



1. Introduction

There are two basic forms of electronic component construction, those with leads for through-hole mounting and those for surface mounting. Through-hole mounting gives a very rugged construction and uses well established soldering methods. Surface mounting has the advantages of high packing density plus high-speed automated assembly. With all components, excessive forces or heat can cause serious damage and should always be avoided.

2. Axial and radial leaded diodes

The following general rules are for the safe handling and soldering of axial and radial leaded diodes. Special rules for types may apply and, for these, instructions are given in the individual data sheets.

2.1 Handling

- Avoid perpendicular forces on the body of the diode
- Avoid sudden forces on the leads or body. These forces are often much greater than allowed
- Avoid high acceleration because of any shock, e.g. dropping the device on a hard surface
- During bending, support the leads between body or stud and the bending point
- During the bending process, axial forces on the body must not exceed 20 N
- Bending the leads through 90° is allowed at any distance from the body when it is possible to support the leads during bending without contacting the body or welding
- Twisting the leads is allowed at any distance from the body or stud only if the lead is properly clamped between body or stud and the twisting point
- Straightening bent leads is allowed only:
 - ◆ if the applied pulling force in the axial direction does not exceed 20 N
 - ◆ the total pull duration is no longer than 5 s

2.2 Soldering

- Avoid any force on the body or leads during or immediately after soldering
- Do not correct the position of an already soldered device by pushing, pulling or twisting the body
- Avoid fast cooling after soldering
- The maximum allowable soldering time is determined by:
 - Package type
 - Mounting environment
 - Soldering method
 - Soldering temperature
- Distance between the point of soldering and the seal of the component body

The maximum permissible temperature of the solder is 260 °C; this temperature must not be in contact with the joint for more than 5s. The total contact time of successive solder waves must not exceed 5s.

The component may be mounted up to the seating plane, but the temperature of the plastic body must not exceed the specified storage maximum. If the PCB has been preheated, forced cooling may be necessary immediately after soldering to keep the temperature within the permissible limit.

2.3 Mounting

If the rules for handling and soldering are observed, the following mounting or process methods are allowed:

- Preheating of the printed-wiring board before soldering up to a maximum of 100 °C
- Flat mounting with the diode body in direct contact with the printed-wiring board with or without metal tracks on both sides and/or plated-through holes
- Flat mounting with the diode body in direct contact with hot spots or hot tracks during soldering
- Upright mounting with the diode body in direct contact with the printed-wiring board if the body is not in contact with metal tracks or plated-through holes

2.4 Repairing soldered joints

Apply the soldering iron to the component pin(s) below the seating plane, or not more than 2 mm above it. If the temperature of the soldering iron bit is below 300 °C, it may remain in contact for up to 10s. If it is over 300 °C but below 400 °C, it may only remain in contact for up to 5s.

3. Surface mount devices (SMDs)

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. Consequently, when designing 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 component 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 on one substrate (a mixed print).

The mix of components affects the soldering methods that can be applied. A substrate having SMDs mounted on one or both sides, but no through-hole components is likely to be suitable for reflow or wave soldering. A double-sided mixed print that has through-hole components and some SMDs on one side and densely packed SMDs on the other normally undergoes a sequential combination of reflow and wave soldering. When the mixed print has only through-hole components on one side and all SMDs on the other, wave soldering is usually applied.

To help with your circuit board design, this guideline gives an overview of both reflow and wave soldering methods and is followed by some useful hints on hand soldering for repair purposes, and the recommended footprints for SMD discrete semiconductor packages.

3.1 Reflow soldering process

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

- Applying solder paste to the PCB
- Component placement
- Reflow soldering.

3.1.1 Applying solder paste to PCB

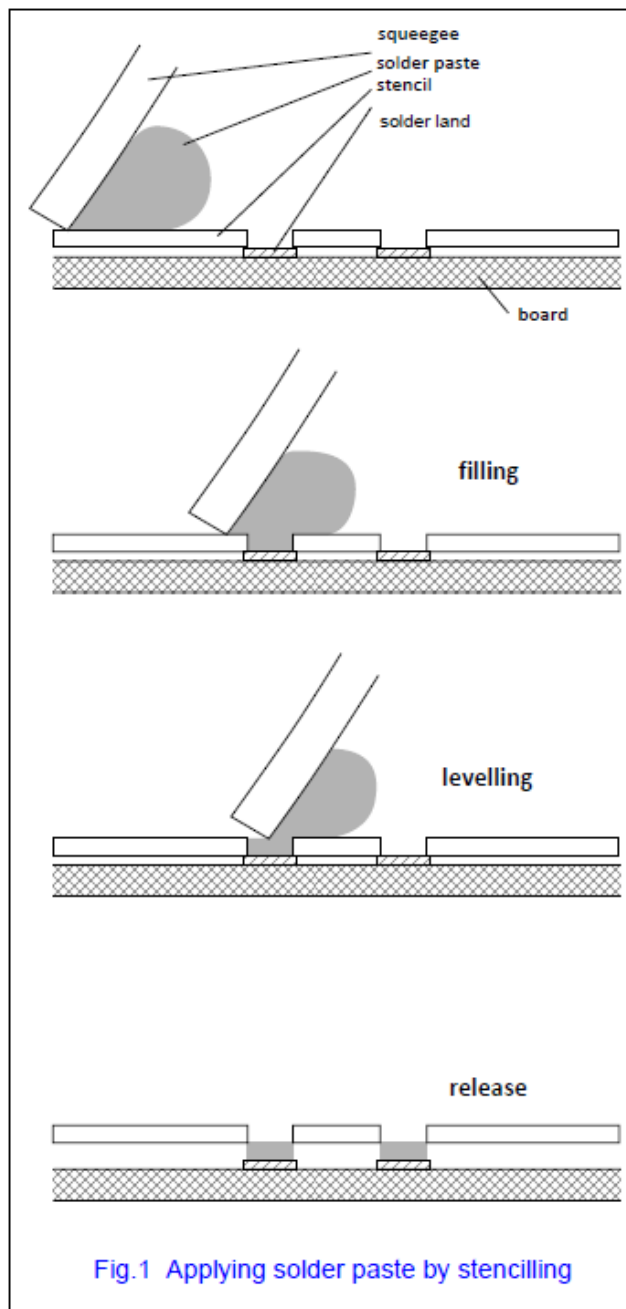
Solder paste can be applied to the PCBs solder lands by one of either three methods: dispensing, screen or stencil printing.

