

General Information on Solder Paste

General Information on Solder Paste

1. Type of solder paste

There are various different types of solder paste available today. These products are generally categorised in the following manner by the type of flux medium contained in the paste.

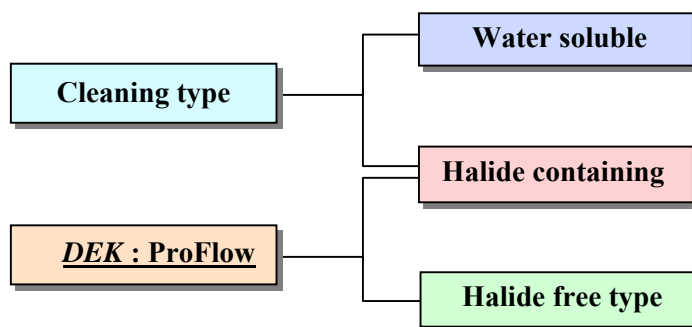


Fig. 1

Water soluble solder paste has to be cleaned with pure de-ionised water. Since Koki no-clean solder pastes, both the halide containing and halide free solder pastes are all rosin based, solvent such as HCFC, or saponifier would be recommended as a cleaner. Most of our halide free solder pastes however, are quite difficult to clean no matter how they are cleaned, and may leave white residues. The main component, which tends to remain on the substrate after cleaning, is normally thixotropic materials. Because easy-to-clean thixotropic materials tend to be deficient in effectively controlling slump resistance of the solder paste, not much attention is paid to the cleanability when selecting thixotropic materials, as more importance is attached to the slump resistance feature.

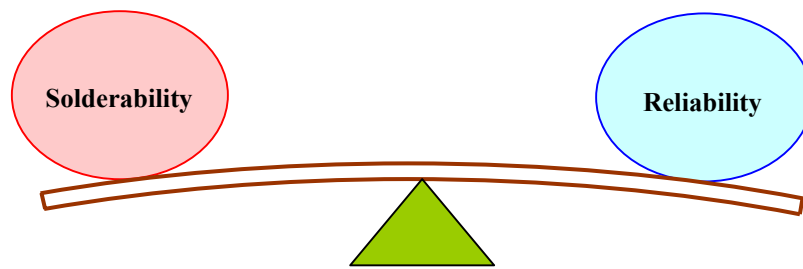
Koki specialises in no-clean solder pastes, Halide containing solder paste and Halide free solder paste.

Increased package density combined with a complete ban of CFCs greatly influenced the industry into the implementation of no-clean processes. As requirements of the process for both workability and reliability are becoming stricter and more precise, no-clean solder pastes have been developed accordingly.

It is often considered that no-clean solder paste must be halide free. It must be clearly understood that even halide containing solder paste can also be used for a no-clean process, because to clean or not, must be determined in accordance with the reliability requirements of the finished products.

In fact, the use of halide containing (0.2%) solder paste for no-clean production is much more popular in Japan than that of halide free paste.

As a general comparison, halide containing paste has better workability, such as solderability, but often exhibits inferior reliability to a halide free type, such as insulation resistance. This can be attributed mainly to its higher activation strength. The seesaw can express the relation between solderability and reliability.



Therefore, so far as a no-clean soldering process is concerned, with of high reliability in mind, it is desirable for the paste to be halide free, but hopefully with as high solderability as halide containing paste to ensure as wide a process window as possible.

It is however true that the main difficulty for no-clean halide free solder paste is improvement in activation. Most halide free solder pastes contain organic acids as an activator instead of halide so as to retain a certain activation strength. Although a variety of organic acids are available, generally, the less the molecular weight of the organic acids, the more activation it has. Since the activation strength of organic acids themselves is much weaker than halide, relatively active and large amounts of organic acids tend to be formulated in the flux system.

However, such high activation organic acids tend to have a tendency to absorb moisture. Thus there is a danger that flux residues left on the substrate could become ionised, reacting with moisture / water, which deteriorates the electrical properties such as surface insulation resistance and electromigration.

Activation systems in Koki solder pastes employ a combination of less hygroscopic organic acids and specially developed non-ionic activators.

Although these special activator systems do not become ionised as they do not have the property of dissociation and are electrically very stable and safe, exhibiting as high an activation strength as halogen. Since the activation temperature of non-ionic activator is relatively high, its combination with carefully selected organic acids provides a longer activation time at the reflow stage enhancing solderability, and ensuring extremely high reliability at the same time.

Solder paste is involved in the following production stages.

Storage → Printing → Mounting → Reflow → Inspection → Cleaning

At each stage, solder paste is required to exhibit good performance on the following characteristics:

Stage	Characteristics to be checked
Storage	<ul style="list-style-type: none"> • Stability of property (viscosity, solderability etc.)
Printing	<ul style="list-style-type: none"> • Fine pitch printing fine (0.5mm) / super fine pitch (0.4mm) • Stencil life • Stencil idle time (abandon time) • Rolling property • Squeegee separation • Print speed (normal speed 20~40mm/sec.→ high speed 200mm/sec.) • Viscosity change • Clogging in apertures • Smearing
Mounting	<ul style="list-style-type: none"> • Tack time • Tack force • Slump resistance
Reflow	<ul style="list-style-type: none"> • Bridge (short circuit) • Solder bead • Micro solder ball • Tombstoning • Wetting
Inspection	<ul style="list-style-type: none"> • Visual cleanliness (flux residue) • In-circuit tester probe testability
Cleanability (<i>cleaning type pastes</i>)	<ul style="list-style-type: none"> • Visual cleanliness (flux residue) • Ionic contamination

Table 1

No supplier, including Koki, has succeeded in developing a solder paste that will perfectly fulfil all the above requirements.

Thus, we have a variety of products that enable us offer the best-suited product depending on the key requirement(s) from customers.

The following products are what we call ‘main stream products’:

- | | |
|-----------------------|---------------------|
| • SE(S)4-M953 | SE(S)4-M953i |
| • SE(S)48-M954 | SE(S)48-M955 |

We normally recommend M953i for the initial trial, but when the printing process is likely to be frequently stopped or interrupted for some time, you may prefer offer M953. This is because, M953i is designed to be very slump resistant and the thixotropic property is quite high, a certain time period of suspension of printing will allow M953i to recover its viscosity/thixotropy and make the paste stiffer, and, as a result, it might adhere to the squeegee / blade and reduce stencil idle time. More recently M954 should be offered as a general-purpose paste, with the recently developed M955 offering improved printing quality and increased activation.

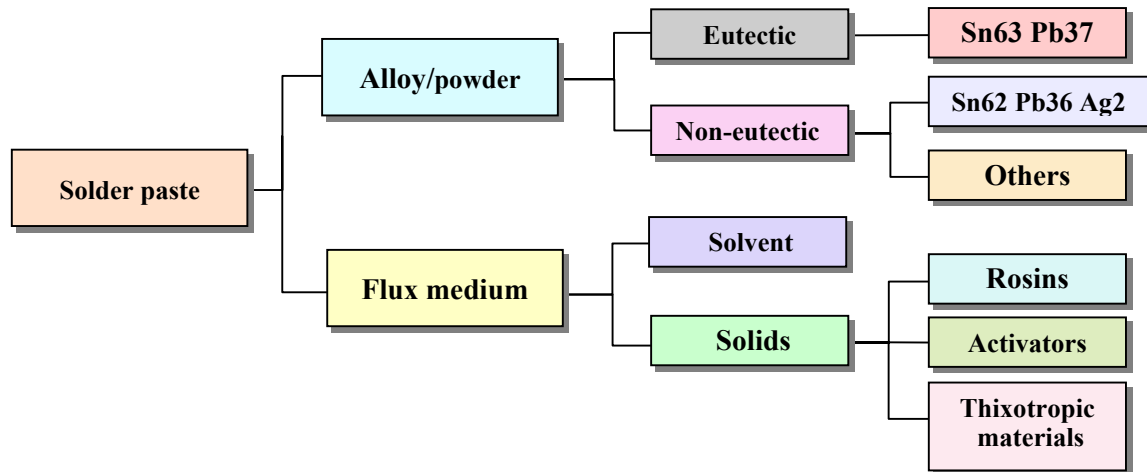
All solder pastes have advantages and disadvantages, so it is critical to verify exact requirements and application in order to offer the best-suited product. In order to meet specific requirements, particularly high-speed printing, better wetting, ICT probability, the following products are available.

- **High speed printing solder paste SE(S)48-M855**
- **Powerful wetting solder paste SE(S)48-M1000-2**
- **ICT testable solder paste SE(S)48-M600**

Koki research and development laboratories are continually developing new solder pastes formulations to cope with the ever increasing quality and product build demands of the electronic industry.

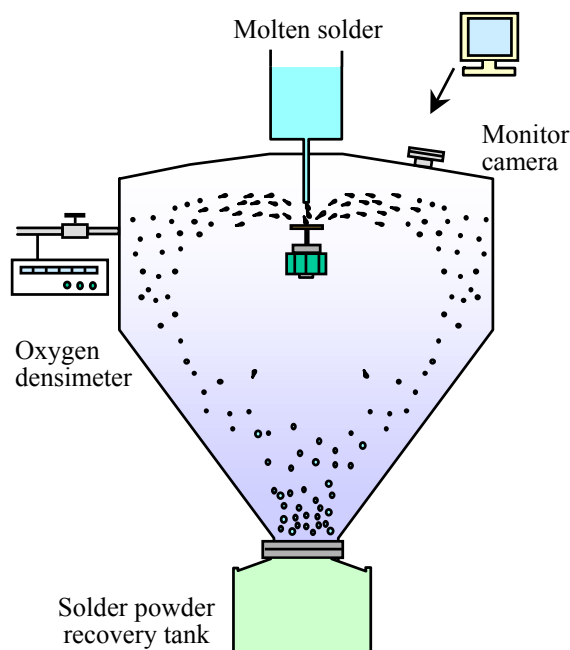
2. Composition of solder pastes

No-clean solder pastes are basically composed of Alloy and Flux.



2.1 Alloy

There are various methods of manufacturing solder powder, such as gas atomisation and centrifugal atomisation. Koki manufactures solder powders by an atomisation method.



Features of this method are;

- 1) Suitable to the manufacture of fine particles.
- 2) Easy to control the oxide film formation on the particle surface.
- 3) Low oxidation level of solder particles.

The powder manufacturing chamber is approx. 5m high and 3m in diameter, which is purged with nitrogen, resulting in a very low oxygen density.

Ingot solder is melted in the solder pot situated at the top of the tank and molten solder is dripped directly on to a high speed spinning spindle. When solder drops hit the spindle, it splashes toward the wall of the tank and before it reaches the wall, the solder becomes spherical and solid.

Solder powder obtained at this production stage ranges from 1~ 100µm. Many factors, such as

solder dripping speed, spindle speed, oxygen density, determines the shape, particle diameter/distribution and oxygen content determines the distribution of particle size.

After this stage, solder powder is brought to classification stages.

Koki has adopted two different powder classification processes. The first classification takes place by using a nitrogen blower. At this stage, utilising the different weight of each particle, particles smaller than those required are removed. The rest of the powder is then moved to mesh sieving to eliminate particles larger than those required.

A series of processes take place under the nitrogen blanket.

Koki basically offer two particle types, either 20~40 μm or 20~50 μm . The former powder is mainly for super fine pitch application (0.4mm/16mil) and the latter is for footprint patterns including (0.5mm/20mil pitch).

There are two key issues on solder powder.

Firstly, particle distribution directly affects the rheology/printability of solder paste, such as the rolling behaviour, separation from the stencil, slump resistance etc. The correct particle size shall be selected according to the minimum pitch and population density of the fine pitch components.

For the purpose of improving the printing performance, we are evaluating the following new powders.

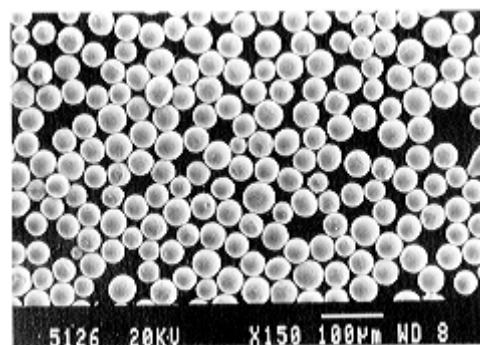
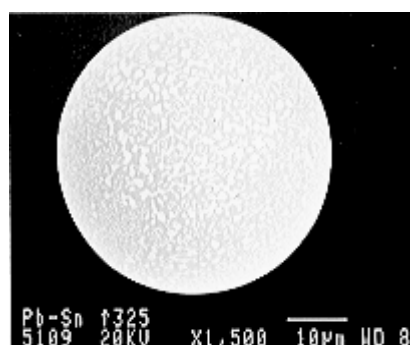
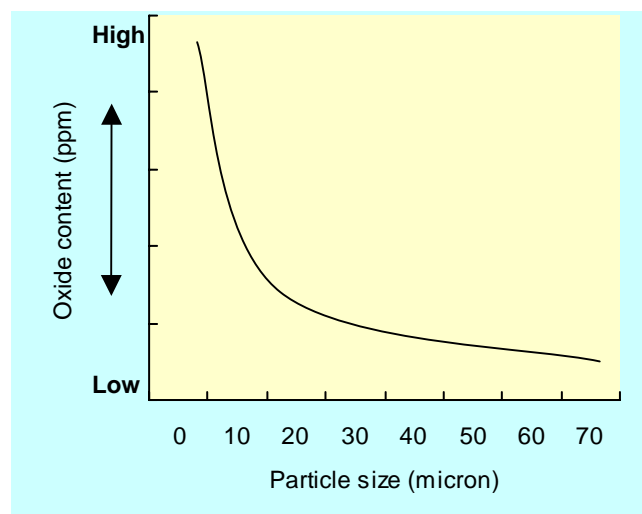
- 15~25 μm
- 20~45 μm

Secondly, oxide content should be carefully controlled as low as possible. In order to minimize the oxide content, the inclusion of particles smaller than 20 μm must be as low as possible. This is because when the particle size decreases the relative surface area increases. As indicated in Fig. 2, particles smaller than 20 μm have a high oxidation level.

If large numbers of high oxide content particles are contained in the solder powder, it can result in microsolder balls when reflowed.

Since a variety of solder pastes are available today, it is important that the correct paste is chosen for the intended application. Careful attention needs to be paid to the particle size distribution and the rigidity of the classification, as these severely affect both the printing performance and solderability.

Fig. 2



2.2 Flux medium

In addition to the solder particles, flux medium is another critical factor for solder pastes, and has a large influence over the entire properties and performances of the solder paste.

Fundamental roles of the flux medium are very similar to those of a wave soldering flux, which are;

1. Elimination of oxide film on substrates

The flux chemically melts and removes the oxide film formed on the surface of electrical components, substrates and solder

2. Prevention of re-oxidation

The substrates, components and solder powder exposed to the heating environment in the reflow oven rapidly oxidise. Solids in the flux soften into a liquid state, effectively covering those soldering parts and prevent them from re-oxidation.

