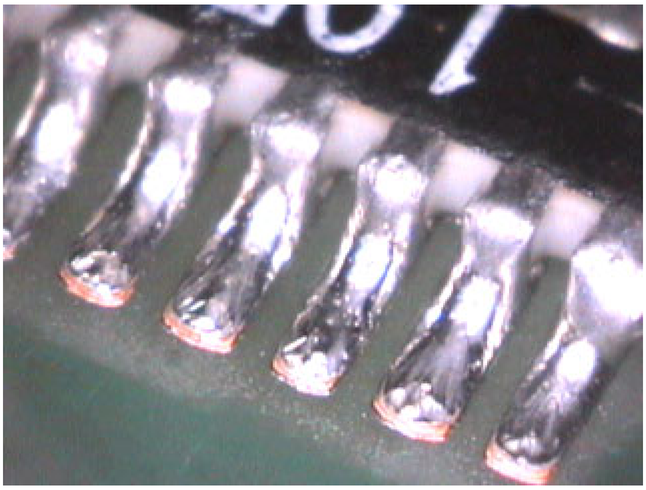
SMT NOTES

for CHIP-R and MLCC



Soldering Joint Criteria

The components and the printed boards must fulfil the requirements for good solderability and wettability even after an (accelerated) ageing treatment. The components are tested according to IEC 68-2-58 under the following conditions :

Property to be tested	Temperature of the solder (°C)	Dwell time (second)
Wettability	235 ± 5	2 ± 0.2
Permanent wetting	260 ± 5	5 ± 0.5

- Speed of immersion : 25 ± 2.5 mm/sec
- Speed of withdrawal : 25 ± 2.5 mm/sec
- Depth of immersion : Such that the surfaces to be examined are fully immersed.

Most reflow soldering techniques utilize a solder paste that can be applied by screen-printing, stencil-printing or dispensing. Solder paste is a suspension of pre-alloyed solder powder particles in a flux vehicle, to which special agents have been added.

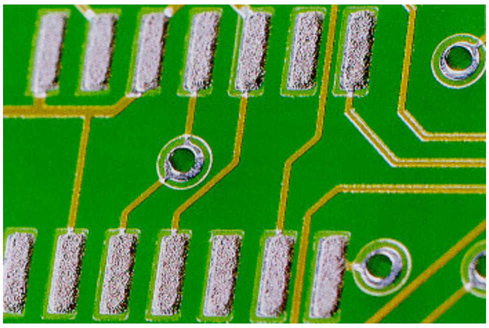
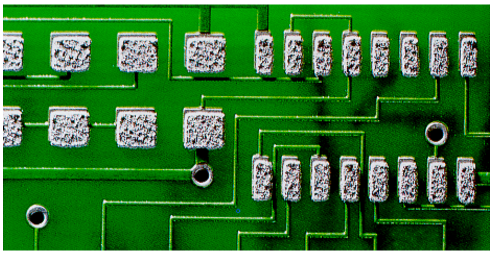
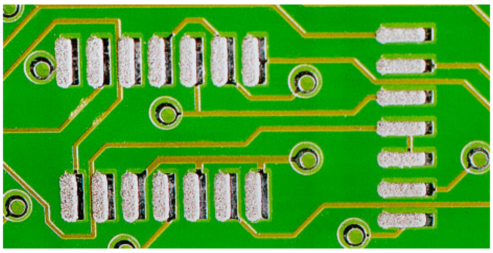
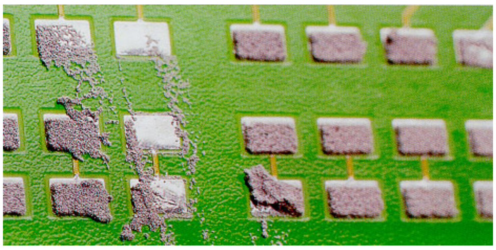
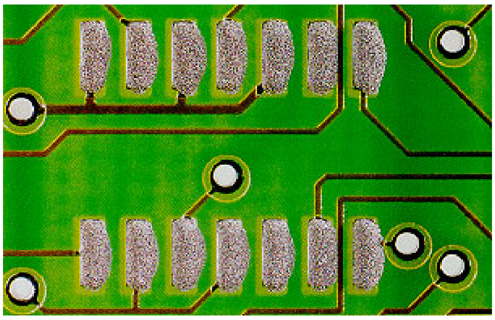
Solder balling is a common problem in reflow soldering using solder paste, because during the melting process the solder powder particles do not completely coalesce, so that isolated spheres of solder are left on the non-wettable parts of the substrate.

The solder balling test determines the reflow properties of the pre-alloyed solder powder particles in the paste.

Preferred	**Acceptable**	**Rejected**
The molten solder from the paste has melted into one solder ball.	The molten solder from the paste has melted into one large solder ball surrounded by a small number of very small solder particles (≤ 50um)	The molten solder from the paste has melted into one or more solder balls surrounded by a lot of small solder particles
		

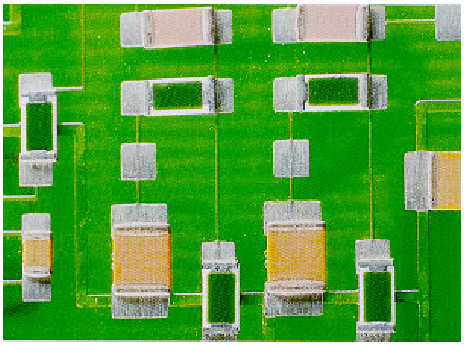
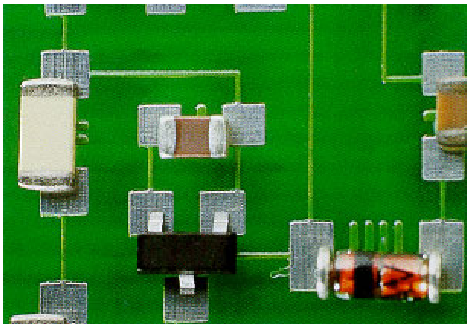
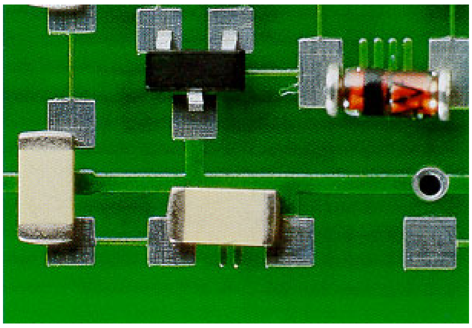
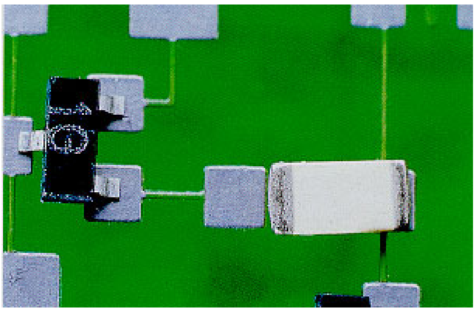
■ Screen or Stencil Printing of Solder Paste

