Wave Soldering Problems

What is a good joint?

The main function of the solder is to make electrical interconnection, but there is a mechanical aspect: even where parts have been clinched or glued in position, the solder also serves to strengthen the joint. As a reminder of what is said in How joints are made, it is generally agreed that:

- There should be a visual appearance of good wetting, with the correct **amount** of solder and a **sound and smooth surface**
- All soldered joints on an assembly should give a uniform impression independent of their location on the board.

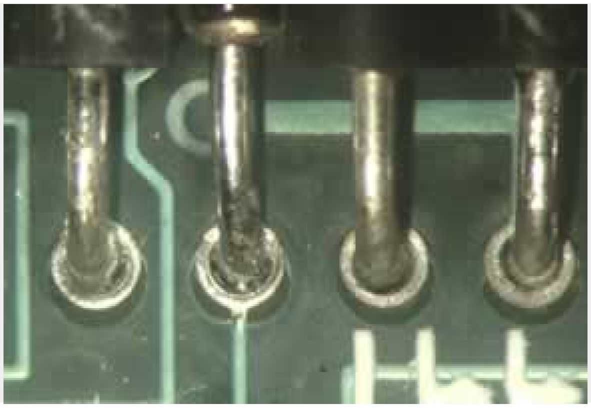
Satisfactory solder joint

Non wetting of lead

In wave soldering, the form of the joint is determined by mechanical and process conditions, and is not limited by the amount of solder available, which is essentially infinite. Solder should flow evenly over the surfaces to be soldered and run out thinly towards the edges of the joint, with a contact angle <30°, unless the solder fillet is small and the contact angle constrained by the closeness of the edge of the solder land, as may be the case with small SM components.

For through-hole pins, there is usually also a requirement that the solder fillet to the lead should be visible on the top surface, with the solder having been pulled upwards by capillary attraction. This is not so much for reasons of joint strength or connectivity, but to ensure that there are no defects in the plating (such as cracks) which indicate potential unreliability.

Poor topside solder fillets



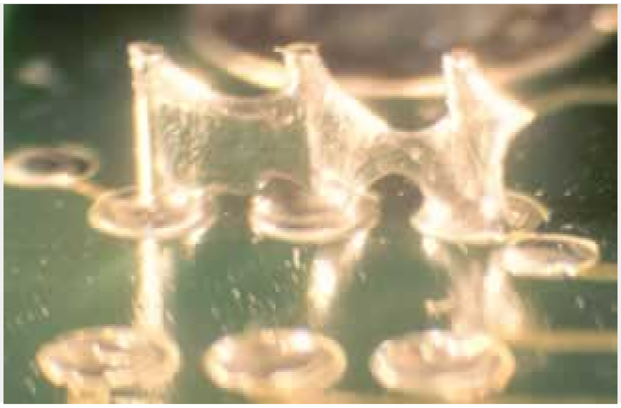
Process faults

Our list of typical wave solder defects can be divided into three main categories:

Too much solder

- solder bridging
- covered pins
- solder peaks
- solder on gold fingers
- solder balls

Multiple solder shorts



Not enough solder

- shadowing
- missing – solder skip
- missing – not fluxed
- contamination/residues
- voids and blowholes

Pin-hole/blow-hole in solder joint

Mechanical problems

- flooding
- lifted components (pushed up)
- cracked joints (moved after soldering)
-

Through hole component lift

Apart from the mechanical problems, many of these defects are related to the quality of the wetting that is achieved, and the way in which the solder flows away from the joint during the peel-back that happens in Zone 3. For wave soldering, vital requirements are:

- freedom from dross on the solder
- critical cleanliness of the equipment
- correct set-up for the specific circuit
- effective control of flux quantity
- maintenance of flux and solder purity
- accurate control of preheat conditions and pot temperature.

Solder balling

Solder balling occurs both with wave soldering and reflow soldering, and may occur intermittently even in the best regulated processes. Its mechanism can be extremely complex, with the root cause lying in a number of areas. There is also considerable interaction, and factors that do not produce balling on their own may do so in combination. For example, soldering in nitrogen, which changes the surface tension of the molten solder, has often been reported as leading to an increased incidence of very small solder balls.

