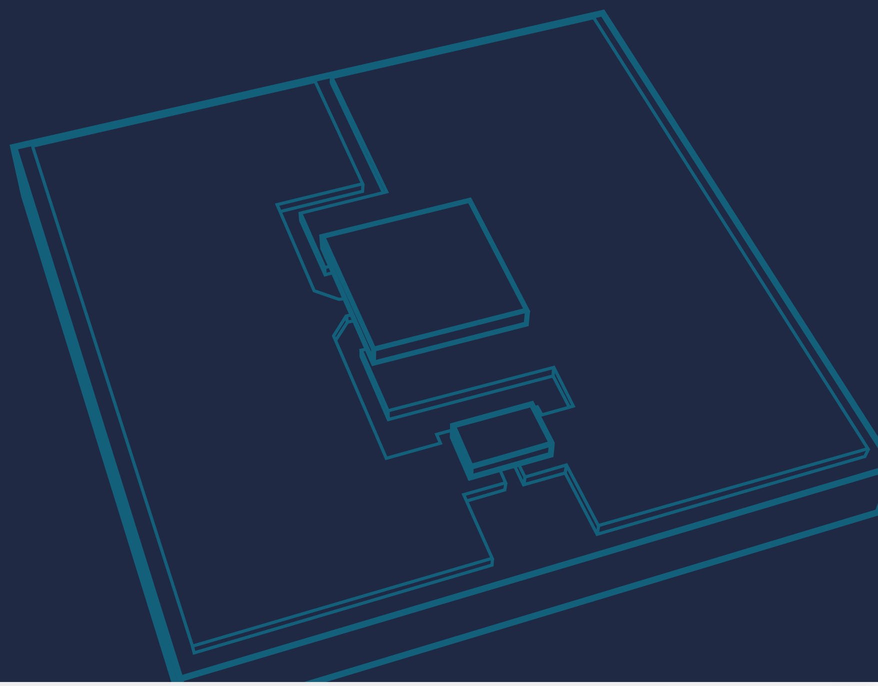
Crystal IS
an Asahi Kasei company



AN015

Proper Soldering Techniques for Crystal IS Klaran LEDs

This application note provides specific guidelines for the soldering of Crystal IS Klaran LEDs. Included is an overview of acceptable process and materials for soldering Klaran LEDs, and information on other processes and common issues that can cause damage.

Soldering Process and Materials

Soldering is the process in which metals are bonded together by using an additional metal that has a lower melting temperature. This differs from welding in that the two metals being bonded do not melt as part of the process.

The soldering process consists of several factors:

1. The part being attached (most importantly, its specified solder temperature profile)
2. The solder material (with consideration of melting temperature, mesh type, form of application, and flux type)
3. The heating equipment
4. Cleaning the assembled PCB

There are a variety of metal alloy combinations and forms for the solders that are used for attaching electrical components. For the purpose of attaching surface mount Klaran devices, we suggest using solder paste.

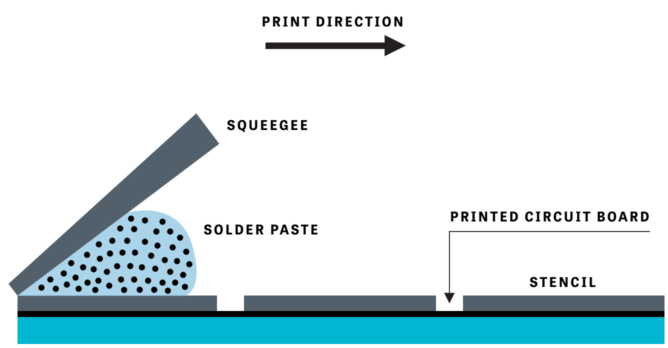
One of the main considerations when selecting solder paste is the solder paste's melting temperature. The recommended soldering guidelines can be found on the datasheet for the particular UVC LED. These guidelines will show a temperature range for which the solder should be liquidous. This is the highest temperature range that the melting point of your selected solder should fall within.