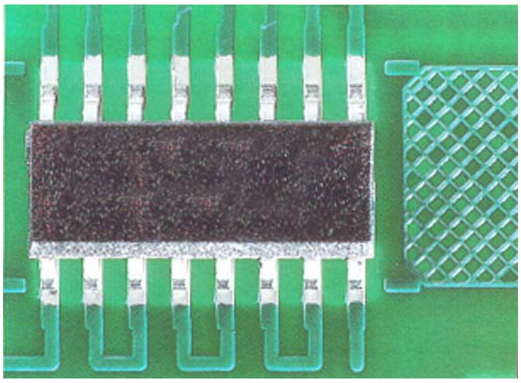
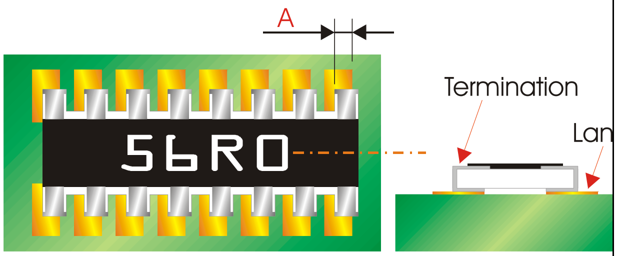
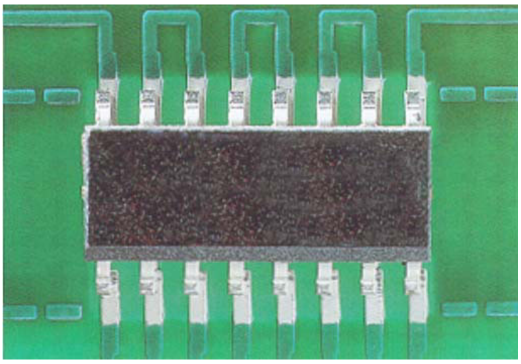
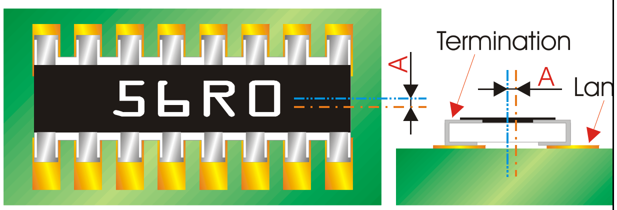
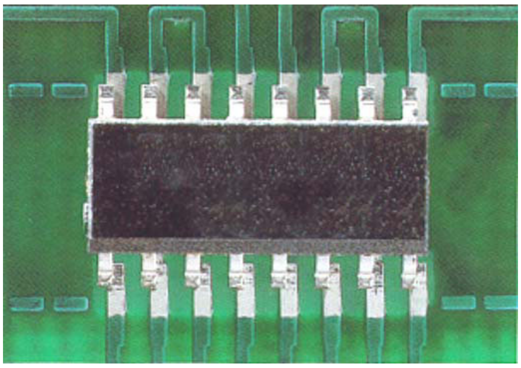
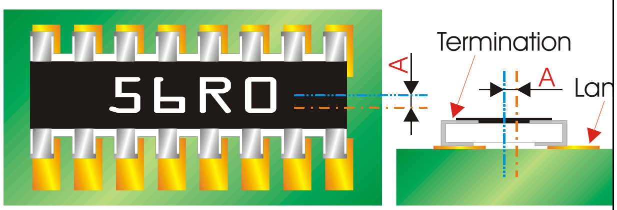
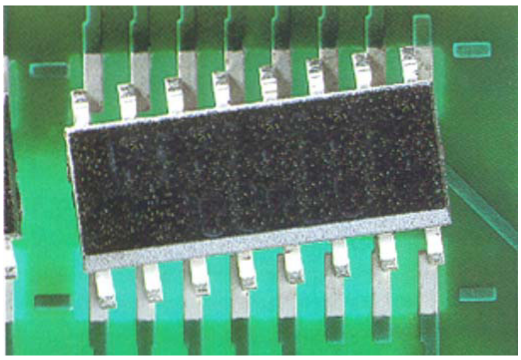
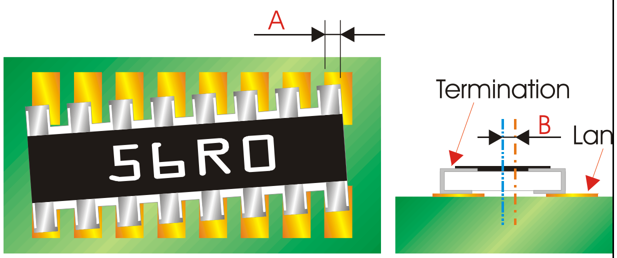
In reflow soldering the quantity of solder at each soldered joint depends on the amount of solder paste deposited on each solder land. In standard cases, the amount of solder paste should be 0.8 mg/mm^2 of surface area of the solder lands. For fine pitch (≤ 0.8mm) multi-lead components, the amount of solder paste should be lower, in the range of 0.5 mg/mm^2.

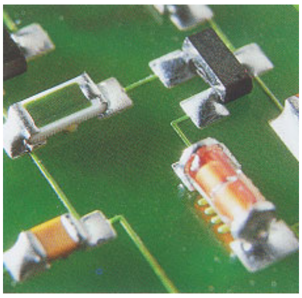
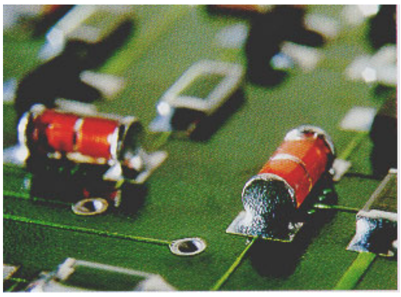
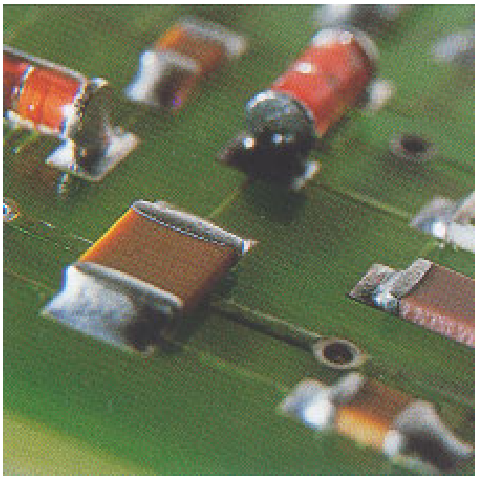
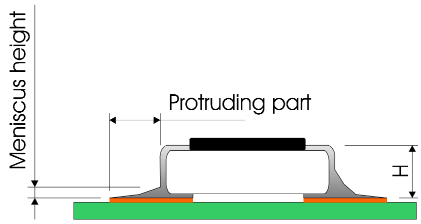
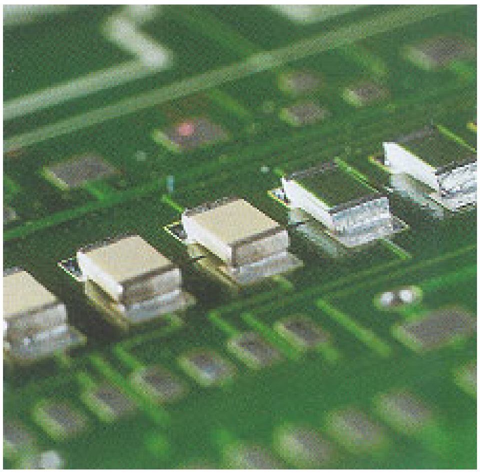
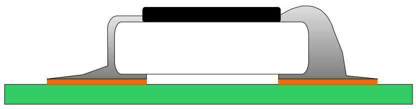
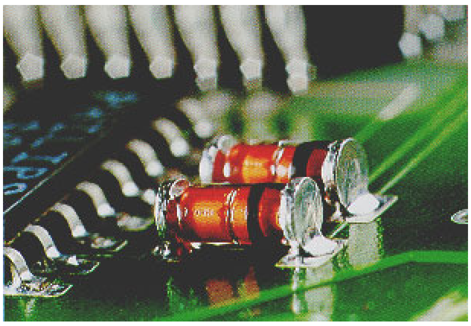
Solder paste printing shifted in x/y direction	Criteria
	**Preferred**
	**Acceptable** For Components like 0805, 1206 : Displacement ≤ 0.2 mm. For multi-lead components like array resistor with a pitch ≤ 0.8 mm : Displacement ≤ 0.1 mm.
	**Rejected** For Components like 0805, 1206 : Displacement > 0.2 mm. For multi-lead components like array resistor with a pitch ≤ 0.8 mm : Displacement >0.1 mm
	**Rejected** Contamination of paste, too little paste and no paste
	**Rejected** Paste outflow more than 0.2mm outside the solder land caused by too much solder paste or too great a placing force, (for multi-lead components with pitch ≤ 0.8 mm : outflow more than 0.1mm)