Wave soldering solder balls

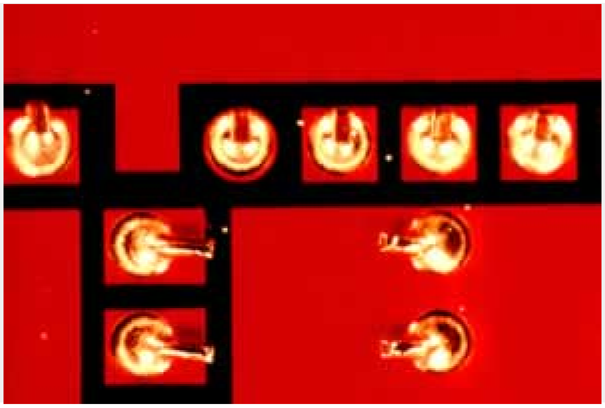
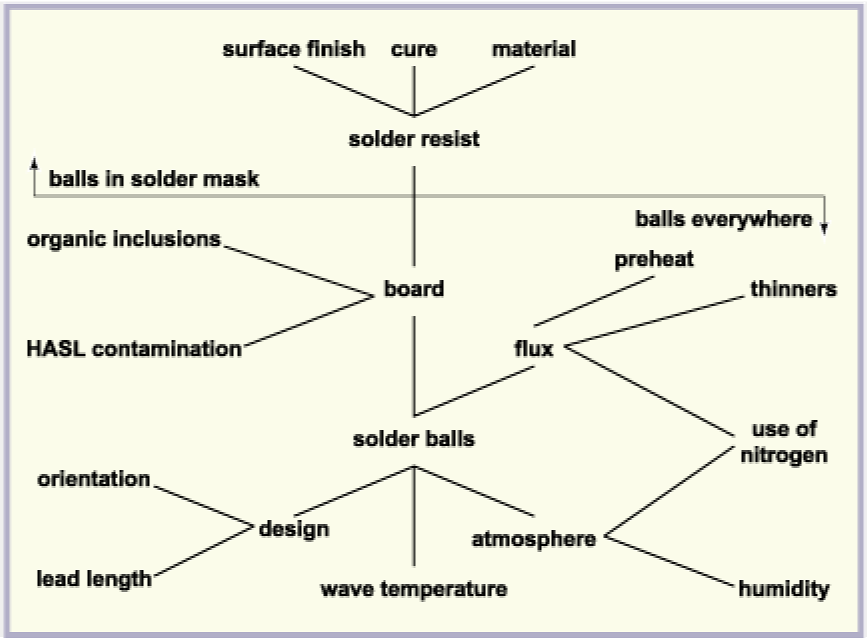


Figure 1 tries to indicate the most likely causes for solder balling in wave soldering. Note that clues to the origins of the problem can be gleaned from observing the nature and distribution of the problem. For example, solder balling associated with particular components can be a design issue, whereas balls embedded in the solder mask, that leave discoloured marks on the board when removed, indicate solder mask incompatibility.

Figure 1: Causes of solder balling



In the case of wave soldering, a rough surface is preferred, especially with low solids fluxes: trapped flux is able to reduce the surface tension of the solder as it peels away from the board, so rough finishes on solder masks generally display fewer solder ball problems than smooth.

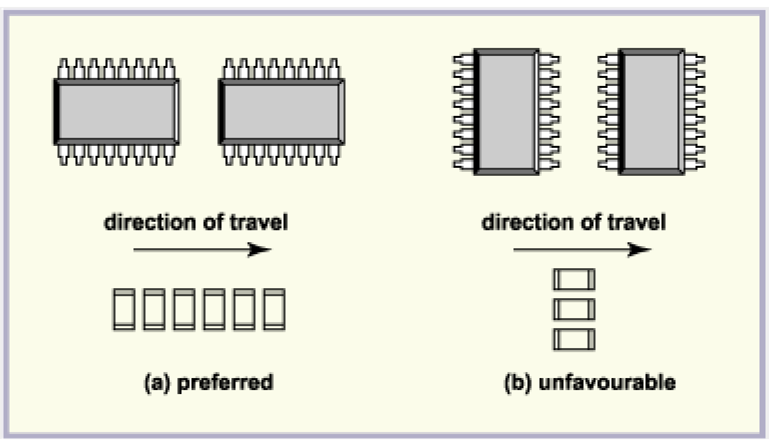
Cases have also been reported of increased solder balling caused by insufficient cure of the solder mask. The effect here is probably related to producing boards with different surface tension characteristics.

