**What is PCB?**

A printed circuit board or PCB, is a plate or board used for placing the different elements that conform an electrical circuit that contains the electrical interconnections between them.

The most-simple printed circuit boards are the ones that contains copper tracks or interconnects only on one of its surfaces. These kinds of boards are known as 1 layer printed circuit board or 1 layer PCB.

The most common PCB's manufactured today are the ones that contain 2 layers, that is, you can find interconnects in both surfaces of the board. However, depending on the physical complexity of the design (PCB layout), the boards can be manufactured of 8 or more layers.

**PCB Layout Rules:**

* 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 which causes the functional failure of the product.
* 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 ounces/square-feet PCB is as shown below:

IPC Recommended track width:

|  |  |  |
| --- | --- | --- |
| **Current/Ampere** | **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 |

* Many safety standards call for a minimum of 8mm clearance between 40V mains and other isolated signal tracks. For non-main voltages, IPC recommend the electrical clearance between adjacent tracks.

IPC Recommended Electrical Clearance:

|  |  |  |  |
| --- | --- | --- | --- |
| **Voltage** | **Coated Board** | **Uncoated**  **(up to 10k feet)** | **Uncoated**  **(over 10k feet)** |
| 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 |

**PCB DESIGN RULES:**

*Track Width and track spacing:*

 If the minimum track spacing needed is 0.3mmm, use 0.5mm instead. This will help to ensure that the yield is high during the production of the PCB.

*Reduce the number of holes and via:*

Whether you send the PCB for CNC (computer controlled PCB drilling) or you drill the holes yourself, you should try to reduce the number of holes and via on the PCB. Since the time utilized by the machine will determine the price of the PCB, the lesser holes and via will reduce the cost of the PCB. The rule is to keep it simple and as basic as possible.

*Size of the PCB:*

You should try to make use of the smallest area of the PCB as possible taking into consideration the track width, spacing and other constraints. Always try to minimize the size of PCB during the layout of the PCB. The size of the PCB is critical being the fundamental of the PCB Design Rule.

*Reduce the number of types of holes sizes:*

Try to design with common hole size as much as possible. This is to avoid changing the drill bits used by the machine. By having many types of hole sizes, the machine will have to change the drill bits and this takes time. The time taken will determine the cost of the PCB. Common drill bits sizes are 0.7mm, 0.8mm, 0.9mm, 1.0mm, 1.2mm, 1.5mm, 2.0mm and 3.0mm.

**PCB TERMINLOGIES:**

*Single sided PCB:*

A single-sided PCB is a board where the copper [footprint pads](https://www.tempoautomation.com/blog/best-footprint-pad-layout-guidelines-for-your-pcb/) are on only one side (top) and the traces or wiring is either on the same side or on the other (bottom) which has one layer of conductive material.

*Double sided PCB:*

Double-sided PCB refers to the use of both the top and bottom surfaces of a circuit board stack up for mounting components which has two layers of conductive material.

*Clearance:*

Clearance is defined as the shortest distance between two points on a PCBA.

*Via:*

A via provides a means of passing current from one layer to another.

*Routing:*

PCB trace routing is the process of specifying the copper traces that connect the board elements. These connections are typically based on the netlist created during schematic generation.

*Tracks or Traces:*

In PCB tracks or traces carries the current/ signal between components. As you can imagine complex a PCB board is more number of tracks needed to cover the complete functionality of a board. In complex board copper tracks will be laid on different layers and connected to respective components on the board.

*Routing width:*

Appropriate track width is very important factor when designing PCB’s ,Specially for those circuits which works on Higher voltage or current. Correct thickness of copper traces ensures safe current transfer and protects the board from overheating issues.

PCB trace width calculator: <http://circuitcalculator.com/wordpress/2006/01/31/pcb-trace-width-calculator/>

*Solder mask:*

Solder mask is a layer of protective coating applied to PCB. This is mainly used to protect the copper traces from Oxidation.  Solder mask provides aesthetics to manufactured board as this is the reason behind different colors of PCB. Out of which Green solder mask is most commonly used.

*Copper Weight:*

This term is used to indicate thickness of copper foil on each layer of a PCB. It's typically expressed in ounces of copper per square foot.

*Ounces:*

The most common unit of measure for the copper thickness on a PCB is ounces(oz).

1 oz = 1.37 mils (Thousands of an inch)

Guidelines for min spacing by Copper Weight:

Allowing as much space as possible between copper elements will yield a more robust final product.

|  |  |
| --- | --- |
| **Cu Weight** | **Recommended space b/w Cu** |
| 1 oz | 3.5 mil (0.089mm) |
| 2 oz | 8 mil (0.203mm) |
| 3 oz | 10 mil(0.254mm) |
| 4 oz | 14 mil(0.356mm) |

The important characteristics to consider when selecting a trace width include:

* The current capacity of the trace (how much current will flow through it)
* The allowable spacing between traces
* The size and pitch of the pads that the trace will be connecting to

Calculating spacing between PCB traces for various voltage levels:

<https://www.smps.us/pcbtracespacing.html>

*Creepage:*

Similar to clearance, [creepage measures distance](https://www.protoexpress.com/blog/high-voltage-circuit-boards/) between conductors on for high voltage PCB. However, instead of measuring distance in air, it measures the shortest distance along the surface of the insulation material.

PCB creepage calculator:

[*https://pcbdesign.smps.us/creepage.html*](https://pcbdesign.smps.us/creepage.html)

*Silkscreen:*

The silkscreen is a layer of ink trace used to identify the PCB components, marks, logos, symbols, and so on.

*Layer Stackup:*

 The printed circuit boards can be made of several layers.

|  |  |
| --- | --- |
| CAD Layer (conductive and nonconductive) | CAD Layer description |
| 1 | Top silkscreen/overlay ( nonconductive ) |
| 2 | Top soldermask ( nonconductive ) |
| 3 | Top paste mask ( nonconductive ) |
| 4 | Layer 1 ( conductive ) |
| 5 | Sustrate ( nonconductive ) |
| 6 | Layer 2 ( conductive ) |
| ... | ... |
| n-1 | Sustrate ( nonconductive ) |
| n | Layer n ( conductive ) |
| n+1 | Bottom paste mask ( nonconductive ) |
| n+2 | Bottom solder mask ( nonconductive ) |
| n+3 | Bottom silkscreen/overlay ( nonconductive ) |