Through hole soldering defects and the solutions

TABLE OF CONTENTS

INTRODUCTION

Bad solder joints that require touch-up are a complex subject. First of all, we must judge it is caused by poor design, bad soldering technique, bad soldering materials, improper pre-treatment or unsuitable equipment. In addition, technical and inspection standards often lead to unnecessary touch-up, but they are not included in our discussion since the soldering operation and quality standards required by each electronic industry are different,

A lot of solder joints that are considered bad, in fact, are actually good. However, there are too many widely recognized inspection standards, which wrongly emphasize the beauty of solder joints and ignore their functions, thus resulting in a huge and unreasonable touch-up cost in this industry. Remember that touch-up does not always improve quality.

Here, we assume that there is no problem with PCB's design, soldering materials selected and pre-treatment before soldering and only discuss technical problems during the soldering process.

Special problems with soldering and suggested solutions will be discussed in this course. While many soldering problems may recur, problems faced by each electronics company are still not exactly the same, so there will be no so-called Standard Answer. Here we provide years of experience for customers' reference, but users still have to treat individual problems appropriately.

https://www.1clicksmt.com/product/lead-free-wave-solder-Machine-e-therm-rw-series.html Dual wave solder machine machine RW-450	https://www.1clicksmt.com/product/Ant-i1.html Offline selective soldering machine Ant-i1
	

1. Trouble-shouting outline

When problem occurs, the first thing that must be checked is the basic conditions of the manufacturing process. We summarize them as the following three factors.

1. 1 Bad materials

These materials include such chemicals for soldering as flux, oil, tin, cleaning materials, and PCB cladding materials such as anti-oxidation resin, temporary or permanent solder mask and printing ink.

1.2 Bad solder joints

This involves all solder joints' surfaces, such as components (including surface-bonded parts / SMT parts), PCBs and electroplated PTHs, etc. must be taken into consideration.

1.3 Improper equipment

These include improper machines, equipment and maintenance and such external factors as temperature, conveyor belt speeds and angles, as well as the depths of immersion and so on which are variables directly related to machine. In addition, ventilation, air pressure, voltage and more factors must be analyzed.

Every problem is different in its own way and should not be lumped under one head. Following is a series of standard inspection steps which can help you find out the root cause.

Step 1: When soldering, the smallest variable should be machines, so the first thing is to check them. In order to realize the correctness of your check, independent electronic instruments can be used as auxiliaries such as thermometers to detect temperatures and multi-meters to calibrate the parameters accurately. Try to find out the most suitable working conditions from the actual operations and records. Note: in any case, do not depend on adjusting the equipment to overcome temporary soldering problems because such adjustments may lead to bigger problems.

Step 2: Check all the soldering materials, such as flux's specific gravity, transparency, color, ion content and purity of tin-lead alloy. This is a continuous work accompanied by both regular inspection and random sampling. All these are helpful to ensure their quality.

Step 3: Poor solder joints of PCBs and components are the biggest factor causing soldering problems. To study the soldering problem of PCB, we must first fix or isolate the other variables which may occur, and then discuss them one by one. For example, if soldering defects occur on pins, other variables should be locked first, and only those pins with soldering defects can be thoroughly compared and analyzed. Through this way of tracking, the source of problem will be clear soon.

Step 4: Check the quality of the PTHs, punching, drilling and other defects. We may use amplifying equipment to see if the PTH surface is smooth, clean or has any other impurities or breaks or the thickness of electroplated layer is standard or not. In the process of tracing soldering problems, the principle and concept should be correct. In addition, steps are very important. How to find out the problem effectively by comparison and analysis is the biggest problem for electronic engineers.

2 NON-WETTING & POOR WETTING

If the solder joints are good or not depends on whether the parts to be soldered can get well wetted, accordingly the quality of solder joint is determined by the quality of wetting. Basically, solder can wet and weld copper pads or other metals, and the surfaces of these metals should not be covered by oxide or other impurities (such as dust, organic compounds, etc.). Bad wetting is often caused by the unclean surface of the parts to be soldered.

Non-wetting and dewetting are different from each other because of their different forming process, so they must be discussed separately. Non-wetting means that when soldering, the solder cannot completely cover the surface of the part to be soldered and the surface of the metal to be soldered is exposed, which is especially easy to happen on bare copper board (Figure 2-1). The red bare copper as seen from the figure overspreads the whole solder side, and its periphery is mainly formed by the cohesion of tin.