A bridge too far!

In most factories, the majority of defects on wave-soldered boards are solder bridges, formed because contact with the solder wave is lost before a sufficient amount of solder has drained from the joints. Often these bridges are linked to particular designs and components, for example through-hole multi-pin components such as connectors and on trailing leads of surface-mounted integrated circuits.

With a row of pads, it has proven easier to avoid bridges when they emerge from the wave in single file, rather than all of them together in a broad front. In examining the impact on solder bridging of pad design and of component orientation, Comerford concluded that “Bridges occurred three to ten times as frequently on integrated circuits oriented perpendicular to the line of travel”. The board should therefore be laid out with all multi-lead packages oriented perpendicular to the wave (Figure 2). With SM components, this is also the optimum orientation to avoid shadowing, where solder fails to reach certain joints because the component body impedes solder flow.

Figure 2: Preferred alignment relative to direction of travel



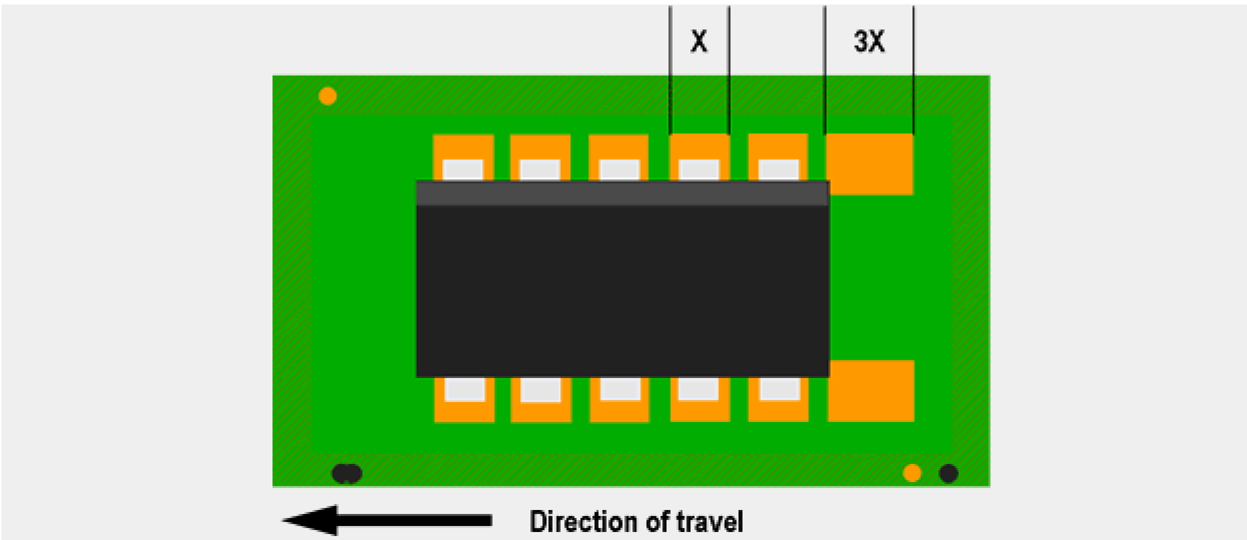
(a) favourable alignment
(b) alignment resulting in formation of bridges

Solder-thieving or ‘robber pads’

An observation first made with through hole components was that, as a row of leads or footprints leaves the wave in a single file, the peelback seems to jump from lead to lead, until the last two emerge, when a bridge tends to form between them. The problem with any series of pins, especially when close together, is “Which pin does the last pin snap to?”

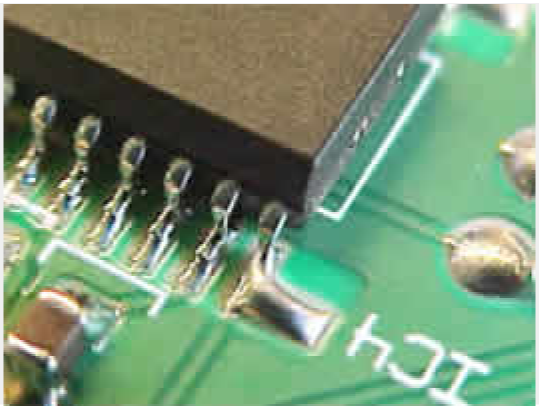
The drainage problem is solved by a combination of aligning the lead array correctly with respect to the solder and providing somewhere for surplus solder to go, by placing a somewhat larger dummy footprint, called a ‘solder thief’ or ‘robber pad’ (Figure 3), at the end of the row, so that it will draw the bridge to a place where it does no harm.

