

How to pick the right Trace Width for PCB design?

by: [venkatesh pampana](#) May 22,2021 135 Views 0 Comments Posted in [PCB Design Tutorial](#)

pcb basics

PCB layout

PCB Design Tutorial

Summary: This article walks through some important guidelines and tips for picking right trace widths while routing the PCB.

OVERVIEW

For everyone who designs a PCB, at some point they are faced with the decision of defining trace width. Many times, novice pcb designers will simply use the default width in the PCB layout software like Eagle, KiCad etc. While it mostly works fine for simple circuits, however, for circuits with medium to high complexity, some additional factors have to be considered. This is especially important when dealing with power circuits and high frequency circuits. Picking wrong trace width may sometimes lead to overheating, electromagnetic interferences etc. Excessively large traces consume valuable PCB space and increase the cost of PCB. This article walks through some important considerations to keep in mind for picking right trace widths while routing the PCB.

What factors are important for trace width?

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

- The current capacity of the trace
- Space availability and manufacturing limitations
- Trace termination
- Impedance

Trace Amperage (current carrying capacity)

For any given trace on the PCB, there is a maximum amount of current it can handle before it fails. Passing large current through a trace will cause it to dissipate power as heat (like a resistor), and given enough current (and time) the trace will be destroyed by either burning through, or de-laminating from the PCB there by breaking the trace. Most often we think of traces as zero resistance connecting wires between the components, but this is certainly not the case. All traces will have some sort of resistance and it is important to consider this when selecting widths. Knowing the resistance of a trace and the maximum current that will be passed through it, will help inform which width to use. So it is good to ask following questions before deciding trace widths:

- What is the thickness of the copper layer?
- Is the track on the outer layers (top or bottom layer)?

Is it on one of the inner layers where heat can't escape as easily?

How long is the track?

What is the thermal coefficient of the dielectric?

Calculating the resistance of traces could be a tedious job. Fortunately, there is a handy tool available online (link: <http://circuitcalculator.com/wordpress/2006/01/31/pcb-trace-width-calculator/>) that can help guide the choice of trace width when considering amperage. Screenshot of calculator is given below.

Inputs:

Current	1	Amps
Thickness	1.4	mil <input type="button" value="v"/>

Optional Inputs:

Temperature Rise	15	Deg C <input type="button" value="v"/>
Ambient Temperature	25	Deg C <input type="button" value="v"/>
Trace Length	35	mm <input type="button" value="v"/>

Results for Internal Layers:

Required Trace Width	0.601	mm <input type="button" value="v"/>
Resistance	0.0295	Ohms
Voltage Drop	0.0295	Volts
Power Loss	0.0295	Watts

Results for External Layers in Air:

Required Trace Width	0.231	mm <input type="button" value="v"/>
Resistance	0.0766	Ohms
Voltage Drop	0.0766	Volts
Power Loss	0.0766	Watts

In the calculator, you have to enter design specifications such as the thickness of copper, the expected maximum amperage that will pass through the trace, the length of the trace, and the acceptable increase in temperature due to the resistance of the trace. After entering these values you will be presented with a calculated trace width. It is important to note that this value is a minimum width required to meet the design criteria inputs.

The thickness parameter mentioned in the calculator is referring to the thickness of the copper layer of PCB and not the thickness of PCB itself. Copper layers used on PCBs come

in fairly standard thicknesses - typically measured in ounces (oz) per square foot or Mil. For example, the most commonly used 1 oz copper board is about 1.4mils. General rule of thumb is that the thicker the copper layer, the lesser the trace width required. PCB manufacturers offer boards of various thicknesses. For example, PCBWay, unlike other popular pcb manufacturers, offers boards over wide range from 0.5 - 13 oz copper thickness.

Generally, the traces on the inner layers are required to be thicker than outer layers as the heat can't escape easily from the inner layers. This is relevant for multilayer PCBs with more than 2 layers.