Dispensing is flexible but is slow, and only suitable for pitches of 0.65mm 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 300µm and wider.

Stencil printing is like 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 to 200µm 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.

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



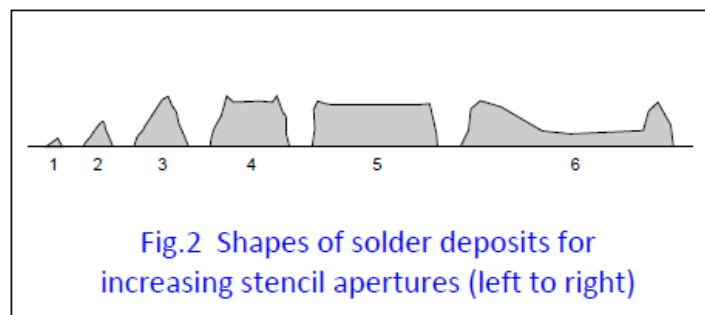
3.1.2 Stencil printing

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

The amount of solder paste used must be enough 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). 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. 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 enough 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. Generally, component placement should be within four hours after the paste printing process.

3.1.3 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 must be used. The footprints given in this note were designed for application by both types of squeegee.

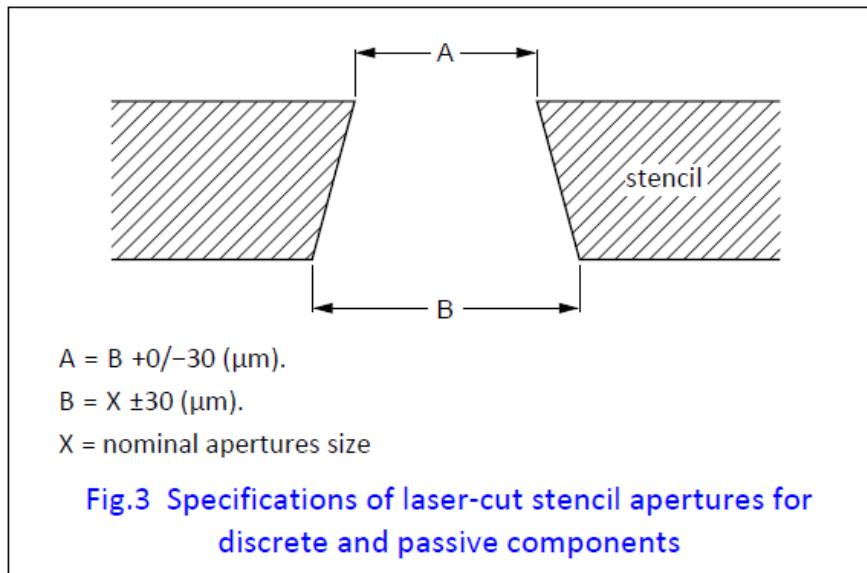
3.1.4 Stencil apertures

Stencil apertures can be made by either:

- Electro-forming
- Etching
- Laser cutting

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

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



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.

3.1.5 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
- Enough tackiness to hold the components
- Removal of oxides during reflow soldering

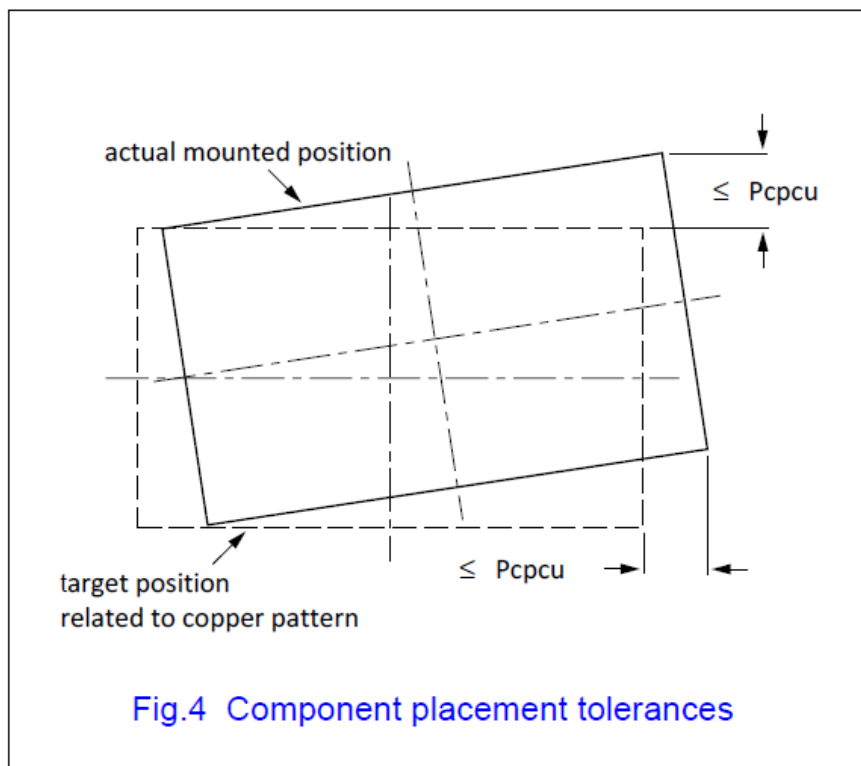
A suitable solder paste type may have the following compositions:

