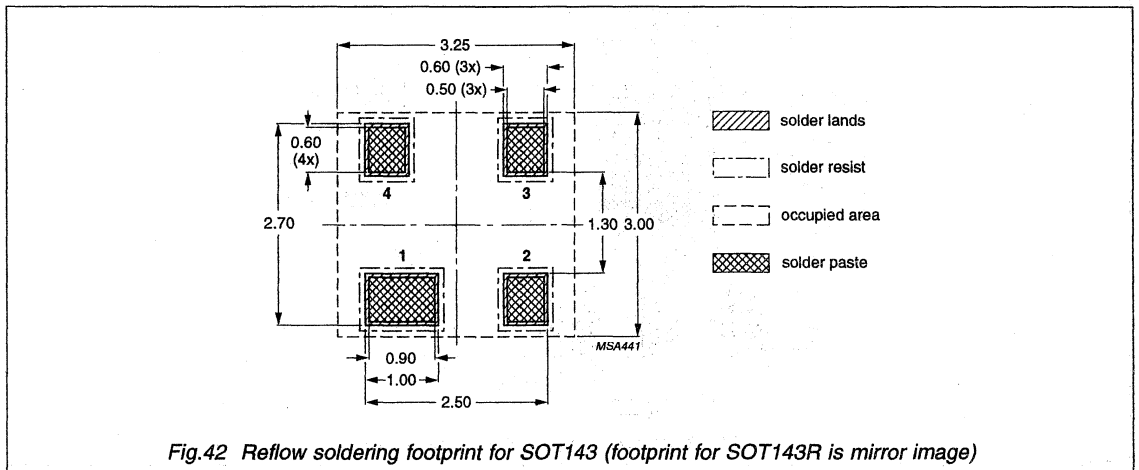
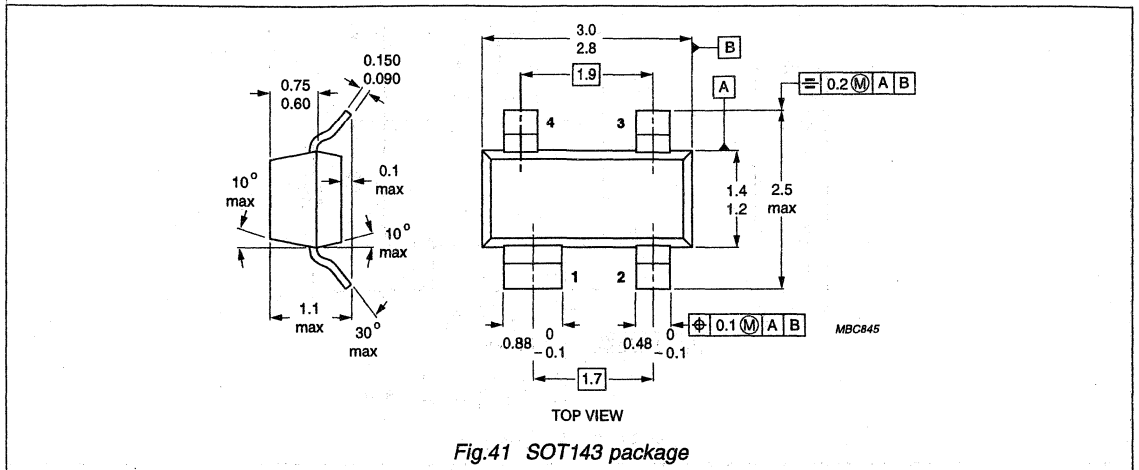


Fig.40 Wave soldering footprint for SOT89 (not recommended for wave soldering, see note on page 5)