■ Placement Accuracy

In principle all components must be positioned exactly onto the solder lands. As a minimum condition, half the width of the component termination has to be placed on the solder land. Displacement in the y direction and/or displacement caused by rotation cause the total displacement. This demand is a practical value, which guarantees a reliable joint and can be visually inspected in an unambiguous way.

Components shifted in x/y direction	Criteria
	**Preferred**
	**Acceptable** Half or more of the width of the component is situated on the solder land. Only acceptable when the conductor is covered with insulating lacquer.
	**Rework** Less than half the width of the component is situated on the solder land.
	**Rework** The metallisation must at least be partly positioned on the solder land.

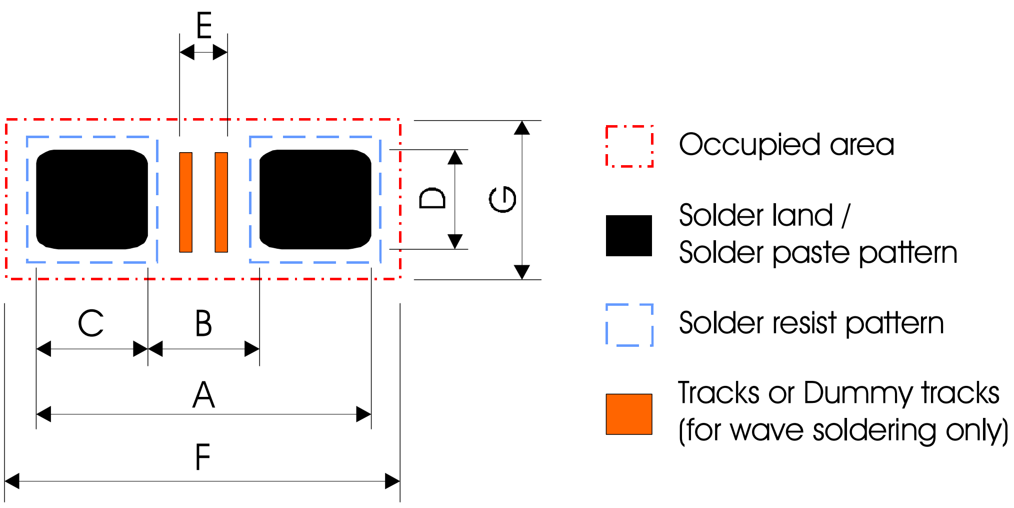
Components with short leads (SO,SSOP, QFP) and Array and Network of Resistor / Capacitor	Criteria
	**Preferred**  Distance A ≥ 3/4 the width of the termination
	**Acceptable**  Distance A ≤ half the width of the termination
	**Rework**  Distance A > half the width of the termination
	**Rework**  Distance A and B > half the width of the termination

Visual inspection of soldered joints	Criteria
 	**Preferred** The height of the meniscus is equal to the component height. The solder fillet is concave.
	**Acceptable**  For Components with a termination height H < 1.2mm : The height of the meniscus must be at least 1/3 of the height H For components with a termination height H ≥ 1.2mm : The height of the meniscus must be at least 0.4mm.
	**Rework**  For components with a height H of less than 1.2 mm, the height of the meniscus is less than 1/3 of the height H. For components with a height H over 1.2mm, the height of the meniscus is less than 0.4mm
	**Rework** The height of the meniscus is less than 0.4mm or the solder fillet is convex.

Guidelines for Footprint Design

The first step in circuit board design is to consider ***how the surface mounted board will be manufactured***. This is because the manufacturing process determines the necessary dimensions of the solder lands, the minimum spacing between components, the area underneath the SMD where tracks may be laid down, and the required component orientation during wave soldering. Therefore a footprint related to the manufacturing process with all this information is an essential tool for SMD circuit board design.

A typical SMD footprint, as shown in following figure is composed of :



These footprint details depend on the following parameters :

- Component dimensions and tolerances as given in the component data ;
- Board dimensional accuracy ;
- Placement accuracy of the component with respect to the solder lands on the board ;
- Solder paste position tolerances with respect to the solder lands (for reflow soldering only) ;
- The soldering process parameters ;
- Solder resist position tolerances with respect to the solder lands ;
- Solder joint parameters for reliable joints.

■ Footprint design for discrete CHIP-R

◆ Reflow Soldering

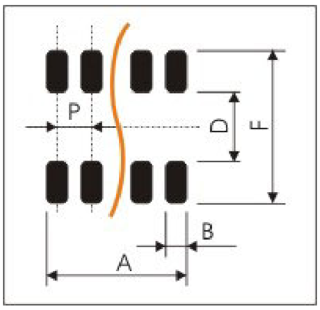
SIZE	Footprint dimensions in mm							Processing remarks	Placement Accuracy
	A	**B**	**C**	**D**	**E**	**F**	**G**		
0201	0.75	0.30	0.30	0.30	0.20	1.10	0.50	IR reflow soldering	± 0.05
0402	1.50	0.50	0.50	0.60	0.10	1.90	1.00		± 0.15
0603	2.10	0.90	0.60	0.90	0.50	2.35	1.45		± 0.25
0805	2.60	1.20	0.70	1.30	0.75	2.85	1.90		± 0.25
1206	3.80	2.00	0.90	1.60	1.60	4.05	2.25	IR or hot plate soldering	± 0.25
1218	3.80	2.00	0.90	4.80	1.40	4.20	5.50		± 0.25
2010	5.60	3.80	0.90	2.80	3.40	5.85	3.15		± 0.25
2512	7.00	3.80	1.60	3.50	3.40	7.25	3.85		± 0.25

◆ Wave Soldering

SIZE	Footprint dimensions in mm							Proposed number & Dimensions of dummy tracks	Placement Accuracy
	A	**B**	**C**	**D**	**E**	**F**	**G**		
0603	2.70	0.90	0.90	0.80	0.15	3.40	1.90	1x (0.15x0.80)	± 0.25
0805	3.40	1.30	1.05	1.30	0.20	4.30	2.70	1x (0.20x1.30)	± 0.25
1206	4.80	2.30	1.25	1.70	1.25	5.90	3.20	3x (0.25x1.70)	± 0.25
1218	4.80	2.30	1.25	4.80	1.30	5.90	5.60	3x (0.25x4.80)	± 0.25
2010	6.30	3.50	1.40	2.50	3.00	7.00	3.60	3x (0.75x2.50)	± 0.25
2512	8.50	4.50	2.00	3.20	3.00	9.00	4.30	3x (1.00x3.20)	± 0.25

