Power Semiconductor Soldering: Techniques for Precision and Durability

Objectives

This document deals with the overall process of soldering power electronic components onto printed circuit boards (PCBs). Soldering is a multidisciplinary science and includes metallurgy, process optimization, machine building, and thermal management. Industry has widely adopted reflow-soldering for surface mount technology (SMT) and wave-soldering for through-hole components (THT) for mass production, while in lab-scale and maintenance, selective soldering remains an often-used technique.

Each of these processes, sketched in **Figure 1**, is described along with suggestions to achieve high-quality solder joints. A special focus is placed on solder alloy materials as the demand to get to a lead-free process brings along several challenges to consider.

Applications

This application note is relevant for any power electronics application involving power semiconductor devices on PCB.

Target Audience

This document is intended for engineers involved with and interested in the process of soldering semiconductor devices to PCB.

Contact Information

For more information on this topic, contact the Littelfuse Power Semiconductor team of product and applications experts at PowerSemiSupport@Littelfuse.com

Introduction

In electronics manufacturing, soldering is understood as a process used to join two or more metal components together by melting a filler metal, known as solder, into the joint. The solder typically is an alloy of different metals that melts at a lower temperature than the workpieces. This enables the solder to flow into the joint and create a strong, electrically, and thermally conductive bond as it cools and solidifies. Depending on the temperatures and filler-materials in use, a classification of soldering is done into one of the following processes:

- **Brazing**, a high-temperature process using brass-fillers to form very strong bonds in mechanical applications
- **Hard soldering**, with temperatures exceeding 450 °C, mostly using a blowtorch to melt the solder and widely found in metalwork
- **Soft soldering**, taking place at < 450 °C, which is the process used in joining components on PCBs

The solder alloys used in the electronic industry are rapidly transitioning from tin/lead (Sn/Pb) to lead-free (Pb-free) solders to meet environmental regulations and the push for greener electronics manufacturing. Laws and regulations, which vary by country and application, control many of these requirements like REACH and RoHS. Over the past decades, there has also been a significant shift from through-hole technology (THT) to surface mount technology (SMT). Although these transitions occurred for different reasons, their overlap has revealed challenges, particularly in integrating Pb-free soldering within SMT assemblies. This application note also addresses the optimization of Pb-free soldering conditions for Littelfuse surface mount devices (SMD). The information summarized in this document aligns with the international standards IPC-JEDEC J-STD-020, JEDEC J-STD-033D, and IEC 61760-1.

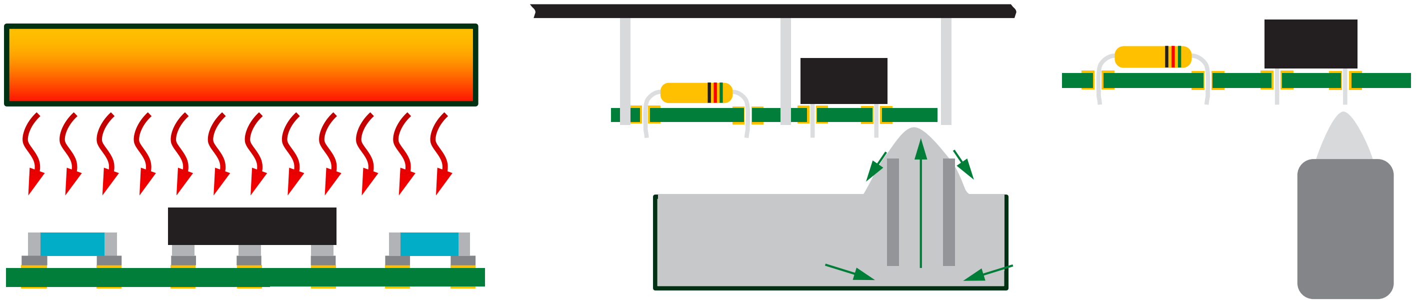

Figure 1. Solder processes covered in this application note

Pb-free Solders and Resulting Challenges

Pb-free solders are usually tin/silver (Sn/Ag), or variations of Sn/Ag solder like Sn99-Cu0,7-Ag0,3. The primary difference affecting the soldering process is the higher melting point of Sn/Ag solder compared to Sn/Pb solder. In terms of quantification, the nominal eutectic melting points are 220 °C for Sn/Ag solder and 183 °C for Sn/Pb (SN63-PB) solder. The off-eutectic Pb-free solders have even higher melting points. The combination 95 Sn/5 Ag solder melts at 240 °C, and there are tin/silver/copper (Sn/Ag/Cu) and other alloys that have melting points that range from 216 °C to 226 °C.

