

Profiling for successful BGA/CSP Rework

Our thanks to Metcal for allowing us to reprint the following.

This paper discusses how to successfully profile a printed circuit board when reworking Ball Grid Array and Chip Scale Packages.

Board Profiling for Production vs. Rework

If we consider profiling a PCB for reflow in production, we are generally concerned with achieving an even Delta T (ΔT) across all of the joints being soldered. This is accomplished by evenly heating both the top and bottomside of the whole board, generally through a multi-zoned convection reflow oven. The capacity of the oven would be appropriate to the size or thermal mass of the PCB and the production throughput requirements.

When we consider reworking an array package such as a BGA or CSP, it is recognized that process control is of paramount importance to achieve a successful result. The normal way to approach the rework is to try and emulate the production reflow profile for the individual component being removed or replaced. The nature of the operation dictates that we only wish to reflow the component being reworked and therefore we have to heat a specific area of the circuit board. Selective heating of an area of board can inadvertently cause process failure even when all the parameters *appear* to be correct. This note discusses the influential factors and the potential pitfalls to be avoided

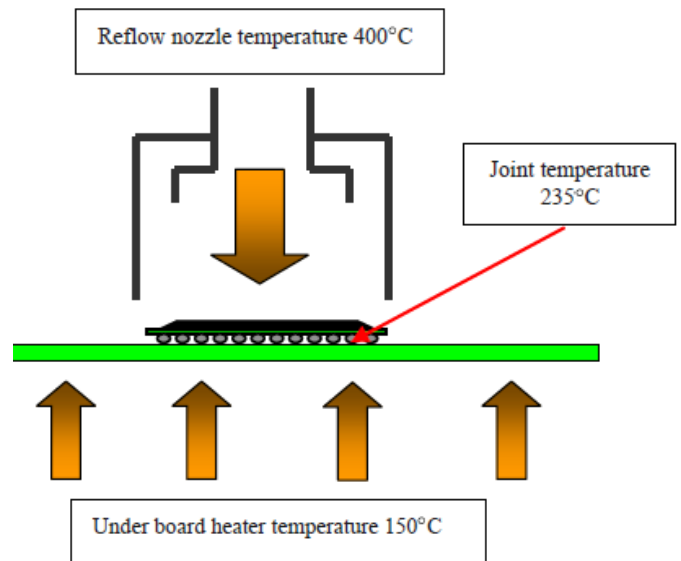
Process Yield Problems

In some cases, particularly on larger PCB assemblies, failure may still occur, giving low yields when all of the conditions for rework are apparently correct. The requirements for a solder profile are generally well understood. They are normally dependent on the materials being used (solder paste or flux) and would typically be as follows.

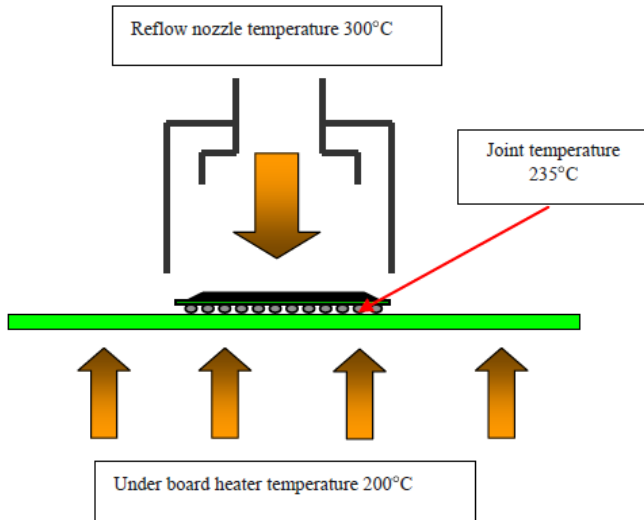
Zone	Time Duration (s)	Target Temperature °C
Pre heat	60 to 90 sec	100 to 120
Soak	60 to 90 sec	155 to 175
Soak 2	60 to 90 sec	210-220
Reflow	30 to 60 sec	230 to 240

On any reflow soldering system that heats both sides of the PCB assembly, the required solder joint temperature is a function of how much heat is applied to both the top and bottom side. It is quite possible to achieve the same solder joint temperature with a number of heat settings. The following 2 examples illustrate this.

