PCB Design: Best Practice

By Michael Bell

Purpose

This E-Workshop supplemental document provides strategies and guidelines for printed circuit board (PCB) design.

Contents

Abbreviations

SI	International System of Units (Metric)
PCB	Printed Circuit Board
EDA	Electronic Design Automation
EMI	Electromagnetic Interference
HV	High Voltage
RF	Radio Frequency
IC	Integrated Circuit

Design Guidelines & Best Practice

When designing, it is best to follow the following guidelines as closely as possible to give a consistent and well drawn schematic.

Stay Within Manufacturer Tolerances

When designing your PCB, make sure to design it within manufacturer tolerances. If you are unsure of what manufacturer you plan on using, it is wise to find a couple and use the loosest tolerance so that you have the option later on to choose a specific manufacturer. Most manufacturers have the same tolerances, however, it pays to always double check.

Snap Grid and Spacing

When designing a PCB layout, utilize the snap grid and keep distances consistent. Make sure to space components out leaving room for the component to be soldered on easily with a soldering iron or solder paste.

Clearly Label with Silkscreen

When spacing out components on the PCB, adjust the placement to prevent overlap in silkscreen. Make sure all reference designators are visible and unambiguous in their placement.

Trace Width

The width of your traces will play a large part in how much current can flow through them. Select trace width based on application and current/resistance requirements. Below is a table of IPC recommended track widths for a 1 oz copper PCB. The values are calculated based on the current required to raise the temperature of the trace by 10 degrees celsius.

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

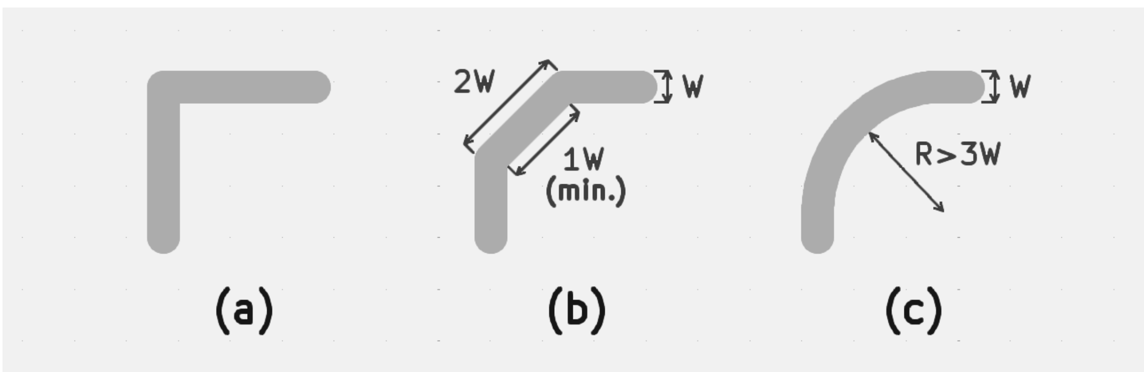
Current/A	Track Width(mil)	Track Width(mm)
1	10	0.25
2	30	0.76
3	50	1.27
4	80	2.03
5	110	2.79
6	150	3.81
7	180	4.57
8	220	5.59
9	260	6.60
10	300	7.62

Trace Spacing

The spacing of your traces depends on the purpose of the traces. In applications where signals are being sent, it is best to follow the 3W rule, where you space two parallel traces 3 times the trace width apart. This will reduce interference (crosstalk) between the two traces.

Trace Bends

When you need to make a bend in a trace, it is important to keep in mind the type of signal the trace is carrying. For low frequency signals and applications where trace length or resistance is not a concern, a 90 degree (a) turn is acceptable, and can sometimes give the PCB a unique look. Typically, a 45 degree miter (b) would be employed for general use in digital and analog signaling as it reduces overall trace resistance (shorter length) and has better high-speed signaling characteristics. For high-voltage power or radio-frequency signals, a round bend (c) would be used as it has the best high-speed characteristics, maintains a consistent characteristic impedance, and, for high-voltage, does not have any sharp corners for charges to build up on.