SMD Footprint Design and Soldering Guidelines



SMD Footprint Design and Soldering Guidelines

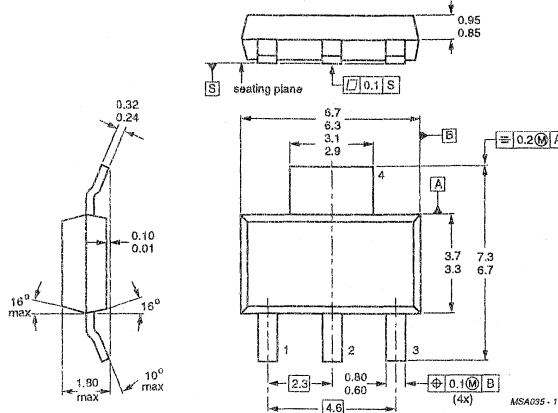


Fig.44 SOT223 package

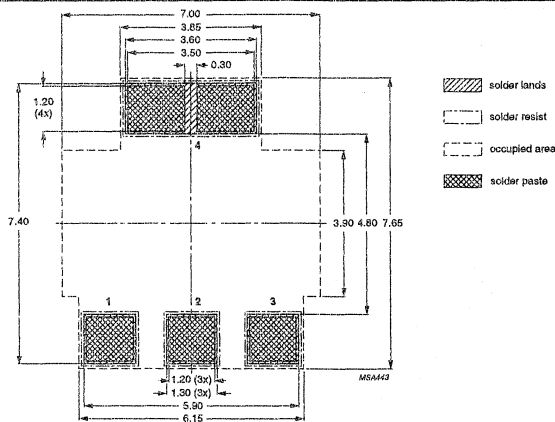


Fig.45 Reflow soldering footprint for SOT223

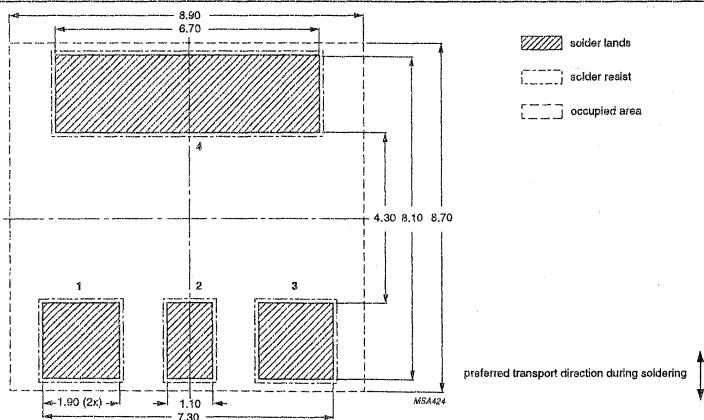
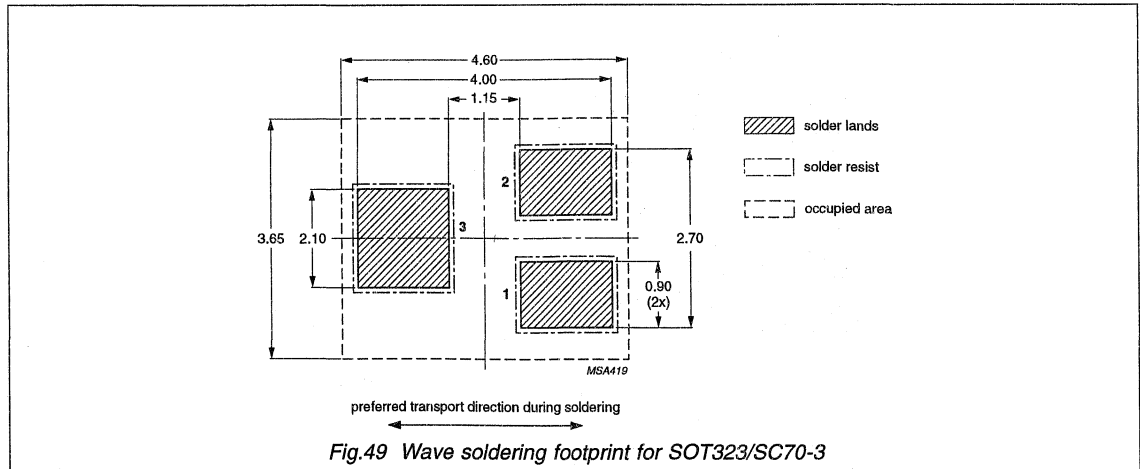
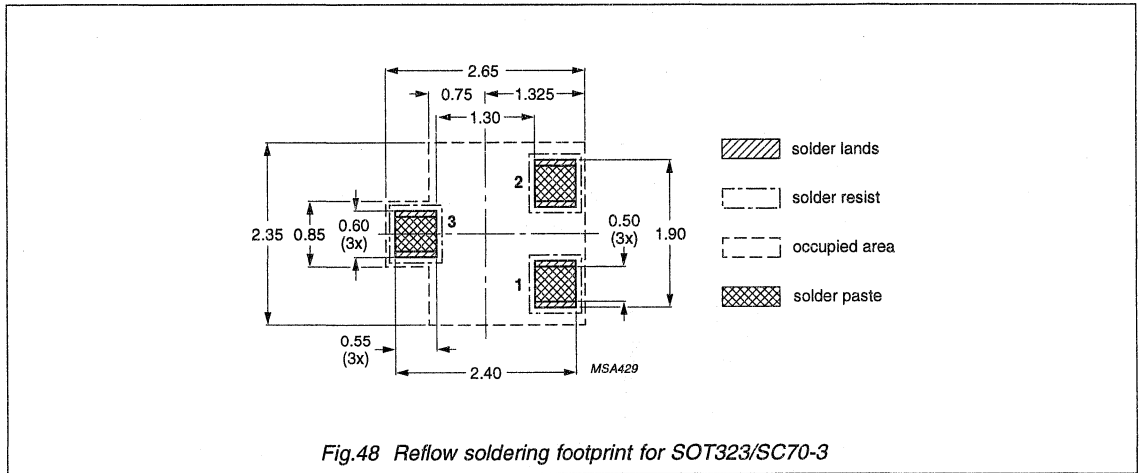
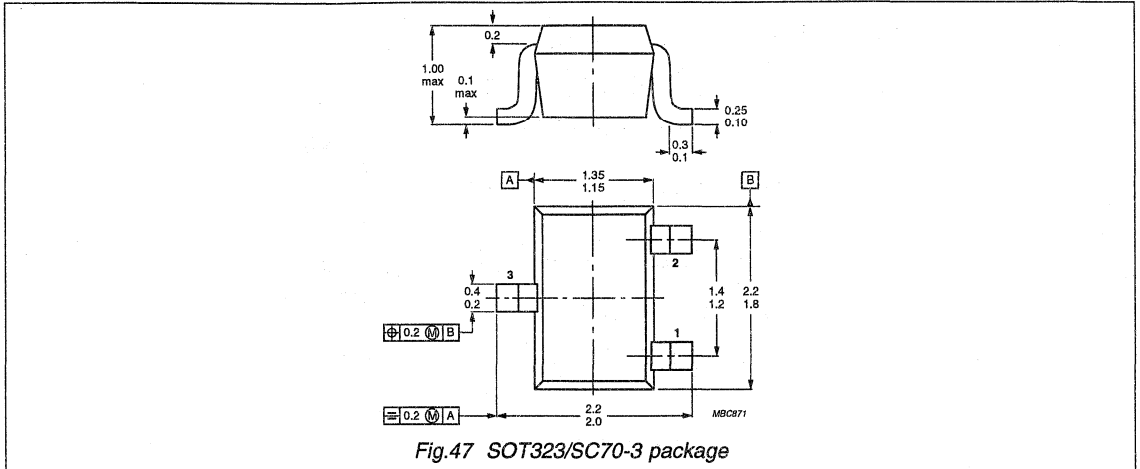