From another point of view, dewetting means that when soldering, tin has fully covered the whole solder side, which means that the surface is well wetted (Figure 2-2). Now eutectic reaction happens between the metal to be soldered and the tin-lead alloy. But, when the surface soldered cools down, its wettability begins to decrease and tin’s cohesion begins to increase. So, a part of liquid tin on the originally flat tin surface is pulled open because of the unbalanced tension between them, and balls or beads are formed via solidification in a dihedral-angle way. At this time, only a small amount of tin-lead alloy can truly achieve the metal-metal combination with the solder side thickly. Those thin tin surfaces seem to cover the whole bare copper surface from the viewpoint of naked eyes, but non-wetting which cannot be seen by naked eyes is often detected under the high-energy microscope. Section 3 will provide more detailed information about dewetting.

Non-wetting is not acceptable in soldering operations because it seriously reduces the durability and extensibility of solder joints and also the electrical conductivity and thermal conductivity of solder joints. Up to now, there is still no mathematical procedure that can accurately calculate the acceptable range of soldering defects. So once this happens, it must not be accepted. From another point of view, non-wetting happens mostly due to the fact that flux cannot remove the oxidation film on the surface to be soldered thoroughly before soldering. In addition, the length of soldering time and temperature will cause the occurrence of non-wetting.

Cause of non-wetting

The defects mentioned in this section are the result of one or more of the following factors:

2.1 External contamination

Both PCBs and components may be contaminated. Contaminants include oil, paint, wax, grease, etc. These contaminants are collectively referred to as impurities (dirt) and can be removed by appropriate cleaning methods.

★ Traditional vapor degreaser has successfully been widely used, but we must choose electronic-cleaning solvents which will not harm PCBs to avoid harmful residues.

★ We may use a water-based cleaner, but must make sure that these impurities are soluble in water.

★ Some external contamination is from solder mask on the PCB surface. If wrong stencil printing program is used or printing ink is spilled when drying, it is very difficult to remove the solder mask after drying. When such mask touches the solder side, it can only be removed by friction or tool. This kind of mechanical force to remove the mask is widely used, but also relatively easy to bury some tiny particles in the PCB surface. Further explanations will be given below.

2.2 Embedded Particles

Foreign substances embedded on the surface of materials to be soldered may also affect the wettability. On a soft metal surface, it is easy to insert hard objects into the metal surface with a grindstone or grinder. These non-metallic materials are obviously not soldered with tin-lead alloys and cannot be removed with flux. Brushes made from some synthetic materials also face similar problems.

The best treatment for this case is the chemical etching of the entire surface to remove nonmetallic impurities. These etching agents are very strong chemicals and must be properly controlled. It is the best to ask the PCB supplier for detailed operation method.

2.3 Silicone Oil

Although silicon oil is an external contamination as mentioned above, it is specifically discussed here because of its unique properties. Silicon compounds are used as lubricants or adhesives because of their strong adhesion. Once contaminated with a silicon compound, even a thin layer, it cannot be removed effectively by any solvent, so silicon compound is considered a poison for soldering.

Silicon contamination is caused by many reasons. Plastic bags use silicon as mold release agent, so is considered to be a major contamination source. Recently, a lot of safe plastic bags have improved this problem, but we still have to pay attention to their selection.

Another source is the coolant applied before soldering. Special attention should be paid to the use of silicon compounds in the factory, although they are far away from the soldering process. However, due to the infection caused by human hand, soldering process will be covered by silicon compounds quickly.

So far, there is no way to remove silicone oil and the only way is to keep PCBs clean and control it tightly as possible as you can.

2.4 Heavy tarnish layer

The oxide film on the solder surface over PCBs cannot be completely removed by flux, so is also one of the sources for poor wetting. As long as the metal surface is exposed to air, the oxide film will form. However, proper and reasonable oxide film can be easily removed by flux. However, PCB’s improper storage or manufacturing process and burn-in process will cause serious oxidation, leaving the flux powerless.

Here are some simple solutions for your reference:

Usually, flux with higher activity has stronger cleanability and can help remove the sticky oxidation film. But, active flux can only be used for some special cases and is not suitable for all PCBs. After the use of this super-spec flux for soldering, more attention should be paid to the influence of its residues on PCB quality. So we must have a strict quality assurance and tracking system to ensure product life.

PCBs can be sprayed or rolled with tin (pre-tin) first by strong flux, and then cleaned with water or solvent. This means that active flux is used to remove severe oxidation film first and then coated with tin to prevent oxidation.