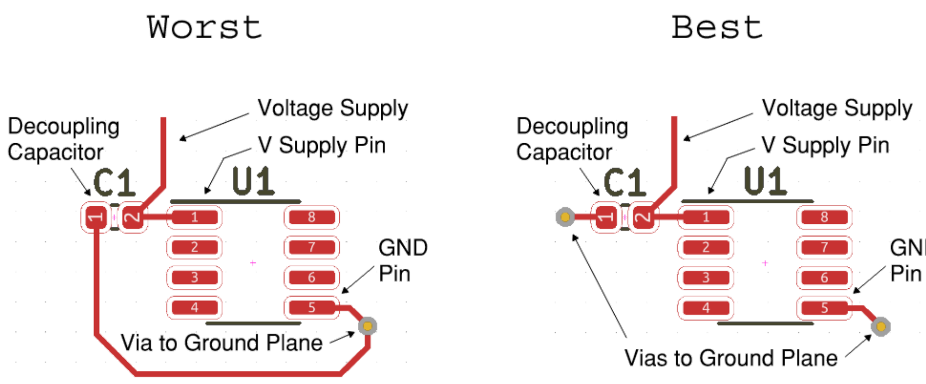


Trace Junctions

Avoid junctions of more than 3 traces. When making a 3-way junction, avoid acute angles. Acute angles create signal reflections at higher frequencies, and generate etchant “acid” traps during manufacturing. When choosing a junction, the 90 degree (a) is perfectly acceptable for general use, but as the signal speed gets faster, the mitred (b) or ‘Y’ junction (c) may improve high-speed signal characteristics and signal integrity.

Place Decoupling Capacitors Close to ICs

When you design a PCB layout for a schematic containing decoupling capacitors for ICs, always place the decoupling capacitor as close to the IC power input pin as possible, and connect it directly to a ground plane using a short trace or via.

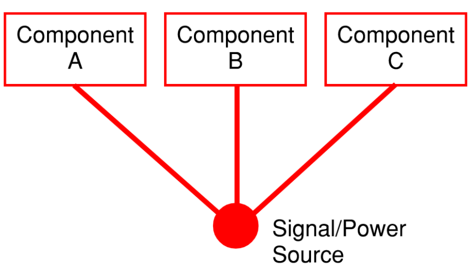
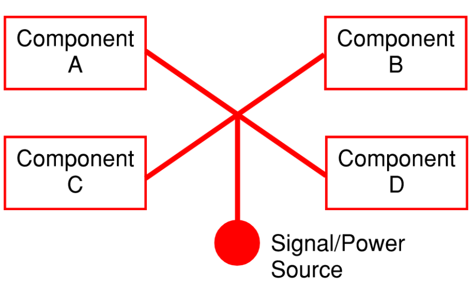


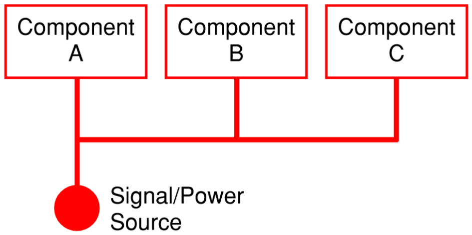
Power and Ground Planes

If you are designing a PCB with 4 or more layers, the two inner layers should be a copper layer for power and ground. This will reduce the number of traces needed to distribute power, reduce EMI, and create a shorter path to both power and ground through vias.

Power and Signal Distribution

There are three methods for power distribution: single point source, star source, and multipoint source. Each has their strengths and weaknesses. Below is a table to summarize:

<table>
<tr><th>Diagram</th><th>Description</th></tr>
<tr><td>
</td><td>Single Point Source
The power traces of all the components only meet at a single reference point. A single point is considered the best for power distribution. It can be harder to implement for complex or large/medium designs however.<table><tr><th>Power</th><th>Signal</th></tr><tr><td>Best</td><td>Okay</td></tr></table></td></tr>
<tr><td>
</td><td>Star Source
Maintains trace length between components. The star connection is often used for complex high-speed signal boards. From the center, all the signals can go to any region of the board with minimal delay between the areas.<table><tr><th>Power</th><th>Signal</th></tr><tr><td>Okay</td><td>Best</td></tr></table></td></tr>
</table>



Multipoint Source

Multipoint sources are considered the worst connection type as it creates differences in the voltage and impedance coupling between components. This connection type introduces noise in nearby circuits when used with high-speed switching and signal connections.