Fig.46 Wave soldering footprint for SOT223

SMD Footprint Design and Soldering Guidelines



SMD Footprint Design and Soldering Guidelines

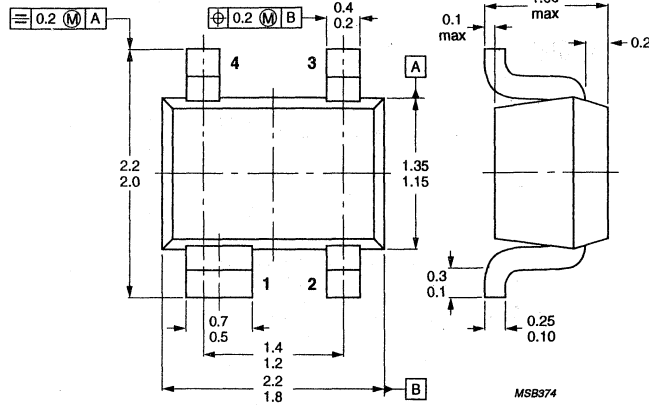


Fig.50 SOT343 package

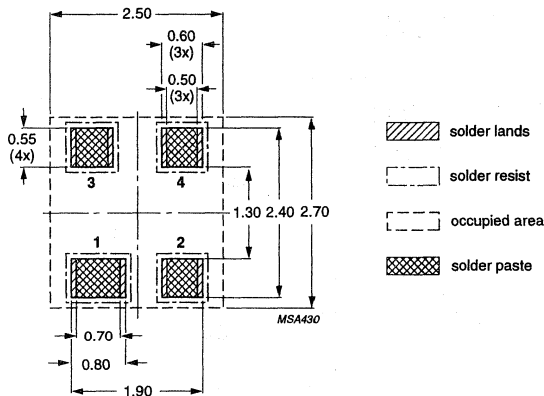


Fig.51 Reflow soldering footprint for SOT343 (footprint for SOT343R is mirror image)

