Guide to Wave Soldering Problems for PCB

What is Wave Soldering?

Wave soldering is a type of soldering process used during the bulk manufacture of printed circuit boards. The wave soldering process allows manufacturers to quickly and reliably solder large printed circuit boards. The process gets its name from the wave of solder that each board gets passed over. Using a wave of solder as opposed to individual solder points produces solder joints that are mechanically and electrically reliable.

The wave soldering process can be used for both the conventional through-hole method of PCB assembly and also the newer surface-mount method.

So what's in a wave solder machine, and how does the process work?

At its core, a standard wave solder machine starts with one component: a heated tank of solder that is has been brought to and maintained at the required temperature for the specific soldering process that is to take place. Inside the tank, the technician sets up a wave of solder, then passes each printed circuit board through the tank so that the top of the wave of solder just makes contact with the bottom of the printed circuit board.

There are two major issues that designers need to be aware of when it comes to designing printed circuit boards that are going to be soldered using the wave soldering method.

- Pad spacing: probably the bigger of the two issues, pad spacing refers to the amount of spacing that exists between the pads that need to be soldered — if they're too close together, liquid solder can flow between the two pads, which may cause not only the two connected pads to short-circuit, but possibly the entire PCB.

- Solder resist: having a layer of solder resist on the printed circuit board isn't as much of an issue anymore, as nowadays a solder resist layer is included in the blueprint as standard practice. Still, double-checking that a solder mask or solder resist layer is part of your PCB's blueprint is always a good thing to do, and helps prevent unfortunate accidents.

Once the board to be soldered has been checked for pad spacing and a layer of solder resist, flux can be applied. Flux helps ensure that the areas of the board that need to be soldered are kept clean and free of oxidation. Two different ways of applying flux are used, depending on what's needed:

- Spray flux, which can be applied as a fine mist and followed up with a jet of compressed air to remove any excess.

- Foam flux, which is applied to the board from a tank of regular flux. A plastic cylinder with tiny holes is submerged in the tank of flux. Once the cylinder has been well and truly submerged, a metal chimney can be fitted overtop of it. Air can then be forced through the cylinder to get foam to rise out through the chimney. The bottom of the PCB can then be coated in this foam flux.

Once the underside of the board has been treated with flux, it's time for the board to be preheated. Because of the way solder is applied as part of the wave soldering process, printed circuit boards that have been wave soldered are subjected to huge amounts of heat — far more than if they'd been manually soldered. Without preheating circuit boards can end up with all different types of soldering defects — all due to the thermal shock that printed circuit boards can be subjected to.

To help overcome this, boards that need to be wave soldered are heated up slowly to the required temperature — bringing them up to the temperature needed for the process in as steady a manner as possible to minimize any chances of thermal shock.

So what are some of the defects that can happen to wave-soldered boards if they're not preheated?

Types of Soldering Defects

Just because there's machinery involved in the wave-soldering process does not mean that this process is any less prone to errors than soldering each join by hand. Whether you're using a solder tank or a hand iron, you need to treat soldering like the precise science it is, carefully controlling where and what you solder. Otherwise, you'll up with any one of a number of soldering defects like the ones listed below.

Insufficient Hole Fill

Insufficient hole fill is problem that occurs on printed circuit boards that have holes drilled for components that are then mounted to the board through these same holes. Essentially, insufficient hole fill means that not enough solder has filled the holes drilled for the components, meaning that they will not "stick" properly to the board once the solder cools. Holes can be insufficiently filled for a number of reasons:

- The flux was applied incorrectly and did not penetrate through the board, meaning that the solder was not activated and could not bind the components properly.
- The temperature of the top side of the board isn't high enough to turn to solder molten so that it can rise through the holes.
- There was not enough of the board in the wave of the solder. If not enough of the board makes contact with the wave, too little solder is pushed up into the through-holes.

The best way to fix these issues is to run through another series of pre-solder checks. Check the type of flux you're using. Check how it's being applied — make sure that there's a sufficient volume of it and enough coverage so that the entirety of the PCB is covered. This will also help with the second problem: insufficient pre-heat.

Since solder flows to heat, the solder won't flow as well if your PCB's through holes have not come to the right temperature. While "the right temperature" depends on your company's operating standards, the general rule of thumb is between 300 and 340 degrees Fahrenheit (roughly 150 to 170 degrees Celsius). The whole assembly needs to reach this temperature *before* the board makes contact with the wave. If you're not sure whether your through-holes are reaching the correct temperature, take a profile of the wave solder process — it should look either exactly alike or very similar to your board's reflow processes.

While you're taking a profile of the wave solder process, you can also check your board height — that is, how much of the board is being exposed to the wave at any one time. Try to have at least ½ of the board thickness running in the solder wave at any one time. This will allow the hydrostatic pressure of the wave to push the solder up into the through hole.

Lifted Components

Lifted components (also known as tombstones) are components that have lifted off of the board during the soldering process. There are several common causes of lifted components, including:

- Incorrect lead lengths, which can cause components to lift from the board when they enter the solder bath.
- Attempting to perform wave soldering on a flexible PCB, which will bend while the rest of the components stay flat, causing them to lift off of the board.
- Using components that each have different requirements when it comes to temperature they need to be soldered at and what solder needs to be used.