Chemical solvent can be used for etching, which means that, strong-acid solution, after appropriate dilution, is used to wipe the oxidized circuits. After appropriate cleaning, plug in the components and solder them immediately. PCBs with the use of such solution on their surface, if not soldered immediately, will show more serious oxidation. Soldered PCBs should be listed for tracing.

Contaminated flux, low activity or wrong operation cannot remove the oxide film effectively, so they should be included in the assessment.

Tin-lead alloy

★ Too short soldering time or too low preheating temperature may leave the flux not enough time to remove the oxide film. If you can extend the soldering time and strengthen the preheating effect, it is absolutely helpful to remove the oxide film.

Check composition of the tin-lead alloy in tin oven (see Table 2-3, 2-4 and 2-5). Establish standard soldering flowcharts and operation instructions so that operators can follow the rules and reduce the variables due to human error.

Matters affected by impure substances in the solder stick are as follows	
Stibium	Even a small amount of stibium is added, solder stick's tension will increase and become hard and brittle, while the melting point increase.
Copper	Solder stick will be hard and brittle when it contains more than 0.2% the copper. As the content increases, the melting point increases.
Bismuth	High-purity solder stick contains 0.001% the bismuth. The melting point will decrease if the content increases.
Zinc	Zinc is the most harmful. Even containing 0.001% the Zinc, solder stick's luster will disappear and liquidity goes low.
Iron	If containing Iron, solder stick's luster will become bad and adhesion will goes low.
Aluminium	It is just as harmful as Zinc, especially its' liquidity will be worse.

Table 2-4

TYPICAL SPECIFICATION OF ALLOY 60/40			TYPICAL SPECIFICATION OF ALLOY 63/37		
IMPURITY ELEMENT		MAXIMUM IMPURITY PERCENT BY WEIGHT	IMPURITY ELEMENT		MAXIMUM IMPURITY PERCENT BY WEIGHT
Tin	Sn	60±0.5	Tin	Sn	63±0.5
Lead	Pb	Bal	Lead	Pb	Bal
Silver	Ag	0.015	Silver	Ag	0.015
Aluminum	Al	0.005	Aluminum	Al	0.005
Arsenic	As	0.030	Arsenic	As	0.030
Bismuth	Bi	0.100	Bismuth	Bi	0.100
Cadmium	Cd	0.001	Cadmium	Cd	0.001

Copper	Cu	0.050	Copper	Cu	0.050
Iron	Fe	0.020	Iron	Fe	0.020
Stibium	Sb	0.2-0.5	Stibium	Sb	0.2-0.5
Zinc	Zn	0.005	Zinc	Zn	0.005
Total all others	TAO	0.080	Total all others	TAO	0.080

Table 2-5

3 DE-WETTING

De-wetting is similar to non-wetting (see Section 2), both of them are unacceptable defects with regards to soldering quality. In this case, solder has already wetted surface of the object to be soldered, but after a period of time, part of the solder can't adhere to the surface and accumulates in the state of liquid (Figure 2-2). Now, cohesion of the solder will form these accumulated liquid solder into "small drops", making the solder surface uneven. When wetting, eutectic reaction may occur with some metals. For example, if copper, iron, gold, silver, or other metals are soldered with tin-lead alloy, another layer of new alloy (Intermetallic Compound Layer, please refer to the soldering principle) will appear in the interface between metals and tin-lead alloy. The new alloy layer's composition is different from the original metal's composition, also different from original tin-lead alloy's composition. The leveler and evener the new alloy is, the higher the soldering fastness is and the better the wettability is.

There are a lot of reasons to cause de-wetting, mainly the soldering surface is contaminated (oxidized) and solder cannot fully adhere for a uniform Intermetallic compound.

PCBs' de-wetting, excepting to the contaminated solder side of bare copper board, is mainly from tin-plating process. The problem is not the surface of tin-plated layer, but the interface between bare copper board and plated tin. Please see Figure 3-2 for this kind of situation. The problem also happens with tin-plated board, because when soldering, tin's cohesion is responsible for pulling back evenly. If the solder side is contaminated and thus cannot be wetted, the surface tension of the solder side will be uneven, some tin's flow force will be greater than its cohesion, thus tin falls off, causing uneven surface.

When de-wetting happens, re-soldering is not helpful, as most of the contaminated surface has been buried by the tin and the flux cannot pre-clean it, so the wetting function cannot be achieved.

The following suggestions can help solve the problem. Strip the solder off from the solder side and even more clean the oxide film on the surface before re-soldering.