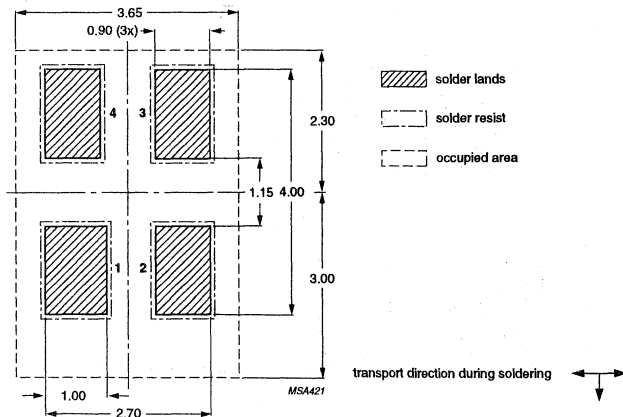
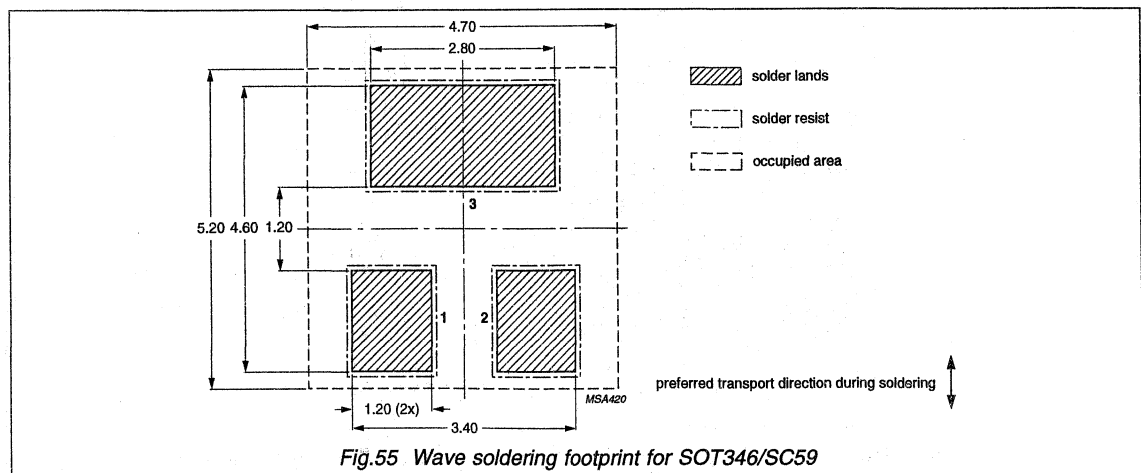
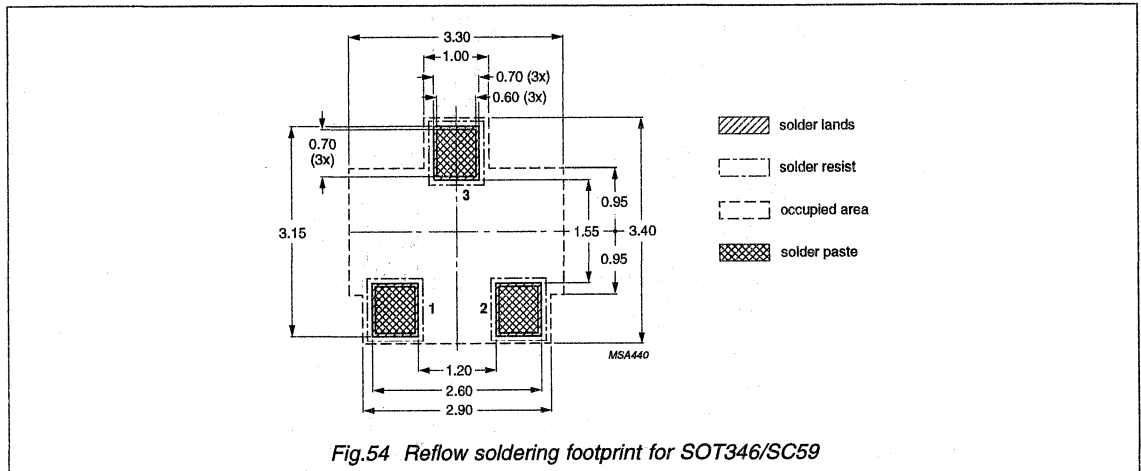
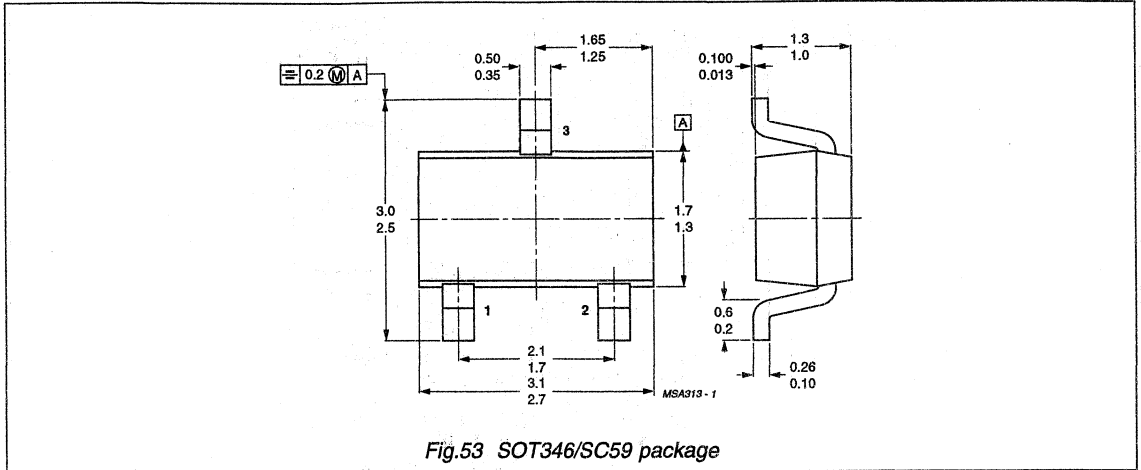