3. Reduction of surface tension of the solder to enhance wetting

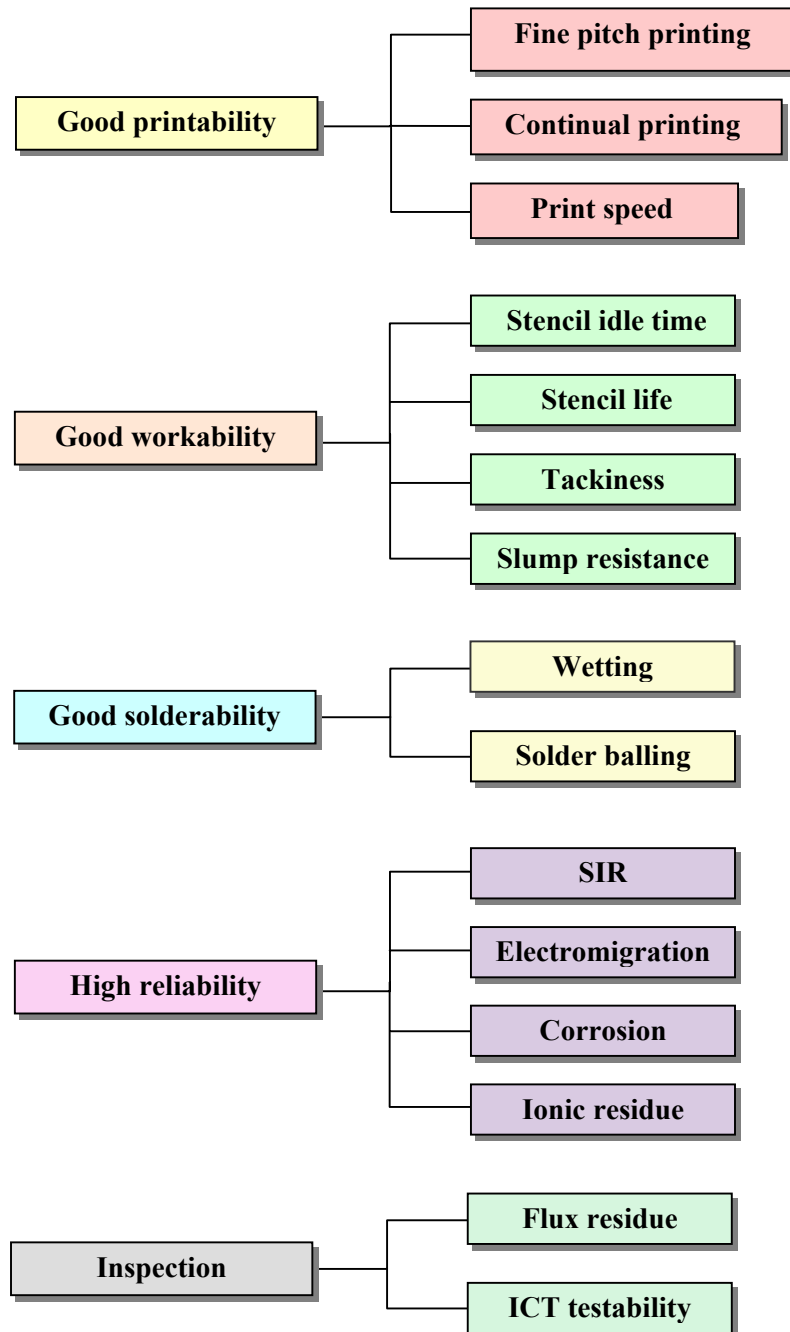
The flux temporarily reduces the surface tension of the molten solder and helps increase the contact area (wetting) with substrate and components.

4. To provide rheology and viscosity in order to make the solder powder printable.

	Component	Responsible for ;	Characteristics
Rosin	<ul style="list-style-type: none"> • WW rosin • Rosin hydride • Disproportionated rosin • Polymerised rosin • Phenol denatured rosin • Ester denatured rosin 	<ul style="list-style-type: none"> • Printability • Solderability • Slump resistance • Tackiness • Colour of residue • ICT testability 	<p>These rosins soften during the pre-heat stage (softening point is around 80~130°C) and spread to the surface of the solder particles and substrate.</p> <p>Rosins Koki normally use are all from natural rosin. Depending on how it is processed, color, activation strength and softening point vary.</p> <p>In order to control workability (slumping, tackiness, etc.) properties, and residue (color, flow, ICT testability) properties, 2~3 different rosins are normally formulated together.</p>
Activator	<ul style="list-style-type: none"> • Amine hydrochloride (halide) • Organic acids • etc. 	<ul style="list-style-type: none"> • Activation strength (solderability) • Reliability (SIR, electromigration, corrosion) • Shelf life 	<p>Especially, these are determinative elements in the strength of oxide removal.</p> <p>Along with softening and liquefying of rosins, these activators wet to metal surfaces and react with oxide substances.</p> <p>Activator is directly influential to the electrical and chemical reliability.</p>
Thixotropic agent	<ul style="list-style-type: none"> • Honey wax, • Caster oil hydride • Aliphatic amide • etc. 	<ul style="list-style-type: none"> • Printability • Viscosity • Thixotropic index • Slump resistance • Odour • Cleanability 	<p>These agents help make the solder paste resistant to sheare stress during printing and recover viscosity after the solder paste is deposited on the pad.</p> <p>Koki employ such agents to improve smooth release of paste from stencil apertures for better printability.</p>
Solvent	<ul style="list-style-type: none"> • Glycoether • Polyhydric alcohol • etc. 	<ul style="list-style-type: none"> • Stencil life / tack time • Slump resistance • Odor 	<p>Depending on boiling point, evaporation rate varies and determines open time and slump properties of solder pastes.</p> <p>Boiling point of the solvents Koki normally uses are about 220~290°C.</p>

3. Practical use of solder paste

In general, the following factors are to be considered when developing or selecting a solder paste.



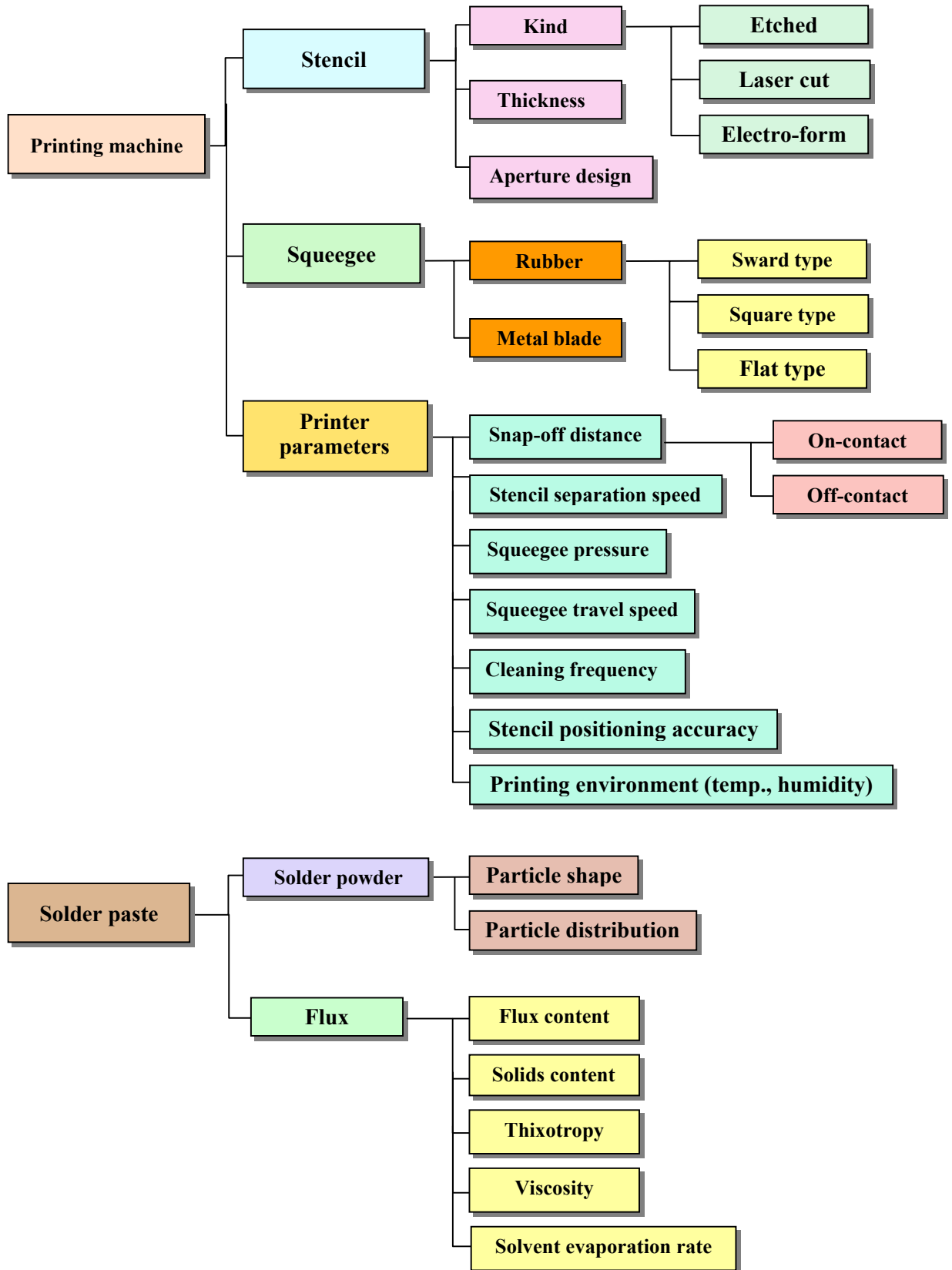
3.1 Printing machine

As the electronics industry develops, there is a growing demand to increase the package density of components on a substrate area. Along with this, performance and quality requirements for solder pastes have become more demanding.

With regard to fine pitch printing, what type of printing equipment and parameters to apply are one of the most crucial factors as well as the performance of the solder paste in order to achieve a high level of printability. This means that even if you have chosen the potentially high performance paste, it is very likely that you will end up with poor printing results because of inadequate parameter setting of

the printing machine or wrong selection of printing instruments, such as squeegee, stencil etc.

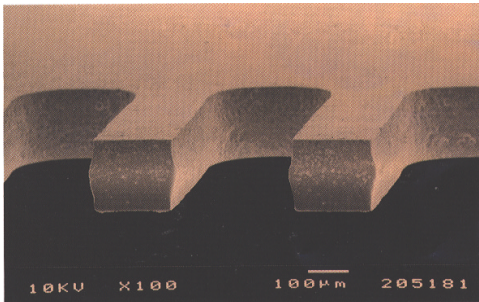
Typically important factors for good printability are ;



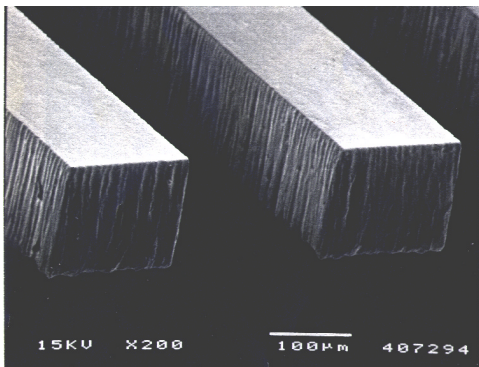
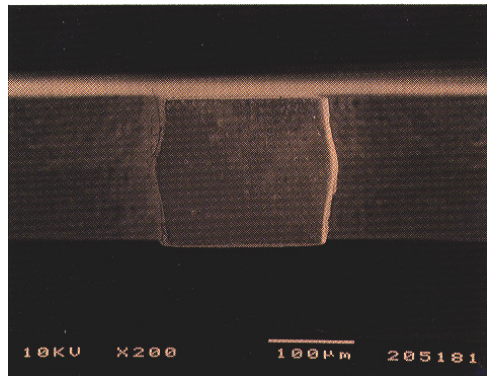
3.1.1 Stencil

There are typically three different types of stencil depending on how it is produced.

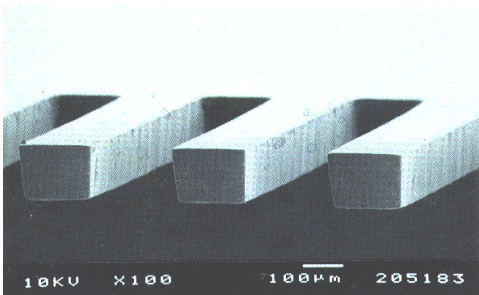
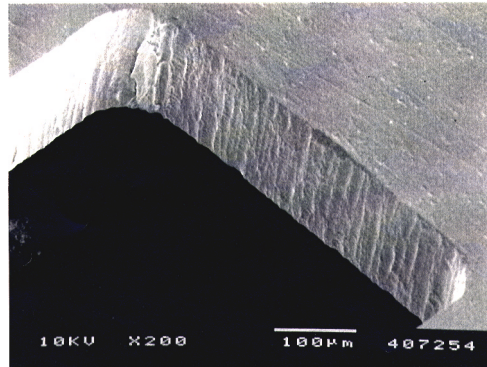
- 1) Chemically etched stencil
- 2) Laser cut stencil
- 3) Electro-form stencil



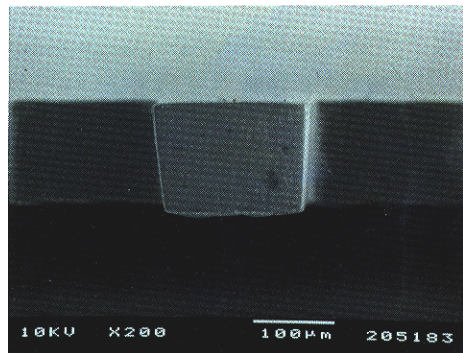
Chemically etched stencil



Laser cut stencil



Electro-form stencil



Chemically etched stencils have been most widely used because they are less expensive and durable. It however, may not be suited for achieving good printability with fine pitch below 0.5mm due to poor surface finish of the aperture walls.

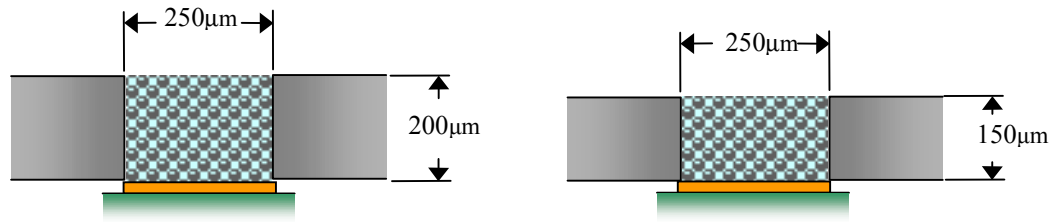
Laser cut stencils have better finishing of the aperture walls in comparison to the chemically etched stencil, but still has some rough surfaces in the aperture walls which may make it difficult to achieve good printing results with ultra fine pitch, such as 0,4mm and micro BGA pads, such as 0.25~0.3mm diameter.

Though the manufacturing cost is high, the electro-form type stencil is best suited for fine/ultra fine pitch printing because of its superior finish of the aperture walls and assures easy release of solder paste filled in it.

Which stencil to use must be carefully selected depending on the process required. We recommend the use of electro-form stencils to ensure quality and consistent printing results whenever fine pitch printing is required.

Thickness of stencil shall be determined according to the minimum size/pitch of stencil apertures. Generally, the thinner the stencil, the better printability it exhibits. This is because thinner stencils gives less shear stress to the solder paste when it is being released from the apertures.

Example



Recommended stencil thickness vs. pitch

Minimum pitch (mm)	Stencil thickness (µm)
> 0.65	150~200
0.5	150
0.4 (0.3mm dia. MBGA)	120~150

It is desirable that the aperture size shall be reduced in relation to that of the PCB pad, taking into consideration stretch of the stencil, tolerance of stencil positioning accuracy, slump of solder paste etc.

Example 0.5mm pitch QFP pattern

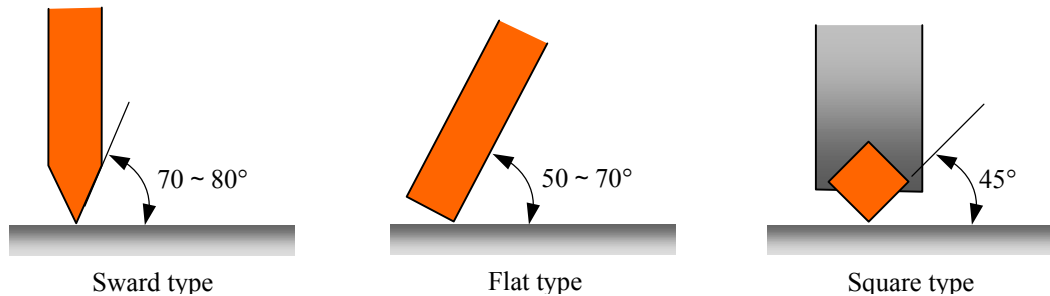


Since how each stencil aperture has been designed could have a major influence on various soldering defect at the end of the production line, such as bridging, solder beading, tombstoning, insufficient solder etc., it must be very carefully designed and controlled.

3.1.2 Squeegee

Squeegees can be classified into either rubber or metal type.

The rubber squeegee can further classified into square flat, and sward type.



