Ceramic Monoblock Surface Mount Considerations

Introduction

CTS ceramic block filters, like many others in the industry, use a fired-on thick film silver (Ag) metallization. The tin (Sn) component in solder alloys commonly selected for component attachment to a printed circuit board (PCB) can interact with the silver metallization during the solder reflow process. The molten tin will bond with the silver in the metallization layer as shown in Figure 1, depleting the thickness of the silver metallization layer, a situation referred to as "scavenging." If the silver scavenging is severe, operational failure of the filter can occur. The silver scavenging by the tin is exacerbated by excessive heat and dwell time. This document discusses means to minimize this and other potential application issues.

Solder Paste Selection

Ceramic filters are usually installed using solder paste. To minimize silver scavenging from the filter metallization it is recommended that a silver-bearing alloy be used. Two common silver-bearing alloys are 62Sn/36Pb/2Ag and 62Sn/36Pb/1.4Ag. If lead-free solder is required a 96.3 Sn/3.7Ag eutectic solder may be used. Non-silver-bearing solder alloys have been used successfully. However, their use is discouraged, and strict adherence to the other soldering guidelines outlined in this document is required.

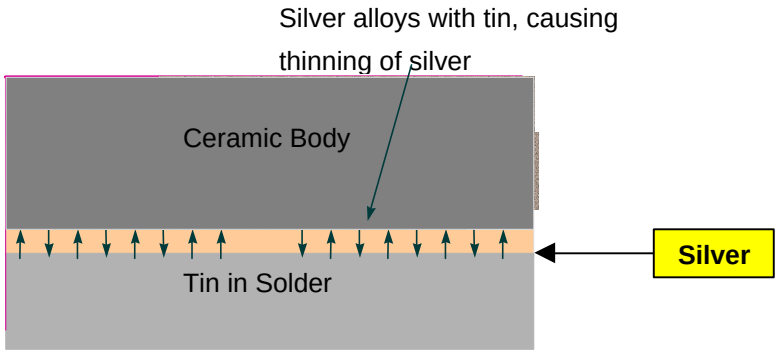

Figure 1 The use of silver-bearing solders will reduce the possibility of silver scavenging from the filter.

Solder Paste Application

CTS silver application process is such that the metallization near the edges of the filter is not as thick as the metallization in the center of the metal surfaces. Figure 2 shows a detail of this phenomenon.

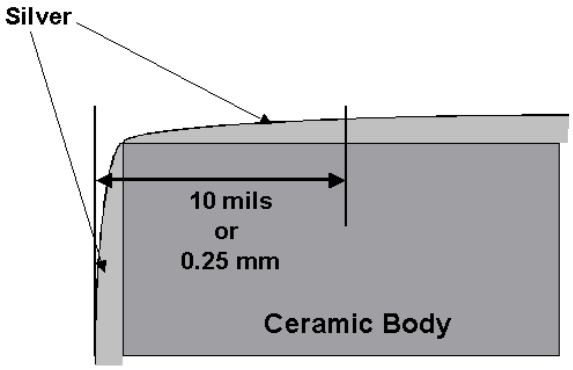

Figure 2 The metallization process causes thicker metal deposits near the center of the component

We recommend that solder masks and solder application methods be designed for the PCB that will prevent solder from contacting the edges of the filter. A clearance of 0.030” is desirable and 0.010” is an absolute minimum.
It is not necessary to solder the entire bottom surface of the filter to ground. This will cause the filter to float on the solder, and the resulting movement could cause input or output short circuits. This effect can be significantly reduced by creating a grid (or checkerboard) resist pattern in the large ground area beneath the filter. The resist grid reduces the surface cohesive forces and prevents the filter from floating as illustrated in figure 3.