Example 1



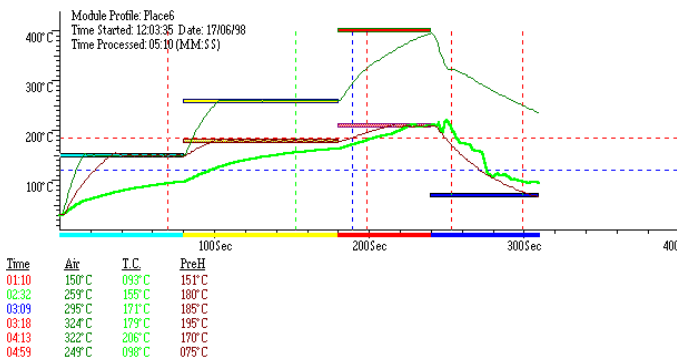
Example 2



The following is an explanation of the kind of faults that can occur when everything appears to be set correctly (i.e. perfect screen print, no board damage, and accurate placement.) but the yields are low.

Practical Case Study

The graph below shows a trace of a profile taken from under a 492 PBGA on a PC motherboard where the thermocouple trace appears to be normal – hitting the desired pre-heat, soak and reflow temperatures. In this case with either solder paste or flux, a very low pass rate was achieved.



The second graph shown is the final profile that was used, and as you can see, the 2 traces are very similar but the heating has been done in a totally different way. In this case, we have increased the number of zones to 4, reduced the top air temperature significantly while increasing the underboard heat to the whole board and utilizing an underboard centralized heating zone.

The route of the yield problems was due to the plastic BGA warping during reflow. An excessive source temperature on the topside of the PCB caused this.

Warpage

A major cause of defects is when either the PC board or the component warp during reflow. Even minor deformation can lead to defects such as open solder joints and solder shorts. A normal PBGA stands off the PCB by about 0.020" and lifting by even 0.005" across the device is enough to cause an open circuit. Another factor is that the larger the device, the more prone it will be to these problems. Even if the board/component survives the process with no apparent defect, then the joint will be constantly under strain as the board returns to its normal shape - causing long term reliability problems.

Fig 1

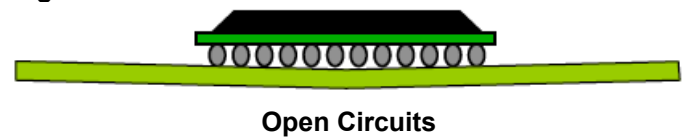


Figure 1 shows unsoldered joints caused by PCB warpage; this can also be caused by component warpage. The joints in the center of the component have lifted above the PCB and were not able to make the solder connection. This is more typical when using only flux dip as there is no solder paste to make up the void area. Another characteristic is that the opens will generally be towards the center of the device.

Fig 2



Figure 2 shows solder shorts. In this case the deformation has caused the solder paste and the solder balls to combine into solder shorts. This is more typical with solder paste, and is normally characterized by shorts on the device periphery, especially in the corners when the device becomes “dog eared” during reflow.

Solution to Problem

The solution to the problem of warpage is generally related to the profile settings, particularly insufficient underboard pre-heat temperature combined with too high a top temperature. Some boards are however badly balanced thermally, that also can lead to problems.

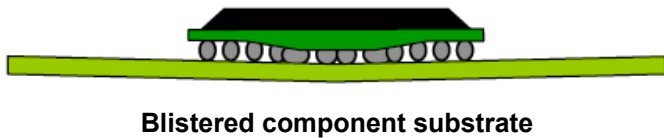
On large PCBs underboard support can also be a factor with the PCB sagging under its own weight. Note: Downward pressure should not be applied by either the reflow-head or vacuum, as this will exacerbate any problems.

Component De-lamination

Another cause of solder shorts is de-lamination of the component substrate during reflow. Plastic packages are generally hydroscopic, meaning that they absorb moisture during exposure to the atmosphere. If the device is then rapidly heated, the moisture expands creating a cavity inside the device. This effect is known as “pop-corning”.

This defect is characterized by a blister on the underside of the component due to the internal expansion. Under x-ray inspection you would normally see solder shorts towards the center of the device. Figure 3 shows the effect of “pop-corning”

Fig 3



Solution to Problem

The solution to this problem is to slowly pre-bake the components and boards prior to reworking them. This will allow any moisture to escape slowly. Some companies already specify this as a pre-requisite before any rework is carried out.

Users should also ensure that only correctly stored devices are used for both assembly and rework (The components are generally supplied in a dry pack that should be re-sealed after opening).