Figure 3: SOIC pad layout with ‘solder thieves’



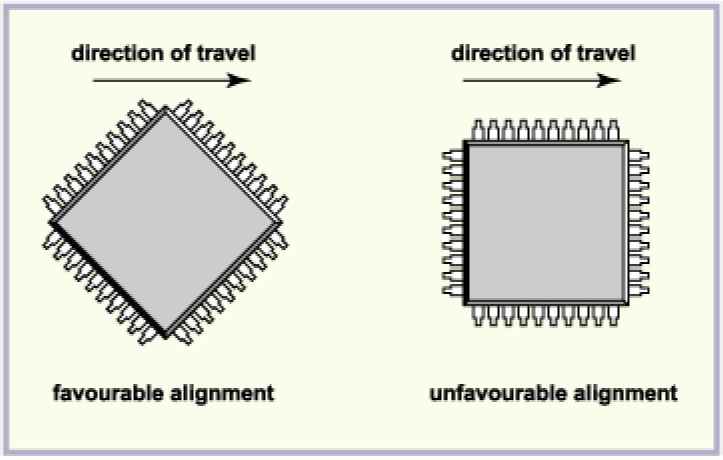
Robber pads are usually two or three times the width of the component pin but the same length, and added beyond the end component pins as in Figure 4. In some cases either edge could be the leading edge of the board in wave soldering, so thieving pads would be provided at both ends.

Figure 4: A wave soldered SOIC with a thieving pad



For components with terminations on all four sides, two of the sides will be prone to shorts and shadowing, no matter which way round the component is rotated. One way of reducing bridges (it improves drainage by increasing the effective lead spacing by 41%), and of helping solder reach the joints on what would be the shadow side, is to position these components at 45° to the direction of travel (Figure 5).

Figure 5: Mitigating the shadow effect



Unfortunately, not every design can be tackled in this way, nor is every designer willing to produce a 45° design! Alternative approaches for situations where yield issues result from these types of components but it has not been possible to redesign the product are to:

- use a pallet to hold the board at 45° (this needs to be loaded and unloaded, so increases production costs, and will also limit the size of the largest board that can be handled)
- improve drainage by using an angled solder pot
- use reflow for QFPs and selective wave soldering elsewhere.

Avoiding bridges is also more difficult with fine pitch packages. One suggested design approach, which is feasible down to 0.8 mm pitch, is 'long and thin' – thin to avoid shorting, and long to provide enough solder to wet the pad and wick up the lead:

- make pads the same width as the maximum lead width
- use the outer dimension of the device, from toe to toe, or the data sheet maximum value, and extend the pads beyond this *as far as possible* – by 1.3 mm if you have the space
- don't forget the usual thief pad at the end of the row.

Mechanical methods

Lack of board 'real estate' means that it is not always possible to adopt best practice designs. An alternative way of reducing the bridging problem is to use a physical barrier of a material that will not wet with solder. Even a relatively thin layer can deter the formation of a bridge. Both solder mask (Figure 6) and glue (Figure 7) have been used for this, but care has to be taken with both materials to ensure that they are not deposited on the pad and that the solderability of the pads is not impaired by any bleed or residue.

Figure 6: Solder mask overprinting reduces shorts with the same size pad

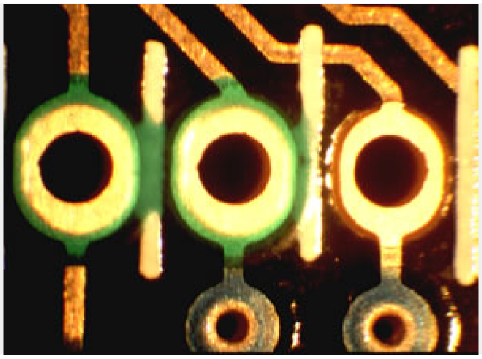
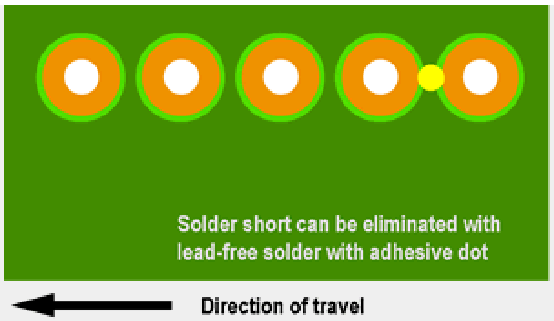


