



Name : Solder Paste 63/37

Model : Sn63Pb37

Net Weight : 500g/bottle

High Quality No Clean Solder Paste Sn63Pb37

1. Product Features

- 1.1. The printing has good rolling and tin dropping properties, and can also finish fine printing for less than 0.4mm pad;
- 1.2. Continuous printing, its viscosity changes very little, stencil operation life is long, more than 12 hours will not dry, still keep good printing performance;
- 1.3. After printing for several hours, it remains the original shape, no collapse and the placement of component will not offset;
- 1.4. It has excellent weld ability and can show proper wet ability in different parts ;
- 1.5. It can adapt to the requirements of different grades of reflow oven, no need to complete the welding under the environment of nitrogen filling, and can still show good welding performance in the wider reflow furnace temperature range. Two

sets of furnace temperature setting modes can be used with "warming up insulation" or "gradually warming up";

1.6. After welding, the residue is very few, the color is very shallow, and has a large insulation resistance, which will not corrode PCB and no need to clean;

1.7. It has good ICT test performance without causing misjudgment.

1.8. Use for paste in hole process ;

2. Solder Paste technical Parameter

Parameter of Flux		
Flux grade	ROL1	J-STD-004
Cl content	<0.5 wt%	Potentiometric titration
Surface insulation impedance (SIR)	Before Humidification heating	>1x10 ¹³ Ω
	After Humidification heating	>1x10 ¹² Ω
Impedance of Aqueous Solution	>1x10 ⁵ Ω	25mil Comb plate
Copper Mirror Corrosion Test	Qualified(non Penetrating Corrosion)	40°C 90%RH 96Hrs
Test paper test of silver Chromate	Qualified(without color change)	Conductive bridge meter
Residues Dryness	Qualified	IPC-TM-650
		IPC-TM-650
		In House

Parameters of Solder Paste		
Metal content	85 ~ 91wt% (±0.5)	Gravimetric method (optional)
Flux content	9 ~ 15wt% (±0.5)	Gravimetric method (optional)
Viscosity	Canning	200Pa.s±30 Malcolm(10rpm,25°C)
	Needle cylinder	100Pa.s±20 Malcolm(10rpm,25°C)
Thixotropy index	0.55±0.55	T3,90% metal for printing
Expansion rate	>90%	T4,87% metal for syringe
Collapse test	Qualified	In House
Tin bead test	Qualified	Copper plate(Sn63, 90% metal)
Adhesive force (Vs Exposure time)	48gG (0 hour)	J-STD-005
	56gG (2 hour)	
	68gG (4 hour)	
	44gG (8 hour)	
Lift time for stencil	>12 hour	IPC-TM-650 ±5%
Quality guarantee period	Six Month	In House
		0 ~ 10°C Sealed storage
* Details information refer to the product's acknowledgement of the product.		

3. Application

3.1 How to choose this series of solder paste

Customers can choose the corresponding alloy composition, tin powder size and metal content according to the requirements of their products and processes, Powder size normal choose T3(mesh-325/+500, 25~45µm), about Fine pitch, you can choose more fine Tin powder.

3.2 Preparation before use of solder paste

1) Returns warm

Solder paste is usually storage in fridge within 0~10°C. So when you take out the solder paste from fridge, the temperature of solder paste less than room temperature too much. If the bottle cap is opened without "Returns warm", it is easy to condense the water vapor in the air and adhere to the tin pulp. In the reflow oven (the temperature is more than 200°C), the moisture is rapidly vaporized because of the strong heat, causing the phenomenon of "tin explode", producing tin beads and even damaging the components.

How to returns warm: Don't open the cap of the bottle, put the solder paste bottle in the room temperature more than 4 hours.

Note: 1. Don't open the cap without full "return warm" 2.

Don't shorten the time of "return warm" by heating.

2) Mixing

Solder paste should be fully mixing before use after "return warm".

Purpose: Make the flux and tin powder evenly distributed and make full use of the performance.

Mixing method: By hand or By machine;

Time of Mixing: 3 minutes by hand; 1 minutes by machine;

Effect of Mixing: Scrape up some solder paste with a scraper. When the blade is tilted,

If the solder paste can slide down smoothly, it will be ok.

(Proper mixing time is different due to factors such as Mixing mode, equipment and ambient temperature. It should be determined by many tests before use.)



3.3 Printing

More than half of the poor welding problems are related to the printing process, so please pay more attention on printing is needed.