Even when eutectic Sn/Ag solders are used, dissolved plating materials like silver (Ag), tin (Sn), and gold (Au) can often increase the melting point to reach temperatures as high as 240 °C. Furthermore, solder can exist in multiple phases and one of these does not melt below 240 °C. As a result of these variations, the worst-case melting point of all the Sn/Ag Pb-free solders is considered to be 240 °C. This pushes the effective melting point of the Pb-free solders 57 °C higher than the Sn/Pb solders.

Challenges that arise from using lead-free solder alloys include:

- **Higher melting temperatures** which require higher processing temperatures, which in turn leads to increased thermal stress for components and substrates.
- **Reduced wetting ability** of lead-free solders, meaning they do not flow as easily or spread as well on the surfaces being joined. This can lead to weaker joints and increased defects.
- **Formation of Intermetallic Compounds (IMCs)** is more often seen in lead-free solders and leads to forming of brittle intermetallic compounds at the solder joint interface. These IMCs can jeopardize the mechanical strength and reliability of the solder joints.
- **Increased oxidation** is more prominent in lead-free solders during the soldering process, which can affect the quality of the solder joints. This often necessitates the use of more aggressive fluxes or inert gas atmospheres.
- **Thermal fatigue and mechanical reliability** can suffer as lead-free solder joints can be less reliable under thermal cycling and mechanical stress. The higher stiffness and lower ductility of lead-free solders can lead to increased susceptibility to cracking and failure under thermal stress.
- **Compatibility issues** may arise as some lead-free solder alloys may not be compatible with certain components or PCB finishes, leading to potential reliability issues. For example, tin-silver-copper alloys can form brittle intermetallic compounds with certain surface finishes like nickel-gold.

These challenges require careful consideration and adaptation of soldering processes as well as choice of materials to ensure reliable and high-quality solder joints.

Component Considerations

Reflow and wave soldering are suitable for a wide variety of components, both SMT and THT. The key considerations regarding the components to be soldered are:

- **Thermal sensitivity**: Some components, especially certain types of ICs and electrolytic capacitors, may be sensitive to high temperatures. It is crucial to follow the manufacturers' recommended profiles to avoid damage.
- **Moisture sensitivity**: Non-hermetic packages, usually made of plastic mold compounds, may be sensitive to moisture. Over time, ambient air moisture can seep into the mold compound of the package. High levels of moisture inside the material can cause damage during soldering due to rapid heating and evaporation. Hence, components with higher Moisture Sensitivity Levels (MSL) need to be handled and stored appropriately to prevent moisture-induced damage during the solder process. For details, also refer to the JEDEC/J-STD-033 standard, which provides detailed guidelines for handling and storage of MSL–rated components, as summarized in **Table 1**. The MSL of a device can be found in the correlating material datasheet (MDS).

Table 1. Handling and storage guidelines for Moisture Sensitivity Levels (MSL) according to JEDEC standards

MSL Level	Floor Life (Maximum Exposure Time)	Storage Conditions before Use	Baking Requirements
MSL1	Unlimited at ≤ 30 °C / 85% RH	Store in a controlled environment (dry storage)	Not Required
MSL2	1 year at ≤ 30 °C / 60% RH	Store in a sealed moisture barrier bag (MBB)[1] with desiccant[2] and humidity indicator card (HIC)[3]	Bake if floor life[4] exceeded (125 °C for 24 hours or 40 °C for 192 hours)
MSL2a	4 weeks at ≤ 30 °C / 60% RH	Store in MBB with desiccant and HIC	Bake if floor life exceeded (125 °C for 24 hours or 40 °C for 192 hours)
MSL3	168 hours (7 days) at ≤ 30 °C / 60% RH	Store in MBB with desiccant and HIC	Bake if floor life exceeded (125 °C for 24 hours or 40 °C for 192 hours)
MSL4	72 hours (3 days) at ≤ 30 °C / 60% RH	Store in MBB with desiccant and HIC	Bake if floor life exceeded (125 °C for 24 hours or 40 °C for 192 hours)
MSL5	48 hours (2 days) at ≤ 30 °C / 60% RH	Store in MBB with desiccant and HIC	Bake if floor life exceeded (125 °C for 24 hours or 40 °C for 192 hours)
MSL5a	24 hours (1 day) at ≤ 30 °C / 60% RH	Store in MBB with desiccant and HIC	Bake if floor life exceeded (125 °C for 24 hours or 40 °C for 192 hours)
MSL6	6 hours at ≤ 30 °C / 60% RH	Store in MBB with desiccant and HIC	Mandatory bake before use (125 °C for 24 hours or 40 °C for 192 hours)