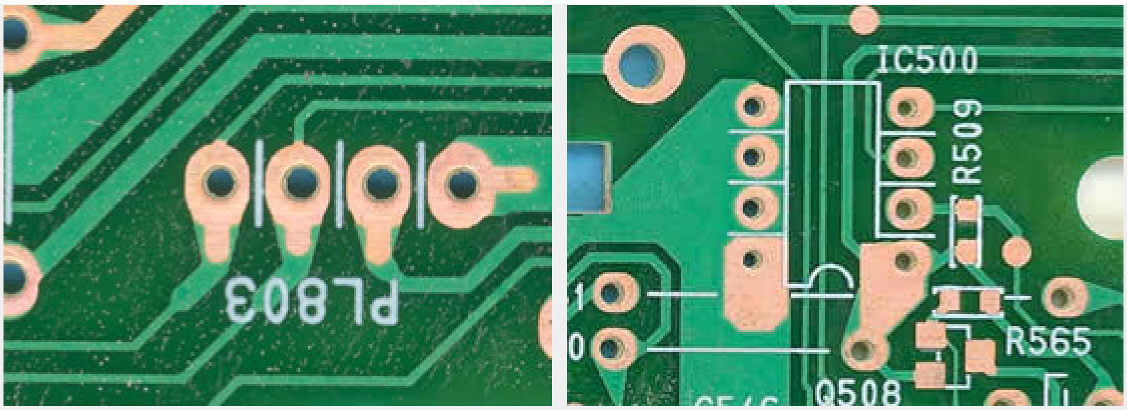
Figure 7: Preventing bridging by using a glue dot



Drainage pads for through-hole parts

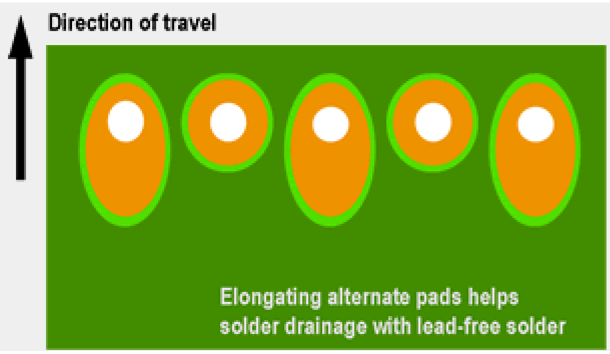
In the same way as for surface-mount components, through-hole parts with multiple leads benefit from having extra pads or pad extensions so that surplus solder can drain away from the joint. Some of the ways in which this can be done are indicated in Figure 8.

Figure 8: Examples of drainage pads that can be used on single- and double-sided designs



Connectors provide a particular challenge when they are presented to the solder pot across the wave, rather than in line, which allows the designer to use conventional robber pads. Unfortunately, on some boards the orientation of the connector is determined by the constraints of the overall application, and the crosswise direction cannot be avoided. Figure 9 shows an alternative approach for improving the clearance of shorts that has proven useful with lead-free solders.

