Knowles (UK) Limited
Hethel Engineering Centre
Chapman Way, Hethel, Norwich, Norfolk
NR14 8FB England

Tel: +44 (0)1603 723300
Fax +44 (0)1603 723301
Email: SyferSales@knowles.com
Web: www.knowlescapacitors.com

Design and coating of circuit board assemblies to limit flashover of MLCCs

Introduction

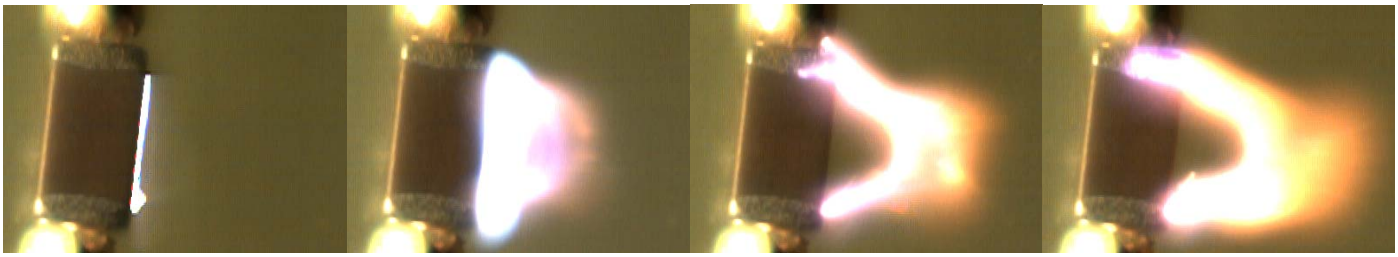
When mounting high voltage rated capacitors (HVMLCCs) on a circuit board it is often necessary to coat the board after assembly to prevent surface arcing between opposing terminals.

The practice of coating circuit boards is very well known and understood, but it is still possible to see examples of MLCCs that fail due to arcing underneath the chip, an area where coating does not always penetrate but which has the shortest arcing distance between the board pads.

A board design to eliminate this may seem obvious, but it is not always used.

Surface Arcing

Surface arcing, arc-over, flash-over, corona discharge, these are all terms for the same thing, an undesirable high voltage discharge which can cause interference and/or component failure. Surface arcing occurs when the dielectric strength of the surface environment is exceeded. High potential difference between the opposing terminations can lead to partial ionisation of the air which can then break down completely and allow a spark to discharge, this spark is visible, audible and the associated corona discharge will create electrical noise.



The arc inception voltage is the voltage at which arcing will commence, there are several factors which can affect this.

1. Humidity. At levels of high humidity surface arcing is more prevalent, this explains the sometimes seasonal nature of arcing problems and also why there are more issues in geographical areas of high humidity.
2. Surface Contamination. Solder balls, flux residue and other contaminants which have been deposited during manufacturing, processing and assembly.
3. Dielectric Type. Higher dielectric constant materials are more susceptible, also higher capacitance values. C0G/NP0 is not usually affected.
4. PCB design. Oversize pads, vias underneath capacitors, buried layers and geometries which inhibit good cleaning can all contribute to arcing problems.

It is also common for arcing to occur along joint lines where two surfaces are in contact, such as where a HVMLCC sits on a board.

The flux used when soldering the board is also worth a mention. Solder paste containing water soluble flux is commonly used as it is very aggressive, giving very good soldering results, and allows the use of water based cleaners with their perceived environmental advantages.

However water soluble fluxes tend to have organic acids that continue to be very aggressive if not completely removed and under high electric fields promote metal migration across the chip and board surfaces. Cleaning problems are most likely be evident where a chip is mounted hard against a board surface trapping flux and it is evident that this is the prime location for high voltage arcing to occur.

Coating

Environmental factors such as humidity are difficult to control in service, which is why most vendors' high voltage MLCCs will usually require the PCB to be conformal coated after assembly.

Some vendors will offer MLCCs with coating already applied, but in these cases it is important to review what the coating material is and how it performs. MLCCs that are supplied ready coated with conformal coatings or with organic surface protectorants (OSPs) are unlikely to have the coating withstand the conventional reflow and cleaning operations used to mount the components and should be used with care.