Notes:
(1)**Moisture Barrier Bag (MBB)**: A sealed bag that prevents moisture ingress, typically used with desiccants to maintain low humidity levels inside the bag
(2)**Desiccant**: Material placed inside the MBB to absorb moisture and maintain low humidity
(3)**Humidity Indicator Card (HIC)**: A card placed inside the MBB that changes color to indicate the level of humidity inside the bag
(4)**Floor Life**: The maximum time components can be exposed to the ambient environment, ≤ 30 °C / 60% RH, without requiring baking

- **Package type**: Surface mount components come in various package types such as SOIC, QFP, or D2PAK, each designed for compatibility with reflow soldering. Ensure the package type is suitable for the specific reflow process used.
- **Component placement**: Proper alignment and placement of components are essential for successful reflow soldering. Automated pick-and-place machines are typically used for precise component placement.

Reflow Soldering

Reflow soldering is a process used to attach SMT components to a PCB by melting solder paste that has been pre-applied to the pads and component leads. The process involves heating the assembly in a reflow oven, following a controlled temperature profile to achieve a reliable electrical and mechanical connection. The steps involved in reflow soldering are:

1. **Solder paste application**: The process begins with the application of solder paste onto the PCB. Solder paste is a mixture of powdered solder, usually lead-free alloys, and a flux that aids in the soldering process. The paste is typically applied using a stencil to ensure precise placement on the pads where components will be mounted.
2. **Component placement**: SMT components are then placed onto the PCB, aligning their leads or terminals with the solder paste deposits. This is usually done using automated pick-and-place machines, which ensure accurate and efficient positioning of the components.
3. **Reflow soldering**: Reflow heating involves subjecting the PCB, now populated with components and solder paste, to a controlled heating process in a reflow oven. The heating cycle typically includes several stages:
   i. **Preheat**, where the temperature is gradually raised to avoid thermal shock and activate the flux in the solder paste to remove oxides from the components' leads and PCB pads.
   ii. **Soak**, where the temperature is maintained at a certain level to ensure uniform heating and further activate the flux.
   iii. **Reflow**, where the temperature is increased to a peak level to melt the solder and form metallurgical bonds between the components' leads and the PCB pads; and
   iv. **Cooling**, where the assembly is gradually cooled to allow the solder to solidify and create strong, reliable joints. The temperature is regulated over time to follow the reflow soldering profile.

The process is briefly pictured in **Figure 2**.

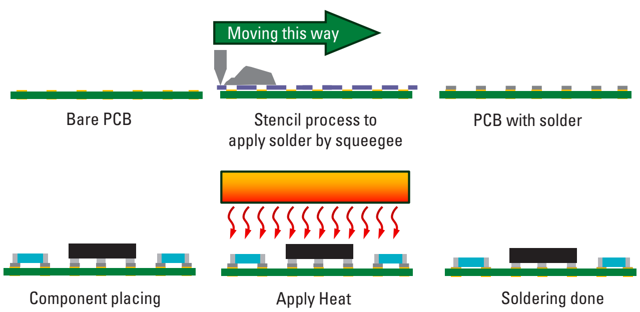

Figure 2. Reflow soldering overview

After reflow soldering, visual inspection of the PCB is advised to verify solder joint quality and detect defects such as solder bridges, voids, or insufficient amount of solder. Automated optical inspection and X-ray inspection are commonly used. Functional testing may also be performed to ensure the assembled PCB works as intended.