Do not damage bare copper or other materials of the PCBs if stripping off with chemical solvent.

You may also use high-temperature air-knife to molt solder (that is, PCBs are dipped in molten solder and taken out and then strong air knife is used to blow the solder flat), and then flux with high activity is used for soldering.

If this problem occurs with pins, repeated solder immersion or soldering can be helpful.

4 SOLDER BALLS

Solder balls and solder webbing are formed in different places. Most of the solder balls occur on the component side of PCBs (Figure 4-1), while solder webbings occur on solder side. Because of the cohesive force of solder itself, the appearance of these solder particles present spherical. They usually adhere to the surface of the PCBs as the flux solidifies, and are sometimes buried on the surface of PCBs' plastic such as solder mask or printing ink, because these inks, when soldering, will have a softening process and are easy to stick to the solder balls.

The reaction mechanism that pushes a solder ball out of the PCB surface is very similar to the formation of blow holes (see Section 2 of Chapter 2), except that the time for gas to form is different. In the case of solder ball, a large amount of gas in the hole is quickly formed and eager to volatilize, but the molten solder on top of the hole has not yet solidified, so the solder ball is easier to rush out from the top, but less likely to form blow holes or empties from the bottom. On the contrary, in terms of blow holes, gas generation in the hole is slower and less, when it wants to volatilize, the solder at top of the solder hole has been solidified, so it can only rush out from the bottom of the molten solder, thus forming empties. More details will be given in Section 3 of Chapter 2 in this course as to how and from where gas is generated in the hole.

Most solder balls are formed because the un-dried flux volatilizes or the flux's moisture content is too high when PCBs are being soldered. When gas contacts the high-temperature molten solder instantaneously, its volume expands, causing the outbreak of solder, at the same time, the solder is ejected, thus forming solder ball.

4.1 Causes of solder balls

In many formulations of flux, more or less water will be introduced, but this small amount of water will not cause a solder ball. If a solder ball suddenly happens, it can be caused by one of the followings:

Insufficient preheating of PCB which may result in undried flux on PCB surface.

Too high water content in flux formulation.

Defective PTH.

Too high humidity around the factory

4.2 Sources of both dampness and moisture

Excessive dampness or moisture during soldering may result from the following reasons:

A fully-filled flux bucket (200 or 201), if exposed to rain, will collect moisture around the opening. It will suck moisture into the bucket from the loose opening when the temperature changes. So it is important to have a sheltered warehouse and check at all times if the opening of the flux bucket is closed.

During foaming, air compressor will carry a large amount of water and oil into the foaming tank, so it is necessary to install a water filter (or trap), maintain and check the air compressor at any time.

Be aware of any wet parts or tools in the manufacturing process and avoid them as possible as you can.

Working with an air knife, in addition to helping warm-up, will also prevent finger from bringing moisture back into the foaming tank.

When solder ball occurs, the patching process is the same as the webbing's, except that the surface of the part is blocked by many other parts, making it more difficult to remove the solder ball by brushing. Care must be taken to inspect the solder balls under the parts as they are often hidden and difficult to detect (see Section 10).

Solder ball is a defect that can occur at any time in the soldering process, causing serious damage to reliability. Prevention is the only reliable way to avoid it.

5 COLD SOLDER JOINTS

Cold solder joint is defined as the solder joint surface not being smooth which is like a broken glass. Cold solder joint is formed by the mutual movement between parts and PCB in the process of solder joint solidification (Fig. 5-1). This mutual movement affects the crystallization process of tin-lead alloy and reduces the strength of the whole alloy. If cold solder joint is serious, there may even be tiny cracks or fractures on the surface of solder joints.

5.1 Causes of cold solder joints

The causes of cold solder joints are as follows:

Vibration of conveyor belt.

Unbalanced bearing or motor rotation.

Too strong exhaust equipment or electric fan.

PCB has passed through the outlet of conveyer track but the solder is not dry yet.

Negligence by touch-up workers.

After PCB is soldered, we can solve the problem of cold solder joints by keeping the conveyer track stable and let tin-lead alloy get perfect crystallization in the process of the solidification. If cold solder joint occurs, it can be repaired by touch up. If cold solder joint is serious, we can consider a re-soldering.

The vibration of parts may affect the solidification of solder joints, resulting in uneven or incomplete appearance of solder joints. Our quality control staff must establish a set of appearance standards of solder joints, so that the soldering operators can have a basis for judgment.