Design

To combat surface arcing pads should be well spaced with well radiused (not sharp) corners and creepage distance maximised. However, it is important to consider that IPC-7351 (Generic Requirements for Surface Mount Design and Land Pattern Standard) & IPC-610 (Acceptability of Electronic Assemblies) usually will demand a heel fillet on the mounting solder joints, so the PCB pads will always tend to represent the shortest flashover distance, extending further under the chip than the chip termination itself. When considering flashover issues, it is worth considering if the flashover is solely due to the MLCC, or if the pad creepage is the determining factor.

Probably the most important factor in reducing the issue of board interface flashover is to stop the chip lying flat against the board surface so that there is less chance for contaminents being trapped, more chance for cleaning agents to penetrate and do their job and no interface joint line for tracking to follow. Incorporating a gap between the chip and board also allows better penetration of coatings ensuring optimum resistance to arcing inception.

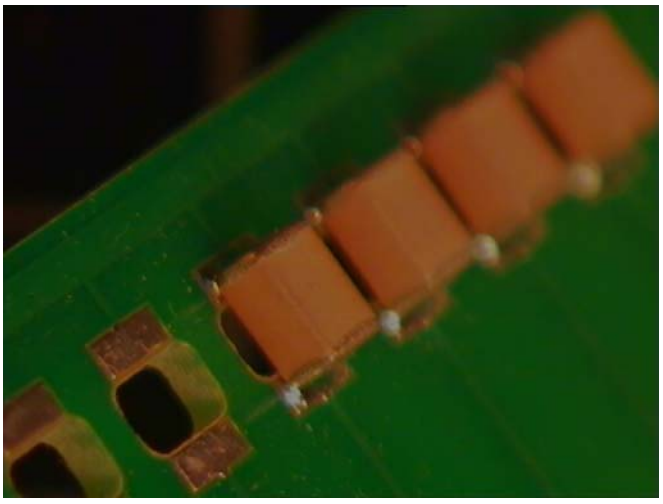
Finally, it is also important to note that buried layers running under MLCCs can also induce surface tracking, even if they are insulated by the board material, and should be avoided wherever possible. If they are unavoidable, then the distance between the top side of the board and the buried layer should be maximised.

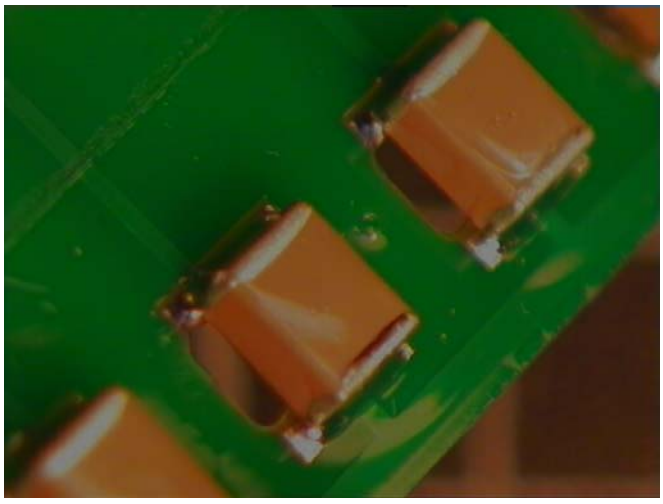
The board designs below have been shown to reduce the instances of flashover between the opposing terminals of HVMLCCs when combined with typical conformal coating procedures.

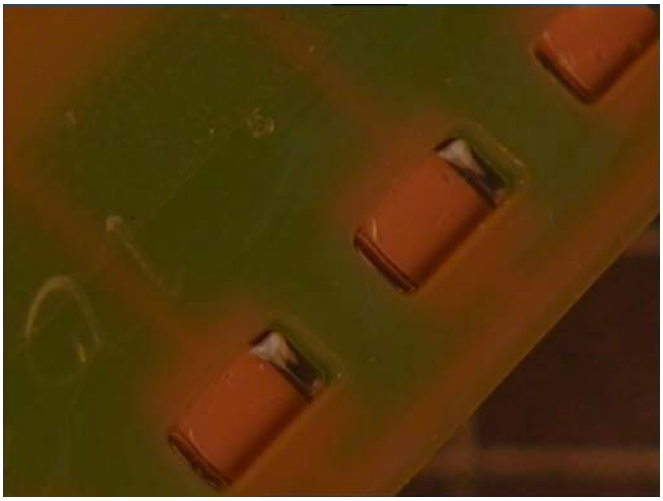
1. Slot the board under the MLCC

This is a standard design for high voltage and is the best solution to combat high voltage flashover.