Reflow Soldering Temperature Profile for Pb-free Solders

The higher melting point of the typical Pb-free (Sn/Ag) solders require soldering temperatures to safely reach 240 °C. The recommended soldering profile must assure that 240 °C is reached for a minimum time, considering the temperature tolerances and varying component size and mass.

The PCB industry has extensive experience with Sn/Pb solders, which typically have peak reflow temperatures of 215–245 °C while exceeding 183 °C for 20–60 seconds. If 57 K are added to these temperatures for ideal reflow and wetting in case of Pb-free solders, the temperatures would be so high that they could damage components, burn, or harden fluxes, and oxidize some solderable surfaces. The typical reflow temperature range for Pb-free (Sn/Ag) solder is 240–250 °C, with 40–80 seconds over 220 °C. Although the duration above the melting point is longer for Pb-free solders, the higher temperatures and the narrower process window for safe operation make the reflow process more critical and challenging to manage.

Due to these facts, the recommended Sn/Pb reflow temperature range is less critical, and therefore, minor deviations in the temperature of equipment or components generally do not create soldering problems. In contrast, the Pb-free solder reflow temperatures are considerably higher, which requires precise reflow temperature limiting and monitoring to avoid failures.

Although most SMD-style semiconductor components are suitable for Pb-free solders, the maximum soldering temperatures or suitable assembly processes should be confirmed. The solder profile can also be influenced by high component density, heavy copper foils, and other unique conditions. The actual temperature of small or large components should be considered when defining the reflow profile. It is also important to check the maximum temperature ratings of components when using Pb-free soldering, especially for SMT components, where the entire component generally reaches the peak reflow temperature.

It is not possible for a power semiconductor device manufacturer to provide a general reflow profile recommendation for a customer in charge of board assembly. Reflow furnace settings depend on the number of heating and cooling zones, type of solder paste/flux used, board and component size as well as component density. The actual temperature setting needs to be above the liquidus temperature of the solder paste in order to form reliable solder joints. A standard surface-mount reflow soldering process according to J-STD-020 [1] is recommended. **Figure 3** depicts a typical Pb-free (Sn/Ag) reflow profile according to JEDEC-standard J-STD-020.

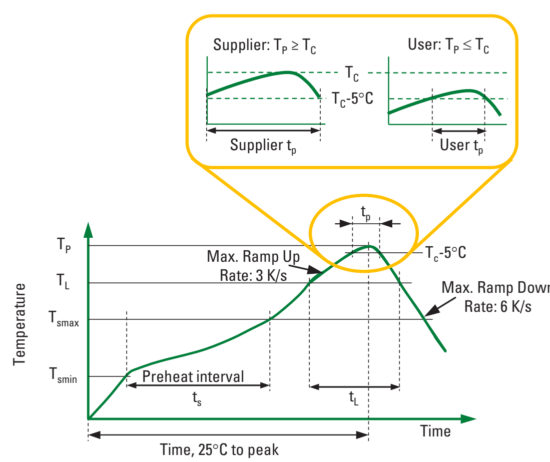

Figure 3. Typical reflow soldering profile for lead-free solder based on J-STD-020

The reflow temperature during assembly should be such that the peak package body temperature, T_P does not exceed the JEDEC standard's classification temperature, T_c of the Pb-free process given in **Table 2**. The reference reflow profiles for soldering application are given by the solder paste manufacturer. They usually provide a suitable starting point for further optimization. The general requirements for electronic-grade solder alloys and fluxes can be found in the IPC-7530 or IEC/TR IEC 60068-3-12.

The maximum reflow processing boundary of an SMD, the critical package body peak temperature, T_c, is determined by classification profiles that must not be confused with the actual reflow soldering profile that has to be applied in the board assembly.

The component qualification procedures conducted at Littelfuse are in accordance with the J-STD-020 standard. They follow the worst-case reflow profiles a given device and its package must withstand, according to the standard's classification tables and temperature profiles mentioned in **Table 2**. These should not be construed as process reflow profile specifications for any specific application.

The classification temperature, T_c, of the lead-free solder process depends on the package thickness and volume of the SMD package as listed in **Table 2** [1].