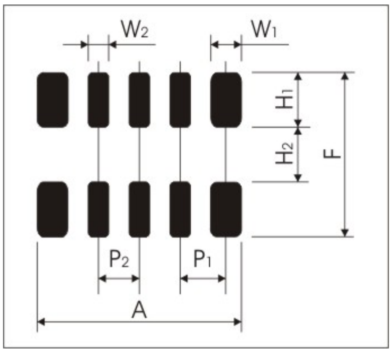
■ Footprint design for Array Resistor :

Type	**0603*4**	**0402*4**	**0402*2**
Symbol / Item	**WA06X / WA06T**	**WA04X**	**WA04Y, WA04P**
A	2.85 +0.10/-0.05	1.80 +0.15/-0.05	1.20 ± 0.05
B	0.45 ± 0.05	0.30 ± 0.05	0.40 +0/-0.05
D	0.80 ± 0.10	0.50 ± 0.1	0.50 ± 0.05
P	0.80	0.50	0.65
F	3.10 ± 0.30	2.00 +0.40/-0.20	1.50 +0.20/-0.10



■ Footprint design for 10P8R Network Resistor :

Symbol	WT04X
W_1	0.35 ± 0.05
W_2	0.50 ± 0.05
H_2	0.80 ± 0.10
P_1	0.70 ± 0.05
P_2	0.65 ± 0.05
A	3.20 ± 0.10
F	2.80 +0.40/-0.20



■ Footprint design for discrete MLCC :

◆ Reflow Soldering

SIZE	Footprint dimensions in mm							Processing remarks	Placement Accuracy
	A	B	C	D	E	F	G		
0402	1.50	0.50	0.50	0.50	0.10	1.75	0.95	IR or hot plate soldering	± 0.15
0508	2.50	0.50	1.00	2.00	0.15	2.90	2.40		± 0.20
0603	2.30	0.70	0.80	0.80	0.20	2.55	1.40		± 0.25
0612	2.80	0.80	1.00	3.20	0.20	3.08	3.85		± 0.25
0805	2.80	1.00	0.90	1.30	0.40	3.08	1.85		± 0.25
1206	4.00	2.20	0.90	1.60	1.60	4.25	2.25		± 0.25
1210	4.00	2.20	0.90	2.50	1.60	4.25	3.15		± 0.25
1808	5.40	3.30	1.05	2.30	2.70	5.80	2.90		± 0.25
1812	5.30	3.50	0.90	3.80	3.00	5.55	4.05		± 0.25
2220	6.50	4.70	0.90	5.60	4.20	6.75	5.85		± 0.25

◆ Wave Soldering

SIZE	Footprint dimensions in mm							Proposed number & Dimensions of dummy tracks	Placement Accuracy
	A	B	C	D	E	F	G		
0603	2.40	1.00	0.70	0.80	0.20	3.10	1.90	1x (0.20x0.80)	± 0.10
0805	3.20	1.40	0.90	1.30	0.36	4.10	2.50	1x (0.30x1.30)	± 0.15
1206	4.80	2.30	1.25	1.70	1.25	5.90	3.20	3x (0.25x1.70)	± 0.25
1210	5.30	2.30	1.50	2.60	1.25	6.30	4.20	3x (0.25x2.60)	± 0.25

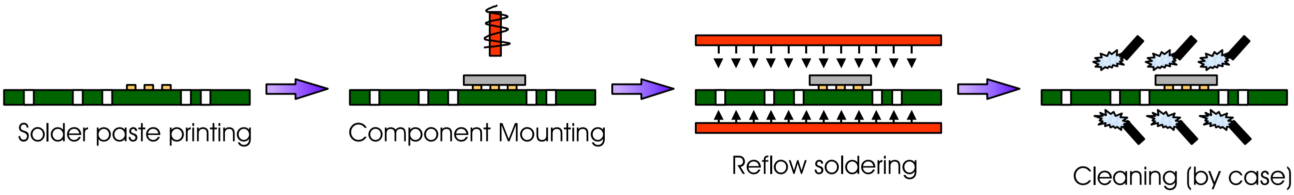
■ Stencil thickness

The following relation between the solder land dimensions (C x D) and the solder paste apertures dimensions must be taken into account of solder for each solder joint :

Solder land dimensions	Solder paste apertures	Max. Stencil thickness
$C \geq 0.6$ & $D \geq 0.6$ $0.4 \leq C < 0.6$ & $D > 0.9$ $C > 0.9$ & $0.4 \leq D < 0.6$	$(C - 0.1) \times (D - 0.1)$	0.20 mm
$0.4 \leq C < 0.6$ & $0.6 \leq D \leq 0.9$	$C \times (D - 0.1)$	0.20 mm
$0.6 \leq D \leq 0.9$ & $0.4 \leq D < 0.6$	$(C - 0.1) \times D$	0.20 mm
$0.5 \leq D < 0.6$ & $0.5 \leq D < 0.6$	$C \times D$	0.20 mm
$C < 0.4$ & $D > 0.6$	$(D - 0.03) \times (C - 0.1)$	0.15 mm
$C < 0.4$ & $D \leq 0.6$	$(D - 0.03) \times (C - 0.05)$	0.12 mm

Reflow Soldering

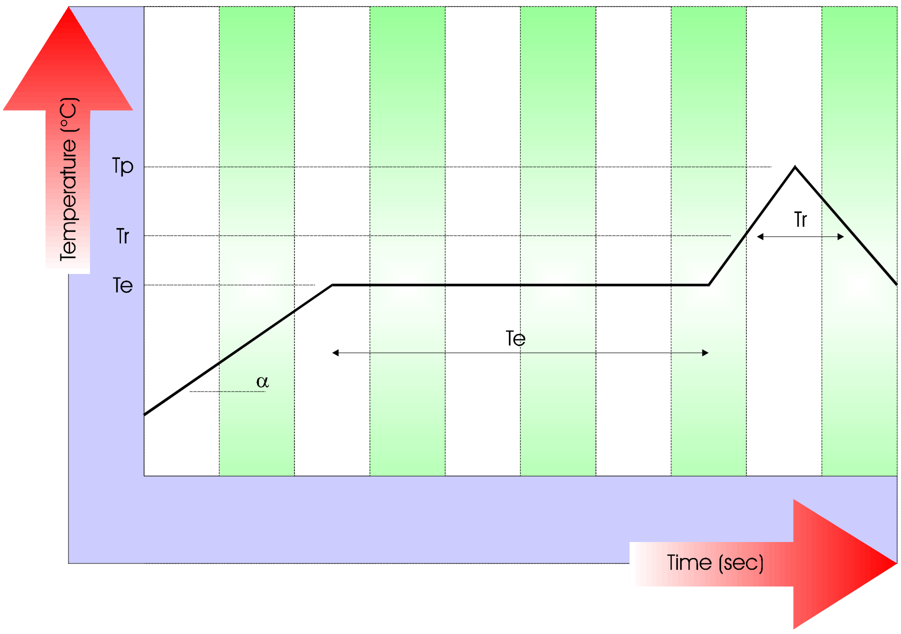
The most common method used for reflow soldering in mass production is an infrared or convection oven. In special cases vapour phase reflow is more suitable. In some situations it is required to assemble only one or two SMDs (e.g. a very fine pitch QFP after the assembly of more standard components, or for repair purposes). For this application reflow methods like resistance soldering, hot gas soldering, or laser soldering can be used. The following figure show the process of reflow soldering :



Temperature profile

In cases where a combination of small and large SMDs must be reflowed, the situation is complex. When the solder paste for small components (e.g. MLCC, Chip-R) start to melt, the temperature near the large components (e.g. IC) is far below the melting point. As soon as the paste for the large components start melting, the temperature of the small components can have reached damaging levels.