Fig.52 Wave soldering footprint for SOT343 (footprint for SOT343R is mirror image)

SMD Footprint Design and Soldering Guidelines



SMD Footprint Design and Soldering Guidelines

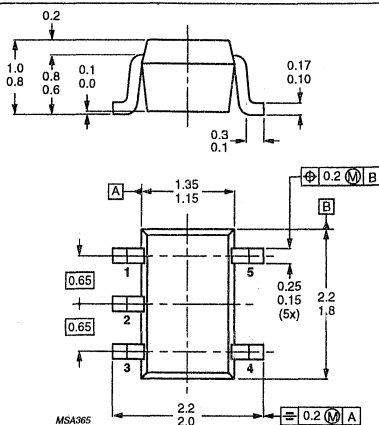


Fig.56 SOT353/SC70-5 package

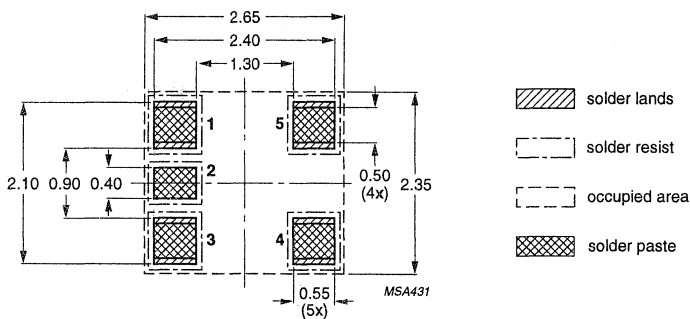


Fig.57 Reflow soldering footprint for SOT353/SC70-5

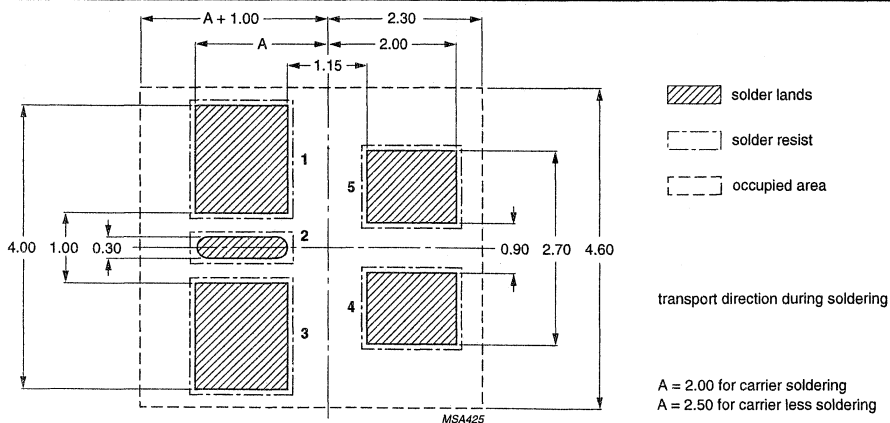


Fig.58 Wave soldering footprint for SOT353/SC70-5

SMD Footprint Design and Soldering Guidelines

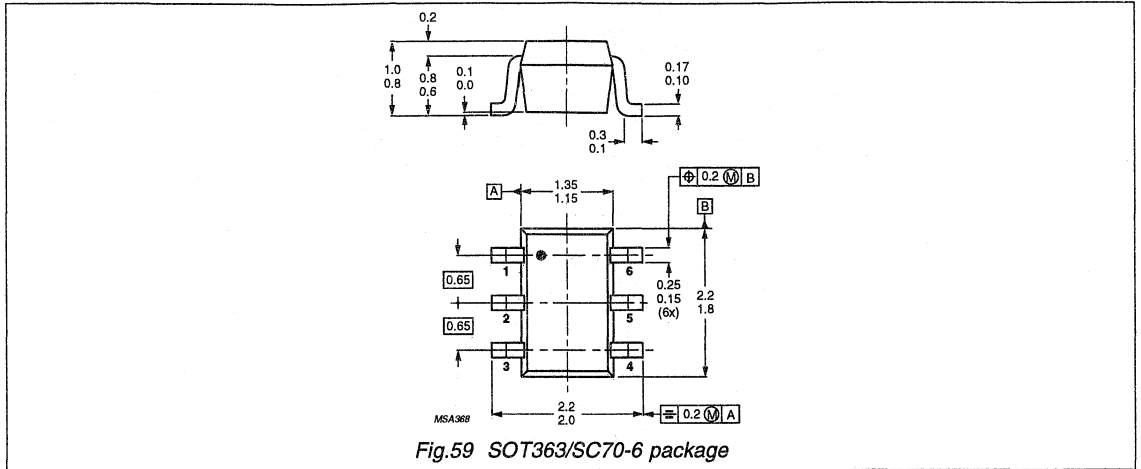


Fig.59 SOT363/SC70-6 package

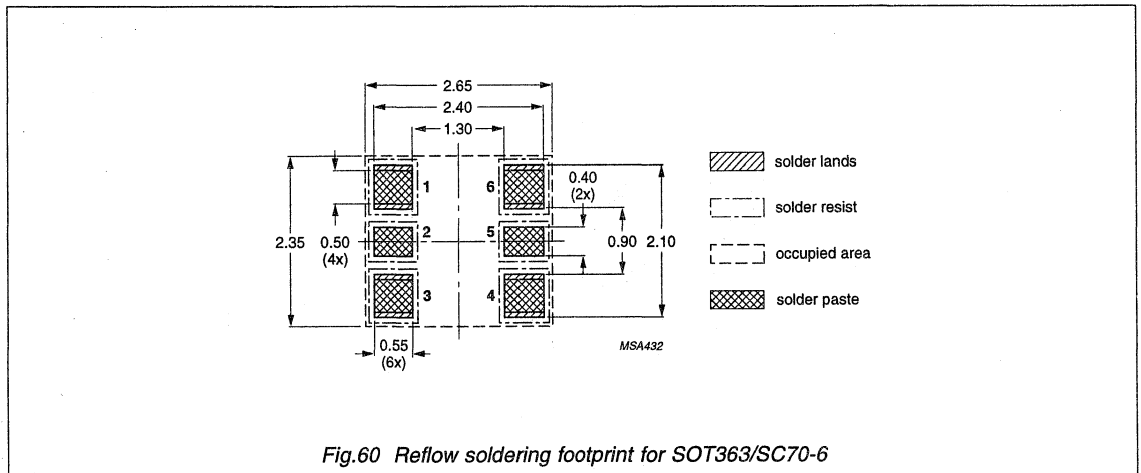


Fig.60 Reflow soldering footprint for SOT363/SC70-6

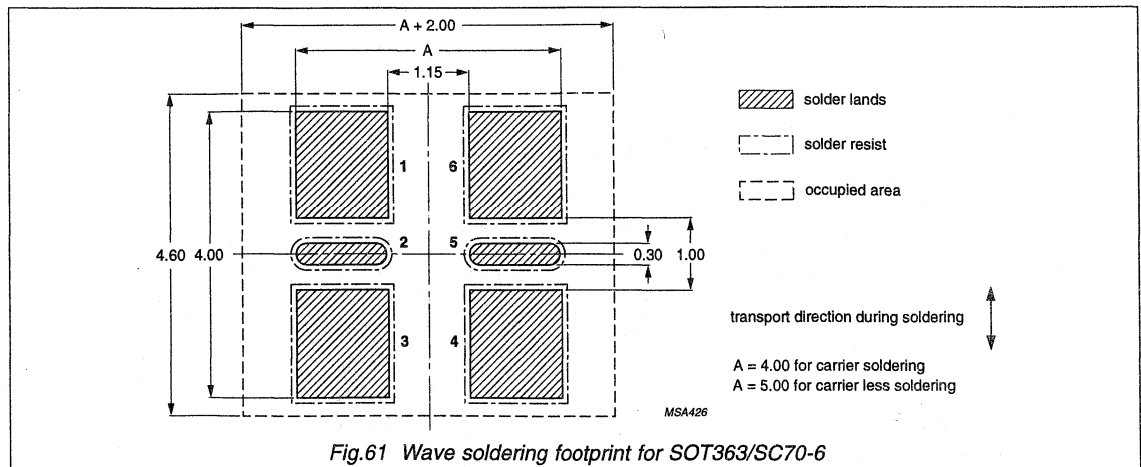


Fig.61 Wave soldering footprint for SOT363/SC70-6

SMD Footprint Design and Soldering Guidelines

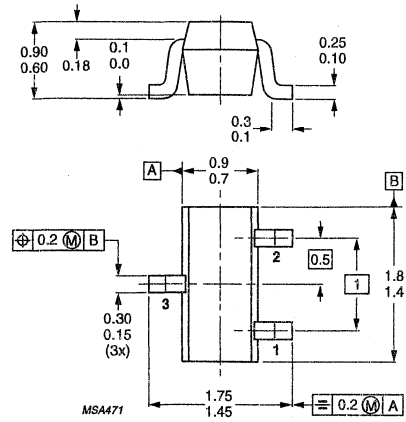


Fig.62 SOT416/SC90 package

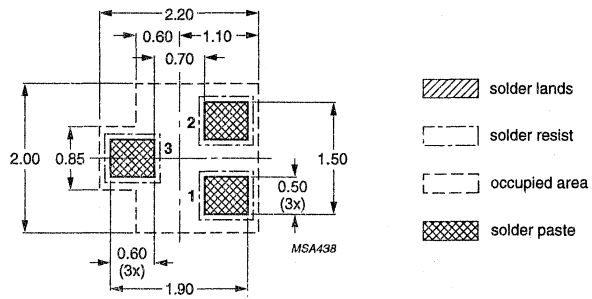


Fig.63 Reflow soldering footprint for SOT416/SC90

Note: The SOT416/SC90 package is not recommended for wave soldering.