Faults caused by non-simultaneous reflow

The reason that manufacturers strive to attain simultaneous reflow is that having a large T across a component can lead to rework and production defects. Some factors can be directly influenced by the profile settings while others are determined by the physical makeup of the PCB assembly. As a manufacturer of rework machines we can guarantee uniformity of the reflow head temperature to approximately 5C giving a total T of about 10C.

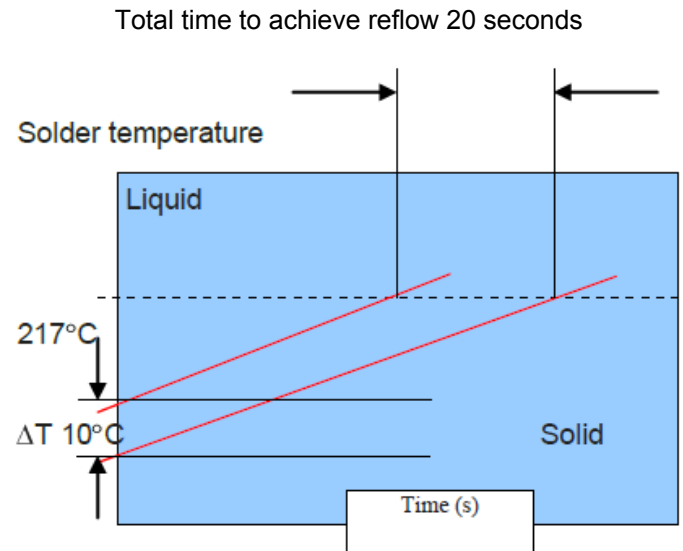
The first factor that will influence how “simultaneous” the reflow is will be profile temperatures at the point of reflow. More specifically, how quickly we pass from solid to liquid state of the solder.

Example – Reflow input temperature too low

The following diagram shows a graph section of a component’s solder joints going from solid to liquid at a melting temperature of 217C. We are assuming that the achieved T across the component is 10C.

The rate of temperature rise (ramp rate) will determine the duration of reflow from start to finish. This is directly proportional to the input temperatures set on the profile. In this case the ramp rate is 0.5C per second.

Graph of ramp rate at 0.5C/Sec



Potential problem

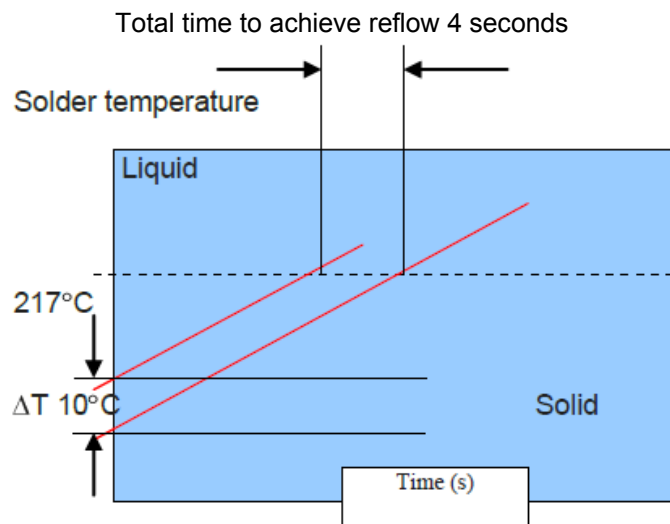
Solder reflows on one side first causing solder shorts



Solution to problem

The solution would involve changing the profile input-temperatures to achieve a higher ramp rate. This would in turn produce a more simultaneous reflow as illustrated in the graph below.

Graph of ramp rate at 2.5C/Sec



Factors that influence ΔT

There are other factors that will influence ΔT on any PCB assembly and these are often out of the control of

the rework machine manufacturer. Position of the component on the PCB is a major influence, if a component is placed at the edge of a board, the temperature of the outer 2 sides will generally be greater than the temperature of the inner sides. This is due to greater heat dissipation towards the center of the PCB. Another factor is the area to which some solder balls are connected. It is quite common on a BGA for a center array of solder balls to be connected to a ground plane specifically to dissipate heat from the component die. These balls may require extra time to reflow.

Conclusion

Profiling for rework is not always as straightforward as it may seem and it is important to deliver the correct amount of heat to create a successful profile. Knowing and understanding the potential defects will enable you to achieve consistently high yields.