To prevent this large temperature difference it is advisable to use a temperature profile as that described in following figure. It is important to know that the temperature differences are the only reason for the typical shape of the temperature profile.



In many cases the following figures can be used :

- The temperature rise in the first zone of the oven (α) should be ≤ 10°C because of possible solder paste spattering.
- The temperature in the equalise zone (Te) should be ≤ 160°C and the time in this zone (Te) ≤ 5 minutes (and < 1minute if possible) because of possible solder paste degradation.
- The peak temperature (Tp) in the last heating zone should be at least 213°C with a max. of 280°C (for double-sided reflow the max. temperature is 265°C).
- Tp : 213°C is chosen to guarantee good soldering for 80Sn/20Pb metallisations, and 280°C is the max. temperature for the glass epoxy board.
- The time that the solder temperature is above the melting point (Tr) should not be longer than 1 minute because of possible decreasing joint strength by intermetallic layers.

Recommendation of soldering profile

The robust construction of chip resistors allows them to be completely immersed in a solder bath of 260°C for one minute. Therefore, it is possible to mount Surface Mount Resistors on one side of a PCB and other discrete components on the reverse (mixed PCBs).

Surface Mount Resistors are tested for solderability at 235°C during 2 seconds. The test condition for no leaching is 260°C for 60 seconds. Typical examples of soldering processes that provide reliable joints without any damage are given in below.

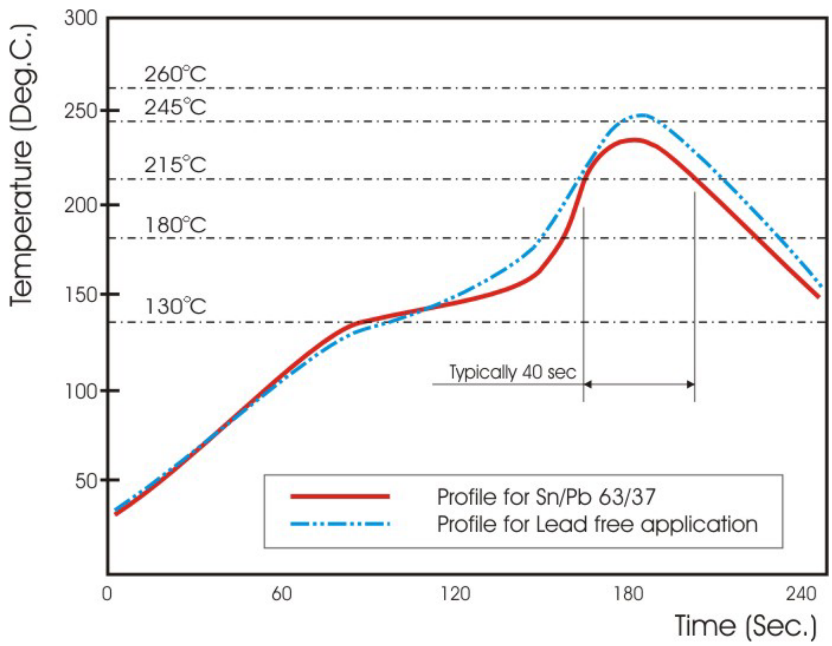
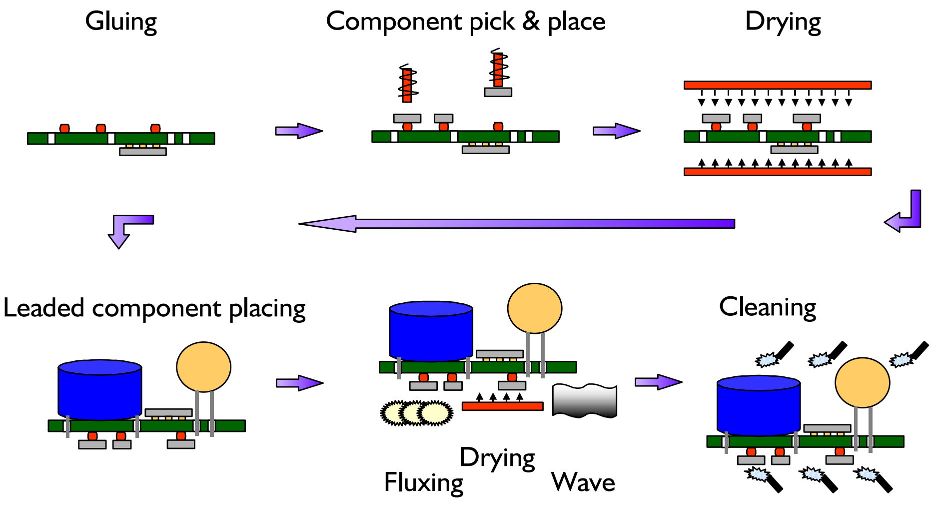


Figure. Infrared soldering profile for Lead-Free Chip Resistors

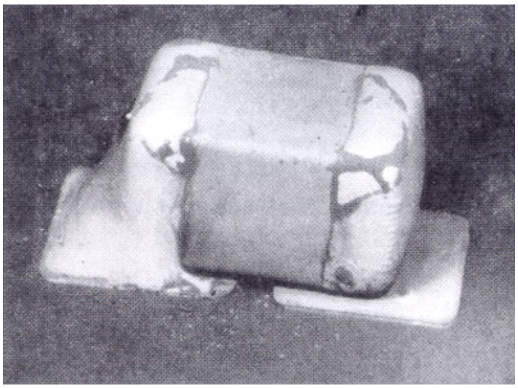
Wave Soldering

Wave soldering was introduced to the electronic industry in the mid-fifties. Since then wave soldering machine have evolved from simple soldering units to fully automated systems.

For mass production purposes, wave soldering is very often preferred to reflow soldering. For mixed print assemblies (leaded components on the topside, SMDs underneath) on single-sided boards, wave soldering is the only reliable soldering technology. The typical wave soldering process is shown as following :



Wetting behaviour of SMDs

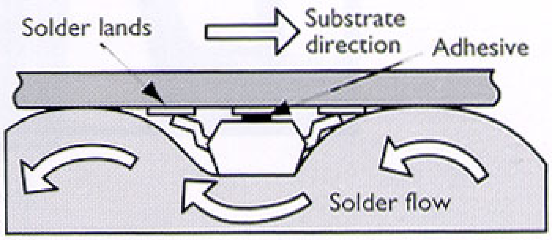
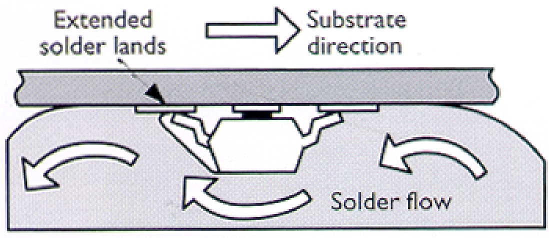


SMD wave soldering places two opposing demands on the solder wave :

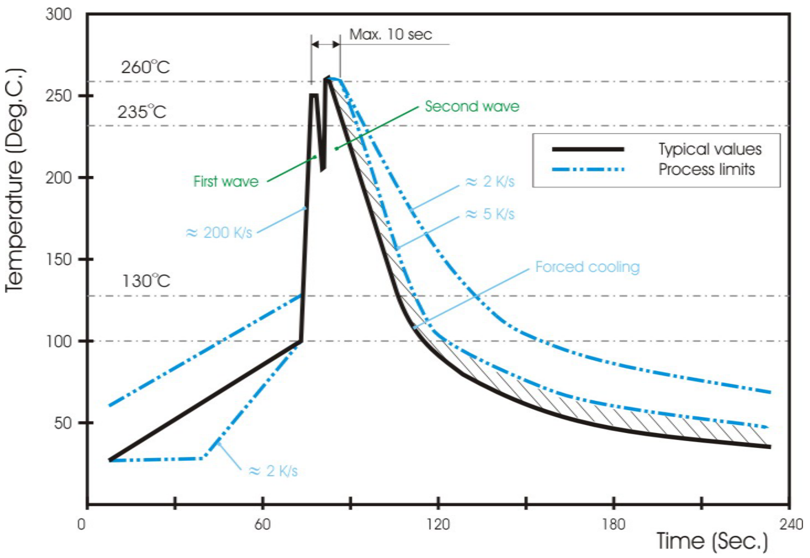
a. The wave must reach all components terminations ;

b. Bridges between adjacent narrowly placed leads must be avoided.