It eliminates completely the issue of trapped contaminents and with appropriate cleaning ensures the chip surface is completely clean. When conformal coating is applied, it ensures that the chip can be coated around all 4 sides, offering the best solution to prevent flashover.





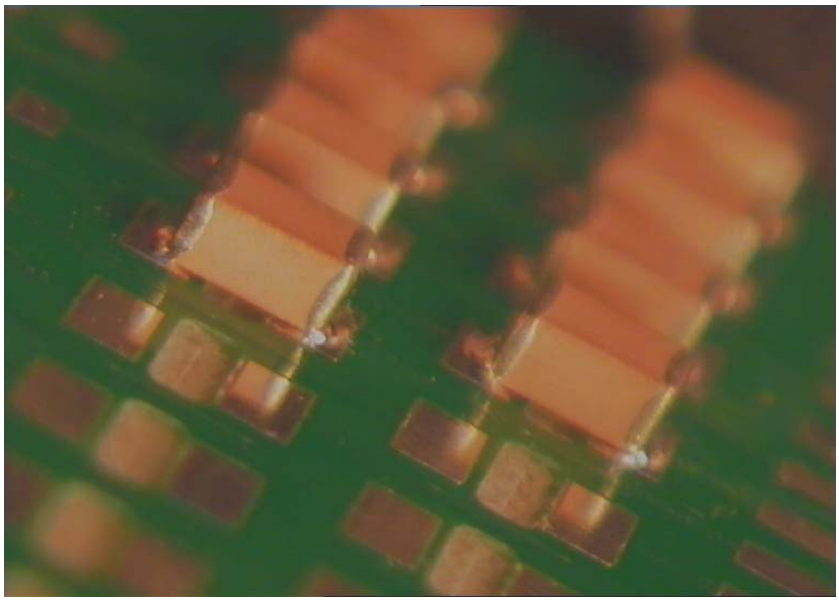


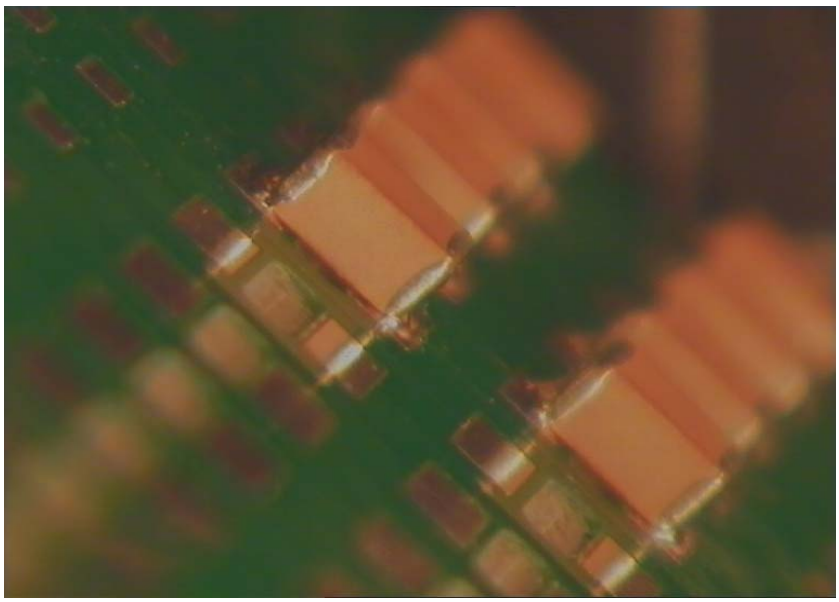
The boards should be routed out completely between the opposing pads and extending either side of the MLCC, so no part of the MLCC beyond the soldered areas is in contact with the board surface.

