Vitronics Soltec

By G. Schouten

General information about designing for solderability

Introduction

The data from component suppliers on lay-out for solderability are in general given with the assumption that only that particular component is used.
The real world shows us that this is seldom the case. This means that in a lot of cases, using the "recommended" lay-out rules, one often get poor soldering results.
The way to improve soldering quality, is to recognise the effect of other components on the same board and add some common sense to the lay-out rules. This information is meant to recognise under which conditions common sense has to be put in.

Designing for solderability

This means that the lay-out is such, that it will promote solder wetting on joints and prevent solder bridging between joints.
It sounds easy, but is often difficult to achieve, due to the fact that both, solder wetting on joints and solder bridging between joints, is obeying to the same physical laws that are involved. The solder joints are formed by capillary forces, acting on the solder during the soldering process.
Capillary forces depend on dimensions and surface tension.

Small capillary gaps have a strong tendency to hold the solder, since the capillary force in these cases is strong. So if component leads are closely spaced, like on SOICs, solder bridging is likely to occur, unless we take some action to promote solder drainage to so called "solder thieves", or "solder collectors".

If other components are closely surrounding components with closely spaced leads, they may promote solder bridging, due to their influence on solder drainage at the point were the board exits the solder wave. This also very much depends on the type and shape of these components as they will interfere with solder drainage conditions.

SMDs in wave soldering

SMDs are designed for reflow soldering. If one uses the lay-out guidelines for reflow soldering and apply those in wave soldering, one could expect problems. So ensure yourself that you are using the proper design guidelines. This remark is of special importance for the SOT-23 pads, which are often too small. Be aware that the advised spacing, or free area, between components is only valid if they are surrounded by the same components. This means that if larger, or higher components are mixed with small components, we must use the free area of the largest component. This free area is necessary to promote wetting and to prevent bridging. If this free area is not specified, one should at least have an area around the component which is equal or larger than the thickness (height) of the component. In case of a mix of high and low components, a free space of at least twice the thickness of the larger component shall be the free area. This is to prevent solder shadowing during the solder process, which might give skipped (non soldered) joints.

The space between adjacent pads of different components shall be at least equal or larger than the thickness (height) of the thicker component.

Information Sheet 035

All SMD components should be placed so that the solder flows easy along the joints. This means that SOICs should be placed lengthways to the transport direction, with sufficient large solder thieves on the trailing edges. The resistors, capacitors, melfs, etc. should be placed perpendicular to the transport direction, so that the solder flows along both connections of the component at the same time.

Solder resist

The thickness of the solder resist layer should for wave soldering never exceed the thickness of the metal plating. If the pads are below the level of the solder resist aperture, skipped or missed joints are likely to occur.

Boards

The boards should have a sufficient free area around the edges, e.g. 5 mm, in order to fix them during soldering, without interfering in joint formation.
Large holes, cut-outs and routed slots should be avoided, or covered during soldering, so that the necessary wave pressure, to achieve good wetting, can be applied.
The boards should be kept flat during soldering. Only in that case the solder process can be reproducible.

Conventional leaded components

The design rules for conventional leaded components mounted in through holes, is not completely the same for boards without hole metallisation compared to boards with metallized holes.
For filling the plated through holes with solder we need the capillary between lead and hole. Depending on the thermal mass of the joint this gap should be 0.4 mm up to twice the lead diameter

The design for non metallized holes is often more critical than for metallized holes. The reason is that air is not solderable, but with non plated holes we must always bridge the air gap between the pad and the lead in order to make a full joint. As long as this air gap is smaller than 0.3 mm, we can make a closed joint, if also some other requirements are fulfilled.

These other requirements are:

- The solderability of the lead and the solder pad on the board must be perfect.
- The hole edge must not be damaged, e.g. by the inserted lead.
- There may be no foreign particles present between the lead and the hole.
- The component might not cover the hole on the component side and so block escaping vapours during the soldering process.

The pad size must be sufficient to give a good adhesion between pad and board. This does not mean that we need the full pad size to make a solder joint. It is often more desirable to increase the space between adjacent joints in order to prevent solder bridging, because large pads promote solder bridging. This can be done by "cutting" segments from the pads, or by covering these segments with solder resist. However not all solder resist types will make this option a good alternative.

For metallized holes the pad size should be minimised as much as possible, in order to prevent solder bridging between adjacent joints. A pad size of 1.3mm will be sufficient in most cases. The solder joint gets its strength from the hole metallisation. The solder fillet, which is essential for non plated holes, because it is there the only connection, is not of major importance here.

In general the protruding lead length should be limited to 1.5 mm. Larger leads give no stronger joint, because no more solder will adhere, but they will increase the risk of solder bridging. For non metallized holes the protruding length should be at least 0.8 mm. If the leads are bended to the pad, the amount of solder between the lead and the pad is mostly sufficient to guarantee a sound joint.

Do not design joints in perpendicular rows in respect to the transport direction, since this gives an uncontrolled solder separation between the joints and the solder wave during exit from the wave. This will cause solder bridging.

Massive or fat solder joints

The visual quality aspects which are defined in several standards were defined at a time that almost all solder joints were made by hand soldering with a soldering iron.
One of the quality aspects in judging solder joints is the shape and wetting angle of the solder fillet. This control aspects was used for hand soldered joints to assure that no bad solderable joints were "buried" with solder. In machine soldering however it is impossible to get (much) solder on a non or poor solderable pad or lead. So if we sometimes have much solder on joints which are machine soldered, we know that there is no question about the solderability. The "fat" solder joint is caused by the solder drainage conditions and should therefore not reworked, because it is a perfect reliable joint.

Poor solder joints

In most cases poor solder joints are formed due to lack of solderability of the component leads and / or the board. The solder process can not improve solder failures that occur due to lack of solderability.

Final remarks on preventing solder failures

Although one can prevent solder failures very much by using the right design rules, it is very difficult to have it perfect, right after the first design. If a board should be soldered in large series it is often necessary to improve the design and to optimise it for zero defects. The reason is that the behaviour of liquid solder can not be completely predicted. The board lay-out is the most important parameter which may interfere with solder defects. This can simply be proved by checking the solder defects. One will always find them on the same spots. If the solder process was out of control, one would find more random failures over the whole board. The fact that failures are found on specific places points out to the design.