During soldering, the component bodies are immersed in the solder. “Skips” are mainly caused because plastic bodies are not wetted by soldering, creating a depression in the solder wave, which is enhanced by surface tension. This can cause a “shadow” behind the component and prevent solder from reaching the solder lands at trailing side of the SMD. The higher the body, the shorter the leads, and the smaller the footprint, the more severe the shadowing effect. The design of the circuit on the board plays a major part in the soldering result. The conditions for wave soldering are rather critical, especially for multi-lead components with closely spaced leads.

<table>
<tr>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>The shadow effect in wave soldering : Surface tension of the molten solder prevents wetting of the downstream solder land.</td>
<td>Extending the solder land to overcome the shadow effect.</td>
</tr>
</table>

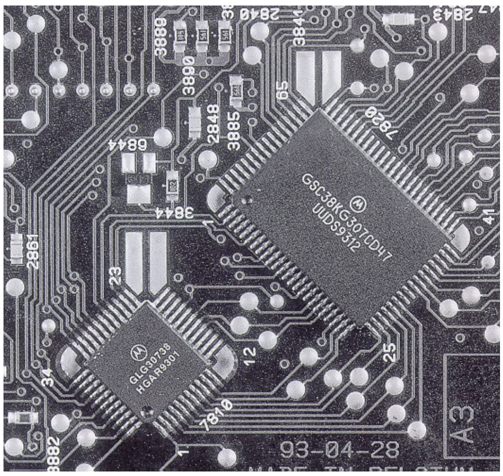
Recommendation of soldering profile



Surface Mountable Resistors are tested for solderability at a temperature of 235°C during 2 seconds. The test condition for no-leaching is 260°C for 60 seconds. Typical examples of soldering process that provide reliable joints without any damage, are given in following figure for wave soldering.

Wave soldering of multi-lead components

The prevention of bridging between the leads of integrated circuit packages is a particular problem. Sometimes it is necessary to shorten the protruding solder lands, and reduce the conveyor speed, or to use **solder thieves**.



Wave soldering is a sell-established process, that has some limitations in SMD applications. For example, an adhesive is required, and care must be taken to ensure that this does not contaminate solder lands, leads or terminations, which would affect solderability and thus quality of the joints. There are some limitations regarding component orientation and conductor spacing. On the other hand, wave soldering is useful for mass production.

Adhesive and Adhesive Application

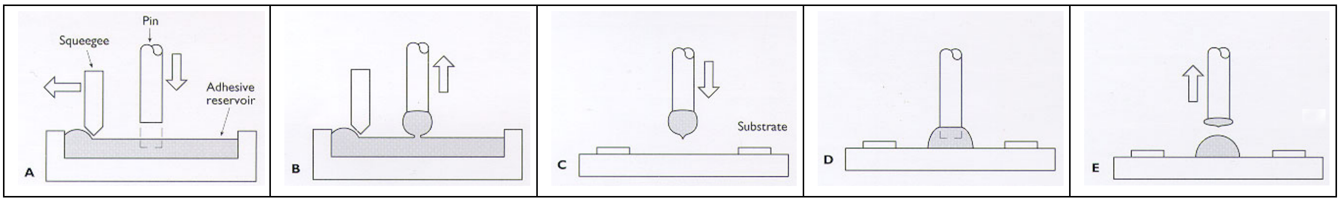
When an assembly has to be wave soldered, an adhesive is essential to bond the SMDs to the substrate. Under normal conditions reflow-soldered substrates do not need adhesive to maintain device orientation, since the solder paste does this. Sometimes and exception occurs in the case of double-sided, reflow-soldered SMD boards. Both adhesive and solder paste are required to hold devices on the bottom of the substrate during the reflow process, if the ration between the weight of the component and the component surface wetted by solder paste is too high.

Application Techniques

There are three different methods of applying adhesive in SMD assembly systems. The choice depends on the required SMD placement capacity, and the types of components used.

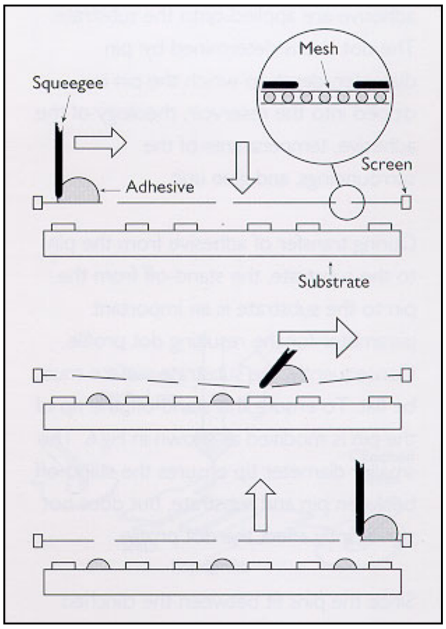
■ Pin transfer

A pin picks up a droplet of adhesive from a reservoir and transfers it to the surface of the substrate. Surface tension causes a portion of the droplet on the pin to form an adhesive dot on the substrate.



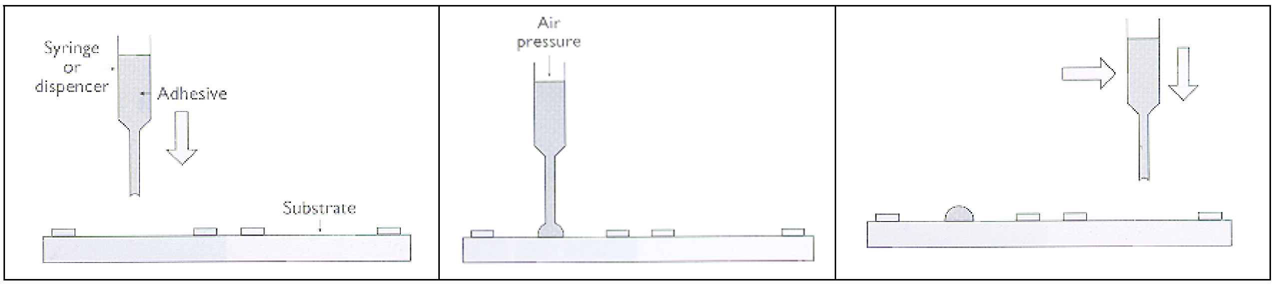
■ Screen or stencil printing

A fine mesh screen, coated with emulsion except for the areas where adhesive is required, is placed over the substrate. A squeegee passing over the screen forces adhesive through the uncoated areas of the mesh and onto the substrate. The same can be done by means of stencil.



■ Dispensing

The dispenser (either a time-pressure system or a rotary pump) is positioned over the printed board by means of a computer controlled X-Y-Z moving system. The adhesive dot size is determined by the amount of time the dispenser is activated.