◆ : Stencil

Similar to most solder paste, lead-free series of solder paste will show excellent performance if high quality stencil and printing equipment are used. Whether used for etching or laser stencil can be printed perfectly. For printing fine pitch, it is better to choose laser stencil. For 0.4mm pitch, stencil with 0.12mm thickness are generally required.

◆ : Printing method

Manual printing or semi automatic and full automatic printing is also ok.

◆ : Printing condition of stencil

The No clean lead free series of solder paste is a non hydrophilic product that is not sensitive to humidity and can still be used under the condition of high humidity (the highest humidity of 75%).

The following are our ideal printing conditions.

It is very necessary to adjust the process requirements accordingly.

Hardness of scraper	60 ~ 90HS (Metal scraper or polycarbamate scraper)
Scraping angle	45° ~ 60°
Printing pressure	(2~4)x10 ⁵ Pa
Printing speed	Standard: 20 ~ 40mm/sec Fine spacing printing: 15 ~ 20mm/sec Wide spacing printing: 50 ~ 100mm/sec
Environmental conditions	temperature: 25±3°C Humidity: 40 ~ 70% Airflow: There should be no air flow at the printing area

◆ : Technical points during printing

- ① Check the scraper, stencil etc before printing
 - ★ Make sure that it is clean, no dust and sundries (if necessary), so as to avoid contamination of solder paste.
 - ★ The scraper should be straight, no damaged
 - ★ Stencil should be straight, without obvious deformation. No residual tin paste or other debris can be found on the stencil.
- ② There should be fixtures or vacuum devices to fix the floor to avoid PCB offset during printing and improve the separation effect after printing.

- ③ The position between the stencil and the PCB will be adjusted to the better. (the gap will lead to tin leakage. The horizontal dislocation will lead to solder paste printing to the outside of the pad.);
- ④ At the beginning of the printing, the solder paste added to the stencil should be in the right amount. Generally it is A5 size stencil is about 200g, B5 is about 300g, A4 is 400g or so. ⑤ With the continuation of printing work, the amount of solder paste on the steel network will gradually decrease, please try to add right amount fresh solder paste;
- ⑥ The separation speed of stencil after printing should be as slow as possible;
- ⑦ In continuous printing, the top and bottom of the stencil should be cleaned every other time (according to actual conditions). Be careful not to leave the water or other impurities on the solder paste and stencil when cleaning;
- ⑧ If the solder paste stays on the stencil for too long (or the solder paste reused for a long time), its printing performance and stickiness may be worse, add a proper amount of the company's special blending flux can be improved;
- ⑨ Attention should be paid to temperature and humidity control in workplaces. In addition, strong air flow should be avoided to avoid volatilization of solvents and affect viscosity; ⑩ The top and bottom of stencil should be thoroughly cleaned before the end of the operation. (pay special attention to the cleaning of the hole);

3.4 Time after printing

When the solder paste is printed, the SMT components should be completed and finish the reflow process as soon as possible, To avoid the drying of the solder paste on the PCB for too long, which affects the SMT component placement and welding effect of the components. Generally recommended that the time should be less than 4 hours after printing.

3.5 Reflow profile (See the graph of the attached page) 3.6 Removal of remnants after welding

The lead-free series of solder paste has little residue after welding, and is very light and transparent. It has very high insulation impedance and no need to clean.

4. Packing and Transportation

Each bottle 500g, wide mouth plastic bottle (PE) packaging, and cover the inner cover sealed package, delivery can be filled with foam box, up to 20 bottles per box, keep the temperature in the box not more than 35°C.

5. Storage

When customers receive the solder paste, they should be stored in the fridge as soon as possible. The recommended storage temperature is 0~10°C

- Too high temperature will shorten life and affect its performance;
- If the temperature is too low (below 0°C), it will produce crystallization and deteriorate the properties;

Under normal storage conditions, the validity period is 6 months.

6. Precautions for health and safety

Note: the following information is provided to users only. Users should know clearly before using it.

Details information please refer to the material safety data sheet (MSDS)