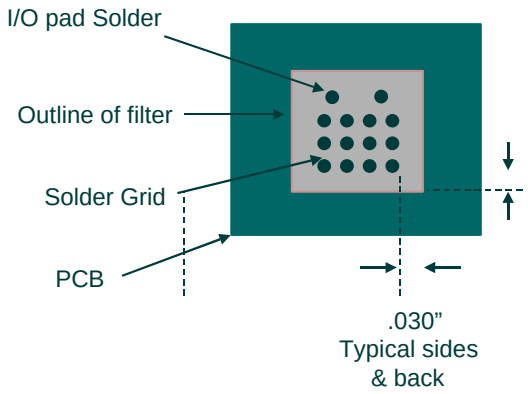

Figure 3 Placement of the solder mask grid pattern

Solder paste should be placed in the grid pattern with beads 0.004”/0.006” high on the ground plane and 0.006”/0.008” high on the I/O pads. The filter may utilize one of two types of shields. The first type is a wrap around shield. This type of shield does not require soldering to the PCB. The second type is an L-shaped shield which should be soldered in place with a .006” - .008" bead of solder. If the filter utilizes a metal shield, the solder bead for the shield should be 0.006”/0.008” high. If differential paste thickness is not possible, 0.006” solder thickness is recommended for all areas.
Since ceramic filters are surface mounted devices it is neither necessary nor advisable to apply excessive solder on the I/O pads to create a solder fillet to the front surface. As mentioned previously, the silver metallization is thinner on the edges of the filter. A fillet formed on the front surface may compromise the metallization on the front edge of the filter.

Re-flow

Re-flow ovens provide greater thermal uniformity when re-flowing metallized components. However, care needs to be exercised when creating and adjusting oven re-flow profiles so that the component is not subjected to temperatures that would accelerate silver scavenge. This is particularly challenging when the large thermal mass of these filters is considered. The best technique for administering this process is to attach a thermocouple adjacent to the filter and run a temperature profile of the filter as it is re-flowed. The temperature of the filter body must not exceed 265ºC and should not dwell above 240ºC for more than 45 seconds. See figure 4.

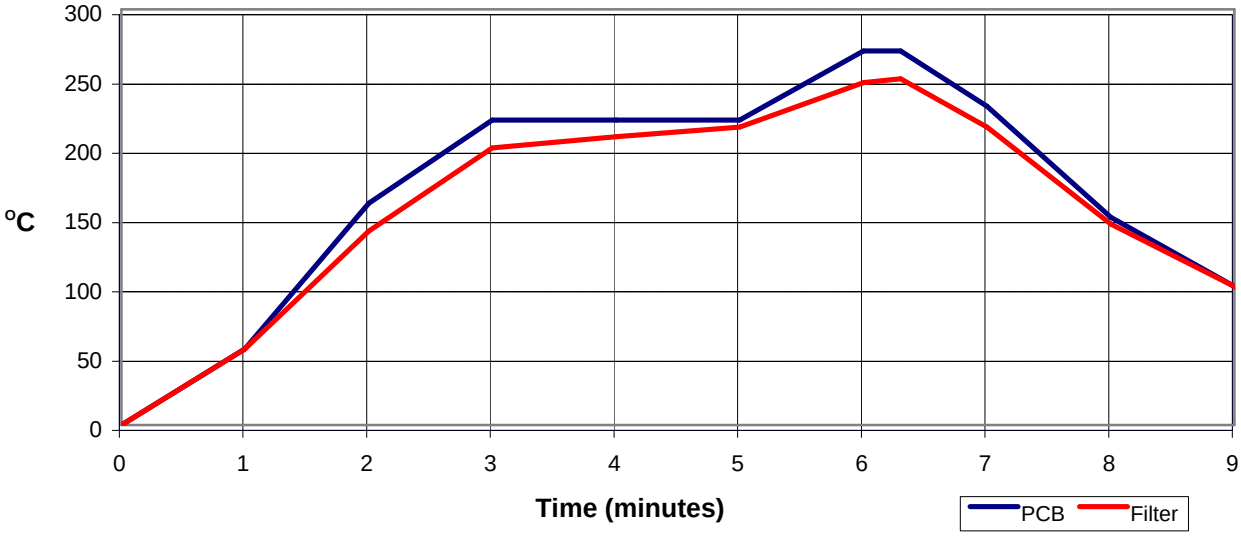

Figure 4 Reflow oven solder temperature profile

This maximum temperature exposure is especially critical for filters with shields attached. The shields may detach at temperatures in excess of 270ºC. A slightly longer "soak" period just below the flux activation temperature will help compensate for the thermal mass of the filter. If PCB's are assembled in an array (multiple boards on a panel) care should be taken when de-paneling the boards to avoid excessively warping and twisting the area of the board containing the filter. Warping and twisting the filter area could lead to fractured solder joints.