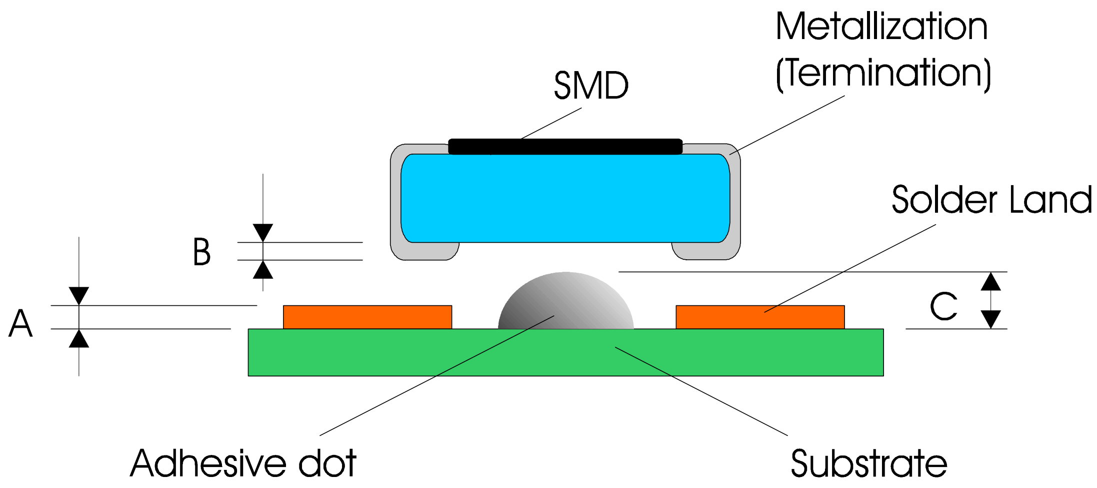


The next table summarises the principal benefits and disadvantages of the automated adhesive application systems :

Method	Advantages	Disadvantage
Pin transfer	■ Compact system ■ Simple, fast process ■ Little maintenance ■ Simultaneous dot placement ■ Accepts pre-loaded mixed-print board ■ Controls adhesive quantity	■ Needs flat board surface ■ Open system subject to outside influences ■ Can not use high yield point adhesive
Screen / Stencil printing	■ Simultaneous dot placement ■ Simple process ■ Uniform dot height	■ Needs flat board surface ■ No obtrusions on surface ■ Screen maintenance required ■ Open system ■ Dot height limited by screen emulsion or stencil thickness
Dispensing	■ Handles irregular surfaces ■ Accepts mixed-point boards ■ Controls adhesive quantity ■ Closed system not subject to outside influences ■ Accepts most adhesives	■ Requires more maintenance ■ Equipment is bulky ■ Much slower system

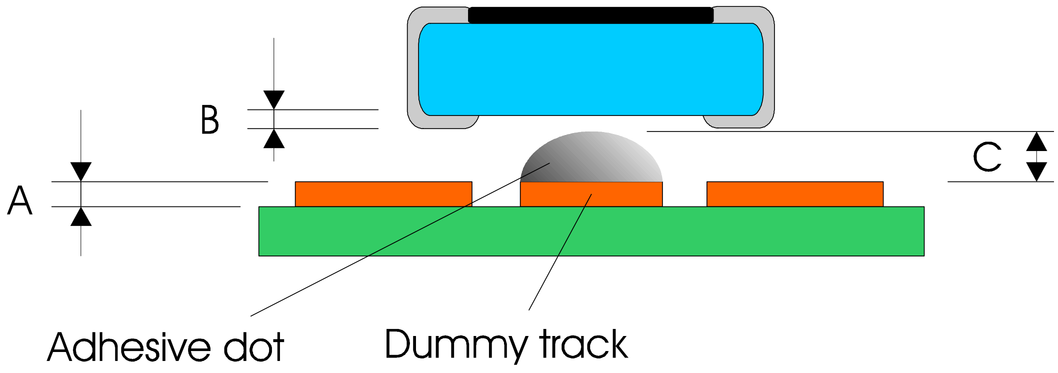
Adhesive requirements

The height and volume of adhesive dots applied at each position are critical for two reasons : t**he dot must be high enough to reach the SMD** ; and there **must not be any excess adhesive**, since this can pollute the solder land and prevent the formation of a good soldered joint.



Criteria for dot height of adhesive : C > A+B

One solution to the problem of height variation of solder lands is to route a dummy track of height A under the device, as shown in following. "C" then depends only on the thickness of the SMD metallisations "B".



As component weight increases, adhesive green strength becomes more important. Under these conditions an adhesive with specific theological properties and high yield point is needed. Epoxy resins can be adapted to these requirements. The important point is to match adhesive properties with type of substrate, device, and placement method.

PCB design guidelines

Mounting density

Chip resistors are designed for handling by automatic chip placement systems.

The temperature rise in a resistor due to power dissipation, is determined by the laws of heat - conduction, convection, and radiation. The maximum body temperature usually occurs in the middle of the resistor and is called the hot-spot temperature and depends on the ambient temperature and the dissipated power.

The hot-spot temperature is important for mounting because the connections to the chip resistors will reach a temperature close to the hot-spot temperature. Heat conducted by the connections must not reach the melting point of the solder at the joints. Therefore a maximum solder joint temperature of 110 °C is advised.

The ambient temperature on large or very dense printed-circuit boards is influenced by the dissipated power. The ambient temperature will again influence the hot-spot temperature. Therefore, the packing density that is allowed on the PCB is influenced by the dissipated power.

Example of mounting effects

Assume that maximum temperature of a PCB is 95 °C and the ambient temperature is 50 °C. In this case the maximum temperature rise that may be allowed is 45 °C. In the graph below (fig.1), this point is found by drawing the line from point A (PCB=95 °C) to point B (Tamb = 50 °C) and from here to the left axis.

To find the maximum packing density, this horizontal line is extended until it intersects with the curve, 0.125W (point C). The maximum packing, 19 units/50*50mm^2 (point D), is found on the horizontal axis.

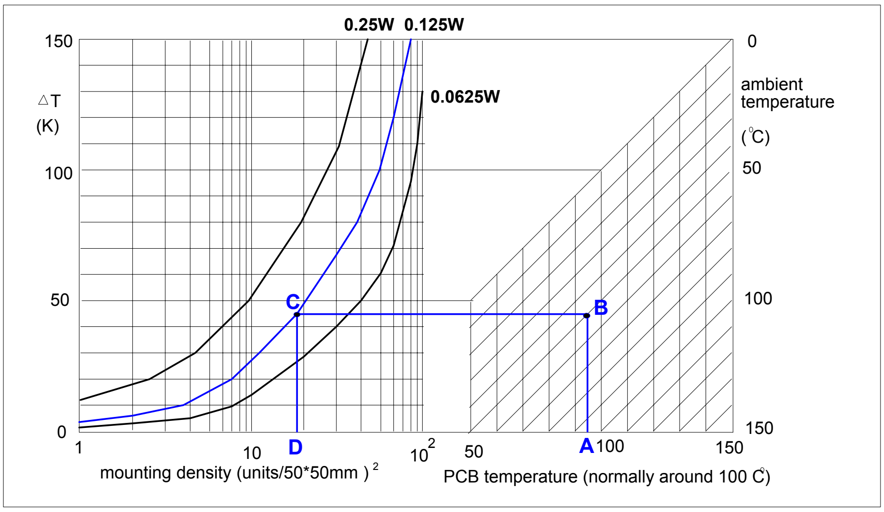


Fig.1 PCB temperature as a function of applied power, mounting density and ambient temperature

PCB design

To take full advantage of design features embodied in the new resistors, equipment manufactures can also take steps to minimise thermal stress:

1. By using forced cooling with a fan

2. By using a board material with a high thermal conductivity (such as ceramic)

3. By using a heat sink to reduce the temperature coefficient of the PC board

Figures 2, 3 and 4 shows the effect of using the heat sink. Figure 2 plots the temperature rise as a function of dissipated power at the hot spot, solder spot and the reverse side of the PC board without heat sink. Figure 3 shows same measurements with the board mounted on a 400mm^2 Heat sink, and figure 4 shows an example of the temperature rise reductions that can be expected when a heatsink is added on the underside of an FR4 PC board.