- Sn62Pb36Ag2
- Sn63Pb37
- Sn60Pb40

3.2 Component placement

The position of the component with respect to the solder lands is an important factor in the outcome 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 (see Fig.4).



A maximum placement deviation (P) of 0.25mm 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.

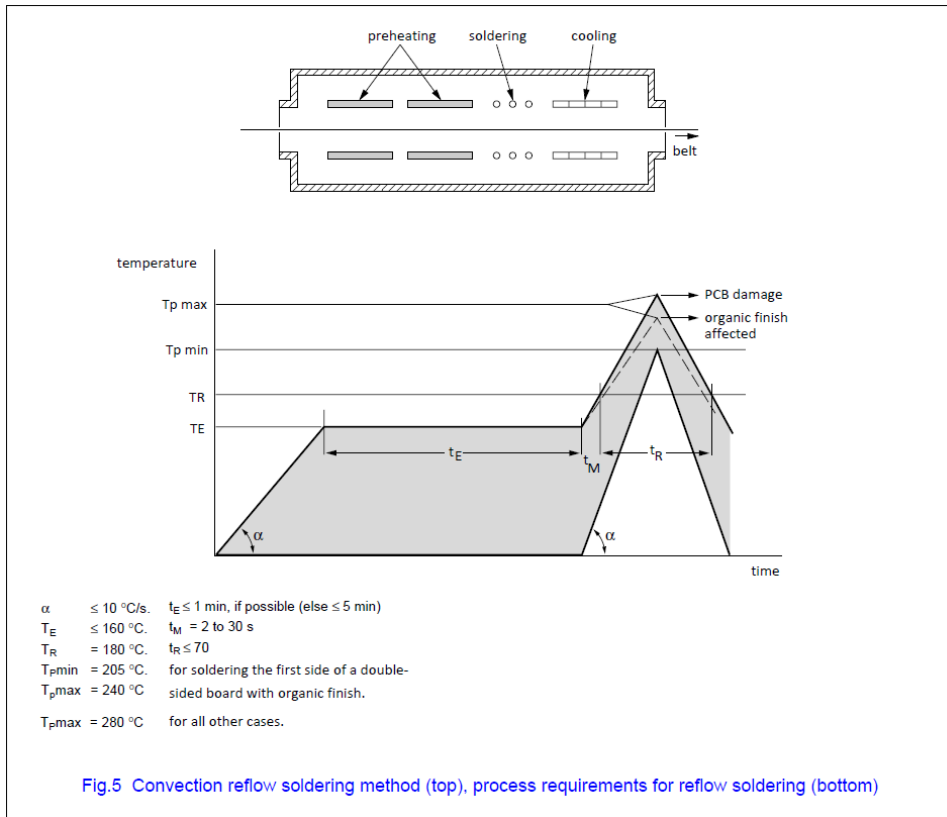
3.3 Convection reflow soldering

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 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. Fig. 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.

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 must 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 must be mounted at the coldest and warmest location and temperature profiles measured.

3.4 Double-wave soldering process

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

- Applying adhesive
- Component placement
- Curing adhesive
- Wave soldering process

3.4.1 Adhesive application

To hold SMDs on the board during wave soldering, it is 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 and position of the dots per component
- Nozzle outlet diameter
- Adhesive type

3.4.2 Adhesive volume

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 (see Fig.6).