6 INCOMPLETE FILLETS

There are many names for incomplete fillets that circulate in the industry of electronics, such as blow holes, pin holes, drop-outs, or empties. In the following course (Chapter 2), we will discuss these defects separately and integrate our terms according to their different characteristics. Therefore, incomplete fillets can be classified into Unfilled Holes or Poor

Solder Rise.

6.1 Unfilled Holes

There is no solder coated around the single-layer board, double-layer board, multi-layer board and solder joint for 360^0. (Figure 6-1).

6.2 Poor Solder Rise

Solder is not wetted completely to the top of the hole wall. This occurs only on double-layer or multiple-layer boards (PTH) (Fig. 6-2). When new PCB design is completed and is soldered the first time, to track the problem will be very complex. If design is complete and the production of PCBs and soldering process are very stable but these problems are suddenly found, the followings should be checked and improved one by one. However, before discussing the followings, the inspection of both machines and materials must be completed first, for example, temperature, speed, flux, lead-tin alloy, etc. If the problem is not caused by the machines and materials, inspection should be carried out according to the followings.

The causes of unfiled holes can be classified as follows:

Bad solder joints of parts and PCB itself.

Solder mask flows into the PTH or sticks to the copper pad surface (for single-layer boards).

Incorrect ratio of hole to pin.

Unstable solder wave or vibrating conveyor belt.

Poor Solder Rise is often caused by followings:

Bad solder joints of parts and PCB itself.

Cracks or residues exist over the PTH.

PTH is contaminated.

Solder mask flows into the PTH.

Flux's activity is lost due to over-heating.

If these problems occur with a certain batch of parts or PCBs, you may check and compare other batches of parts or PCBs to find out the difference, or check whether there is a change in the manufacturing process of the upstream manufacturer.

If problems are repeated with certain parts, it may be that the original design did not consider the heat balance problem. That is, the temperature distribution of PCB solder joint is not balanced. From this point of view, it can be seen that the multiplayer board needs special preheating treatment, especially heat should be transferred to the top of the through-hole, because in the case of incomplete solder joints, the multilayer board will not be acceptable.

As for the height of the solder level in a PTH, each factory has its own standard based on the requirements of its products. In the case of double-layer boards, IPC specifies that height of the solder level from top of the through-hole is allowed to reduce by 25% (based on the thickness of PCB). But the PTH wall must be completely wetted even the height is reduced by 25%. As to the multiple-layer boards, a lot of manufacturers require that PTH must be filled completely.

7 Excess solder

Excess solder is defined as having too much solder around the solder joint as we are unable to determine whether it is a standard solder joint (Fig. 7-1). Too much solder hides the curvature of wetting between solder joint and PCB, and may cover part of the pins where should be exposed, making it invisible to the naked eye. Excess solder doesn't make the solder stronger or more conductive, it just wastes solder.

Each electronics factory must have a set of instructions for its own product or soldering process, which must specify the maximum amount of solder to be consumed per solder joint. The surface, top and bottom of the solder joint must be well wetted in shape of a cone with a standard curvature (solder strip).

7.1 Causes of excess solder

Incorrect depth of soldering.

Not enough pre-heating or solder temperature.

Improper selection of flux activity or gravity.

Bad solder joints of PCB and parts.

Unnecessary grease is mixed into the soldering process.

Solder composition is substandard or solder is heavily contaminated.

If the excess solder is found, it must be removed. The most efficient method is to re-solder the PCB, but the PCB must be allowed to sit for 4~6 hours, so that the strength of PCB's resin structure can restore. If soldering is performed twice too fast, it will cause heat damage.

The occurrence of excess solder can affect the reliability of products seriously. It may mask defects in solder and cause headaches in mechanical strength, electrical properties or appearance. No soldering standard allows severe excess solder to occur.

8. ICICLING

The term "icicling" can describe the shape of solder joint very aptly (Fig. 8-1). The reason for this is that when the molten solder touches the object to be soldered, it will cool down rapidly due to a fast loss of heat and be drawn into a sharp shape like icicling before reaching the wetting state. They often occur in wave soldering, dip soldering and touch up processes. Places where icicling happens include solder joints, solder surfaces, pins, and even solder-dipping equipment or tools.

Based on different soldering processes, the causes can be classified as follows:

8.1 Touch up

When soldering manually with a soldering iron, "small flag" may happen over solder joint and tip of the soldering iron. This situation is due to uneven heat conduction which results in sharp cooling of solder, meaning that soldering iron's temperature is not enough.

