N.D. Mehta, Assistant Professor, VGEC Chandkheda

Design Consideration of PCB for High Voltage isolation and Current Capacity

PCB Design Layout Rules Recommendation

In the PCB design of electronics circuit, it is important that one plan and has a checklist of the do's and don'ts before proceeding to do the printed circuit board layout. The understanding of the circuit is critical to the design, for example one needs to understand the maximum current and voltage that are carried by each conductor in order to determine the track width of the conductor and the type of PCB that will be used.

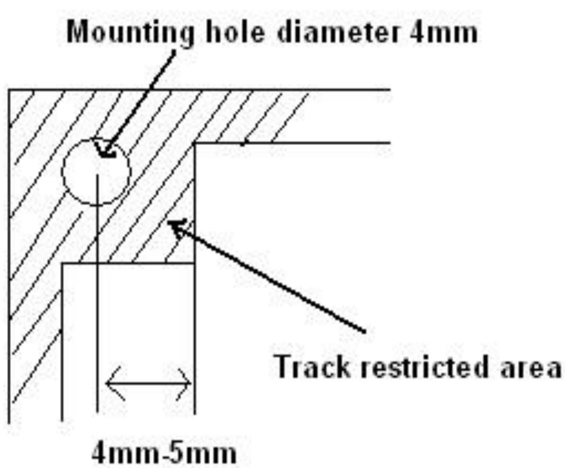
The voltage difference between each track will determine the clearance between each conductor. If the clearance is not enough, chances are that the electrical potential between each track will cause spark over and short circuit the PCB.

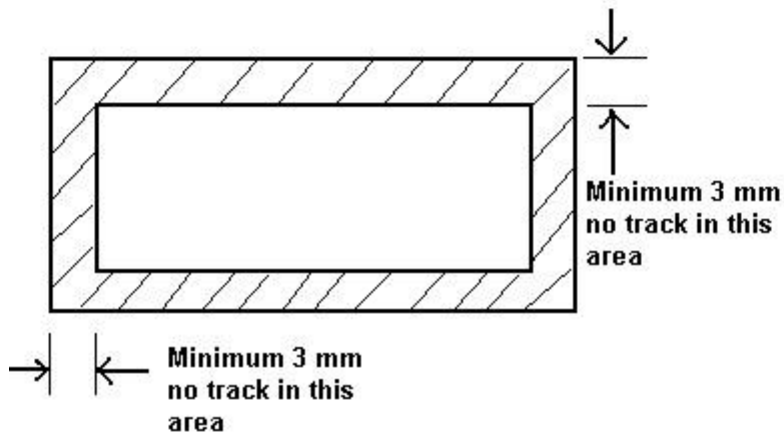
This will cause functional failure to the product and the safety of the users that are using the product will be compromised. It is therefore critical for one to understand some of these basics requirements before one proceed to design the PCB.

Tracks Restricted Area

Tracks should not be located on the areas that can cause them to be peeled off easily. One of the restricted areas is holes on the PCB which are used to mount screws or PCB spacers. These holes are usually used to secure the PCB to a casing or to secure it in a fixed place.

The edges of the PCB should not have any tracks as these areas are usually used to transport the PCB from one process to another process by using a conveyor belt. These edges are places where the possibility of scratches and cracking of the PCB happens. The recommended areas that should not have any track is as shown in the diagram below assuming a hole diameter of 4 mm which is used to mount a PCB spacer.





Tracks Restricted Areas

Conductor Thickness and Width

The PCB conductor thickness and width will determine the current carrying capacity of the track. The IPC standard for the conductor thickness and width of the common 1 oz/square-feet PCB. However, it is always advisable to use a bigger value due to the tolerance and variation of the PCB processes. If higher current carrying capacity is required, a 2 oz/square-feet or 3 oz/square-feet type of PCB is preferred. Many electronics hobbyist prefer to solder a thick cooper conductor on the PCB track to increase the current carrying capacity of the track.

IPC Recommended Track Width For 1 oz cooper PCB and 10 °C Temperature Rise

Current/A	Track Width(mil)	Track Width(mm)
1	10	0.25
2	30	0.76
3	50	1.27
4	80	2.03
5	110	2.79
6	150	3.81
7	180	4.57
8	220	5.59
9	260	6.60
10	300	7.62

N.D. Mehta, Assistant Professor, VGEC Chandkheda

For a high voltage PCB, you'll need a board material that's specifically designed to tolerate an overvoltage event, as well as the regular high V operating conditions. There are a few material options you to consider:

FR4 Laminate: FR4 high a very high dielectric breakdown. However, it is more porous than BT epoxy and polyimide, which makes it easier for the board to become contaminated. It also has a weak edge structure, and as the edge cracks, the dielectric value will decrease. Aging is a likely problem, especially for electronics near the edge. FR4 also has no recovery or protection from carbonization that occurs during overvoltage events.