Table 2. Classification temperature (T_c) for Pb-free solder process

Package Thickness	Volume [mm3] < 350	Volume [mm3] 350 – 2000	Volume [mm3] > 2000
< 1.6 mm	260 °C	260 °C	260 °C
1.6 mm – 2.5 mm	260 °C	250 °C	245 °C
>2.5 mm	250 °C	245 °C	245 °C

Different temperatures and time durations for the Pb-free solder process, according to **Figure 3**, are explained, along with their values [1], in **Table 3**.

Table 3. Parameters for Pb-free solder process

Profile Feature	Pb-free Assembly
Preheat/Soak Zone Temperature Min (T_{smin}) Temperature Max (T_{smax}) Time (t_s) from (T_{smin} to T_{smax})	 150 °C 200 °C 60–120 seconds
Ramp-up rate (T_L to T_P)	3 K/second max.
Liquidous temperature (T_L) Time (t_L) maintained above T_L	217 °C 60–150 seconds
Peak package body temperature (T_P)[1]	T_p must not exceed the classification temperature T_c in Table 2
Time (t_p)[2] within 5 °C of the specified classification temperature (T_c)	30 seconds
Ramp-down rate (T_P to T_L)	6 K/second max.
Time 25 °C to peak temperature	8 minutes max.

Notes:
(1) The tolerance for the peak profile temperature (T_p) is defined as a supplier minimum and a user maximum.
(2) The tolerance for the time at peak profile temperature (T_p) is defined as a supplier minimum and user maximum.

The preheat time and temperature are determined primarily by the flux system. The flux system must clean and remove oxidation from the surfaces to be soldered, which includes PCB and component leads. Solder pastes often include both flux and volatile organic filler materials. Preheating removes excess volatile materials while the fluxes are doing their work. Typical preheat conditions are 60 to 120 seconds between 150 °C and 200 °C, but since the preheat cycle depends on the flux system, it is best to adhere to the preheat cycle recommended by the flux or solder paste manufacturer. The maximum heating rate is typically limited to 3.0 K/second, but most reflow systems heat considerably slower because of the mass and preheat requirements.

The maximum cooling rate is typically 6.0 K/second and is limited to prevent damage to solder joints or components caused by mechanical stresses generated by differences in the coefficient of thermal expansion (CTE) and mismatch between the components and the PCB. In general, it is advantageous to cool down near the maximum rate. The maximum cooling rate keeps the solder from crystallizing, which improves the mechanical properties of the solder. The solder finish should be shiny after cooling, a dull finish being evidence of crystallization. Cooling rates can be more critical with Pb-free solder reflow because the temperature difference between solidification and room temperature is much larger than with Sn/Pb solders. The temperature difference for Pb-free solders is (240–25) °C = 215 °C and for Sn/Pb solder is (183–25) °C = 158 °C. A gradual cooling rate change allows these soft solders to creep and release the mechanical stresses which are proportional to the product of the thermal expansion coefficient, temperature differential, and total component length. Therefore, larger components and components with CTE values significantly different compared to the PCB's CTE are more susceptible to damage due to cooling.

Important note:
The TO-263 and TO-268 surface mount packages from Littelfuse require special considerations due to their larger size, to ensure optimal solder bonding between the base and the substrate, which supports high power dissipation.

- **Soldering technique**: The recommended soldering techniques are either reflow soldering with convection heating or vapor phase soldering, both of which require a preheating step. For convection heating, Littelfuse recommends a suitable cover gas flow with gas temperature as close as possible to the maximum allowed package temperature. The advantage is a very homogeneous heating of the whole substrate or board and less stress for all devices. The gas should preferably be forming gas, but pure nitrogen is also possible. This enables the use of solder paste with a small amount of flux, which remains a requirement for good solder joints to power devices. Infrared heating is less suitable due to the size and thickness of the packages and the thickness of the copper bases, which absorb very little radiation.
- **Soldering profile**: The soldering profile should consider:
  - Heating and cooling ramps to not exceed 2 K/s and 1 K/s respectively
  - Pre-heating at a maximum temperature of 160 °C for a time of 60 s is necessary.
  - The allowable peak surface temperature of the device is 250 °C for 10 s maximum.