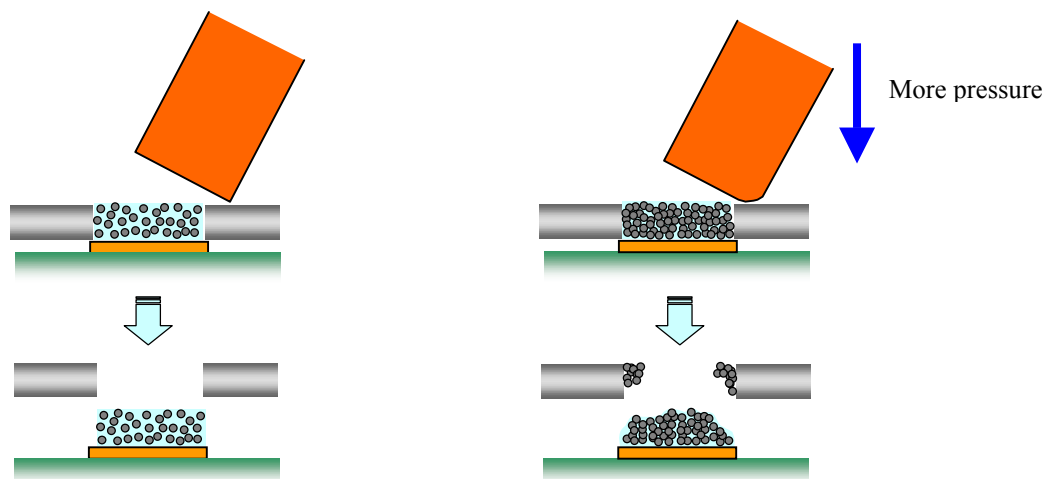
Whilst not easy to judge which type is the best, squeegee angle will directly affect how the solder paste rolls across the stencil. Proper rolling of the solder paste is essential to realise adequate filling into each stencil aperture.

A sward type normally has the angle around $70 \sim 80^\circ$. As the down force is relatively weak, it may be suitable for low viscosity solder paste.

A square squeegee has a 45° angle and its filling pressure of solder paste into the stencil apertures is high, thus making it suitable for high viscosity paste. If low viscosity paste is applied with this type of squeegee print smearing may occur.

A flat squeegee, which has a $50 \sim 60^\circ$ angle, is most popularly used as its angle adjustability provides wider adaptability for different kinds of solder paste.

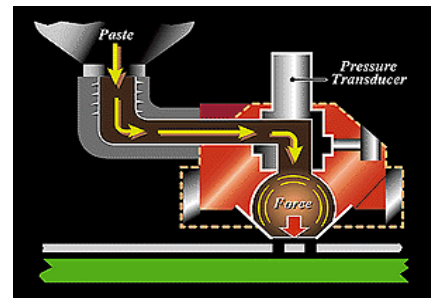
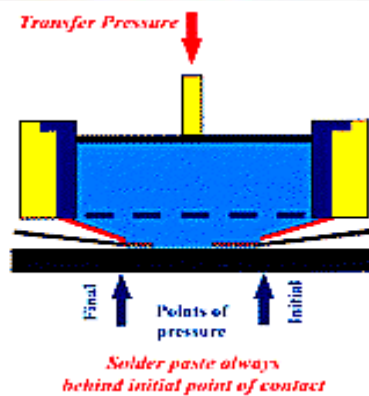
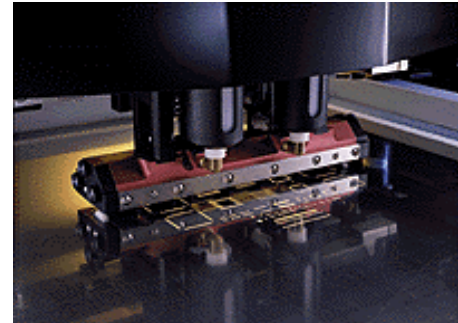
When a rubber squeegee is used, the tip edge must be kept sharp all the time. If it wears, the squeegee pressure is needs to be increased to prevent mapping of the paste. But, if the squeegee pressure becomes too high, it tends to result in pressing excess solder particles into the stencil aperture, as it actually increases down force (or solder paste filling pressure). This leads to increased friction between the solder particles and seriously affects smooth release of the solder paste from the aperture.



Metal blades are also becoming increasingly popular in fine pitch applications because of their resistance to wear, increased working life and do not scoop solder paste from the aperture due to its increased rigidity.

3.1.3 Printer parameters

Today, there are basically two different types of printing processes available, which are conventional squeegee type printing and direct image printing, such as DEK ProFlow, MPM Rheometric Pump Print Head and Ekra Crossflow.



DEK ProFlow
DirEKt Imaging

(HOME)
MPM
Rheometric Pump Print Head

All of the Dek, MPM and the Ekra systems offer direct printing of solder paste from the print head. Such a process makes it possible to print without exposing solder paste to the environment on the stencil and greatly helps prevent deterioration of property, and also realises a clean/safe operation.

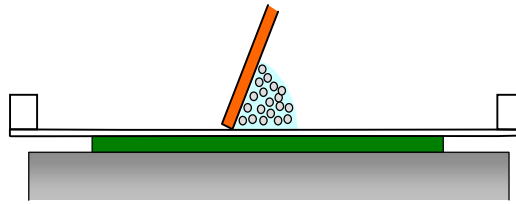
Information given in this section, however, concentrates on the conventional squeegee printing process.

Major functions of the printing machine itself are, stencil separation speed, stencil separation speed control, squeegee speed and pressure control, stencil positioning control etc.

For the purpose of good deposition of the solder paste on to each pad, snap-off distance control is regarded as one of the most critical factors. There are two settings, either 'on-contact' or 'off-contact'.

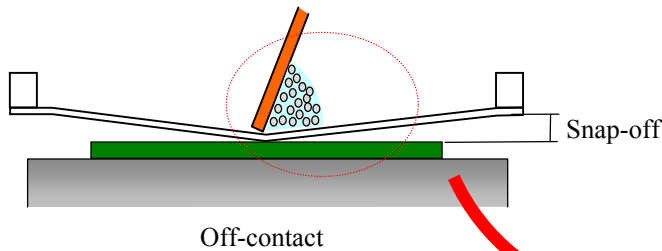
When the printing machine is provided with a stencil (or PC board) separation speed control system, 'on-contact' printing is recommended, especially for fine pitch applications.

This is because on-contact, i.e. there is no distance between the bottom of the stencil and the pad surface, can help prevent smearing of solder paste, which could in turn cause bridging.

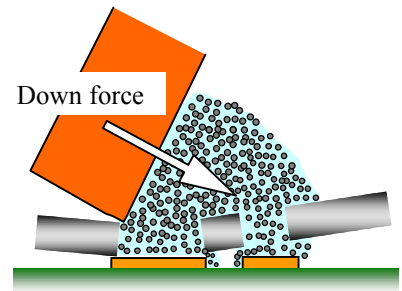


On-contact

‘Off-contact’ should be applied when the stencil (or PC board) separation speed control system is not available, so that the solder paste can be released from stencil apertures smoothly and gradually assuring sharp definition.



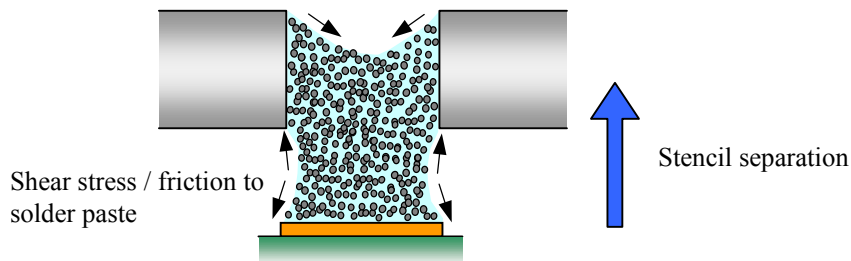
Off-contact



As some distance between the stencil and substrate can cause smearing, careful optimisation of printing parameters, such as squeegee speed, snap-off distance etc., will be required.

Among the printer parameters, stencil separation speed control could be the most important factor in determining the print quality.

Reduction of stencil separation speed reduces friction between the paste and aperture walls, and helps separate the paste smoothly.



Although adjustable, range and steps depend on the printing machine, we recommend it to be as low as possible, such as 0.1mm~0.5mm/sec. As most printing machines are provided with two steps stencil/PC board separation speed control, the stencil will be separated from the substrate at very low speed for the initial distance of 1~2mm and, then, switched to quick motion. Therefore, low separation speed may not affect the print cycle time to a great extent.

Squeegee pressure should be adjusted to the lowest required level. Excess pressure would be a cause of stretch/mispositioning of the stencil, deformation of the squeegee, which may result in flux bleeding and solder paste smearing.

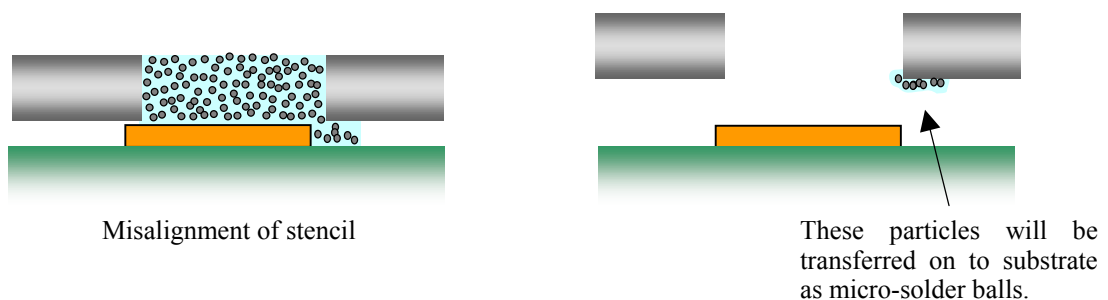
Squeegee travel speed will be determined according to throughput. However, from the solder paste point of view, it is recommended to set it around 20~40mm/sec. for quality printing performance and continual printing, because fast printing induces strong mechanical stress on the solder paste and lowers viscosity and thixotropy.

As a consequence, an excess drop in viscosity and thixotropy causes the solder paste to slump and could cause bridging even at the printing stage and solder beading.

In case fast printing, such as > 100mm/sec. is required, we recommend solder pastes specially developed for this application, SE(S)4-M850 and SE(S)48-M855, which enable printing up to 200mm/sec.

When printing continues to 5, 6, 8....cycles, printing quality deteriorates mainly due to flux bleeding to the bottom surface of the stencil. Once flux starts bleeding, it leads to solder particles and excess solder paste being deposited, which eventually produces bridging in the printed deposit. To prevent this, the stencil should be cleaned periodically. How frequently it should be cleaned all depends on pitches of pattern, nature of solder paste and so on. It is however likely that the finer the pitch becomes, the more frequently cleaning should be carried out to prevent printing and soldering defects.

Alignment of the stencil with the substrate should be very accurate. Otherwise, solder paste smears under the stencil and adversely affects continual printing, and would leave micro solder balls around pads by transferring solder particles from the bottom of the stencil to the substrate.



Controlling the printing environment is also important to realise consistent quality printing. The solder paste is a mixture of solder powder and flux medium, and flux medium is composed of solids (rosins, thixotropic agent etc.) and solvents. Both the solids and solvents are influential over how solder paste behaves during the printing process when environmental temperature varies.

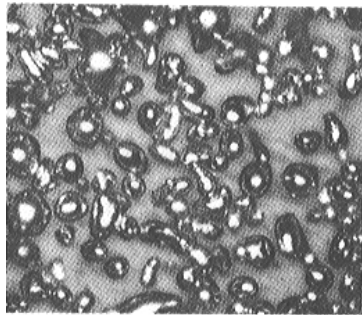
3.1.4 Solder powder

It is commonly understood that solder particle shape and size distribution are important factors in securing successful printing of fine line work.

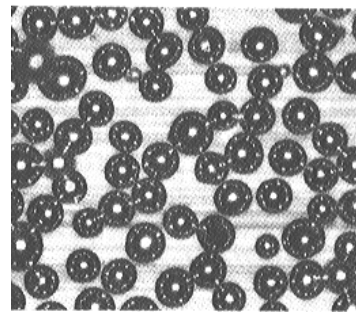
When solder paste was first manufactured, 'irregular' shape solder powder was quite popular.

This was because it was less expensive to produce and there were no fine pitch applications.

But as time went by, along with dramatic innovation of semiconductor technology and the use of fine pitch applications has been accelerated from 0.8 → 0.4mm or even smaller.



Irregular shape solder particle



Spherical solder particle

In such fine line applications, improvement of printing quality is essential to reduce soldering defects, and thus, spherical solder particles have replaced the 'irregular' shaped solder particles, as it provides better rheology.

The finer the pitch patterns and stencil apertures become, the smaller the particle size required. As a reference to determine the suitable solder powder size, it is recommended that the maximum particle diameter shall be 1/4~1/5 of the aperture width.

Until today, Koki offered two powder sizes, either SE(S)4- type powder; 20-50 micron for mainly >0.5mm pitch or SE(S)5- type; 20-40 micron for 0.4mm pitch application.

In order to both improve printing quality and reduce production costs (of 20-40 micron powder), we have started to implement 20-45 micron powder, SE(S)48- type with recently developed products, such as SE(S)48-M855, SE(S)48-M1000-2.

In the future, we will have only one type of solder powder SE(S)48- for normal printing applications.

For super fine pattern applications, such as CSP bumping, flip chip soldering, we now have 15~25 micron solder powder.

Because of its relatively high oxide content, it is highly recommended to combine it with a halide containing flux.

3.1.5 Flux for solder paste

Properties of the flux medium in a solder paste are very influential to its printing properties.

Flux medium for solder paste is a very complex material and different properties of solids content, flux content, thixotropy, viscosity, solvent behaviour etc. synergise each other.

Optimisation of each property is critical to realise quality printing.

The viscosity of our solder pastes for ordinary printing applications are generally ranged from approx.1000 to 2600Ps.

Viscosity of Koki solder pastes are measured by the Malcom viscometer Model : PCU-200 series.

Viscosity of solder paste shall have an influence on its rolling behaviour, smearing, etc.

Thixotropic agents helps make the solder paste resistant to shear stress during the printing process making continual printing possible and recover the viscosity of the paste once deposited on substrates.

Thixotropic agents normally adopted are called ‘wax’.

Not only does it make the paste resistant to mechanical stress by squeegees, but also helps reduce friction between the paste and stencil aperture walls enabling smooth release of the paste.

Solvent evaporation rate is responsible for the stencil life and stencil idle time at the printing stage.

Normally, 1-4 different boiling point solvents (220~290°C) are formulated in the paste mainly to control tack time and slump property. When a fast evaporation rate solvent is formulated in a paste, the paste may dry out on the stencil in a short period of time.

Or, if the printing environment temperature is rather high, for example over 30°C, as solvent evaporates very quickly, the paste starts to stick to the squeegee, clogs in the stencil apertures etc.

More details follow in the ‘Workability’ section.



Malcom viscometer

3.2 Workability

A crucial factor, in addition to printability, solderability and reliability that cannot be neglected when selecting the best suited solder paste for your customer’s production, is whether it has good workability.

3.2.1 Stencil idle time

As substrate patterns become increasingly finer including 0.4 mm pitch, micro BGA (0.3mm dia.) and 0402 chip components, customers are paying more attention to this feature.

Definition of stencil idle time is the ability of the solder paste to be left idle on the stencil and then recover, without kneading, to deliver acceptable prints. The rheological properties of the solder paste, and its inherent ability to transition from its “at rest” viscosity to its “working” viscosity, influence a solder paste’s ability to respond well to idle times or pauses.

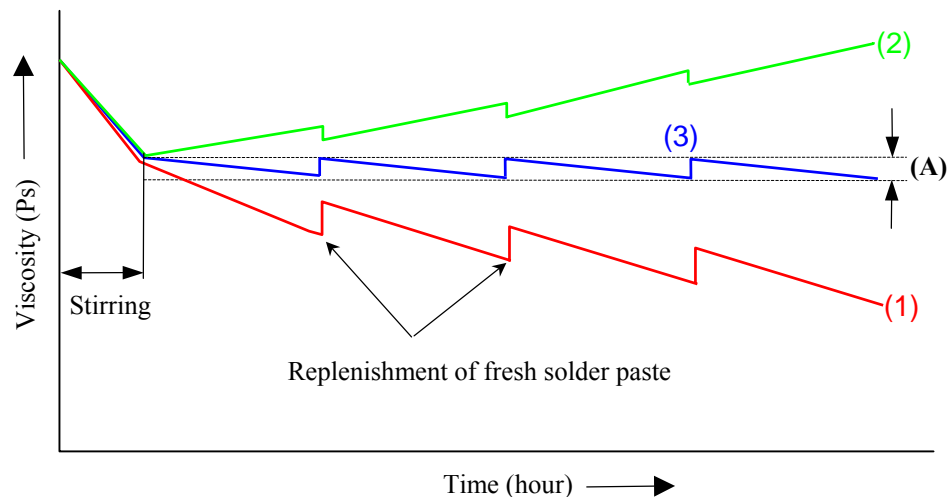
Solder paste which has rather high thixotropy, such as SE(S)4-M953i may have a relatively short stencil idle time, because when it is left at rest for approx. 30 min., the thixotropy will recover and deteriorates the rheology, and will result in poor printing of the first 4-5 prints after printing is resumed.