BT Epoxy: A thermoset resin, BT epoxy has strong sidewalls and is better for applications with planar coils and medium voltage circuits.

Isola, high V laminates: There are several high voltage laminates, Isola is one of the most well-known, and that actually extinguish arcs and leave a non-conductive base layer. While that is an incredible performance advantage in high V applications, understand the design restrictions before you start. These laminates are usually quite pricey, and you can only produce single sided boards or very simple two sided boards.

PCB Design Electrical Clearance

Many safety standards call for a minimum of 8mm clearance between 40V mains and other isolated signal tracks. These safety standards are to ensure that the users that are using the products will be protected from any electrical hazards.

For non-main voltages, IPC recommend the electrical clearance between adjacent tracks. It is important to know the maximum difference in voltage that are applied on the adjacent tracks of a PCB.

Coating the PCB will help to reduce the requirements of the track clearance. However, the quality of the coating as well as the material used are critical to ensure that these requirements are met. Again, it is always advisable to increase the clearance to cater for the variations of the PCB processes.

IPC Recommended Electrical Clearance

Voltage	Coated Board	Uncoated(Up to 10,000ft)	Uncoated(over 10,000ft)
0-50	0.13mm	0.64mm	0.64mm
51-100	0.13mm	0.64mm	1.50mm
101-150	0.40mm	0.64mm	3.18mm
151-250	0.40mm	1.27mm	3.18mm
251-500	0.75mm	2.54mm	12.7mm
>500	0.00305mm/V	0.005mm/V	0.0254mm/V

Creepage, clearance and isolation

N.D. Mehta, Assistant Professor, VGEC Chandkheda

The spacing distance between components that is required to withstand a given voltage is specified in terms of clearance and creepage. A visual representation of the distinction between these terms and their applicability to board-mounted components.

Creepage

Creepage distance is defined as the shortest path between two conductive materials measured along the surface of an isolator which is in between. Maintaining a certain creepage distance addresses the risk of tracking failures over lifetime. The generation of a conductive path along the isolator surface due to the high voltage applied over long time is more related to the RMS value and depends on environmental conditions, which are described by a pollution degree and the material characteristics of the isolator.

To determine the creepage distance the following parameters have to be considered:

1. Working voltage
2. Pollution degree
3. Type of isolation
4. Tracking resistance of isolation materials (CTI value)
5. Circuit type (primary circuit, etc.)

It has to be noted that breakdown of the creepage distance is a slow phenomenon determined by dc or rms voltage rather than peak events or transients. Inadequate creepage spacings may last for days, weeks or even months before they fail.

Clearance

Clearance distance describes the shortest distance between two conductive materials measured through air. Sufficient clearance distance prevents an ionization of the air gap and a subsequent flashover. Similar to creepage distance the pollution degree, temperature and relative humidity influence the tendency for a breakdown. Breakdown along a clearance path is a fast phenomenon where damage can be caused by a very short duration impulse. Therefore, it is the maximum peak voltage, including transients, that is to be used to determine the required clearance spacing.
To determine the clearance distance the following parameters have to be considered:

1. Working voltage
2. Supply voltage
3. Overvoltage category and allowable transients
4. Pollution degree
5. Type of isolation
6. Installation altitude

N.D. Mehta, Assistant Professor, VGEC Chandkheda

7. Periodical transients in primary circuits

Clearances shall be dimensioned that overvoltage transients which may enter the equipment and peak voltages which may be generated within the equipment do not break down the clearance.

Protection with isolation

The general requirements are that a single level of insulation is acceptable if the circuit is not accessible, but wherever there are accessible components, they must be insulated from hazardous voltages by a double- level system, and each level must meet the insulation specifications appropriate to the application.

Five categories of insulation can be defined

Functional insulation (F) is that which is only necessary for circuit operation. It is assumed to provide no safety protection.

Basic insulation (B) provides basic protection against electric shock with a single level; however this category does not have a minimum thickness specification for solid insulation and is assumed to be subject to pinholes. Safety is provided by a second level of protection such as supplementary insulation or protective earthing.

Supplementary insulation (S) is normally used in conjunction with basic insulation to provide a second level of protection in the event that the basic level fails. A single layer of insulating material must have a minimum thickness of 0.4 mm to be considered as supplementary insulation.

Double insulation (D) is a two-level system, usually consisting of basic insulation plus supplementary insulation.

Reinforced insulation (R) is a single-insulation system equivalent to double insulation. It also requires a minimum thickness of 0.4 mm for use in a single layer.