This is only a guideline. The appropriate temperature profile has to be adjusted experimentally for each product.

Wave Soldering

Wave soldering stands as a prominent solution, especially in high-volume production scenarios, facilitating the reliable assembly of through-hole (THT) electronic circuitry. This section provides an overview of the wave soldering processes, along with recommendations for successful soldering and maintaining consistent quality.

Wave soldering is an industrial soldering process primarily employed for the through-hole assembly of electronic circuit boards. In this process, the components on the circuit board pass a stationary wave of liquid solder alloy, creating solder joints to connect the components securely to the board. Wave soldering is also an option to mount SMD components or mixed assemblies. Here, SMD components are typically glued to the PCB to keep them in place during soldering.

The procedure unfolds in several steps:

1. The electronic assembly is prepared before soldering. Through-hole components are placed on the circuit board, and they are securely held in place by fixtures to ensure stability during the soldering process
2. Next, the assembly is treated with flux. Flux serves to remove oxide layers from metal surfaces and facilitates wetting, ensuring high-quality solder connections.
3. The assembly then passes through a preheating zone, raising its temperature to minimize thermal shock and facilitate wetting during the wave soldering process.
4. Afterwards, the design is conveyed over the solder wave, which contains molten solder – typically, a tin-lead alloy or a lead-free alternative. The solder wave creates a uniform solder layer on the metal component leads by flooding them with liquid solder. The assembly is positioned to only expose the bottom side of the circuit board to the solder wave. This way, through-hole components are reliably connected as the solder is applied to their leads and forms reliable solder joints by filling the PCB's correlating holes.

After wave soldering, the assembly goes through a further thermal zone to ensure proper cooling and solidification of the solder joints, minimizing stresses in the connections. The assembly then passes through a cooling zone to lower the overall temperature and ensure stable solder joints. The process is schematically pictured in **Figure 4**.

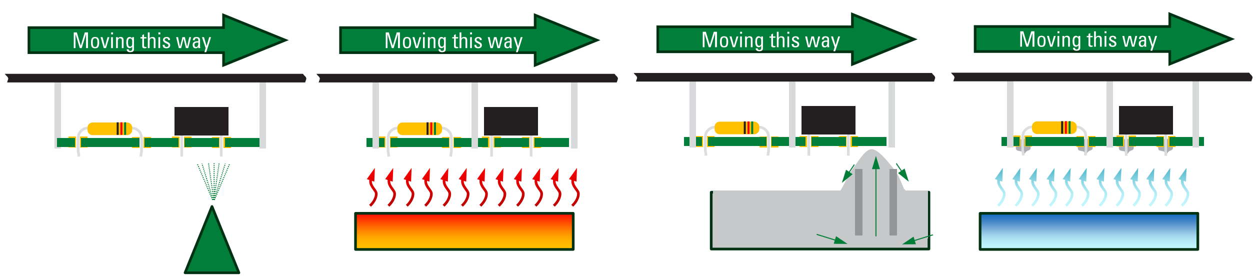

Figure 4. Typical steps in a wave-soldering process

Finally, visual inspection is recommended to verify that the solder joints meet the quality requirements. In some cases, cleaning may be necessary to remove excess flux, which can be done through methods such as spraying, dipping, or other cleaning techniques.

Wave Soldering Temperature Profile for Pb-free Solders

As with the reflow soldering process and for the same reasons, a temperature profile for wave soldering needs to consider ramp-up and ramp-down temperature gradients, maximum temperatures, and intervals for the components to get in direct contact with the heat source. IEC 61760-1 [3] gives an overview on how such a temperature profile can be designed. The correlating diagram is sketched in **Figure 5**.

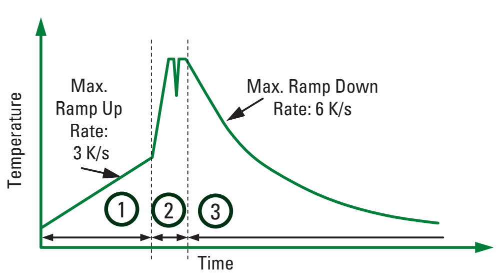

Figure 5. Generic wave soldering temperature profile

The process is separated into three intervals that consist of:

1. A pre-heating time with a limited temperature gradient to get the component from ambient temperature close to the solder temperature without causing too high thermal stress
2. A time where the assembly gets in touch with the liquified solder. This interval is typically considered to not exceed 10 seconds with a solder-bath temperature of 260 °C.
3. The phase of cooling down, again with a limited temperature gradient to keep thermal stress within tolerable limits.