Philips Semiconductors – a worldwide company

Argentina IEROD, Av. Juramento 1992 - 14.b,
(1428) BUENOS AIRES, Tel. (541)786 7635, Fax. (541)786 9367

Australia 34 Waterloo Road, NORTH RYDE, NSW 2113,
Tel. (02)805 4455, Fax. (02)805 4466

Austria Triester Str. 64, A-1101 WIEN, P.O. Box 213,
Tel. (01)60 101-1236, Fax. (01)60 101-1211

Belgium Postbus 90050, 5600 PB EINDHOVEN, The Netherlands,
Tel. (31)40-2783749, Fax. (31)40-2788399

Brazil Rua do Rocio 220 - 5th floor, Suite 51,
CEP: 04552-903-SÃO PAULO-SP, Brazil,
P.O. Box 7383 (01064-970),
Tel. (011)821-2333, Fax. (011)829-1849

Canada PHILIPS SEMICONDUCTORS/COMPONENTS:
Tel. (800) 234-7381, Fax. (708) 296-8556

Chile Av. Santa Maria 0760, SANTIAGO,
Tel. (02)773 816, Fax. (02)777 6730

China/Hong Kong 501 Hong Kong Industrial Technology Centre,
72 Tat Chee Avenue, Kowloon Tong, HONG KONG,
Tel. (852)2319 7888, Fax. (852)2319 7700

Colombia IPRELENZO LTDA, Carrera 21 No. 56-17,
77621 BOGOTA, Tel. (571)249 7624/(571)217 4609,
Fax. (571)217 4549