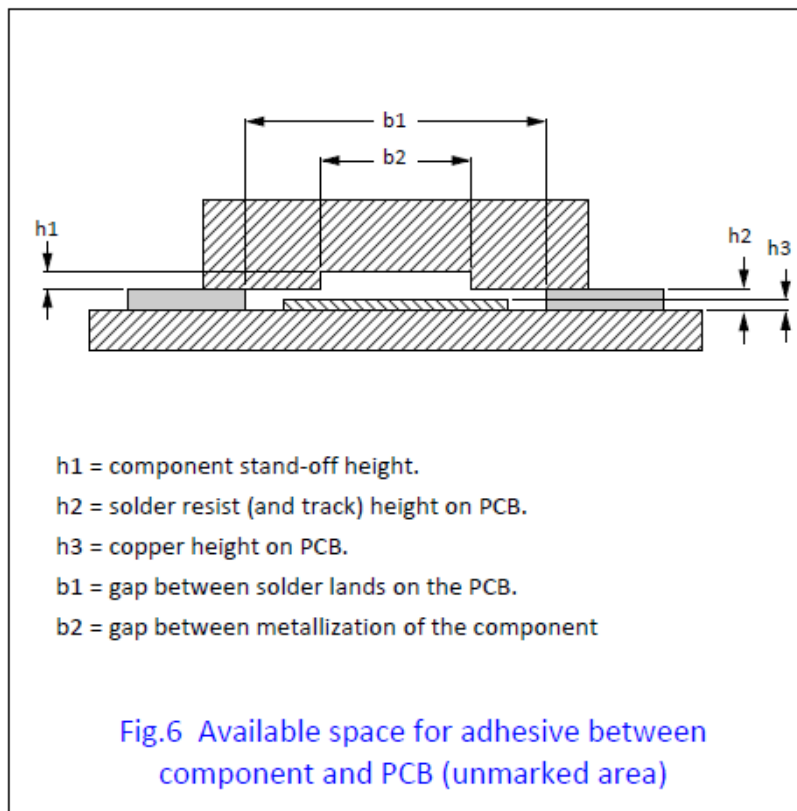
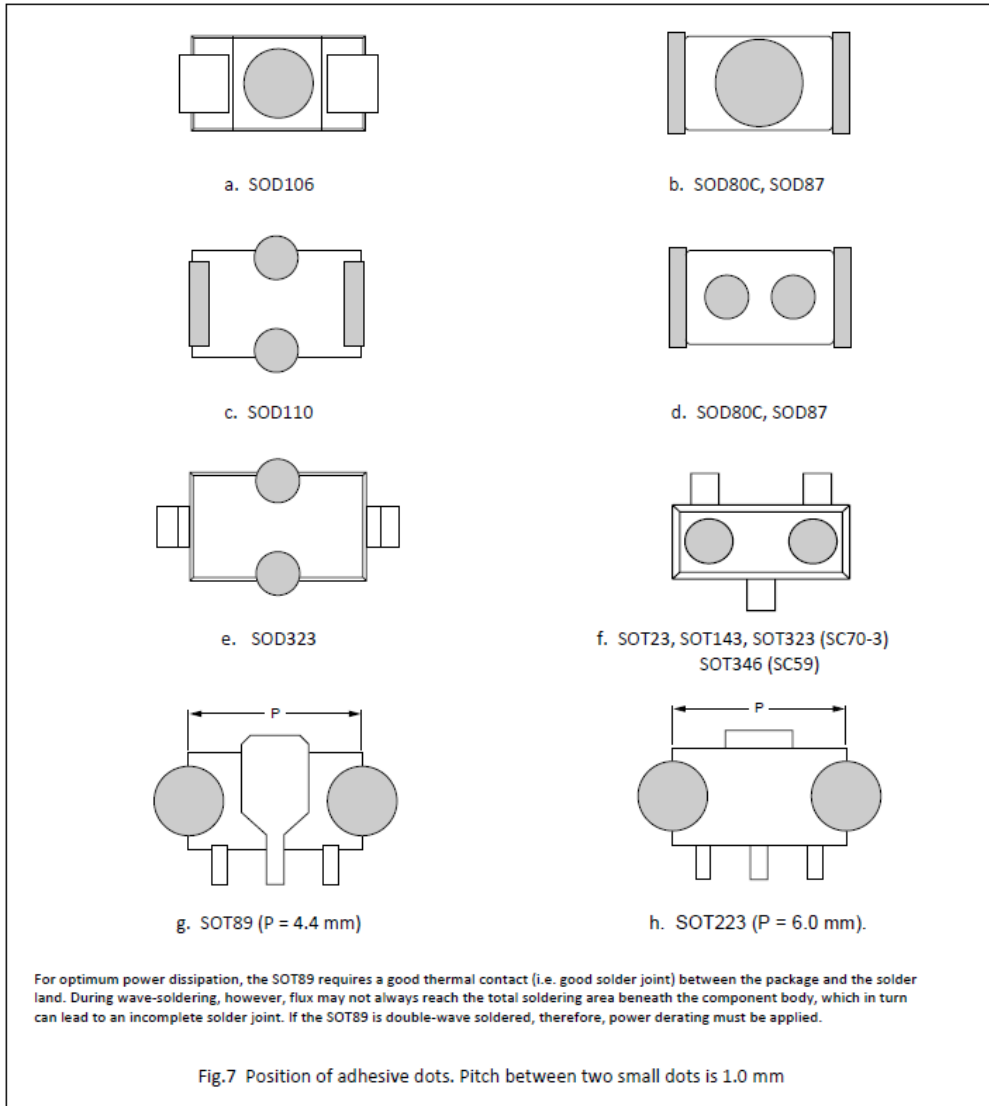


Table 1 gives guidelines for volumes of adhesive dots per package

The spreading in volumes should be within 15%.



3.4.3 Adhesive dots - number, position and volume per component

Figure 7 shows the recommended positions and numbers of adhesive dots for a variety of packages. SOD106, SOT89 and SOT223 packages require much larger adhesive dots compared with those for other components. SOD80C and SOD87 packages can have one large adhesive dot (recommended) or two smaller adhesive dots.

3.4.4 Adhesive nozzle outlet diameter

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

3.4.5 Adhesive types

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.