2. Route a recess in the board under the MLCC

A full slot is not always viable. Buried layers and ground planes within the board may not allow a full slot through the board, but a similar result can be achieved with a routed slot opening up the board space directly beneath the MLCC.

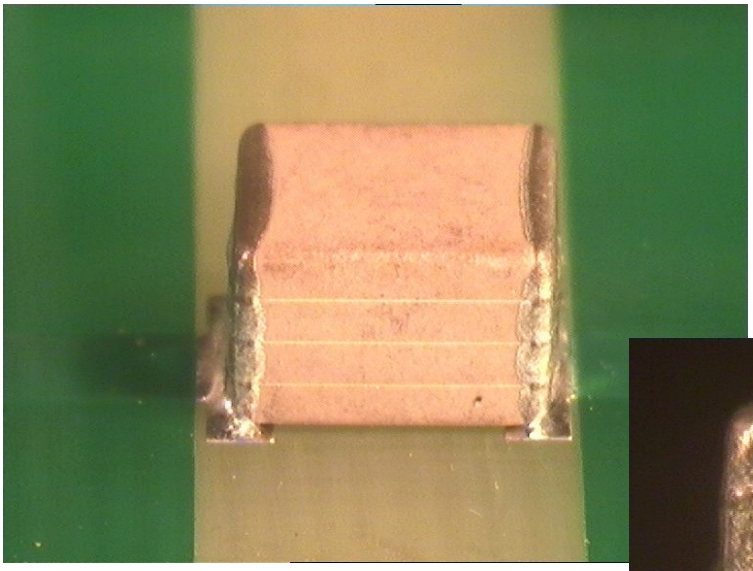
Again, it dramatically reduces the issue of trapped contaminents and helps ensure that cleaning solutions can penetrate properly around the MLCC. Application of conformal coating may need more care to ensure that the board is appropriately orientated to ensure coverage, but the larger aperture measn this is much easier than with a conventionally mounted MLCC.

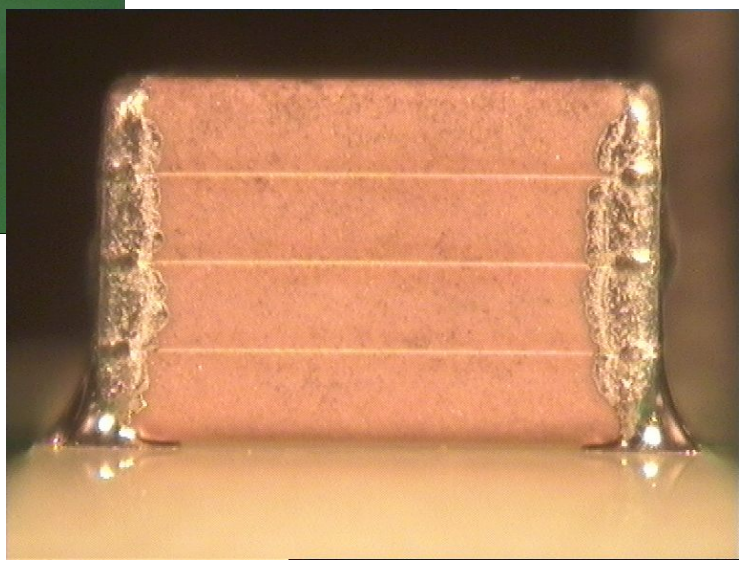




3. Remove the mask layer of the board under the chip

Simply removing the mask varnish layer from the board under the chip can mean the interface is open enough to allow suitable cleaning and penetration of coating under chip without the need to resort to opening up the board itself. Obviously the processing window will be smaller than a full slot, but it's a simple concept that is not always immediately obvious.





Conclusion

Operating MLCCs at high voltage demands consideration is made to the board design and layout as well as the component suitability. With appropriate designs coating can be introduced around all inactive areas of the MLCC and the potential for flashover controlled

More Information

For further information on Knowles capacitor component ranges, or for further technical assistance please contact our Sales Department on +44 1603 723310 or by Email at syfersales@knowles.com