Figure 9: Improving solder clearance by elongating alternate pads



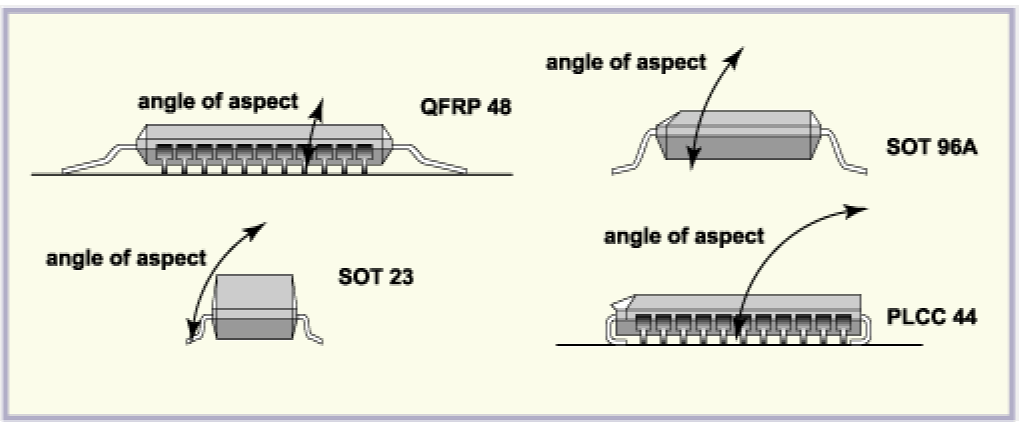
Solder skips

Solder bridges and skipped joints are apparently at opposite extremes of the soldering defect spectrum, but have the common features that their solutions need attention to both process and design, and the cures are often similar, soldering success being improved by using a preliminary chip wave and aligning the pads with the direction of travel.

When soldering bottom-side surface mount components, the requirement is to get solder in contact with the terminations and pads for *long* enough and *intimately* enough for wetting to take place. Where the solder cannot properly wet the interface and form a joint, the result is a type of defect known as a 'solder skip'. Tackling this problem involves both design strategies and machine modifications.

Soldering problems began as soon as surface mount components started to be wave-soldered to polymer circuit boards. Whilst chip components presented few problems, active component formats were not very 'soldering-friendly', SOICs and PLCCs being especially difficult to wave solder. This is because the ends of the leads are too close to the relatively high body mouldings. The 'angle of aspect', formed between the upper edge of the component body and the end of the solderable lead, is about 60° for SOICs and can reach 90° for PLCCs (Figure 10). The solder wave finds it difficult to access these corners, because of the high surface tension of the molten solder. Until wetting takes place, the solder surface in contact with a component is like a balloon pressing against the walls of a room – in a tight corner, at best it will only make contact at the periphery.

Figure 10: The contours of SOICs, PLCCs and QFPs



This lack of contact means that smooth 'lambda' waves, so good at soldering through-hole components, often produce unsatisfactory SM soldering results because there is not enough movement to break the surface tension of the solder at the component lead/pad interface. A similar sort of situation exists where SM parts are closely spaced, making it difficult for the solder to access the joint. This problem is both addressed during board design and tackled during manufacture by using waves with high turbulence and an appropriate angle of attack. The concept of the double wave is shown in Figure 11.

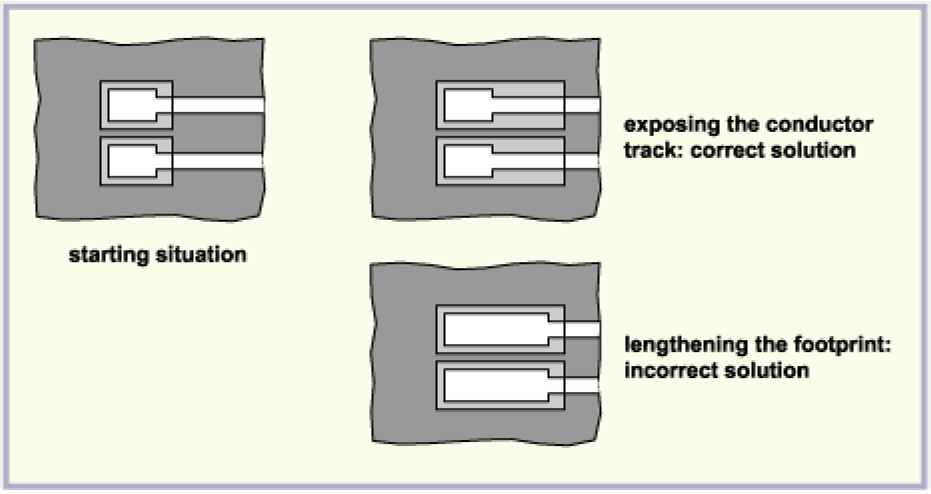
Figure 11: Double wave system: the ‘chip’ wave is on the left

The primary or ‘chip’ wave is a symmetrical wave with an intentionally turbulent wave crest (Figure 12). The high kinetic energy at the point where the solder meets the board ensures that the solder finds its way to every joint on the board. A secondary wave then allows the solder to drain away from the board without leaving behind any bridges or unwanted accumulation of solder.