3.2.2 Stencil life

The length of time that a solder paste can be worked on the stencil and still maintain its properties (printability, print definition, tack, etc.) following multiple printing/idle cycles.

Factors that determine the stencil life are mainly thixotropy and solvent evaporation. For example, if a solder paste does not have good thixotropy (**Line 1**) and is kept printing, the viscosity keeps falling and ends up with lots of soldering defects like bridging and solder beading.

Or, if the solvent used for the solder paste is of quick evaporation (**Line 2**), the viscosity of the solder paste becomes excessively high and starts sticking to the squeegee and clogging in the stencil apertures.



Thus, **Line 3** examples ideal a long stencil life solder paste. The viscosity variation (A) in continual printing is quite small.

Though most of Koki's solder pastes have rather small viscosity drops in continual printing, it is still highly recommended to control the ambient temperature to below 25°C to prevent excess evaporation of solvents.

3.2.3 Tackiness

Tackiness can be classified into two terms, tack time and tack force.

Tack time is the length of time that a solder paste can stay tacky enough to hold a component in position after printing. Tack force indicates the strength of adhesion that the solder paste exhibits in order to hold a component.

We determine the tack time of our solder pastes by using the Malcom Tack tester.

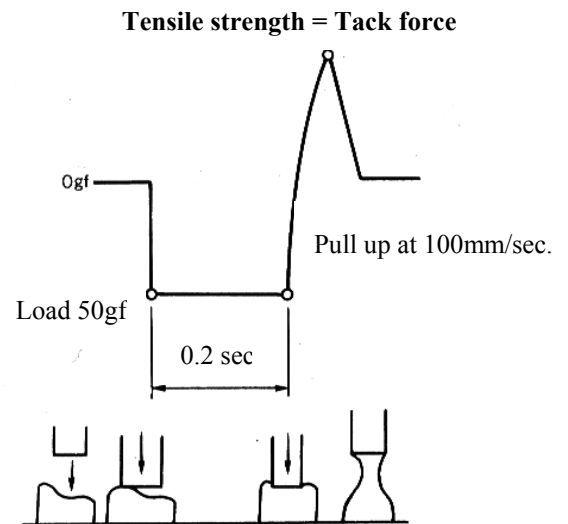
5 deposits of solder paste are printed on a ceramic plate by using a 200µm thick, 6.5mm diameter aperture.

Then, a test probe is pressed onto each deposit for 0.2sec. with a load of 50g and it is then pulled back up at 100mm/sec. and the tensile strength is measured.

By repeating this test, we determine how long the paste stays sticky enough (we standardise minimum adhesion required to hold a component as more than 100gf.).



Malcom Tack tester



What largely contributes to the tack time characteristics, are solvents. Tackiness itself is correlated with stencil life and slump resistance, as the evaporation rate of solvent is the main determining factor. It is normally desirable to use a higher boiling point solvent as it evaporates at a slower rate and preserves long tack time. Some concern is required as to what type of solvent is to be used, because it in turn tends to cause slumping of the solder paste when a high boiling point solvent is formulated. Solvents we normally formulate are of a boiling point ranged from 230~300°C. Furthermore, a combination of 2~3 different solvents is quite normal practice to obtain the best compromise in terms of tack performance, viscosity, odour and compatibility with the rest of compositions.

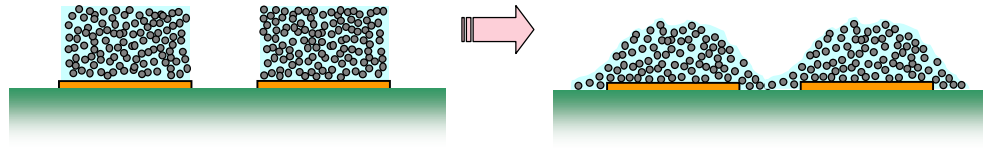
3.2.4 Slump resistance

In the main there are three different modes of slump.

1. Shear slump : Slump which occurs with continual printing by shear stress.
2. Static slump : Slump which occurs at room temperature without any shear stress.
3. Heat slump : Slump which occurs during the heating process, i.e. pre-heating and reflowing.

As the space between pads is decreasing due to the demand for higher packaging density, the slump properties of the solder paste is now regarded as one of the most important factors for the fine pitch solder paste.

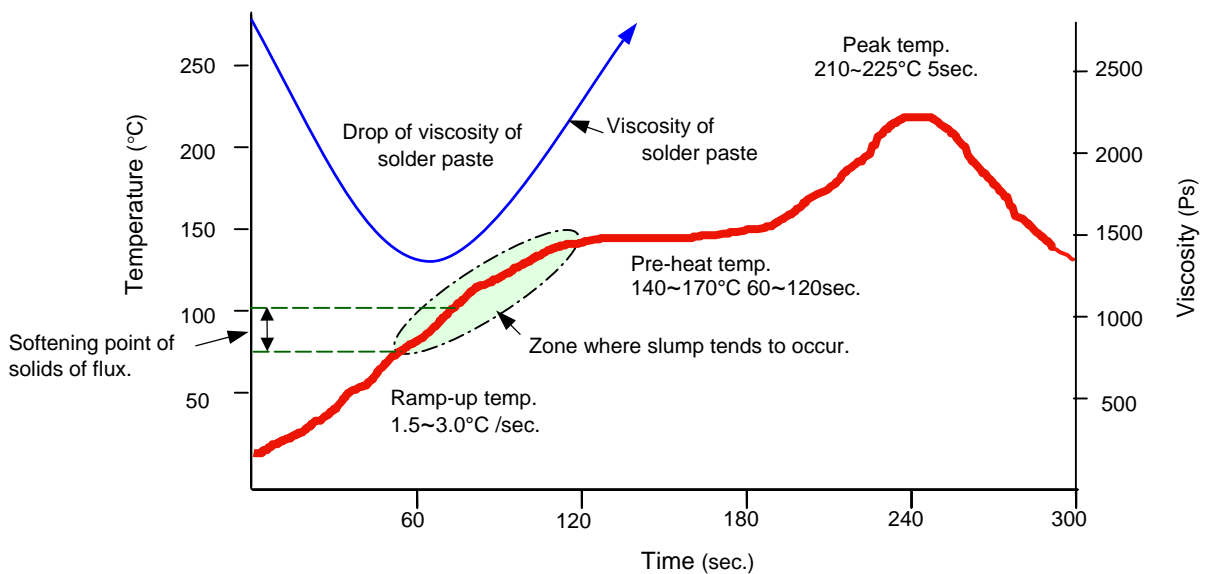
Should the solder paste slump after printing and spread around the pads, it causes bridging with neighbouring pads, micro solder balls, etc.



There are various factors which affect the slump properties of solder pastes, such as flux content, viscosity, softening property of rosins and thixotropic agents, boiling point of solvents etc.

Heat slump resistance is the most critical feature among the above three modes for quality soldering, as most slumping tends to occur during the heating process due to softening of the flux.

When reflowing, the solder paste is exposed to temperature as below.



As the temperature elevates at the first ramp up stage, solids materials (rosins, thixotropic agents, activators) soften dramatically in combination with solvent left in the flux. Then, in proportion with evaporation of the solvent, the viscosity starts increasing.

Generally, soldering defects such as bridging and solder beading are due to paste slump at this stage.

In order to reduce slumping, it is necessary to select a proper combination of different boiling point solvents and thixotropic agents.

When evaluating the slumping properties of solder pastes we normally use ISO (or IPC) standards, which are indicated in the Technical Information sheets of our solder pastes.

3.3 Good solderability

3.3.1 Wetting

In a production process, repair of products gets more difficult and costly when defects happen down stream on the production line. Poor wetting or non-wetting is a very critical defect as it is difficult to detect and can cause a serious claim after products have been delivered to the market.

Products can often be situated in an environment where temperature and humidity keeps changing, and vibration or shocks are continuous. If solder joints are not strong enough, cracks in the solder joints can occur, and as the worst case, electronic components would dislocate.

Thus, wetting of the solder paste is often given the highest priority among the key features required, such as printability, tack time, etc.

‘Wetting’ is a phenomenon where a liquid solder spreads to the surface of the terminal of a component.

When the liquid is put on the surface of the solid, the liquid forms into a certain shape (Fig.-3) according to the property of each liquid and solid. Angle θ where the liquid surface at the contact point of the liquid and solids meet is called the ‘contact angle’. This contact angle is used as a scale for wetting.

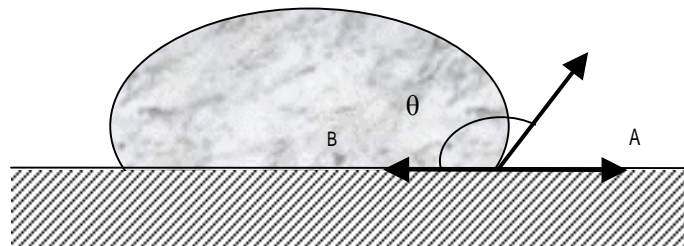


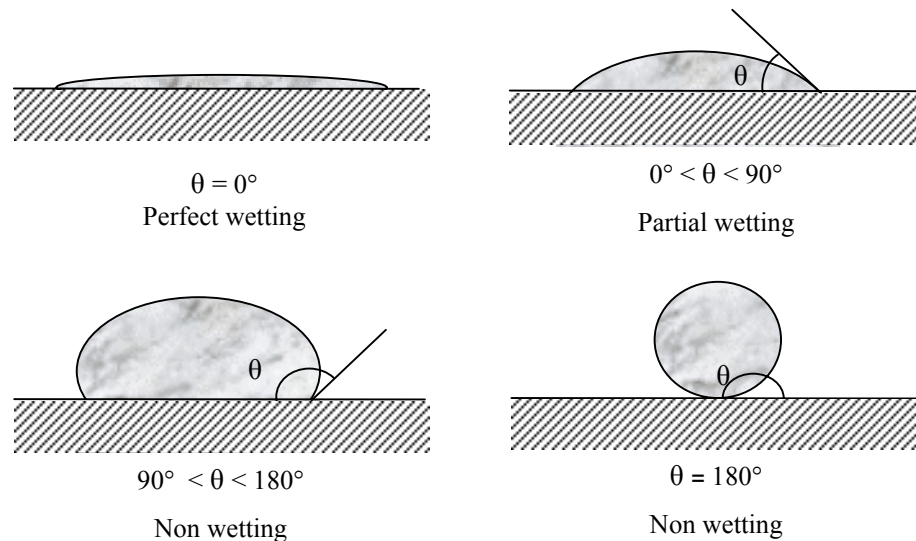
Fig.-3

This wetting balance is expressed below by Young’s equation.

$$Y_A = Y_B + Y_C \cos \theta$$

- Y_A : Surface tension
- Y_B : Surface tension between solid and liquid
- Y_C : Surface tension of liquid
- θ : Contact angle

$Y_C \cos \theta$ is called as ‘adhesion tension’, wetting will be indicated as follows:



When the oxide film is formed on the surface of the substrate, it becomes a major obstacle to achieving good wetting of the solder.

Flux plays important roles in achieving good wetting and its main functions are;

1. Removal of oxide film from the surface of the substrate and solder.
2. Reduction of surface tension of the solder.
3. Prevention of re-oxidation during the heating process.

As explained earlier, flux is composed of rosins, activators, thixotropic agents and solvents. Rosins are responsible mainly for **2** and **3** above, activators are mainly responsible for **1**. Thixotropic agents and solvents are not effective for solderability.

Poor wetting is normally seen at lead wires of IC and QFP components, and tends to occur due to the following causes;

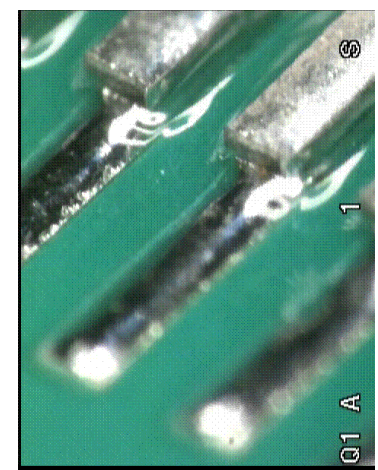
1. Severe oxidation of the base metal, such as copper, phosphor bronze.

When producing leaded package, the leads are first plated with solder, leads formed, then to the designated length.

i.e. Bare alloy is exposed immediately after manufacture of such component and gets oxidised during storage.

2. Employment of difficult-to-solder base metal, such as Alloy 42 (Fe, Ni base metal).

As the pitch of the lead wires have become increasingly small, 0.65 → 0.5 → 0.4mm, the lead wire itself has become thinner. In other words, the fine pitch lead wires can be bent very easily and affect the planarity of the leads.



Poor wetting

In order to provide rigidity to the lead wires, alloys such as Fe and Ni are being used.

However, once the oxidation film is formed on such alloys, it is hard to remove.

Therefore, the activation strength or oxide film removal force of the activators in the flux determines the wetting of solder paste.

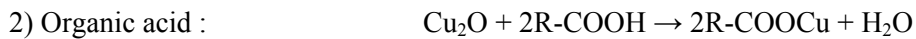
Activators are classified into the following two kinds.

- Halide base (most of them are Amine hydrochloride)
- Organic acids base

Both of them have a properties to react with metal oxide and clean.

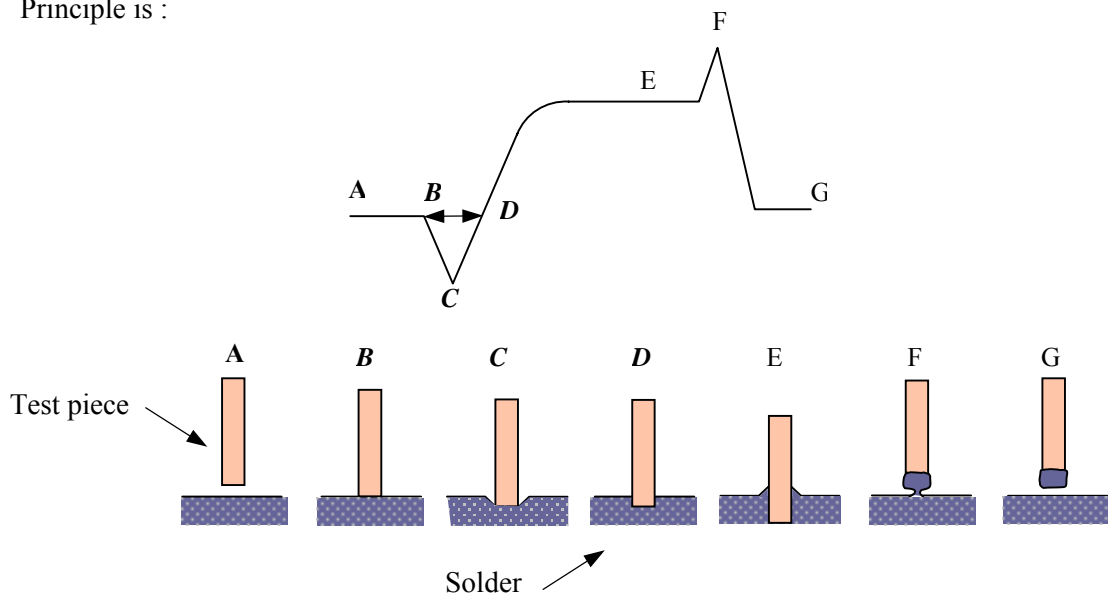
Reactivity differs depending on each component, but in general, halide has stronger activation but has more chance of corrosion than organic acids.

Reaction of each halogen and organic acid with copper oxide, as an example, is expressed as follows:



Efficiency of wetting can be attributed to speed and force in removing the oxide film. Wetting can be measured and indicated as wetting speed by a Meniscograph. The quick wetting is translated as powerful and fast removal of the oxide film. This phenomenon can be well demonstrated by Meniscograph measurement.

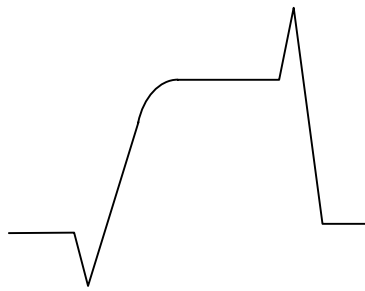
Principle is :



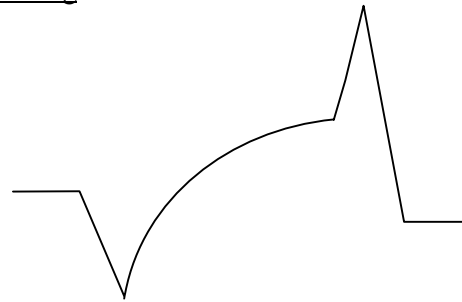
- A : Original position.
- B : Point where tip of test piece comes in contact with solder.
- C : Point where wetting starts after test piece is immersed 0.2mm into solder.
At this point, solder is pushing up test piece.
- D : Solder has started to wet and contact angle has become 90°.
- E : Maximum wetting.
- F : Point where solder is separated from test piece.
- G : Original position.