Denmark Prags Boulevard 80, PB 1919, DK-2300
COPENHAGEN S, Tel. (032)88 2636, Fax. (031)57 1949

Finland Sinikalliontie 3, FIN-02630 ESPOO,
Tel. (358)0-615 800, Fax. (358)0-61580 920

France 4 Rue du Port-aux-Vins, BP317,
92156 SURESNES Cedex,
Tel. (01)4099 6161, Fax. (01)4099 6427

Germany P.O. Box 10 51 40, 20035 HAMBURG,
Tel. (040)23 53 60, Fax. (040)23 53 60 00

Greece No. 15, 25th March Street, GR 17778 TAVROS,
Tel. (01)4894 339/4894 911, Fax. (01)4814 240

India Philips INDIA Ltd, Shivsagar Estate, A Block,
Dr. Annie Besant Rd. Worli, Bombay 400 018
Tel. (022)4938 541, Fax. (022)4938 722

Indonesia Philips House, Jalan H.R. Rasuna Said Kav. 3-4,
P.O. Box 4252, JAKARTA 12950,
Tel. (021)5201122, Fax. (21)5205189

Ireland Newstead, Clonskeagh, DUBLIN 14,
Tel. (01)7640 000, Fax. (01)7640 200

Italy PHILIPS SEMICONDUCTORS S.r.l.,
Piazza IV Novembre 3, 20124 MILANO,
Tel. (0039)2 6752 2531, Fax. (0039)2 6752 2557

Japan Philips Bldg 13-37, Kohnan 2-chome, Minato-ku,
TOKYO 108, Tel. (03)3740 5130, Fax. (03)3740 5077