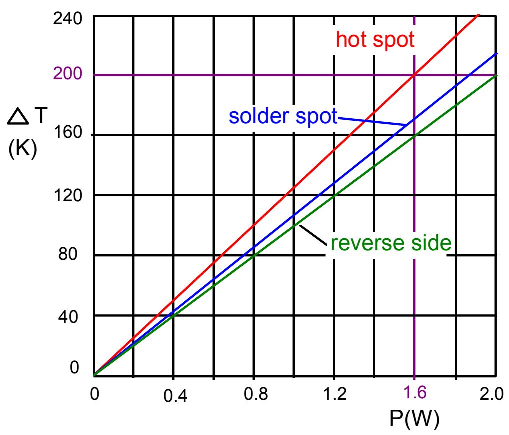


Fig.2 Temperature rise at hot and solder spot as a function of dissipated, mounted on FR4 without use of a heatsink.

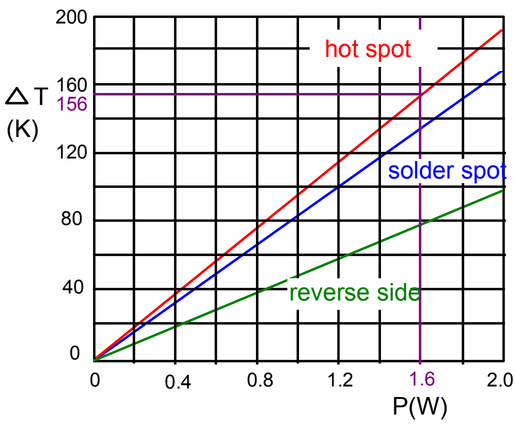


Fig.3 Temperature rise as a function of dissipated power with Philips test substrate mounted on a 400 mm^2 heatsink

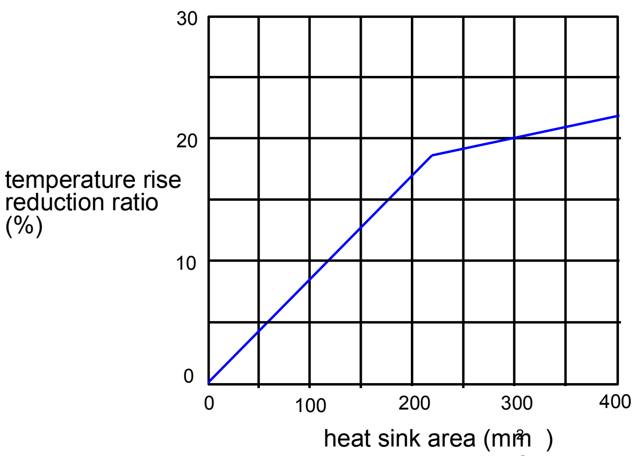


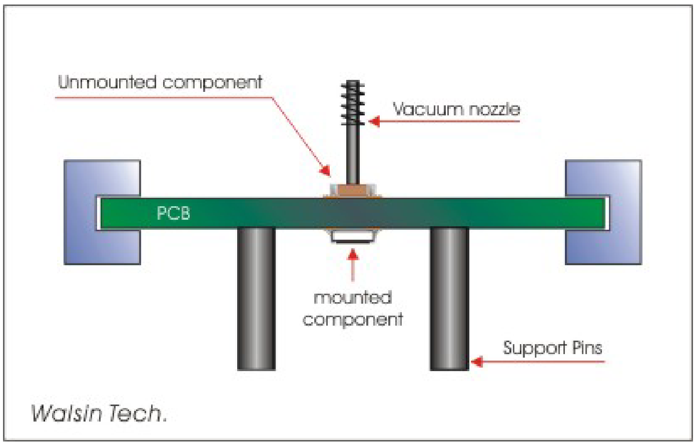
Fig.4 Temperature rise reduction of solder spot as a function of heatsink area

Precaution of handling

a. Precaution for soldering

1. Note that rapid heating or rapid heating or rapid cooling or local heating will easily damage CHIP-R/MLCC.
2. DO NOT give the heat shock over 100°C in the process of soldering. We recommend taking preheating and gradual cooling.

b. Precaution for handling of substrate



1. DO NOT exceed to bend the PCB after soldering this product extremely.
2. Mounting place must be as far as possible from the position that is closed to the break line of PCB, or on the line of large holes of PCB.
3. DO NOT bend extremely the PCB in mounting another components. IF necessary, use Back-up pin (Support pin) to prevent from bending extremely.
4. DO NOT break the PCB by hand. We recommend to use the machine or the jig to break it.

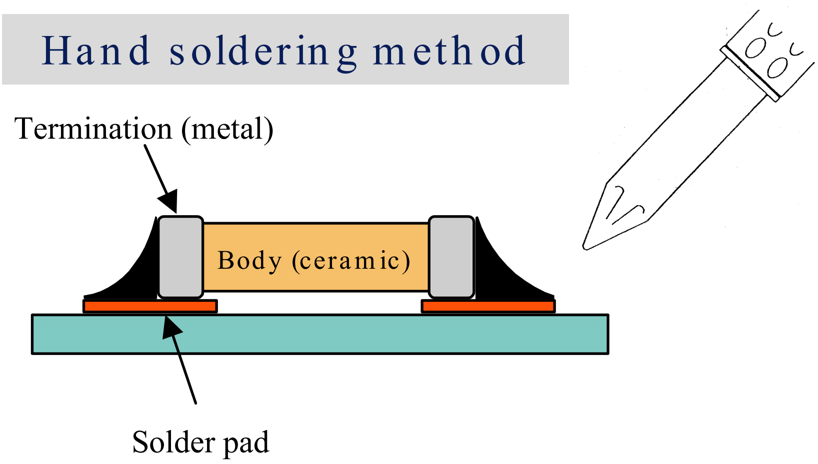
c. Caution of wave soldering

1. We do not recommend the wave soldering to this product, because of solder bridging happens owing to narrow 0.8mm pitch of this product.

d. Soldering gun procedure

Note the follows, in case of using soldering gun for replacement :

1. The tip temperature must be less than 280°C for the period within 3 seconds by using soldering gun under 30W.
2. The soldering gun tip shall not touch this product directly.



The tip of soldering iron should not directly touch the chip component to avoid thermal shock on the interface between termination and body during mounting, repairing or de-mounting process (shall ensure both termination had been molten still before chip component removed).

e. Storage conditions

Note the following, in case of storing this product :

1. Avoid the atmospheres which are high temperature, high humidity, dusty and having corrosive gas (Hydrogen chrolide, Sulfurous acid gas, Hydrogen sulfide etc.) to prevent terminal solderability fromdeclining. Keep the storage conditions less than 40°C and 70% relative humidity, and use up this product in 6 months as far as you can.
2. Avoid direct heat and sunshine to prevent the tape of package from transforming and sticking to this product.
3. Capacitance value occasionally will descend in several percent at the characteristic of high K capacitor material.