The solution is not to always increase the temperature of the soldering iron. Instead, we should use the same temperature, but an iron which can supply more heat. That means we may use an iron with higher heat content and more stable temperature, or larger contact area. The tip of the soldering iron should be kept clean. Proper solder stick and correct touch-up technique are also helpful to solve the problem of icicling.

8.2 Wave soldering and dip soldering

The causes of icicling with automatic soldering furnace are quite complicated. In addition to heat conduction, other problems such as bad solder joints, design, machine and equipment may also result in icicling. Let's discuss them one by one.

A. Heat conduction

Temperature from machine or tool is unbalanced.

Uneven heat transfer may be caused by too large solder area on PCB surface or too dense objects to be soldered due to local heat absorption. Heavier metal parts may absorb more heat.

Bad solder joint

Bad solder joints of parts and PCB itself.

Flux with low activity may not wet parts enough.

Incorrect ratio of pins to holes.

Too large PTH before part is inserted.

Too large solder area on PCB surface may result in slow solidification and high fluidity of molten solder.

Machine & equipment

PCB is soldered too deeply.

Wave solder flows unstably.

The solder surface in a manual or automatic solder-feeding furnace has solder slag or suspended matter.

B. To solve icicling, we must determine its source first.

Heat conduction and problems with machine and equipment can be adjusted by means of detection. Design problems must be solved by improving the original design or overcome by touch up. Bad solder joints must be resolved by other means (see Section 1).

9 BRIDGING

Bridging will cause PCB short circuit. The reason may not be only excess solder (Fig. 9-1). However, the cause of short circuit is not simply the bridging. Problems may occur with the metal wiring under the solder mask, or the parts themselves (To be mentioned in Section 10 of this chapter). A bridging is defined when a short circuit is caused due to the connection between solder joints on PCB surface. The main cause of bridging is PCB circuit design, soldering materials or machine and equipment.

9.1 PCB design

The discharge of solder flow on solder surface of PCB is not considered (PCB design criteria is not followed). So, it is easy to cause solder accumulation and form a bridging when solder flows through PCB.

Molten solder flows before solder joints or other soldered wirings are dried after a PCB has been soldered. So, solder touches adjacent solder joints or soldered wirings.

PCB circuits are designed too close to each other.

Not active enough flux.

Pins are not bent regularly or are kept too close to each other.

9.2 Soldering materials

There are solder, copper or other metal residues on PCBs or pins.

Bad solder joints on PCBs or pins.

Not active enough flux

Tin-lead alloy is contaminated.

9.3 Machine & equipment

Heat is not enough.

Scum appears on the surface of solder wave.

PCB is soldered too deeply.

Touch up may be used to separate the bridging if found.

10 SOLDER AND COMPONENT SHORT CIRCUITS

Short circuit is usually called “Short.” Some short circuits occur in the circuit covered by PCB solder mask or over the parts in contact with each other. Short circuit resulting from these should not be confused with bridging (see Section 9 of this chapter). When a short circuit occurs, the electrical function of the PCB itself is absolutely unable to work. At this time, various automatic test instruments can be used for detection and correction. But if PCB has intermittent short circuit because of temperature change, vibration or impact, it is difficult to detect its position correctly. When encountering such intermittent short circuit, it can be analyzed and checked according to the following different situations.

10.1 Short circuit occurs with wirings covered with solder mask

The main reason for this is that the PCB substrate wirings, when being tin-plated or tin-sprayed, are plated too thickly. When tin-plated boards are soldered, some plated solder will once again molten by 250℃ molten solder and reflow around, and mostly flow to the bottom. When the plated solder is melting, high tension may appear with PCBs and solder mask, leading to the flow of molten solder. This strong momentary push often puts the solder into the adjacent metal wirings, thus causing short circuit. Following are some solutions for your reference.

Minimize the thickness of solder when tin-plating or tin-spraying the PCBs.

When designing PCB circuits, put the wirings as far away from each other as possible.

Standard treatment is to re-coat the solder mask.

10.2 Short circuit occurs between components

This is caused by improper design or processing.

A. Design

The exposed wiring is too close to the top of the solder joint.

Metal parts or lead wires are too close to the exposed wirings.

Parts or lead wires themselves are touching each other.

B. Processing

Two serious vibration of solder waves.

Out gassing of solder happens when soldering.

Solder balls from IR reflow or wave soldering.

If short circuit occurs in components themselves, it is very difficult to find out the cause. Currently, only X-ray can be used to find it, but the speed is slow and cost is high, which is not in line with economic benefits. When short circuit happens, parts are replaced or PCBs are repaired for solution.