Wetting speed is expressed by the distance (time) between B - D.

Ex. Good wetting



Poor wetting



Referential wetting speed of each Halide containing (0.2%) and Halide free flux are approx. 0.5~1sec. and approx. 1.0~1.5sec. respectively.

From these figures, it is observed that Halide is quite efficient in removing the oxide film.

In classifying the activation level of a flux, MIL standard is commonly used. MIL QQ-S-571 classified the flux type R (rosin base), type RMA (mildly activated rosin), type RA (activated rosin base), type LR (low residue fluxes).

MIL standard was discontinued and has been replaced by ANSI/J-STD-004.

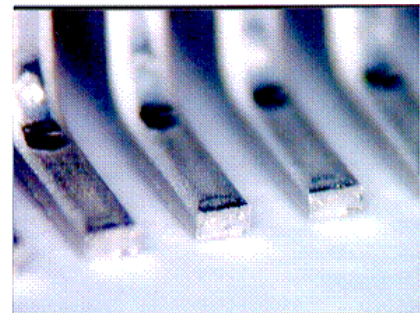
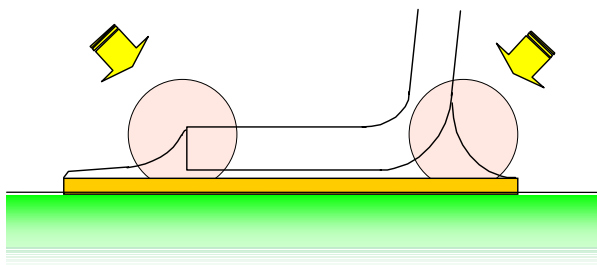
Koki's range of solder pastes can be classified into 3 categories according to ANSI,

- 1) halide free (0% halide) ROL0 (M953, M850, and etc.)
- 2) low halide (0.03% halide) ROL1 (CA/H-00-620M and etc.)
- 3) halide type (0.2%). ROM1 (SE/S5-A310, CH-10-763K and etc.)

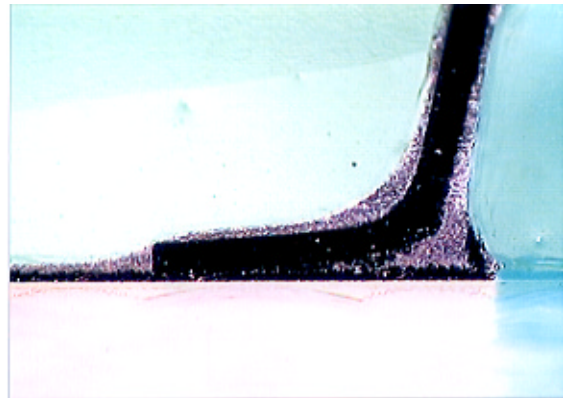
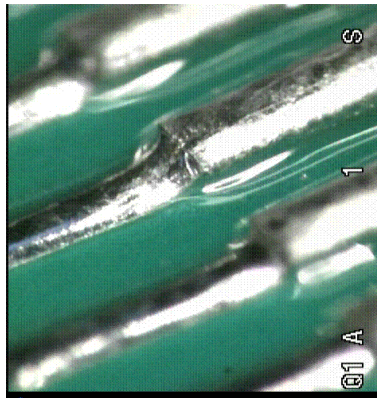
Though it is commonly known that halide containing solder paste can effectively improve wetting (and is popularly used in Japan), due to some concern (or tradition) for electrical and chemical reliability, halide free products are widely used in Europe and the U.S.A.

In order to meet the market demand which is to overcome difficult-to-solve poor wetting problems without any halide and deterioration of reliability, we've developed 'Powerful wetting' no-clean solder paste SE(S)48-M1000-2 (halide free). (*Please refer to the separate 'Product Information' for details.)

When observing 'wetting', the tip and back heel of the lead wire are normally checked.



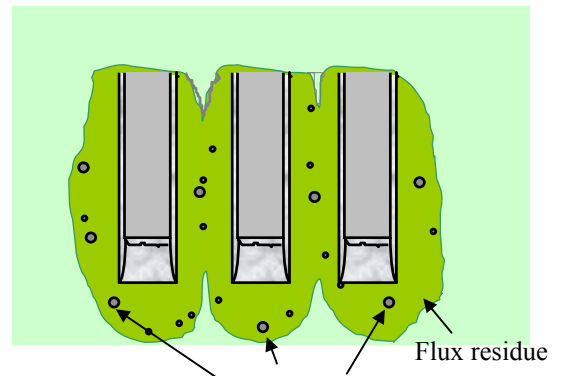
SE(S)48-M1000-2 offers excellent wetting.



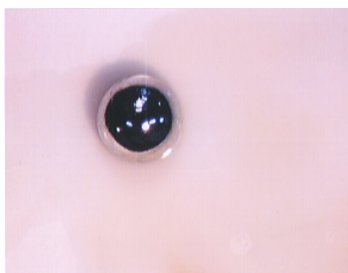
3.3.2 Solder balling

There are two types of solder balling problem, which are Micro-solder balls and Solder beading. Possible causes of micro-solder balls are;

- Initial oxidation of the solder powder
- Oxidation of solder powder during the reflow process
- Slumping of solder paste, after printing, at component placement and during reflow
- Transfer of solder particles from the stencil



Micro-solder balling problems that are attributed to the nature of the solder paste are initially due to the quality of the solder powder.



Solder ball test by ISO

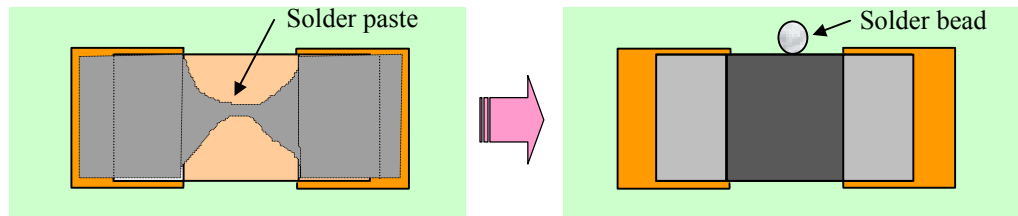
If solder powder is badly oxidised at the production stage (powder manufacture, classification, mixture with flux medium) for some reason, micro-solder balling can occur. As the oxidised particles require aggressive activation to be removed and do not melt at normal reflow temperature, such particles are carried away from the soldering pads along with the spreading of flux and leave satellite solder balls.

At our production, quality (oxidation level) inspection of solder powders is repeated at least twice, immediately after powder classification and mixture with flux medium, according to the ISO standard with a ceramic plate (*See Technical Information sheets of any Koki solder paste.).

Once the solder powder is properly mixed with flux media, oxidation can hardly occur as the flux encapsulates and isolates each particle from air exposure.

Thus, if some micro-solder balling happens at customers' production, have a close watch how the problem has happened having in mind the other possible causes.

Solder beading occurs as solder paste deposited on each pad under chip components becomes soft due to temperature elevation during reflow and flows toward the centre of the component by capillary effect. It finally pops out the side of the component as a solder bead when the temperature reaches the melting point

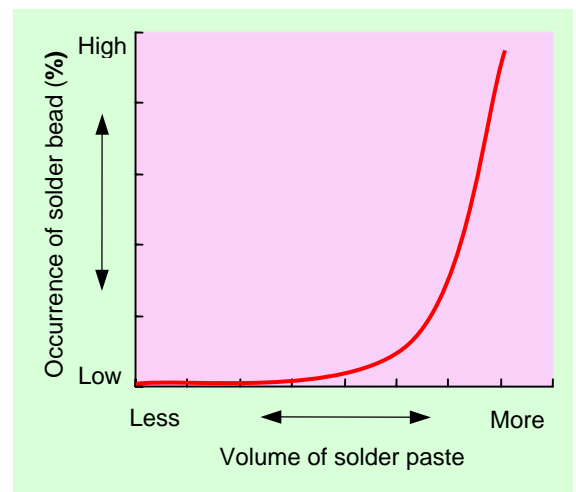


It is quite difficult to overcome this problem simply by changing the flux formulation of the solder paste.

The occurrence of this problem largely depends on the volume of solder paste printed on each pad, it is desirable to control stencil parameters, such as aperture size to pad size, stencil thickness.

From a solder paste point of view, high thixotropy and high viscosity paste, such as M953i and short tack time paste, like M951K, tend to produce less solder beads as these solder pastes have less flow of flux during the reflow process.

But, still some adjustment of the stencil will be required to completely eliminate the solder beads.



3.3 Reflow profile

During the reflow heating process, the state and action of the solder paste changes more or less as follows:

1. First ramp-up :

By elevating the temperature, solvents start evaporating. Evaporation rates differ depending on the boiling point of each solvent employed.

In the same way, rosins and thixotropic materials start getting softer. How each material softens, depends on its softening point. In general, the softening point of solder paste is at around 100°C.

Thus, when the ramp-up rate is too steep, it softens the solids of the flux whilst a large portion of the solvent remains and makes the solder paste watery, and can be the cause of slumping → solder beading and bridging.

2. Pre-heat stage :

This stage is required to volatilise the solvent completely and distribute the heat uniformly to the substrate.

The flux becomes soft like liquid, and uniformly encapsulates the solder particles and spread over the substrate, preventing them from being re-oxidised.

Also, along with the elevation of temperature and liquefaction of the flux, each activator and rosin become activated and start eliminating the oxide film formed on the surface of each solder particle and the substrate.

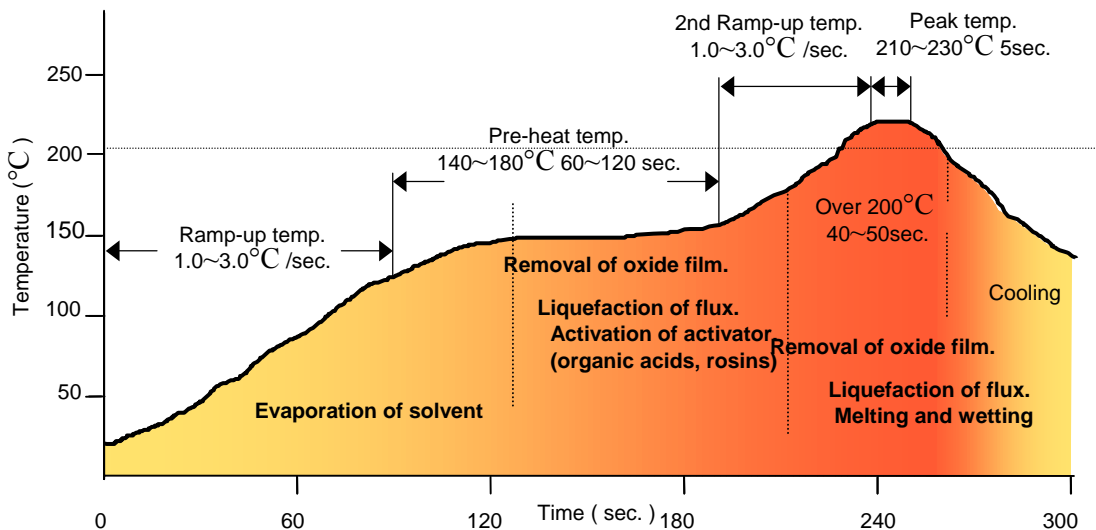
3. Over 200°C :

When solder particles melt by reaching their melting point, in reaction with the flux medium, oxide gets eliminated and soldering takes place.

A reason why the condition ‘over 200°C 30~40sec.’ is recommended is to secure complete melting of the solder and certain wetting time is required in case some high heat capacity components are mounted.

If bridging by heat slumping occurs, it is determined at the first ramp-up to the early soaking stage. What causes bridging here is inadequate heat distribution to each pad and lead wire.

When the temperature of the lead wire becomes higher faster than that of the pad, then molten solder tends to wick up the lead wire. As excess molten solder swells at the heel of the lead it comes in contact with the solder of the neighbouring lead wire. Thus if bridging occurs, it is important to figure out very carefully how it happened.



In general, typically two different types of reflow profile have been popularly used, ‘linear’ type and ‘saddle’ type. Though we normally recommend the above ‘saddle’ type profile, let’s think how each profile has been created.

When surface mount components and reflow assembly process started to be used, population of components was not high and heat capacity difference among components was relatively small. Such simple board configurations allowed an easy-to-design gradual ramp-up/no-soaking zone profile to be used without major problems.

Advancement in downsizing of PC board design led to the requirement of dense population of components and accelerated implementation of high heat capacity package components such as ICs and QFPs.

Increased heat capacity difference among components made it difficult to achieve good thermal equilibrium by using the gradual type profile (or even the saddle type profile) in combination with the conventional IR reflow oven due to the uneven heat mass distribution, different heating efficiency depended on the colour of components and shadow effects.

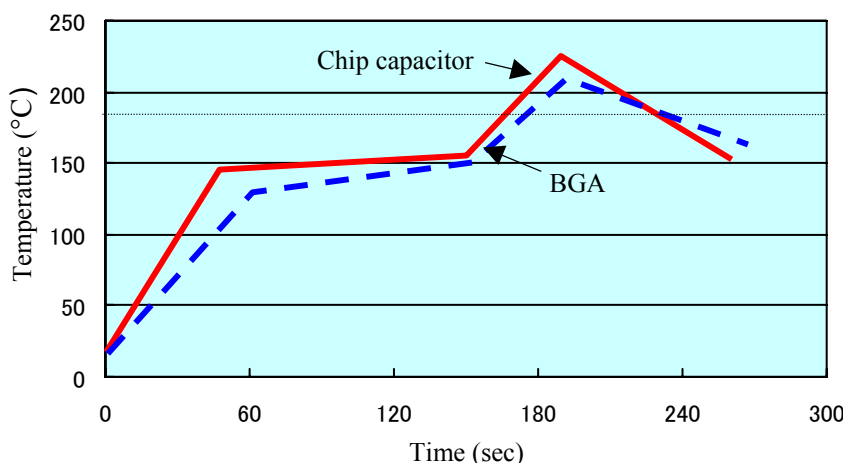
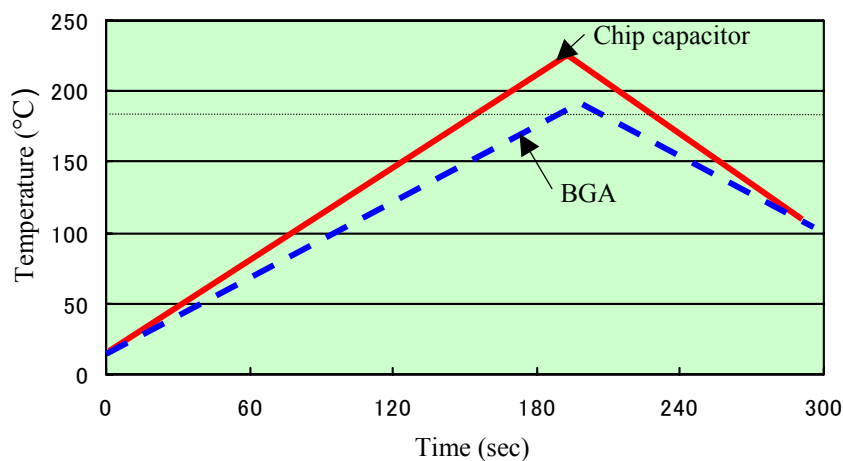
Then, a vapour phase reflow process, which is capable in realising extremely good thermal equilibrium, once become popular. But, due to several problems, such as cracking and tombstoning of components by abrupt heat elevation, toxicity of solvent, abolition of CFC solvent etc., this process became less popular quickly.

Forced air convection reflow processed developed in turn, has won a large popularity because of far better heating uniformity in comparison with IR reflow.

A reason why the saddle type reflow profile has been widely used is mainly and simply for the purpose to achieve a soaking zone, as similar thermal equilibrium at the peak reflow zone to vapour phase process is possible with the help of a forced convection reflow process.

We believe it is very important to achieve better production yields that the priority of how the reflow profile to be designed, will always give consideration to the kinds of components and the design of the substrate itself, rather than how solder paste behaves during the heating process.