Korea Philips House, 260-199 Itaewon-dong,
Yongsan-ku, SEOUL, Tel. (02)709-1412, Fax. (02)709-1415

Malaysia No. 76 Jalan Universiti, 46200 PETALING JAYA,
SELANGOR, Tel. (03)750 5214, Fax. (03)757 4880

Mexico 5900 Gateway East, Suite 200, EL PASO,
TEXAS 79905, Tel. 9-5(800)234-7831, Fax. (708)296-8556

Netherlands Postbus 90050, 5600 PB EINDHOVEN, Bldg. VB,
Tel. (040)2783749, Fax. (040)2788399

New Zealand 2 Wagener Place, C.P.O. Box 1041, AUCKLAND,
Tel. (09)849-4160, Fax. (09)849-7811

Norway Box 1, Manglerud 0612, OSLO,
Tel. (022)74 8000, Fax. (022)74 8341

Pakistan Philips Electrical Industries of Pakistan Ltd.,
Exchange Bldg. ST-2/A, Block 9, KDA Scheme 5, Clifton,
KARACHI 75600, Tel. (021)587 4641-49,
Fax. (021)577035/5874546

Philippines Philips Semiconductors Philippines Inc.,
106 Valero St. Salcedo Village, P.O. Box 2108 MCC,
MAKATI, Metro MANILA,
Tel. (63) 2 816 6380, Fax. (63) 2 817 3474

Portugal PHILIPS PORTUGUESA, S.A.,
Rua dr. António Loureiro Borges 5, Arquiparque - Miraflores,
Apartado 300, 2795 LINDA-A-VELHA,
Tel. (01)4163160/4163333, Fax. (01)4163174/4163366

Singapore Lorong 1, Toa Payoh, SINGAPORE 1231,
Tel. (65)350 2000, Fax. (65)251 6500

South Africa S.A. PHILIPS Pty Ltd.,
195-215 Main Road Martindale, 2092 JOHANNESBURG,
P.O. Box 7430 Johannesburg 2000,
Tel. (011)470-5911, Fax. (011)470-5494

Spain Balmes 22, 08007 BARCELONA,
Tel. (03)301 6312, Fax. (03)301 4243

Sweden Kottbygatan 7, Akalla. S-16485 STOCKHOLM,
Tel. (0)8-632 2000, Fax. (0)8-632 2745

Switzerland Allmendstrasse 140, CH-8027 ZÜRICH,
Tel. (01)488 2211, Fax. (01)481 77 30

Taiwan PHILIPS TAIWAN Ltd., 23-30F, 66, Chung Hsiao West
Road, Sec. 1, Taipei, Taiwan ROC, P.O. Box 22978,
TAIPEI 100, Tel. (886) 2 382 4443, Fax. (886) 2 382 4444

Thailand PHILIPS ELECTRONICS (THAILAND) Ltd.,
209/2 Sanpavuth-Bangna Road Prakanong, Bangkok 10260,
THAILAND, Tel. (66) 2 745-4090, Fax. (66) 2 398-0793

Turkey Talatpasa Cad. No. 5, 80640 GÜLTEPE/ISTANBUL,
Tel. (0212)279 2770, Fax. (0212)282 6707

Ukraine Philips UKRAINE,
2A Akademika Koroleva str., Office 165, 252148 KIEV,
Tel. 380-44-4760297, Fax. 380-44-4766991

United Kingdom Philips Semiconductors LTD.,
276 Bath Road, Hayes, MIDDLESEX UB3 5BX,
Tel. (0181)730-5000, Fax. (0181)754-8421

United States 811 East Arques Avenue, SUNNYVALE,
CA 94088-3409, Tel. (800)234-7381, Fax. (708)296-8556

Uruguay Coronel Mora 433, MONTEVIDEO,
Tel. (02)70-4044, Fax. (02)92 0601

Internet: <http://www.semiconductors.philips.com/ps/>

For all other countries apply to: Philips Semiconductors,
Marketing Communications, Building BE-p,
P.O. Box 218, 5600 MD EINDHOVEN, The Netherlands,
Telex 35000 phtnl, Fax. +31-40-2724825

SCDH47 © Philips Electronics N.V. 1996

All rights are reserved. Reproduction in whole or in part is prohibited
without the prior written consent of the copyright owner.

The information presented in this document does not form part of any
quotation or contract, is believed to be accurate and reliable and may be
changed without notice. No liability will be accepted by the publisher for
any consequence of its use. Publication thereof does not convey nor
imply any license under patent- or other industrial or intellectual property
rights.

Printed in The Netherlands

budgetnr/printrun/ed/pp1

Document order number:

Date of release: 1996 Feb 22

9397 750 00505

**Philips
Semiconductors**



PHILIPS

Let's make things better.