3.4.6 Adhesive smear prevention

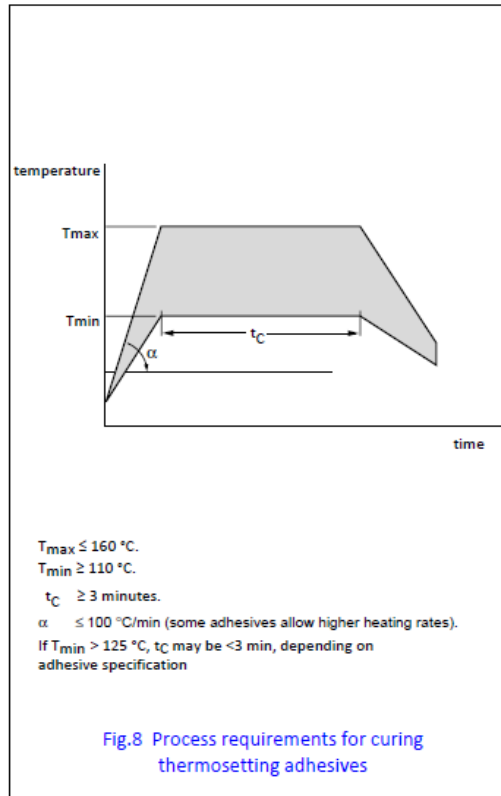
Positioning components on the PCB is similar in practice to that of reflow soldering. To prevent component shift and smearing of the adhesive, board support is important while placing components. This is particularly important when placing the SOD106 package.

3.4.7 Adhesive curing

To provide enough 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.



3.4.8 Adhesive bonding strength checks

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.

Table 2 Bonding strength requirements

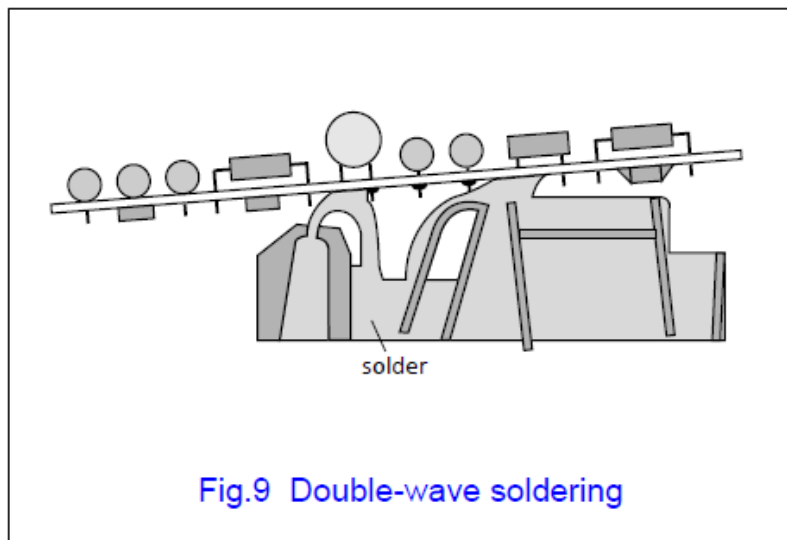
COMPONENT	MINIMUM BONDING STRENGTH (cNcm)	TARGET BONDING STRENGTH (cNcm)
SOD323, SOD110, SOT323 (SC70-3)	110	250
SOD80C, SOD87	200	350
SOT23, SOT346 (SC59), SOT143	150	250

3.5 Wave soldering process

PCBs can be wave soldered after applying adhesive, placing the component on the PCB and curing. The wave soldering process is basically built up from three sub-processes. These are:

- Fluxing
- Preheating
- (Double) wave soldering

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).



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.

3.5.1 Fluxing

Fluxing is necessary to promote wetting both 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 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 (corrosive highly activated fluxes should always be removed)
- The required appearance of the board after soldering
- Customer requirements

3.5.2 Preheating

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

3.5.3 (Double) wave soldering

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 1.6mm thick PCB.

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 (double sided overflow): height with respect to underside of the board (mm) relative stream velocity with respect to the board	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	

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.

3.6 Hand soldering microminiature components

It is possible to solder microminiature components with a light-weight hand-held soldering iron, but this method has obvious drawbacks and should be restricted to laboratory use and/or incidental repairs on production circuits:

- Hand-soldering is time-consuming and therefore expensive
- The component cannot be positioned accurately, and connecting tags may meet the substrate and damage it
- There is a risk of breaking the substrate and internal connections in the component could be damaged
- The component package could be damaged by the iron

3.7 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 about 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

3.7.1 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.