To fix the incorrect lead lengths, look at what leads you're using. If your leads are too long, they can hit the solder bath and be pushed out of the through-hole. To fix this, you can increase the in-wave immersion time, which should lessen the thermal demand on the leads and allow them to settle.

In order to fix flexing of the PCB or other flex-related issues, double-check what kind of PCB you're using and what its thermal tolerances are. Board flexing is commonly seen on large connectors and on large IC packages or sockets. PCBs that flex from the get-go, such as plastic, should not be wave-soldered, as the wave of solder can cause the plastic to bend and lift the components away from the board.

Finally, once you've checked the thermal tolerances of your board, check the thermal tolerances of all your components. Components that have different temperature demands or lead solderability temperatures can also lift on contact with the wave as some components solder down and others are pushed away by the excess heat. Make sure that the components you're using have all of the same requirements, which should help prevent those sorts of issues.

Excessive Solder

Excessive solder is what happens when your board passes through a wave solder tank and takes on too much solder as part of that process. You end up with excess buildup. While this might be fine because it still ends up making an electrical connection between the board and the component in question, it will be hard for you or whoever else is looking at the board to tell exactly what's going on inside the solder itself.

There can be several reasons that you might have too much solder on your connections:

- Having the same types of components face different directions
- Using incorrect lead lengths when it comes to your design processes
- Simply running your conveyor belt too fast

The first cause is an easy enough fix. Make sure that all of your components of one single type — e.g. all of your batteries — face the same way when they go into the wave tank. So if one component is facing "away" from the tank when it goes in, all the other components of that type should be facing that way too.

Fixing lead protrusions (which are caused by incorrect lead lengths in the design process) can also help prevent excessive solder. If the lead is too long, then solder can pool against it and become excessive. To solve this, decide on a lead length that does not extend too far beyond your pad surfaces. For example, NASA uses a lead length of 2.29 mm — just long enough to stick past the pad so it can be soldered.

If your components are all facing the same way and your solder lengths are just as long as they should be, you might need to slow your conveyor belt down. A conveyor belt that goes too fast may end up dumping wave after wave of solder on your board's components as they pass through the solder tank. There's not really an easy fix for this one, unfortunately — have a discussion with your project manager about acceptable conveyor belt speeds.

Solder Balling

Solder balling can occur when small bits of solder re-attach themselves to the PCB — specifically near the leads — as it goes through the wave soldering process. Common causes of solder balling can include:

- The temperature of the solder in the tank being too high.
- Solder falling back into the wave as it separates from the board. This can cause that excess solder to splash back onto the board.
- Gasses released by heated flux can cause liquid solder to get spit back onto the board.

The best way to fix general solder balling problems is in the design of the PCB itself. When it comes time to pick out the solder mask you use in your PCB design, try to find one that has the least chance of allowing solder to adhere to it in the first place. By choosing the best solder mask for your design, you can help make your board

design more robust.

Solder balls that are caused by gasses from heated flux usually stem from an excessive back flow in the air around your wave solder tank or a drop in the nitrogen present in the environment. Check how much air is flowing into the solder tank, as well as the amount of nitrogen present in your solder tank's environment. Keeping a careful eye on both of these will help alleviate solder ball issues caused by bounce-back from heated flux.

When it comes to solder falling back into the wave, there's no one easy fix — it can be dependent on a lot of things: whether there's volatile material still in the flux, how high the solder wave is, etc. The best way to get a bead on what exactly is causing the problem is to run your solder wave with a white card over the top of the wave, but without any boards processing. Run the same test with a couple test boards going through the wave and compare the results.

Solder Flags

Solder flags are small protrusions of solder that poke out from the ends of your leads. Even though they will still enable you to form a proper electrical connection with the other components on the board, they indicate both improper flux application and issues with how the solder is draining from your board.

Common causes for solder flags include:

- Slow drainage of the solder from the wave solder machine, leaving an excessive amount of solder on the lead, which will "flag" out to one side as it drains.
- Trails of solder that look like whiskers — commonly identified as the result of improperly-applied flux.
- Improper storage of your solder leads, which can cause oxidation that your solder will not be able to readily attach to.

Slow drainage is usually caused by poor control between the separation of the solder wave and the board. In most cases, this is because the back flow of the solder wave is not set correctly — make sure yours is set for a "lambda" style wave. If your solder wave's back flow is set properly, the solder will flow at the same speed and in the same direction as the as the board when it separates from the wave. If you need to, you can run it slightly faster. However, running it slowly (or not running it at all) will actually increase solder flags/spikes.

If the issue is oxidation, this can be because of how your leads have been cut, stored, or cut and stored. If your leads are not cut in-house, check with your supplier to see how they cut their leads. Bare-cut ends can cause your leads to oxidize, especially if they're stored for a long period of time. If your leads are being cut in-house, check how they're being cut, how they're being stored and also for how long they're being stored. All three of these factors leave the lead much more open to oxidation, making it much harder to wet with solder when it comes time to run it through the wave.