One of the most common and cost-effective solder pastes that fall within this range is SAC alloy (Tin (Sn), Silver (Ag), Copper (Cu)). SAC alloy has a melting point of 217°C to 218°C which falls into the upper limits of the profile requirements of Klaran LEDs.

Additional variables to consider when selecting solder paste include: mesh size, packaging, and flux type. A paste is made of small round particles of metal alloy mixed with flux; the size of these are referred to as the particle or mesh type and is graded in a range of 1-8 with 3-6 being the most common. The higher the grade type, the smaller the particles in the solder paste. For hand application, the mesh type is not as important; it is necessary to consider if using a screen type of application.

Applying Solder Paste

One of the most common soldering application methods for medium to large scale production is a screening process. In this process, a stencil is placed over the PCB board, solder paste is applied, and a squeegee spreads the paste across the top. When the stencil is removed, a correctly set amount of solder paste is left on the board.



Solder can also be applied with a syringe. A syringe may be used to apply the paste used in a screen-printing application, directly to the PCB by an automated machine, or directly to the PCB by hand. Syringes often come with different size nozzles to allow the amount dispersed to be more closely controlled based on the size of the pad.

Another important component of the soldering process is flux. The main purpose of flux is to remove oxidization from the surfaces to which the solder is being bonded and the solder itself. While mostly inert at room temperature, as flux is heated, an acidic reaction facilitates the cleaning of the surfaces. In a solder paste, flux prevents additional oxidization to the solder simply by keeping it separated from air. By weight, a solder paste is approximately a 50/50 ratio of flux and solder particles, but this is due to the higher density of the flux. By volume, the solder particles make up nearly 90% of the mixture. Flux also helps the solder to smooth out while in a liquidous phase, and this is often referred to as wetting.

Another important component of the soldering process is flux. The main purpose of flux is to remove oxidization from the surfaces to which the solder is being bonded and the solder itself. While mostly inert at room temperature, as flux is heated, an acidic reaction facilitates the cleaning of the surfaces. In a solder paste, flux prevents additional oxidization to the solder simply by keeping it separated from air. By weight, a solder paste is approximately a 50/50 ratio of flux and solder particles, but this is due to the higher density of the flux. By volume, the solder particles make up nearly 90% of the mixture. Flux also helps the solder to smooth out while in a liquidous phase, and this is often referred to as wetting.

There are three main types of flux found in solder pastes: (1) Water-soluble, (2) Rosin based, and (3) No-clean. Each type of flux has different ranges in the levels of acidic activity available. Water-soluble is the most variable and has an available range that covers all levels of activity. Rosin based fluxes have a low activity range and is typically used for easy to solder surfaces. No-clean fluxes have a low to medium range of activity. Any type of chosen flux is acceptable for the bonding of Klaran devices if proper amounts of solder, the correct solder temperature range, and cleaning process is applied.

Heating Equipment

The type of equipment used to provide heat for activating the flux and melting the solder (i.e. the reflow process) is very important. The equipment needs to reach the temperatures required for the chosen solder, and it must do so in a controlled manner because each level of heat provides a necessary step in the process.

A reflow oven is suggested for the attachment of Klaran LEDs. There are two main types of reflow ovens: batch and multi-stage. A batch reflow oven controls the temperature profile by using fast heating elements in a small space and can cycle the air out of the chamber for cooling. As the name implies, batch reflow ovens are used for small batch productions. A multi-stage reflow oven has separate zones set to the temperatures required to match the specified profile, and usually the last zone is used for cooling. A speed adjustable conveyor belt moves PCB boards through the oven allowing larger volume production. Regardless of the type of oven used, it is important to know that the temperature profile for reflowing electrical components is based on the temperature of the device.

Part of setting up a profile correctly is to use a temperature sensor attached to the PCB board in order to test and adjust the profile.

Cleaning the Assembled PCB

Cleaning the assembled PCB is a necessary step and the right approach to cleaning is highly dependent on the type of flux found in your solder paste. Do not use petroleum-based aerosol cleaning agents, ultrasonic cleaning, or perform mechanical cleaning with brushes or swabs on your mounted Klaran LEDs. For water-based flux, use water; for rosin and no-clean fluxes, it is best to use isopropyl alcohol (IPA) but water can also be used. Rinsing with a lab wash bottle provides gentle force to eliminate excess residues left after reflowing of the PCB board.