For example, when the temperature of soldering of two different components, say a chip capacitor and a BGA is measured, peak temperature difference between these components will be large as the temperature elevation rate of each component is different, according to their heat capacity.

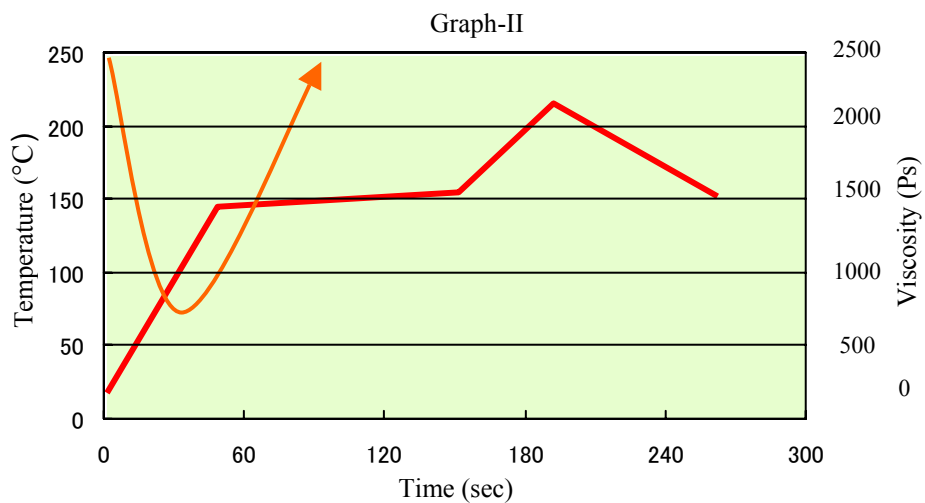
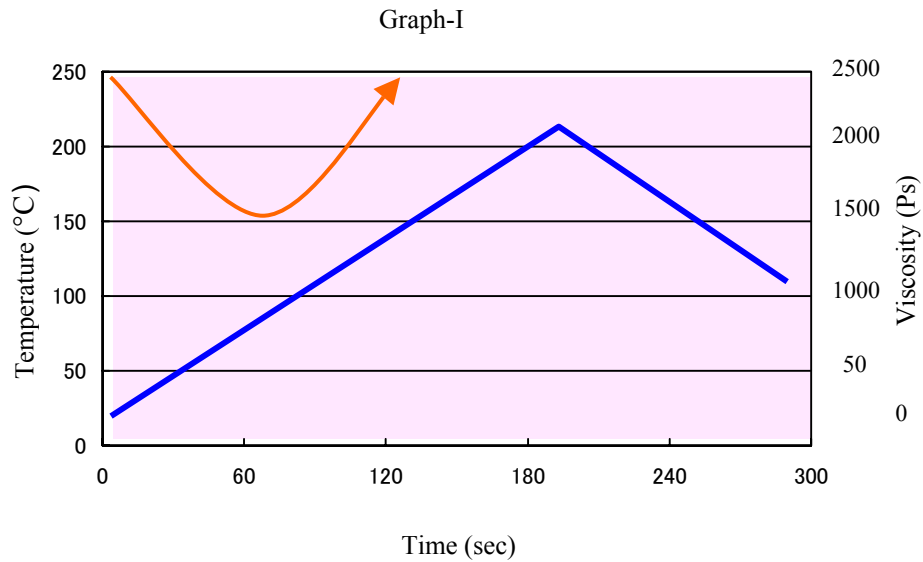


A soaking zone of the saddle type reflow profile, on the contrary, allows the temperature of relatively large heat capacity components to catch up with that of the rest of components before the second ramp-up starts. It makes the temperature difference of each component at the soldering temperature much smaller.

Therefore, in terms of realising better equilibrium among components, it is desirable to have a saddle type reflow profile.

From a solder paste point of view, it is correct that as temperature elevates, solder paste tends to become soft and can cause slumping and result in solder beads, bridging and other soldering defects if the first ramp-up is too steep and/or flux formulation is incorrect.

Slow ramp-up helps evaporate solvents while making the pace of softening of the rosins and thixotropic materials slower, and eventually helps to prevent solder beads, bridging and other defects.



As the above Graph-II indicates, the steep ramp-up can create a steep drop in solder paste viscosity. Therefore, what has been done conventionally for solder pastes to prevent this is to formulate lower boiling point solvents, so that majority of solvent can evaporate before the temperature reaches the

softening point of rosins and other solids materials. This however could result in short stencil and tack life of the pastes.

Having in mind that the saddle type reflow profile is desirable to ensure good solder joint quality regardless of the size/heat capacity of components, we have developed a special thixotropic formulation which ensures extremely high slump resistance even under a relatively steep ramp-up, and made it possible to adopt high boiling point solvents for extended stencil and tack life at a same time. This new thixotropic formulation has been applied to most of Koki solder pastes since SE(S)-M953i was developed. (ex. M850, M954)

Such a flux formulation has succeeded in making the solder pastes much less dependent on the design of the reflow profile, and as a result, to the widen process window of the soldering process.

Our conclusion is that customers can adopt either type of profile, gradual or saddle type, so far as Koki solder pastes and most solder pastes are concerned, if a certain level of heat energy to evaporate all solvents is ensured. But our recommendation as to what design of reflow profile to apply, will be determined by the full understanding of the heat equilibrium of the substrate and components to secure quality solder joints on every component, rather than worrying to much about how the solder paste behaves during the reflow process.

According to our experiences, how to design stencil apertures will have more of a practical influence over the prevention of soldering defects such as solder beading and bridging, than spending a lot of time in changing the design of the profile.

3.5 Reliability

In a no-clean process, as post reflow residues are left on the substrate, the residue must be reliable. As to how high a reliability is required, all depends on the finished products that customers manufacture. When the product is directly related with human lives, such as engine control units, ABS for automotive, medical products etc.

As explained earlier, reliability and solderability are contrary to each other because activators (organic acids, halides) are more or less ionic and corrosive and could lower reliability depending on how they are formulated in the flux system.

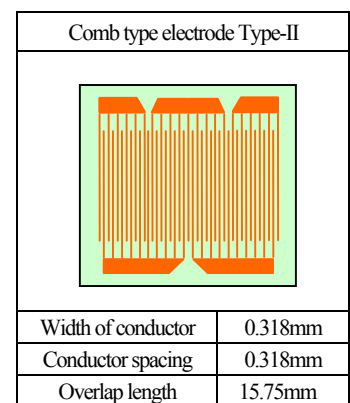
In order to determine, reliability of a product, the following properties are typically inspected.

- Surface insulation resistance

This test is to characterise the flux by determining the degradation of electrical insulation resistance of a rigid comb type electrode specimen due to deleterious effects of high humidity and heat conditions.

We normally follow the IPC-TM-650 method and climate condition is 85°C, 85%RH.

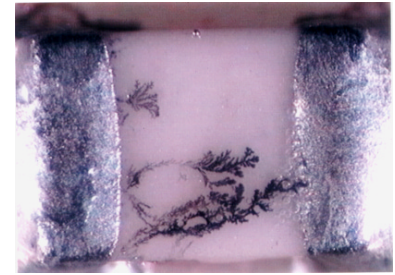
If the flux residue, which is composed of rosins, activators and thixotropic agents, is hygroscopic and gets conductive even partially, insulation resistance drops.



- Electromigration

Electromigration is a phenomenon by which a metallic filament (Ag, Sn, Cu and etc.) grows (dendritic growth) from the cathode to the anode by DC voltage.

When the flux residue, which lies between the terminals, is ionic, and is exposed to a high humidity environment or condensed with water, it causes current leakage and thus leads to dendritic growth.



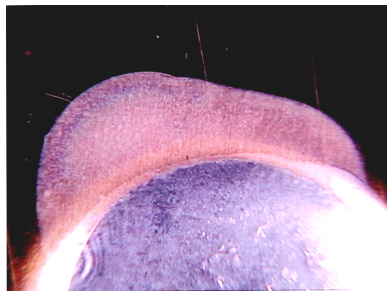
Dendritic growth

The test method is similar to SIR, but with voltage application.

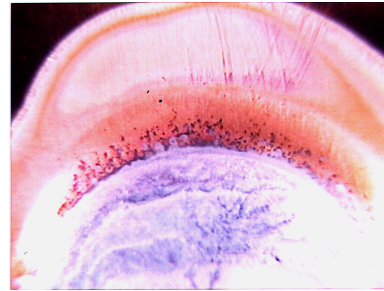
- Corrosion

There are typically two methods of testing the corrosiveness of a flux, which are, the copper plate corrosion test and the copper mirror corrosion test.

Depending on which standard, (JIS, IPC, and etc.) test methods are slightly different. See each standard for detailed test procedures.



No corrosion



Corrosion with black spots

- Ionic contamination

This test method is specified by MIL standard.

By soaking the post reflow PCB in de-ionised water/isopropyl alcohol solution of an ionograph system, ionic residue can be obtained as resistance.

Resistance value is to be converted into NaCl ($\mu\text{g}/\text{cm}^2$).

MIL specifies ionic residue level required for RMA products to be less than $3.1\mu\text{g}/\text{cm}^2$.

However, since this test is actually designed to measure the ionic contamination of not only the flux residue of solder paste, but also the substrate and components as a whole, the value obtained shall be regarded only as referential information.

3.5 Inspection

3.5.1 Flux residue

Flux medium of the solder paste is composed of solvent and solids (rosins, activators, thixotropic agents). Solids content of most of our solder pastes is approx. 60~70% and most of it remains on the substrate after reflow as a flux residue.

Since this amount of solids is required to obtain certain rheology and solderability values, it is technically difficult to reduce, unless nitrogen reflow is applied.

What we normally pay attention to when formulating a new flux medium, is to use rosins which result in as light colour as possible after reflow, so that it can offer a superior cosmetic appearance.

3.5.2 In-circuit tester testability

Increasing demand for downsizing of electronic products has seen electric components getting smaller and smaller and the density of component population, higher and higher. This has made it difficult to secure provision of test pads for in-circuit test (ICT). Since probes directly come in contact with solder fillets when no test pads are provided, how and in what state flux residue is left on the solder joints is an important factor in determining good testability.

Typical factors which cause poor probing, and conventional measures to cope with it are as shown in the table below.

Factor	Measures
Volume of flux residue	As thicker residue deteriorates probing and conductivity, reduce solids content as much as possible.
Spreading of flux	Similar to the above, it is desirable to have a thinner flux residue on the surface of solder fillet for easier contact. Flux should be designed to flow around the solder fillet at reflow leaving as thin a layer of residue as possible.
Hardness of flux residue	Hard flux residue cracks by being struck with a probe. As pieces of the residue stick on to the tip of the probe creating poor conductivity, flux components should be selected to ensure a soft flux residue.
Test equipment (Probe type, contact pressure, contact location and etc.)	Use of single tip pin, increase of contact pressure etc.

It is, however, difficult to realise good testability with the above mentioned techniques for the following reasons.

1. Reduction of solids content

It adversely affects other performances of the solder paste, like printability, tack life, and solderability.

2. Spreading of flux

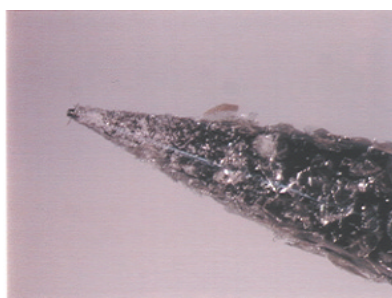
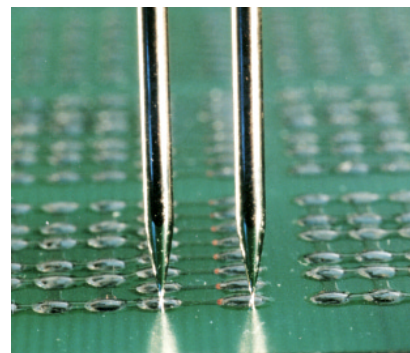
Though spreading behaviour of flux during the reflow process can be controlled by formulating low melting point rosins and thixotropic agents, the softening point of WW (water white) rosin, for example, is approx. 80°C, and if such a flux system was formulated, it would result in serious slumping.

3. Hardness of flux residue

Similar to the spreading property, theoretically, the state of the flux residue can be controlled to be soft by formulating lower softening point rosins. But in reality, in order to give slump resistant properties, higher softening point rosins must be included, such as abietic acid-polymerised rosin. This polymerised rosin has a softening point of approx. 140°C and can be a cause of residue cracking.

Realised after thorough investigation, we came to the conclusion that the flux residue must remain soft to prevent cracking and to let the probe easily penetrate, even at room temperature after reflow.

With the development of SE(S)48-M600, by adopting a new additional flux component, we have succeeded in making its post reflow flux residue as soft as candle wax and non-sticky to the test probes.



Competitive paste



SE(S)48-M600

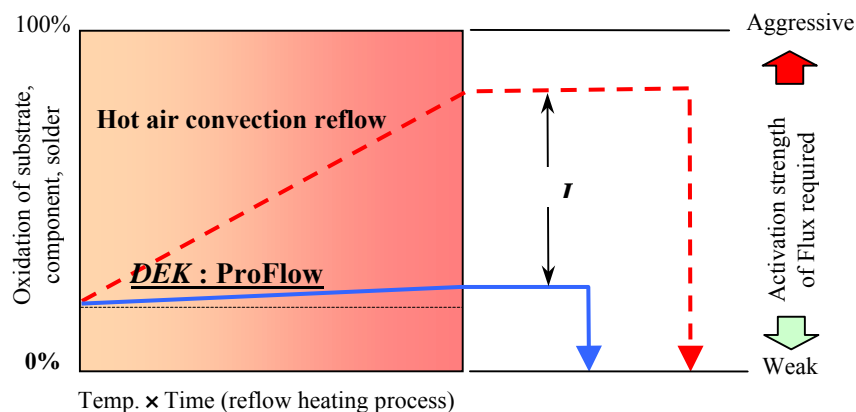
In our range of products, M600 is the only product which offers perfect testability.

3.6 Nitrogen reflow process

Advantages of using a nitrogen reflow process are generally as follows

1. Improvement of solderability
2. Lower flux residue
3. Higher reliability

The difference between normal air reflow and nitrogen reflow processes, is simply whether or not what is going to be reflowed, i.e. substrate, components and solder paste, will be ‘additionally’ oxidised during the heating process of the oven.



It is normal that the substrate and components have initially been oxidised during storage. When they are processed through a hot air reflow oven, which normally gives the most severe oxidation, the activation required of the solder paste to remove the oxide film additionally formed during the heating process, must be very high.

On the contrary, since almost no re-oxidation takes place with a nitrogen reflow process, much weaker activation will be sufficient to ensure quality solder joints.

1. Improvement of solderability

Conventional no-clean solder paste + nitrogen reflow = a) Improvement of wetting
(such as *SS4-M850 and M953*) **b) Reduction of micro solder balls**

A combination of a nitrogen blanket and ordinary no-clean solder pastes provides more activation tolerance and significantly improves consistency of soldering quality by enhancing solder wetting and reducing micro solder balls.

2. Reduction of flux residue

Low solids solder paste + nitrogen reflow = Reduction of volume of post reflow flux residue

As expressed in the above chart, the minimal additional activation allows the solder paste to have a very low solids content, mainly rosins, because the nitrogen blanket protects the solder particles from oxidation.

3. High reliability

Nitrogen use solder paste + nitrogen reflow = a) Higher surface insulation resistance
(such as *SS5-N100K, N110*) **b) Lower ionic residue**
c) Etc.

As soldering takes place in a low oxygen atmosphere preventing additional oxidation, it enables a considerable reduction to the activation strength. As indicated by arrow (A), in theory, this level of reduction in flux activity may be possible while still ensuring quality soldering.

Reduction of volume of activator and/or activation strength literally increases product reliability.

When choosing the right solder paste or considering implementation of inert gas reflow, it is very important to clearly verify why such a special process is required.

3.7 Pin-in-paste (Intrusive Reflow)

It is sometimes the case that a double-sided reflow board is assembled with leaded components. The production procedure normally taken for this application is to reflow both sides of the substrate and insert the leaded components, then use multi-spot / selective wave soldering.

In order to increase production efficiency, the use of pin-in-paste paste processes seems to be increasing.