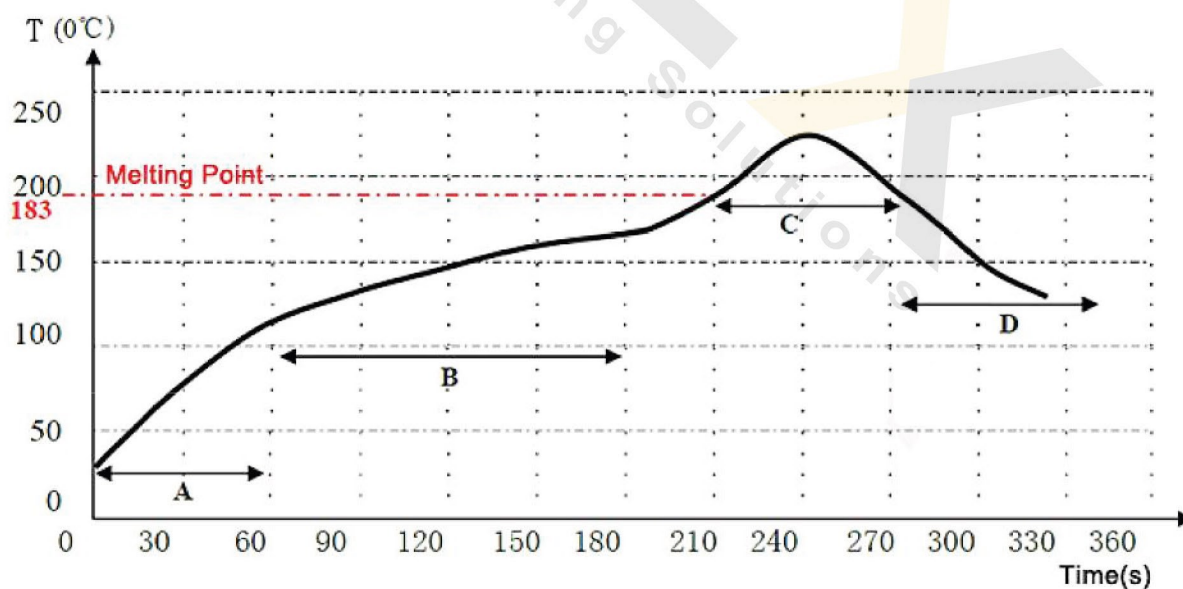
This product does not contain regulated chemicals or organic solvents regulated by the organic solvent poisoning prevention rules, but the necessary precautions are needed to ensure the health and safety of the human body. For products containing lead, their operation should be carried out in accordance with the labor safety and health law and the rule of lead poisoning prevention.

No Clean Series Solder Paste

Reflow Profile

(SUIT FOR SN63PB37, SN60PB40, SN55PB54)

The following is the profile recommended by us use by reflow process, which can be used as reference for setting the temperature curve for reflow. The profile can effectively reduce the vertical flow of solder paste and the occurrence of solder balls, which is applicable to most of products and process conditions.



A. Preheating zone (25-33% of heating channel)

In the preheating area, some volatile solvents in the solder paste are evaporated and the thermal impact on the components is reduced;

- Requirement: The heating rate is 1.0~3.0°C/s
- If the heating rate is too fast, it may cause solder paste mobility and deterioration of components, resulting in tin ball and bridge. At the same time, the components will be subjected to excessive thermal stress and damage

B. Immersing area (33-50% of heating channel)

In this area, flux is active, chemical cleaning operation begins, and the temperature of PCB is uniform before reaching the reflow zone.

- Requirements: temperature: 130~170°C, time: 60-120 seconds, heating rate: <2°C/s

C. Welding area

Metal particles in solder paste are melted and formed on the surface of solder joints under the action of liquid surface tension;

- Requirements: temperature: 210~240°C, time: Above 183°C 40-90 seconds

(Important), above 200°C 20-50 seconds;

- If the peak temperature is too high or the welding time is too long, it may cause the solder joint to darken, the carbonization and discoloration of the flux residue, and the damage of the components.
- If the temperature is too low or the welding time is too short, the wet ability of the solder may be worse and the high quality solder joint cannot be formed. The solder joints of the components with larger heat capacity may even form a virtual welding.

D. Cooling zone

After leaving the reflow zone, the PCB enters the cooling zone, and the cooling rate of the solder joint is also very important. The strength of the solder joint will increase with the increase of cooling rate.

- Requirements: the cooling rate is < 4°C, and the cooling termination temperature is preferably not higher than 75°C
- If the cooling rate is too fast, it may cause damage to the components due to excessive heat stress, and there are cracks in the solder joints.
- If the cooling rate is too slow, a larger grain structure may be formed, resulting in poor strength of the solder joint or shift of the components.

Note:

- The above temperature curve refers to the actual temperature at the solder joint, not the setting temperature of the reflow oven.
- The above reflow profile is for reference only, and can serve as a basis for users to find the best curve in different process applications. The actual temperature setting should be considered in combination with the properties of the products, the distribution of components and the characteristics of the components, and the technological conditions of the equipment. Many experiments should be done in advance to ensure the optimization of the profile.
- This series of solder paste can be heated by the "warming up" heating method, and gradually heating.