3.7.2 Good wetting

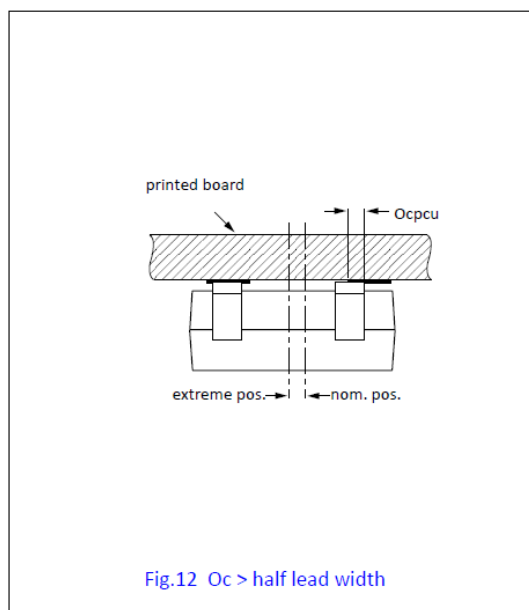
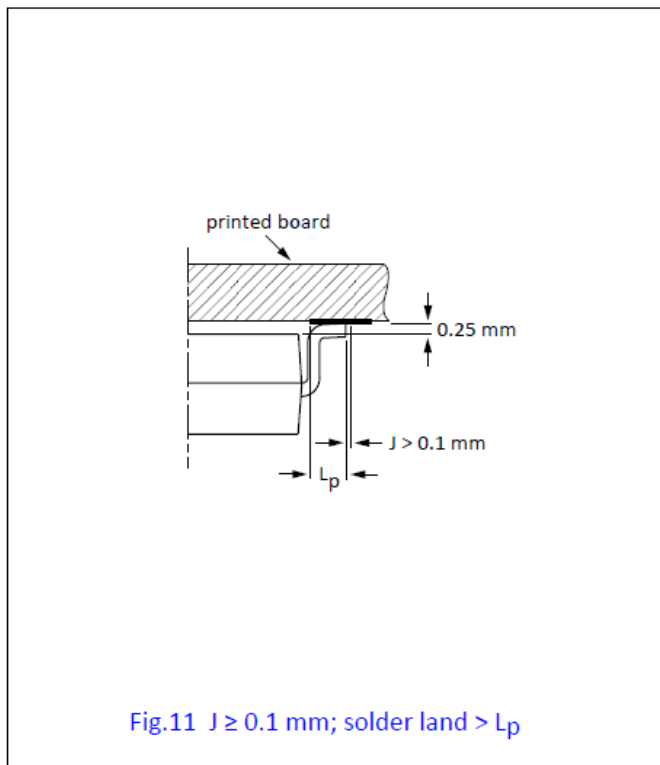
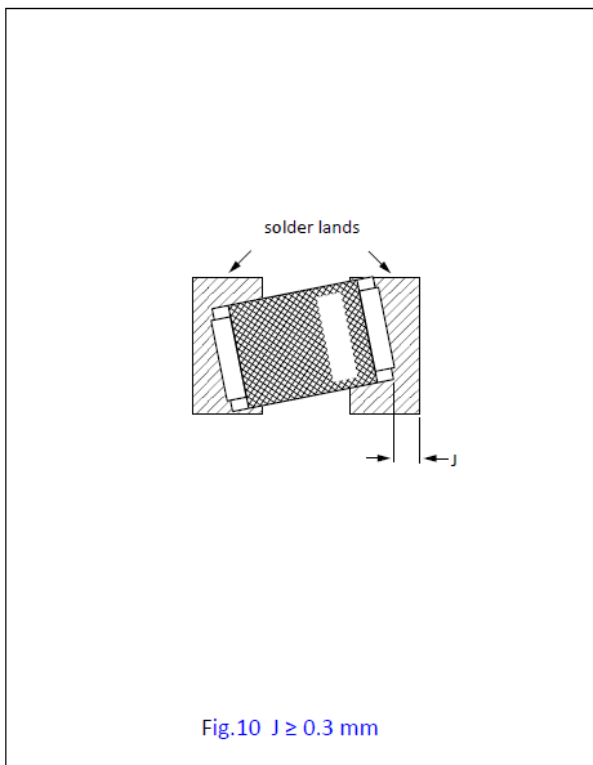
This produces an even flow of solder over the surface land and component lead and thinning towards the edges of the joint. The metallic interaction that takes place during soldering should give a smooth, unbroken, adherent layer of solder on the joint.

3.7.3 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.

3.7.4 Smooth, continuous joint surface

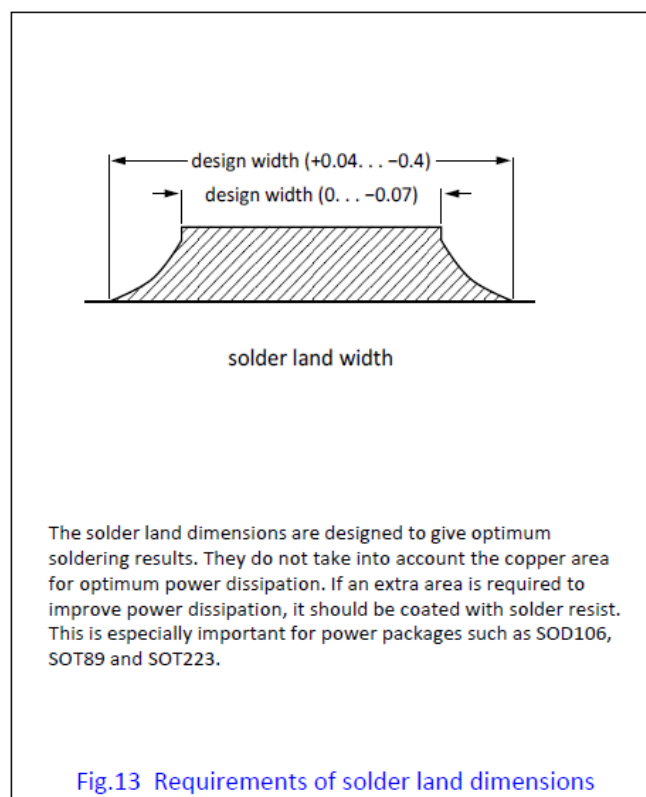
The surface of the solder should be smooth and continuous. Small irregularities on the solder surface are acceptable, but cracks are unacceptable.