The occurrence of two spikes in the diagram is a consequence of the wave soldering technology using two sequential waves. This is commonly used as it enhances the solder joint quality, particularly if the waves move in opposite directions.

As in reflow soldering, several parameters need to be considered, which a power semiconductor manufacturer has no control over. Therefore, developing and verifying an individual temperature profile remains the PCB manufacturer's responsibility.

Selective Soldering

Selective soldering is a precise method used to solder specific components on a printed circuit board without affecting nearby components. This technique is particularly useful for complex PCBs with a high density of components or those that include both surface-mount and through-hole components.

In the case of automated selective soldering, the process represents a miniature version of wave soldering. In contrast, instead of moving the PCB to the wave, a nozzle that generates a solder wave in the shape of a small, pointed dome is brought to the solder-position, as depicted in **Figure 6**. The PCB assembly remains stationary while the nozzle is moved accordingly.

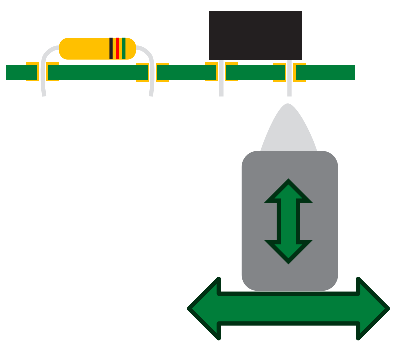

Figure 6. Precise miniature wave-generating nozzle in selective soldering

Due to the identic requirements, this process needs to be regarded as wave soldering with all the correlating features and precautions that need to be considered.

Design Hints to Support Successful Soldering

Soldering, in essence, is a thermal process that relies on accurately delivering liquified metal to the dedicated position and ensuring proper coverage while preventing damage or unintended deposition of material.

Common failures in soldering include:

- **Cold joints** that can cause weak electrical and mechanical connection, often caused by insufficient heat, contaminated surfaces, or moving the joint before the solder solidifies.
- **Solder bridging** is an unintended connection between adjacent pads which can cause malfunction or short circuits.
- **Insufficient solder amount** to form a reliable joint can cause poor electrical and mechanical connections. This might be a consequence of too high a speed in wave soldering or contaminated stencils used to apply solder paste in reflow processes.
- **Lifted pads** that detached from the carrier-PCB, potentially because of excessive heat, too steep temperature gradients, mechanical stress during soldering, or process failures during PCB manufacturing.
- **Tombstoning** happens when an SMD component lifts off the pad on one side but gets soldered on the other side, resembling a tilt tombstone in appearance. This can be caused by uneven heat distribution or mechanical stress during soldering.
- **Solder voids** occur due to poor wetting, insufficient flux, or improper soldering techniques.
- **Overheated joints** are unreliable connections caused by too high temperatures that may cause fluxes to not operate as they are supposed to.
- **Solder splashes** consist of small droplets of solder, scattered on the PCB and potentially causing malfunction. Solder contamination can be one root-cause.

While some of these failures can be prevented by proper processes and process control, PCB layouts can also contribute to enhancing solder quality. Designers should take care to adhere to given design rules for PCB layout.

Especially in power electronics where components carry higher currents and thus both width and thickness of copper traces grow accordingly, best practices in PCB design must consider the impact of these larger masses on thermal development and temperature distributions.

Best practices in design focus on the component's pads and the connection to surrounding copper traces. **Figure 7** is a representation of two pads connected to a larger copper area.

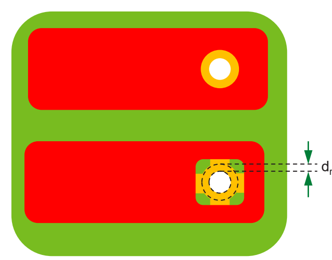

Figure 7. Connecting pads to copper area

While electrically both pads remain part of the same trace, the lower design uses thermal traps to connect the via to the surrounding polygon. During soldering, this prevents larger amount of thermal energy from being dissipated to the copper that effectively works as a heat sink. Thus, the heat remains with the pad and allows the solder to form a reliable joint.