Figure 12: Close-up of turbulent chip wave

From the design perspective, the potential for skipped joints can be reduced by extending the footprint, or exposing a short length of track not covered by the solder mask. These can help lead the solder to a joint close to a high component body (Figure 13).

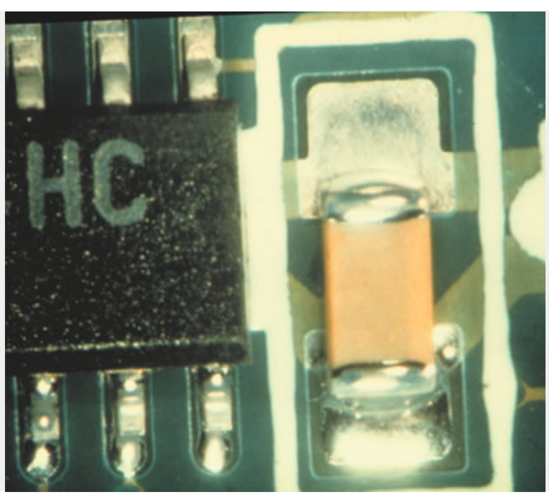
Figure 13: Letting the conductor track lead the solder to the joint



Footprints for MELFs and chips should extend far enough to provide an aspect angle of about 60°. This allows for slight misalignment of the component, so that in no circumstances does the angle get steeper than 45°.

One also has to remember that even the underside of an assembly is not flat, and that the wave comes into contact with a three-dimensional surface. It is therefore possible for areas of the board to be physically prevented from coming in contact with the solder, creating a type of solder skip referred to as 'shadowing' (Figure 14).

Figure 14: Shadowing caused by an adjacent component

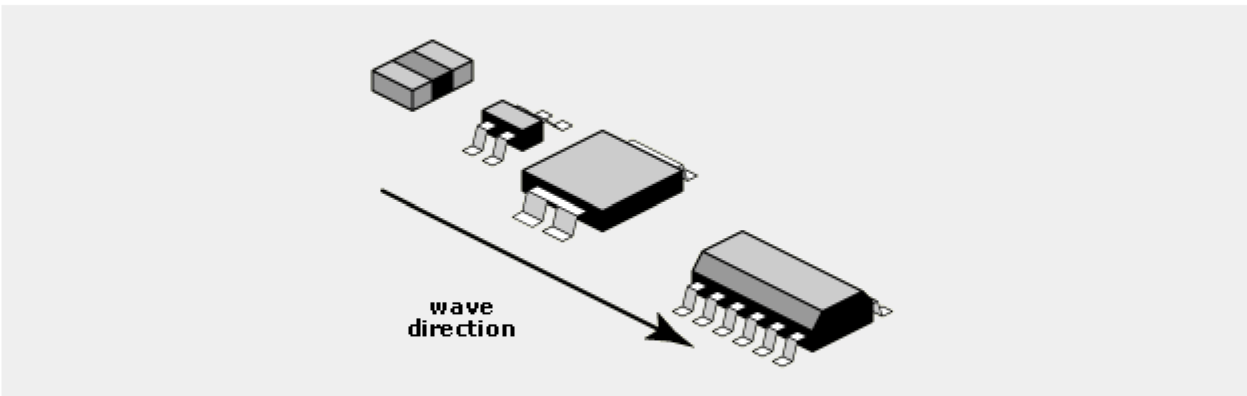


This specific problem can be overcome by altering the direction in which the board approaches the solder wave, but only if the layout engineer has forseen the potential for this kind of defect and has made sure that there is at least one direction in which shadowing will not occur.

A similar effect can happen with closely-spaced pins, as with connectors. Here the solution is to orientate the banks of pins parallel to the direction of flow, so that the pins hit the wave sequentially and the presence of the component body does not affect the joint. When laying out the board, therefore, it is recommended that all DIP and axial components should be aligned along one axis. This has the subsidiary benefits that it makes the board easier to inspect, and slightly reduces the auto-insertion machine time, saving on production costs; having even one axial component on a board oriented in a different axis, means that the axial insertion machine must rotate the board or pallet to install that component.

When surface mount components are to be wave-soldered, again there is a preferred orientation of the component relative to the wave (Figure 15) in order to reduce both shadowing and bridging.

Figure 15: Recommended orientation for SM components



Source : http://www.ami.ac.uk/courses/topics/0170_wsp/index.html