4. 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



4.1 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 (see Fig.13).

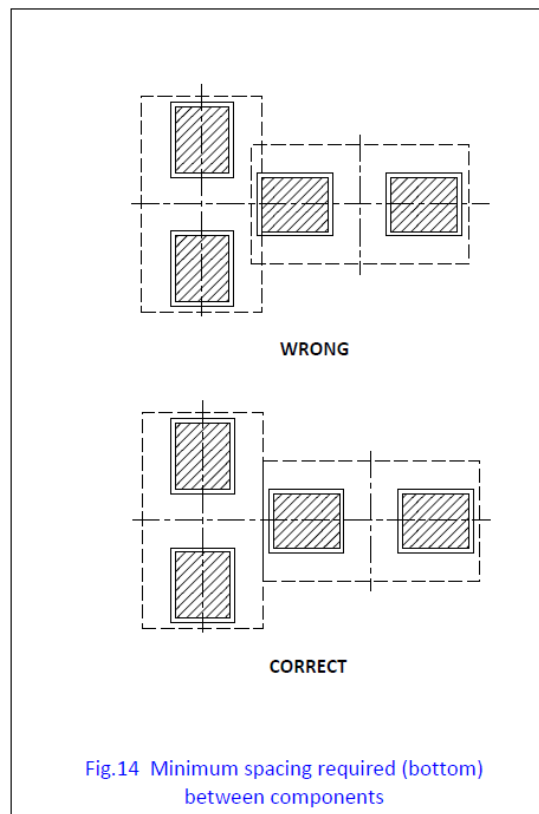
These dimensions are more crucial for fine-pitch components.

4.2 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 0.15mm or 0.3mm 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.

4.3 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 that the occupied areas do not overlap (see Fig.14).

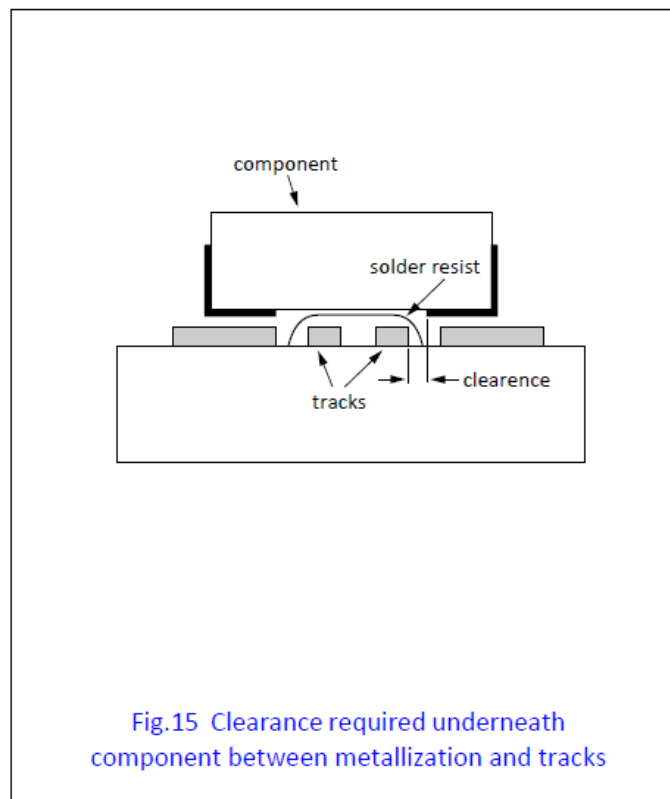


4.4 Solder paste pattern

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 that the occupied areas do not overlap (see Fig.14).

4.5 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 metallization position of the mounted component should be considered (see Fig.15).



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 $\geq 0.1\text{mm}$), for low-voltage applications. The number of tracks in this space is determined by the specified line resolution of the printed board.

4.6 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.

5. Recommended Footprints

The dimensional package outline drawings and recommended footprints for most of our discrete semiconductor packages may be found on our data sheets. Refer also to WeEn Application Note, WAN007, "Surface mount power packages".

Revision history

Rev	Date	Description
v.1	20040901	initial version
v.2	20190701	new format update
v.3	20190806	Format update

Contact information

For more information and sales office addresses please visit: <http://www.ween-semi.com>

Legal information

Definitions

Draft — The document is a draft version only. The content is still under internal review and subject to formal approval, which may result in modifications or additions. WeEn Semiconductors does not give any representations or warranties as to the accuracy or completeness of information included herein and shall have no liability for the consequences of use of such information.

Disclaimers

Limited warranty and liability — Information in this document is believed to be accurate and reliable. However, WeEn Semiconductors does not give any representations or warranties, expressed or implied, as to the accuracy or completeness of such information and shall have no liability for the consequences of use of such information. WeEn Semiconductors takes no responsibility for the content in this document if provided by an information source outside of WeEn Semiconductors.

In no event shall WeEn Semiconductors be liable for any indirect, incidental, punitive, special or consequential damages (including - without limitation - lost profits, lost savings, business interruption, costs related to the removal or replacement of any products or rework charges) whether or not such damages are based on tort (including negligence), warranty, breach of contract or any other legal theory.