As the thermal traps introduce gaps between the via and the surrounding area, care must be taken to remain with a suitable ring thickness, d_r to keep a mechanically stable via.

Using thermal traps to connect larger copper areas, especially with larger-mass power electronic components, remains the preferred solution. It is important to design the traps carefully as they have to cope with higher current densities compared to a solid connection.

When wave soldering is the targeted process, components should be placed in a way that prevents so-called shading.
Looking at the SMD component in **Figure 8**, the effect can be seen.

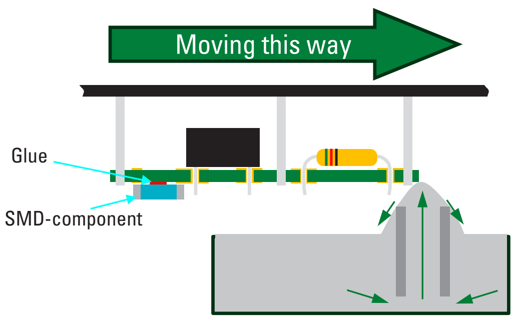

Figure 8. Shading of spots due to poor component placing

To allow for wave soldering, the SMD component is kept in place by glue. Being mounted in close proximity and in line with the through-hole component, the surface tension of the liquified solder may prevent proper wetting of the area. Consequently, the first pad may not be soldered properly, while the second pad receives the desired treatment. This may lead to poor connections but could also result in the tombstoning effect mentioned above.

In case short distances between two parts is a desired feature, arrangements should be considered that prevent pads from being aligned in a straight line. This is a recurring task when placing buffer capacitors for integrated fast-switching circuits such as gate-drivers or power-supply components like DC-DC-converters.

Summary

The soldering of semiconductor devices to PCB has historically involved Sn/Pb solders, but Pb-free solders are increasingly being used to eliminate Pb, as required by environmental regulations. Most Pb-free solders suitable for these applications are Sn/Ag alloys with higher melting points, with correspondingly higher solder temperatures. The higher temperatures are limited by the maximum allowable temperatures of components and fluxes. This creates a narrow temperature window for Pb-free solder. The optimization of the Pb-free solder processes is therefore more critical compared to Sn/Pb soldering. Pb-free soldering can also be more critical for power components and other larger mass components, which take longer to heat.

Although Pb-free solders often use the same flux systems as Sn/Pb solders, and thus have similar preheat cycles and heating rates, additional considerations are necessary for larger components, such as TO-268, TO-263, or power module packages. These components have higher thermal mass and require special treatment to ensure proper solder bonding between the base and substrate for efficient power dissipation. Careful control of ramp rates and uniform heating is essential to prevent thermal stress and ensure reliable solder joints in automated systems.

Pb-free solders are more environmentally friendly and are suitable for soldering most semiconductor components with appropriate temperature profiles in place. For wave- and reflow soldering, the temperature profile is influenced by multiple factors, including the number and configuration of heating and cooling zones, the type of solder, or flux used, the size of the board and components, and component density. Consequently, semiconductor device manufacturers are unable to provide a precise solder profile recommendation. It is the responsibility of the PCB assembly supplier to establish the appropriate soldering profile, considering these variables. Additionally, it is crucial to adhere to the Moisture Sensitivity Level (MSL) specifications during thermal processes. Components with higher MSL ratings must be properly handled, stored in moisture-controlled environments, and baked if necessary to prevent moisture-induced failures, ensuring the reliability and performance of sensitive semiconductor devices.

If these precautions, along with some attention to the PCB design are met, successful soldering of power electronic components can be achieved.

References

[1] Joint Industry Standard on Moisture/Reflow Sensitivity Classification for Non-hermetic Surface Mount Devices, IPC/JEDEC J-STD-020E

[2] Joint Industry Standard on Handling, Packing, Shipping and Use of Moisture, Reflow, and Process Sensitive Devices, IPC/JEDEC J-STD-033D.

[3] International Standard IEC 61760-1, Standard method for the specification of surface mounting components