The procedure is :

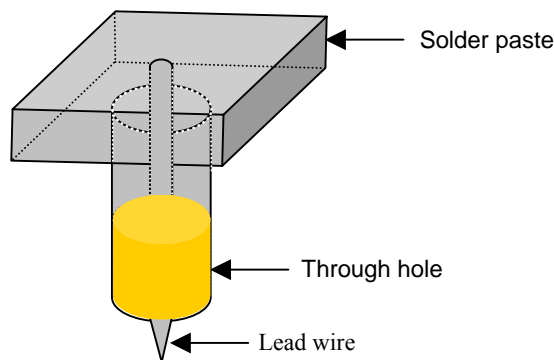
Print paste → mount components → reflow → flip over → print paste (*also on through holes*) →
mount SMT components → *insert leaded components* → reflow

This process is gaining in popularity in Taiwan for the production of notebook computers.

A key issue for successful implementation of the 'pin in paste' process is how a sufficient volume of solder can be secured.

Generally speaking, solder paste is a mixture of solder alloy and flux medium at 1:1 by volume.

How much solder paste to be deposited should be worked out by calculating the volume of the through hole opening minus the lead wire and applying the above factor.

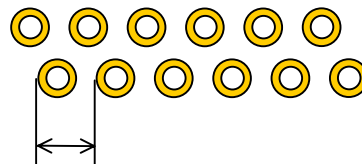


In terms of securing sufficient solder volume, the following factors should also be considered.

1. Design of substrate

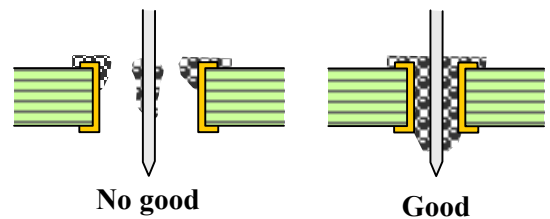
- Pitch of each through hole component

Pitch of through holes, determines how to design the stencil aperture. If the pitch is too close, it will be a major hindrance to enabling the pin-in-paste process as an insufficient amount of solder will be deposited.

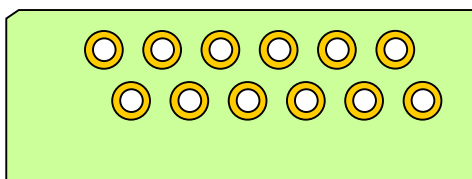


- Diameter of through hole

In the case of the diameter being too large in comparison with the diameter of the lead wire, it is possible that solder will drop out during transportation and mounting.



- Location of through hole



If the through holes are located at the very edge of the substrate, it will be difficult to secure a sufficient solder paste deposit and special consideration must be taken for designing the stencil aperture.

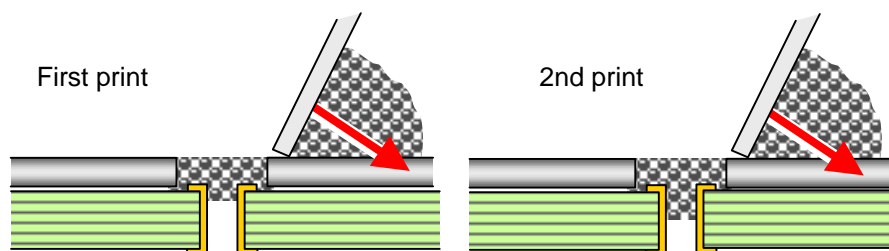
2. Printer

- Printing method

What type of printing method (or machine) to use will have a direct influence on the effective assurance of solder volume

1) Conventional metal/rubber squeegee printing → *Print twice*.

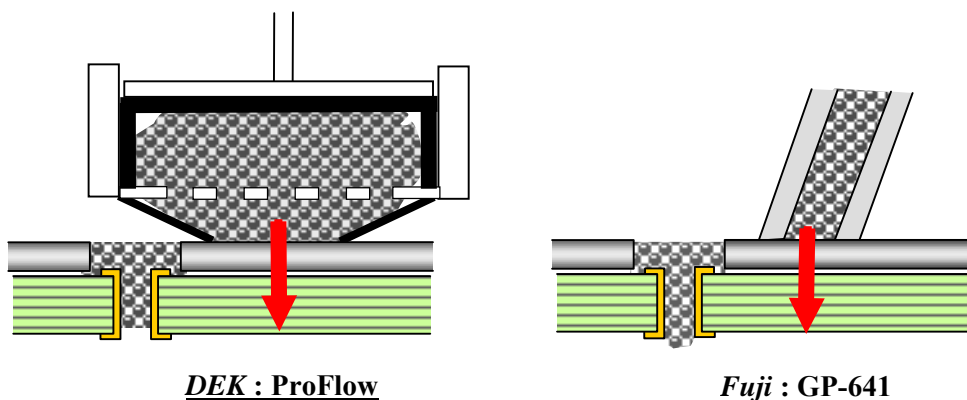
In order to secure a sufficient volume of solder paste, it is desirable to print solder paste not only on the surface of the substrate, but also into each through hole as well. When conventional squeegee printing is applied, printing must be done at least twice so that some solder paste can go into the through hole. However, this multiple printing has disadvantages and tends to cause under stencil smearing, resulting in bridging of fine pitch patterns even initially at the printing stage.



2) Vertical press type printing (Enclosed Head)

- ◇ Dek : ProFlow
- ◇ MPM : Pump print head
- ◇ Fuji : GP-641

On the other hand, vertical press type printers can effectively push solder paste into the through hole whilst ensuring proper solder paste filling pressure to the remainder of the board layout.



There are several customers in Taiwan who have successfully implemented Pin-in-paste processes with the combination of Fuji GP-641 and Koki no-clean solder pastes.

- Stencil

The following factors of the stencil are also crucial in determining the volume of solder to be provided to each through hole.

1) Thickness

Generally, the thickness of stencil widely used is 150 micron. Thicker stencils definitely help increase the amount of solder paste deposited without increasing printing area. For example, when the thickness is increased to 200 micron from 150micron, i.e. >33% increase by volume. Assuming that the flux and alloy of the solder paste are mixed at the ratio of 1 : 1 by volume, by increasing the thickness by 50micron, the volume of solder will be increased by >16%.

2) Aperture size

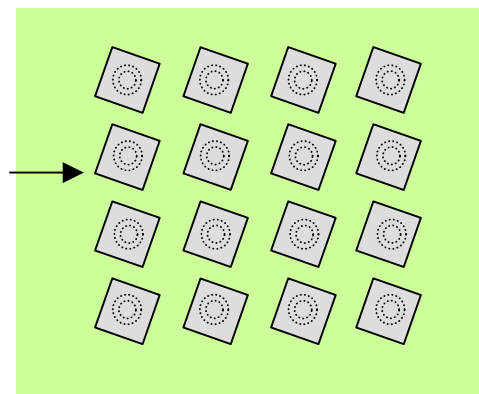
With consideration that solder volume after reflow will be approximately 1/2 of the volume of the solder paste, proper aperture size to ensure sufficient volume of solder shall be obtained by calculating each volume of through hole, lead wire, and solder paste to be filled into the through hole.

3) Design of aperture

Depending on the through hole component to be reflowed, for example, a connector, the pitch between each through hole will be very small, thus a special design of aperture will be required.

Ex. a) PGA socket

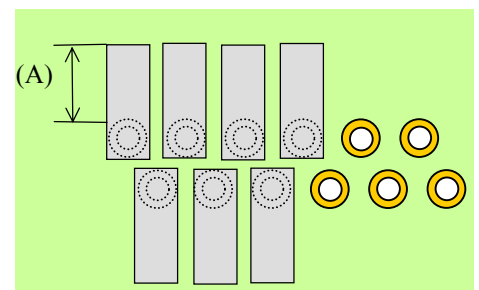
Solder paste (top view)



Ex. b) Connector

When the pitch is small, the design of aperture should be like this sketch to secure sufficient solder paste.

How long (A) should be and its width, shall be carefully determined taking into consideration the minimum volume of solder required and bridging with neighbouring pads.



3. Solder paste

The paste to be applied for Pin-in-paste processes, should exhibit good performance in printing, mounting and reflow as a whole. But, the following two characteristics should be considered in selecting the correct paste for a Pin-in-paste process.

- Viscosity

- 1) At printing stage

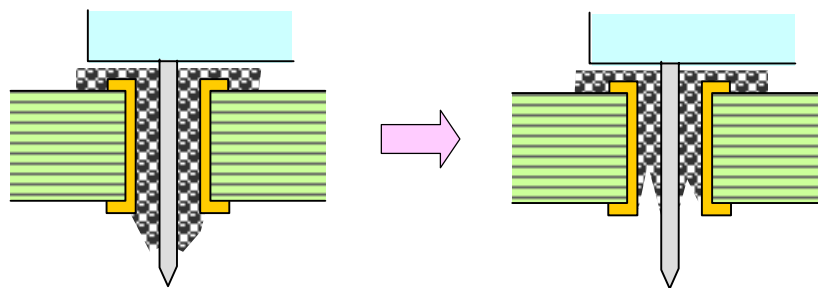
When a conventional squeegee type printer is used, lower viscosity paste will be better, because softer paste can go into the through hole more easily.

For the vertical press type printers, higher viscosity paste should be desirable because when the vertical filling pressure is increased to push more paste into the through holes, a lower viscosity paste could cause bleeding at fine pitch patterns.

- 2) At mounting stage

When the lead wire is inserted into the solder pasted filled through hole, it looks like the illustration below left.

If the viscosity is too soft, when the board is jolted at high speed mounting by the fast movement of the X-Y table, some of the solder paste gets shaken off, and results in insufficient solder.



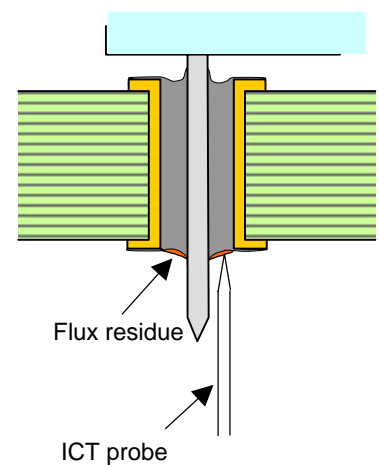
Thus, when such a high speed moulder is to be used, attention should be directed to the table speed.

Koki SE4-M641, which was developed specifically for the use in a Fuji GP-641, has a relatively high viscosity of 2700Ps and has proven that it successfully prevents shake-off of the paste in conduction with Fuji high-speed IP mounting units.

- ICT testability

Since quite large amounts of solder paste are to be deposited on each through hole in the Pin-in-paste process, and consequently, quite large amounts of flux residue is to be left on the solder joint, ICT testability of flux residue is a very important issue, where it is required.

For such applications, we recommend Koki SE(S)48-M600 which was specially developed to achieve 100% yield at the ICT stage.



4. Conclusion

Once again, the most crucial point when implementing pin-in-paste process is how to secure a sufficient volume of solder.

To achieve this process, stencil design and a printing machine must be selected. Once the printing equipment has been decided upon, Koki can provide the best suited solder paste, including one for ICT testing requirements.

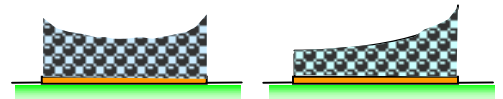
4. Trouble shooting

4.1 Printing

Typical problems / causes / measures are as follows.

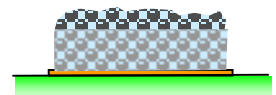
1) Scraping / Scavenging

- Squeegee pressure is too high and the squeegee scoops out the top of the solder paste being deposited.
- Re-adjust the pressure to the minimum required level.



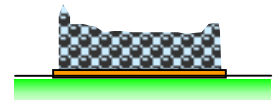
2) Excess paste

- Squeegee pressure is too low and as a result of 'mapping', extra solder paste is deposited on the pad.
- Gradually increase the squeegee pressure and obtain the proper setting.



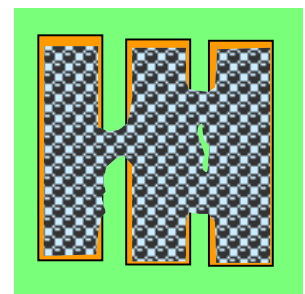
3) Tailing / dog ear

- Stencil separation speed is too fast.
- Adjust the stencil separation speed as low as possible.



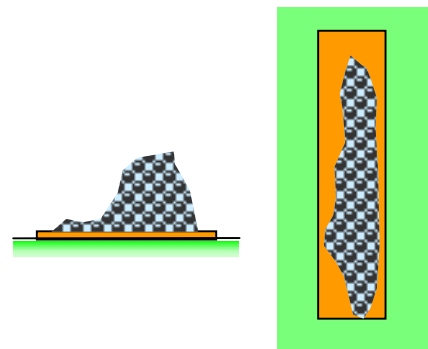
4) Smearing

- In continual printing, smearing will eventually happen. Though how badly the smearing will occur largely depends on the rheology of the solder paste, some parameter adjustment may help to improve the situation.
- Reduce squeegee pressure to the minimum required level (= reduction of solder paste filling force).
- Check if alignment of the stencil apertures with each pad is accurate ensuring a good gasket.
- Make sure the stencil tension is tight enough and gives an on-contact print.
- Reduce the aperture width smaller than that of the pad.
- Lower temperature in the printing environment so that the viscosity of the solder paste increases.
- Increase the frequency bottom side stencil cleaning.
- Reduce squeegee speed if it is unnecessarily fast as it merely breaks down the thixotropy of the paste and makes it very soft (shear thinning).



5) Insufficient deposit

- Clogging of the aperture occurs and insufficient volume of solder paste is released onto the pad.
- Reduce stencil separation speed.
- Reduce squeegee pressure. If the pressure is not properly set, an excess number of particles are stuffed into the aperture and inner pressure / friction among the particles increases, and consequently makes smooth release of the solder paste difficult.



4) Poor squeegee separation

- When printing temperature is quite warm, such as $>30^{\circ}\text{C}$, evaporation of the solvent in the paste is accelerated and results in the solder paste becoming quite sticky.

Control the temperature to be below 25°C .

- If it is possible, adjust the printer parameter so that the solder paste is allow to wet to the stencil at least for a few seconds before the squeegee is raised.

- Check that fresh paste is being used, and not paste that was printed the previous day.

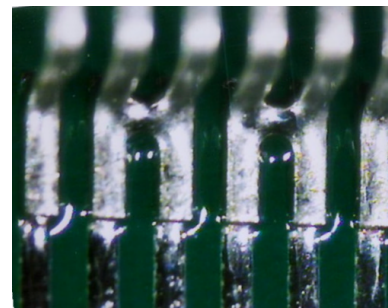
5) Dry-out of solder paste

- If the paste dries out extremely quickly on the stencil, check if the paste is fresh material and also check the printing environmental temperature.

4.2 Soldering defects

1) Bridging

- Excess solder is provided on each pad.
It is desirable to reduce the aperture width taking into consideration stretch of the stencil, slump of the solder paste, and accuracy of stencil positioning.



example

0.5mm pitch / pad width 0.25mm \rightarrow Aperture width 0.235mm

0.4mm pitch / pad width 0.2mm \rightarrow Aperture width 0.185mm



- Misplacement of component / solder paste



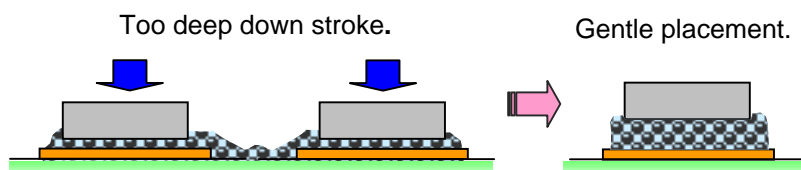
Misplacement of component



Misprint of solder paste

- Excessive component placement pressure

When the downward stroke is too deep, solder paste may be squashed out of the pads and bridge with the solder from the neighbouring pads.



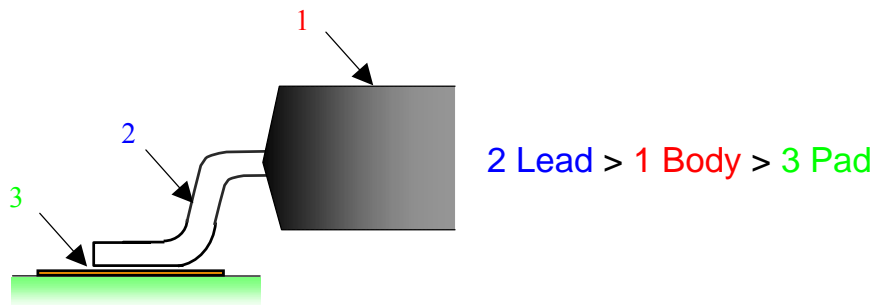
- Slump of solder paste

Excess shear stress to the solder paste by fast squeegee speed may collapse the thixotropy and make the paste “slumpy”. Or if the local printing environment temperature is unusually high >30°C, viscosity drops and can cause slumping.