Notwithstanding any damages that customer might incur for any reason whatsoever, WeEn Semiconductors' aggregate and cumulative liability towards customer for the products described herein shall be limited in accordance with the *Terms and conditions of commercial sale* of WeEn Semiconductors.

Right to make changes — WeEn Semiconductors reserves the right to make changes to information published in this document, including without limitation specifications and product descriptions, at any time and without notice. This document supersedes and replaces all information supplied prior to the publication hereof.

Suitability for use — WeEn Semiconductors products are not designed, authorized or warranted to be suitable for use in life support, life-critical or safety-critical systems or equipment, nor in applications where failure or malfunction of an WeEn Semiconductors product can reasonably be expected to result in personal injury, death or severe property or environmental damage. WeEn Semiconductors and its suppliers accept no liability for inclusion and/or use of WeEn Semiconductors products in such equipment or applications and therefore such inclusion and/or use is at the customer's own risk.

Applications — Applications that are described herein for any of these products are for illustrative purposes only. WeEn Semiconductors makes no representation or warranty that such applications will be suitable for the specified use without further testing or modification.

Customers are responsible for the design and operation of their applications and products using WeEn Semiconductors products, and WeEn Semiconductors accepts no liability for any assistance with applications or customer product design. It is customer's sole responsibility to determine whether the WeEn Semiconductors product is suitable and fit for the customer's applications and products planned, as well as for the planned application and use of customer's third-party customer(s). Customers should provide appropriate design and operating safeguards to minimize the risks associated with their applications and products.

WeEn Semiconductors does not accept any liability related to any default, damage, costs or problem which is based on any weakness or default in the customer's applications or products, or the application or use by customer's third-party customer(s). Customer is responsible for doing all necessary testing for the customer's applications and products using WeEn

Semiconductors products in order to avoid a default of the applications and the products or of the application or use by customer's third-party customer(s). WeEn does not accept any liability in this respect.

Export control — This document as well as the item(s) described herein may be subject to export control regulations. Export might require a prior authorization from competent authorities.

Evaluation products — This product is provided on an "as is" and "with all faults" basis for evaluation purposes only. WeEn Semiconductors, its affiliates and their suppliers expressly disclaim all warranties, whether express, implied or statutory, including but not limited to the implied warranties of non-infringement, merchantability and fitness for purpose. The entire risk as to the quality, or arising out of the use or performance, of this product remains with customer.

In no event shall WeEn Semiconductors, its affiliates or their suppliers be liable to customer for any special, indirect, consequential, punitive or incidental damages (including without limitation damages for loss of business, business interruption, loss of use, loss of data or information, and the like) arising out of the use of or inability to use the product, whether or not based on tort (including negligence), strict liability, breach of contract, breach of warranty or any other theory, even if advised of the possibility of such damages.

Notwithstanding any damages that customer might incur for any reason whatsoever (including without limitation, all damages referenced above and all direct or general damages), the entire liability of WeEn Semiconductors, its affiliates and their suppliers and customer's exclusive remedy for all of the foregoing shall be limited to actual damages incurred by customer based on reasonable reliance up to the greater of the amount actually paid by customer for the product or five dollars (US\$5.00). The foregoing limitations, exclusions and disclaimers shall apply to the maximum extent permitted by applicable law, even if any remedy fails of its essential purpose.

Translations — A non-English (translated) version of a document is for reference only. The English version shall prevail in case of any discrepancy between the translated and English versions.

Trademarks

Notice: All referenced brands, product names, service names and trademarks are the property of their respective owners.

Contents

1.	Introduction	1
2.	Axial and radial leaded diodes	1
2.1	Handling	1
2.2	Soldering	1
2.3	Mounting	2
3.	Surface mount devices (SMDs)	2
3.1	Reflow soldering process	3
3.1.1	Applying solder paste to PCB	3
3.1.2	Stencil printing	4
3.1.3	Squeegee	5
3.1.4	Stencil apertures	6
3.1.5	Solder paste	6
3.2	Component placement	7
3.3	Convection reflow soldering	8
3.4	Double-wave soldering process	9
3.4.1	Adhesive application	10
3.4.2	Adhesive volume	10
3.4.3	Adhesive dots - number, position and volume per component	11
3.4.4	Adhesive nozzle outlet diameter	12
3.4.5	Adhesive types	12
3.5	Wave soldering process	14
3.6	Hand soldering microminiature components	16
3.7	Assessment of soldered joint quality	17
3.7.1	Positioning	17
3.7.2	Good wetting	17
3.7.3	Correct amount of solder	17
3.7.4	Smooth, continuous joint surface	18
4.	Footprint definitions	19
4.1	Solder lands (conductive pattern)	19
4.2	Solder resist pattern	20
4.3	Occupied area of the component	20
4.4	Solder paste pattern	21
4.5	Area available for tracks (conductive pattern)	21
4.6	Component orientation during wave soldering	22
5.	Recommended Footprints	22
	Revision history and contact information	24
	Legal information	24
	Definitions	24
	Disclaimers	24
	Trademarks	24
	Contents	25

Please be aware that important notices concerning this document and the product(s) described herein, have been included in section 'Legal information'.

© WeEn 2019.

All rights reserved

For more information, please visit: <http://www.ween-semi.com>

Date of release: 01 July 2019

Document identifier: WAN006