After cleaning, it is recommended that you perform a bake out process. The purpose of performing a bake out process is to eliminate moisture as well as any outgassing of organic compounds that might be left over from the PCB manufacturing or assembly process. Formal bake out ovens typically operate under vacuum to ensure that moisture or compound vapors do not settle back onto the parts. For the purpose of removing the moisture introduced during the cleaning process, it is suggested to oven bake at 60°C (140°F) for 2 hours for Isopropyl Alcohol

(IPA), 4 hours for water cleaning. The oven should not have a ramp rate higher than 2°C (3.6°F) per minute. Klaran UVC LEDs fall under a moisture sensitivity level (MSL) rating of 1 which means they have an unlimited shelf life if they are kept under 30°C and 85% relative humidity and do not require baking process.

Other Processes and Common Issues

Most common issues that have been seen while attempting to solder Klaran LED devices are related to heat, solder application, and cleaning.

The equipment used to provide heat for the soldering process is important. Please note that hot plates, soldering irons, and air soldering techniques do not provide accurate control of the heat being applied to the solder and contacts. Since most Klaran LEDs should be mounted on metal core PCBs in order to provide improved thermal conductivity, the heat is quickly drawn away from the contact which adds more complications to these heating methods. As a result, overheating could cause damage to the LED package or the chip itself. Uneven heating, or underheating, can cause an irregularly mounted device resulting in poor optics and partial bonding of the contacts. Not only should the allowable soldering temperatures of the Klaran device being attached be considered, but the required temperatures of any other components that may be part of the board assembly should be considered as well. This ensures that, as parts are added to the board, the already soldered parts are not being re-melted multiple times. Multiple flowing of solder joints can cause shifting of the device and poor intermetallic in the solder joint, making it brittle.

Solder paste is suggested because the amount being applied is controllable and it has the proper amount of flux as a premixed component. When applying solder paste by hand, a common mistake is using too much solder paste. For the size pads that the Klaran LED uses the thickness of a screening stencil, and therefore the solder paste would be ~6 mil (~150 µm), or about the thickness of a human hair. Using a syringe to apply a small parallel line of solder paste across each pad provides plenty of product for a good bond. Any other solder alloy that remains within the profile provided by the Klaran LED specification is acceptable.

It is a best practice to clean your assembled PCB, even if using a no-clean flux. Flux residues can be acidic which may damage the board or components, so it is understandable that to clean water-based and rosin, flux is expected. No-clean fluxes leave lower amounts of residue than other types of fluxes. The residue from no-clean fluxes can cause leakage currents across the LED contacts, especially when too much solder paste has been applied.

The process equipment and materials discussed in previous sections of this application note are suggested for the proper attachment of Klaran LEDs to your PCB board. If you are planning to use alternative processes or materials, we welcome you to contact Crystal IS and discuss this with our Application Engineering team.

Disclaimer

The information in this document has been compiled from reference materials and other sources believed to be reliable and given in good faith. No warranty, either expressed or implied, is made, however, to the accuracy and completeness of the information, nor is any responsibility assumed or implied for any loss or damage resulting from inaccuracies or omissions. Each user bears full responsibility for making their own determination as to the suitability of Crystal IS products, recommendations, or advice for its own particular use. Crystal IS makes no warranty or guarantee, express or implied, as to results obtained in end-use, nor of any design incorporating its Products, recommendation or advice.

Each user must identify and perform all tests and analyses necessary to ensure that it's finished application incorporating Crystal IS' products will be safe and suitable for use under end-use conditions. Each user of devices assumes full responsibility to become educated in and to protect from harmful irradiation. Crystal IS specifically disclaims any and all liability for harm arising from buyer's use or misuse of UVC devices either in development or end-use.

WE INVITE YOU TO LEARN MORE ABOUT OUR UVC LEDs.

Crystal IS an Asahi Kasei company

70 Cohoes Avenue, Green Island, NY 12183 U.S.A.
518.271.7375 | www.cisuvc.com | sales@cisuvc.com