Determining the width of a trace based on the current demands is important for most power traces and high power signals, however, except power lines, most traces on PCBs pass the signals that draw negligible current. For these low power signal traces, we should look at other characteristics presented below to determine the width.

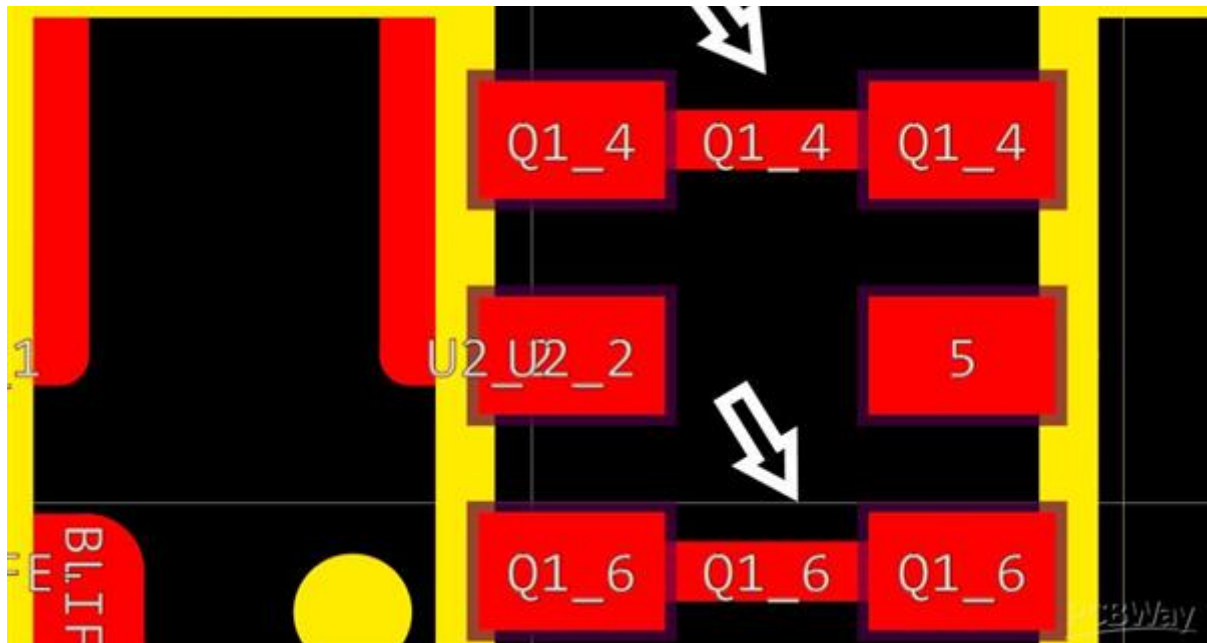
Space availability and manufacturing limitations

The size of a PCB is directly connected to the manufacturing cost of the PCB. Therefore, designers aim to keep PCBs as small as possible. The downside of reducing board size is that it can limit the available space to route traces. For low power signal traces it is generally advised to keep traces small to increase the available space available for routing. However, minimum trace width recommendations from manufacturers should be kept in mind. The minimum trace widths and spacing are manufacturer and process dependent and usually vary between 3mils and 8mils. Usually, the lower minimum widths tend to be much more expensive compared to higher minimum widths. So It is recommended to avoid trace widths less than 6mil, unless it's absolutely necessary. The widths of 6 to 25 mils is typical for most signal traces.

Trace Termination

Sometimes, the point at which a trace meets a pad can also inform the width of a trace. In most SOIC footprints, trace widths are set to the same width as the pad they terminate to. This will avoid violating minimum spacing constraints between adjacent traces. An example scenario is given below:





In the image we can see two traces connected to the pad of the footprints. The width of these traces are both slightly smaller than the width of the component pads. This allows for plenty of clearance between adjacent traces while routing away from the chip. Selecting the trace widths for BGA components can be a little more tricky and is out of scope of this article.

Impedance

Impedance is another determining factor in some applications. At low frequencies (ie. below