Filter Replacement

Removal of Filter

If filter replacement is necessary, it is recommended that a directed source hot air repair station be used. Use tweezers to handle the filter. Do not expose the filter to temperatures in excess of the solder melting point as damage may occur making postmortem difficult.

Installing Replacement Filter

It is helpful that the filter be preheated to 180ºC before reflow. Pre-tinning the filter with a soldering iron is not recommended due to possible damage generated by a concentrated heat source. Use tweezers to handle the filter. Do not place material handling devices inside of the resonator holes as filter damage may occur. The use of the same silver bearing solder paste is recommended for filter rework or replacement. As with the initial reflow, replacement filters should not be exposed to temperatures in excess of 265ºC.

Circuit Board Design

Silver Migration

The application of a DC potential to the input and output terminations of the filter in the presence of humidity may activate a migration of the silver ions from the ground areas to the termination, dropping the DC resistance from several megohms to several hundred ohms within a month of exposure. Therefore, use a DC blocking capacitor along with an inductor or resistor to ground to eliminate any DC electrical potential on the inputs or outputs of the ceramic filter. See figure 5.

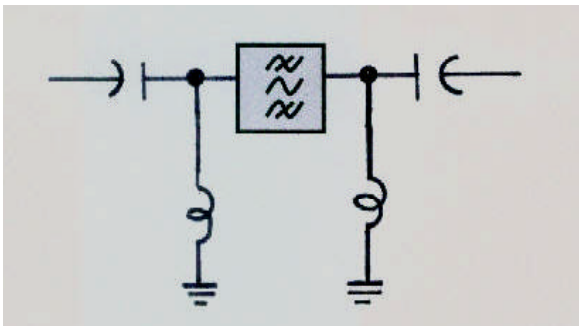

Figure 5 DC bias and silver migration.

Parasitic Capacitance

The I/O pads on ceramic filters are usually large as compared to other surface mount components. When the PCB is laid out to accommodate the I/O pads, care needs to be taken not to place ground planes close to these pads. We recommend that all ground planes below the I/O terminations are at least .020" away from the top layer. If there are ground planes closer than this (as can be the case with multilayer PCBs) it is recommended that the ground plane be relieved beneath the terminations (see figure 6).

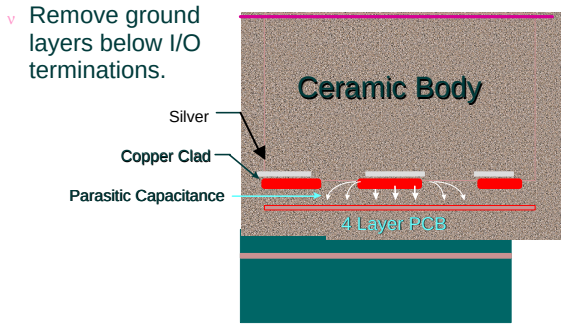

Figure 6 Parasitic Capacitance

Summary

The following checklist is a summary of things that have been discussed and should be used as a reference when designing a process for soldering ceramic filters to printed circuit boards.

- A silver bearing solder should be used.
- Solder should be kept .030" from the edges of the filter to prevent scavenging.
- Solder should be applied in a grid pattern on the ground plane area of the filter.
- The filter should not be exposed to temperatures in excess of 265ºC during reflow.

- Solder paste thickness should be .004”/. 006” in the ground plane area and .006”/. 008” on the I/O pads and shield area. A thickness of .006” may be used as the target thickness in all areas if differential thickness is not possible.
- If a board array is used during manufacturing, care should be taken when singulating the boards not to warp and twist the area where the filter is soldered.
- Do not attempt to form a solder fillet on the front surface of the filter. Scavenging of silver can occur if solder fillets are formed on this surface.