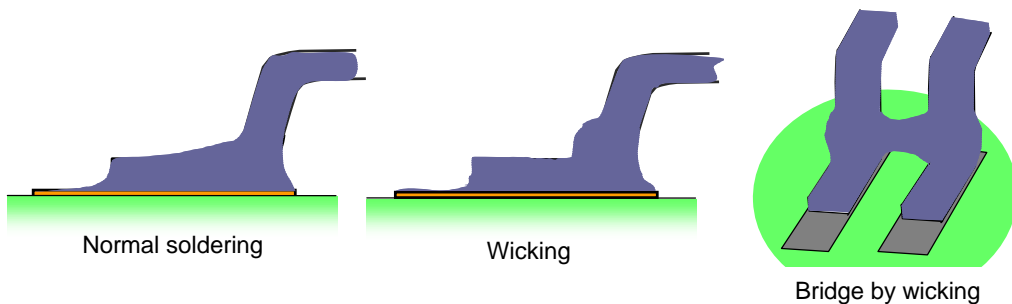
Reduce the squeegee speed to a proper speed and control the printing temperature < 25°C.

- Wicking

When reflowing with a hot air convection oven, temperature relation among **1-component body**, **2-lead wire** and **3-soldering pad** becomes as below.



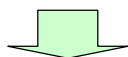
As the temperature of **2-lead wire** reaches to the melting point of solder faster than that of **3-soldering pad**, the majority of molten solder is sucked to the lead wire and swells at the heel then touches with neighbouring solder causing bridging at the between the leads.



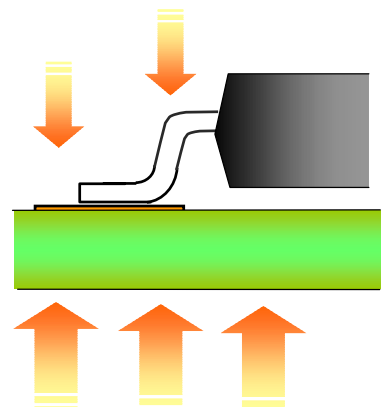
In order to prevent the lead wire from being heated faster than the PCB pad, we recommend reducing the heating energy from the top, whilst increasing heating from the bottom.

Since bridging can be caused at any part of the printing, mounting and reflow stage, pay close attention to where the problem is originating from.

$$2 \text{ Lead} > 1 \text{ Body} > 3 \text{ Pad}$$

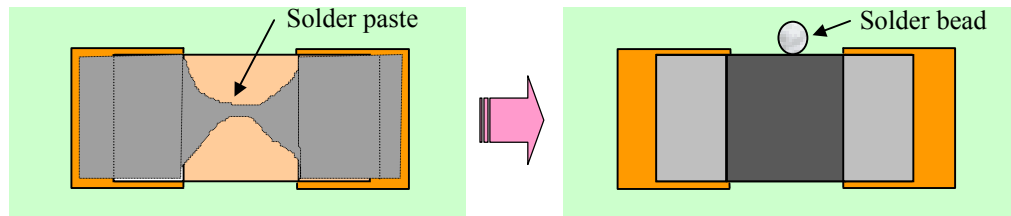


$$3 \text{ Pad} \geq 2 \text{ Lead} \geq 1 \text{ Body}$$



2) Solder beading

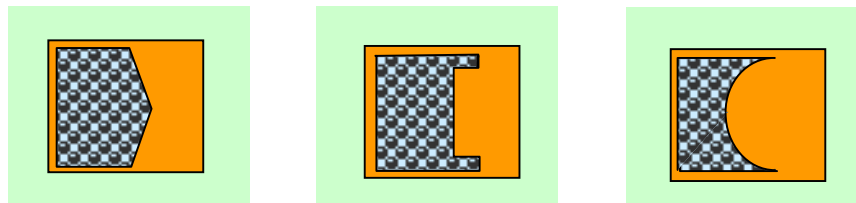
As explained earlier, solder beading occurs when solder paste softens due to heat input, and flows to the underside of chip components.



- The most practical and most effective measure in reducing the number of solder beads, is to reduce the volume of solder paste deposited on each pad.

This can be done either by reducing the aperture size, or reducing the thickness of the stencil, or a combination of both.

It is also quite effective if the design of the aperture can be changed as suggested below.



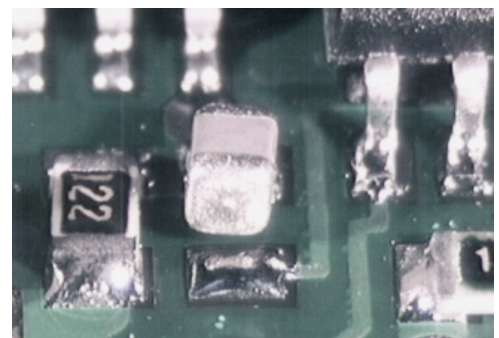
Aperture area can be as small as 70~60% of pad area.

The idea is to minimise the total volume and the volume of solder paste which comes in contact with the components.

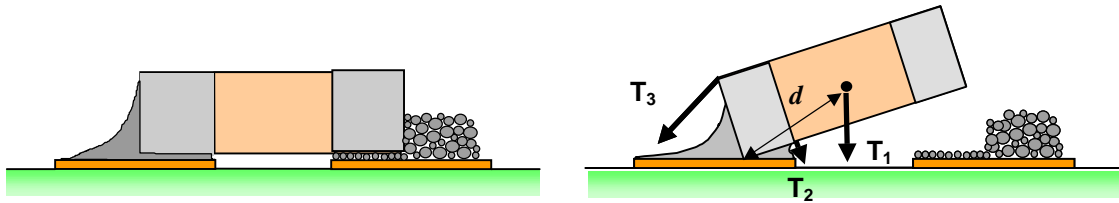
- Offer SE(S)4-M953i as it tends to cause less solder beads due to its high thixotropy and slump resistance.

3) Tombstoning (Manhattan effect)

Though a variety of factors, such as the nature of solder paste, component mounting accuracy, stencil aperture design, reflow profile etc., can cause the occurrence of tombstoning, ultimately, it happens when the below indicated momentum T_3 , becomes greater than the other tension forces.



- Momentum by self-weight of component..... (T₁)
- Momentum by wetting tension of molten solder under component (T₂)
- Momentum by wetting tension of molten solder on the side wall of component..... (T₃)



Condition which causes tombstoning shall be when : $T_1 + T_2 < T_3$

Especially, in the case of smaller components, like 1608, 1005, tombstoning tends to happen more often because the distance from the centre of gravity and the fulcrum d is small.

There are various factors that can cause the problem, such as;

- 1) Inconsistent size of each side of the pads.
- 2) Inaccurate placement of the component.
- 3) Inconsistent solder paste volume on each pad.
- 4) Too much solder paste.
- 5) Differing solderability of each terminal of the component.

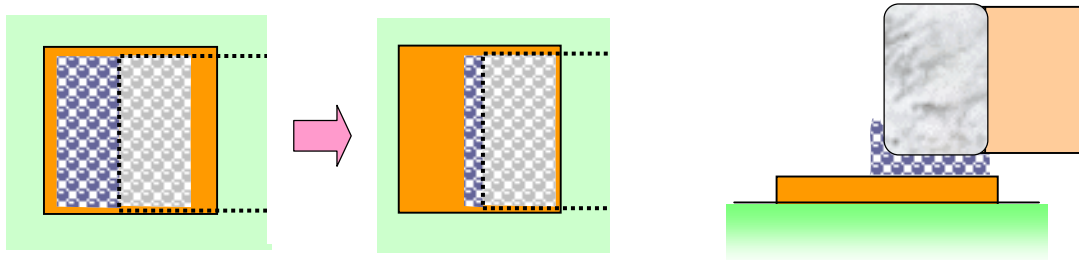
An important point about what causes the problem as a result of, or in combination with these factors, is that when either end of the terminals starts to be pulled by the wetting effect of solder faster than the other end, unbalanced momentum $T_1 + T_2 < T_3$ can be created.

It's been said only 0.2 seconds of time difference in wetting at the each terminal can cause tombstoning.

In order to overcome the problem, the momentum relation must become as below ;

$$T_1 + T_2 < T_3 \quad T_1 \longrightarrow + T_2 \geq T_3$$

- Since the tombstoning occurs because of an imbalance in wetting time and/or wetting force (tension) of each terminal of a chip component, reduction of solder paste deposit volume seems to be quite effective in improving the situation.



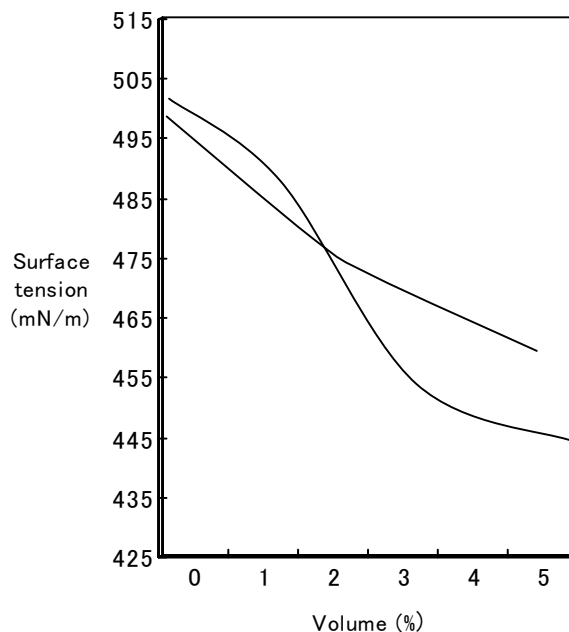
- To adjust the wetting time of the solder to the component.

When the solder is of eutectic (Sn63/Pb37), solder starts wetting to the terminals of chip components immediately after it melts. Thus, if there is some time difference in wetting of the two ends, and one side starts wetting faster than the other, the momentum T_3 caused by this can lift up the other end of component

In order to prevent this, it is desirable to have some Silver Ag (0.4~2%) so that solder transforms from solid to liquid passing the intermediate plastic state. By providing this plastic state, even if only the one end enters the plastic state, wetting does not immediately take place and in the meantime, the other end enters into the plastic state as well, thus balancing the wetting time.

- To reduce the wetting tension of solder.

It is also desirable to reduce the wetting tension of the solder against the component's terminals. This can be done by adding either Bismuth (Bi) or Antimony (Sb). Since Bi, however, tends to deteriorate the bonding strength of solder joints, Sb is regarded as the best candidate, because it increases the bonding strength and has almost no influence over wetting.



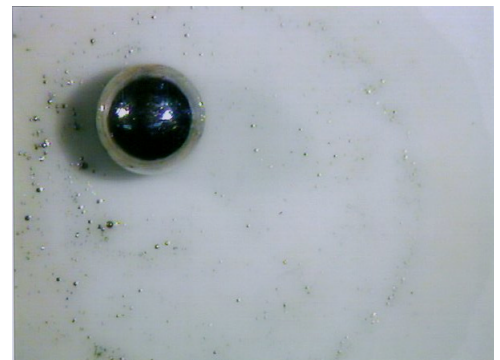
Koki's new type alloy SSA- is composed of ; **Sn62.6, Pb36.8, Ag0.4, Sb0.2**

and offers both the advantages of wetting time adjustment and reduction of wetting tension for the effective prevention of tombstoning.

4) Micro solder balls

Possible causes of micro solder balling are ;

- Initial oxidation of the solder powder
- Oxidation of solder powder during the reflow process
- Slumping of the solder paste after printing, at component placement and or during reflow.
- Transfer of solder particles from the stencil



Therefore, similar to bridging, because micro solder balls can be caused by each production stage of printing, placement of components and reflow, carefully find where the problem has originated from.

If the problem has been caused due to quality of solder paste itself, micro solder balls must be observed at every soldering spot on the PC board and continuously occur on every board as long as the same solder paste is used.

By carefully observing this point, you can realise whether the problem is caused by the solder paste itself or by the production process.

5) Poor wetting

As explained, wetting problems normally happen due to poor quality (severe oxidation) of components, or use of difficult to solder alloys, both the base (Fe, Ni, etc.) metal and plating cover (Ni/Pd, Ni/Au, etc.).

The only measure we can recommend is to try to increase the peak temperature a little within a range that not damage any of the components.

For example, from 210°C to 220°C.

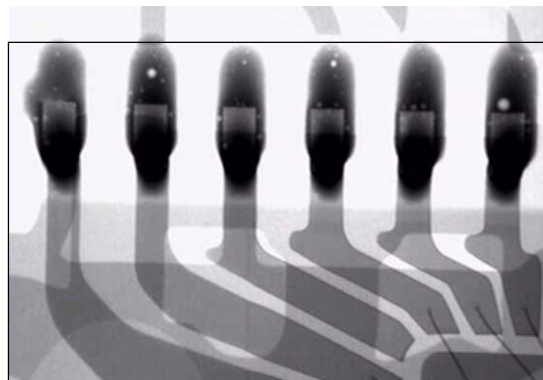
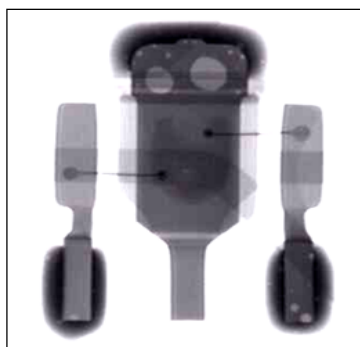
You may offer active solder paste, such as Halide containing paste, if it is acceptable, and if not SE(S)48-M1000-2.

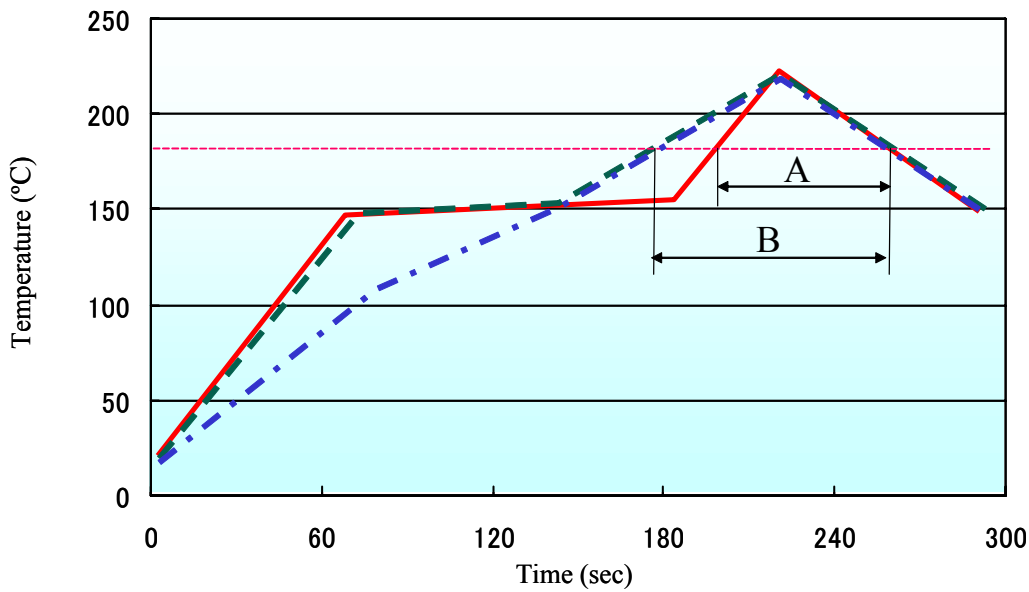
These products do help to improve wetting.

6) Void

Causes and measures are as below.

- 1) Generation of Gas : Volatilization of solvent and solids and generation water vapour.
→ Adjust reflow profile and extend the time above melting point (A → B).
- 2) Evacuation of Gas : Poor fluidity of solder due to low temperature.
→ Increase peak temperature.
- 3) Poor wetting : Oxidation or contamination of component and PC board.
→ Shorten O₂ exposure time by shortening pre-heat, improve cleanliness of component and PC board.





- : Original profile
- - : Recommended profile (Extend time above melting point + reduce heat exposure)
- - : Recommended profile (Extend time melting point)

Conclusion :

It is critical for implementing effective and efficient measures to thoroughly and carefully observe where soldering defect(s) is originated from by checking through the production line from the down stream, and identify whether the problem has been due to quality of solder paste or inadequate process optimization.