Power	Signal
Bad	Bad

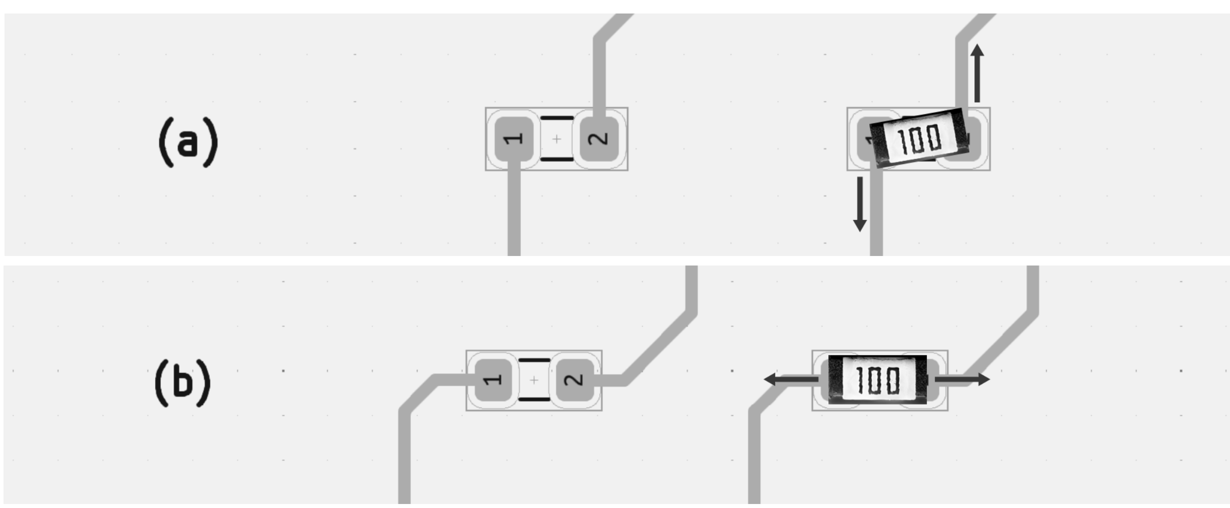
All PCBs will have a combination of these three methods of power and signal distribution.

Differential Pairs

When designing a PCB layout containing differential pairs, keep the trace lengths the exact same. Most EDA CAD softwares automatically maintain the differential pair length, but in some cases you may need to enable that function.

SMD Thermal paths and Reflow Considerations

When soldering surface mount components using solder paste and a hot air station (or reflow oven), the surface tension of molten solder will automatically align surface-mount devices (SMDs) on their pads. This however can cause issues where the thermal relief on pads may create colder areas on the pad where solder may solidify faster. In the case of pads with traces connecting from top and bottom (a), the component may



Useful Resources

Useful Web Pages

Topic	URL
PCB Design	https://blog.mbedded.ninja/pcb-design/
Traces	https://www.autodesk.com/products/fusion-360/blog/trace-width/ https://www.mktpcb.com/pcb-traces/ https://www.microwaves101.com/encyclopedias/mitered-bends
Footprints & Packages	https://www.pcblibraries.com/Products/FPX/UserGuide/download/Footprint%20Expert%20Surface%20Mount%20Families.pdf https://www.topline.tv/SMDnomen.pdf https://www.ti.com/support-packaging/find-packages.html
Decoupling Capacitors	https://www.renesas.com/us/en/document/apn/an1325-choosing-and-using-bypass-capacitors https://www.protoexpress.com/blog/decoupling-capacitor-use/
Manufacturer Tolerances	https://www.pcbway.com/capabilities.html https://docs.oshpark.com/submitting-orders/drill-specs/ https://docs.oshpark.com/services/
Differential Pairs	https://resources.altium.com/p/what-are-differential-pairs-and-differential-signals
Ground Loops	https://resources.altium.com/p/preventing-ground-loops-your-pcb-design
Decoupling Capacitors	https://www.analog.com/media/en/training-seminars/tutorials/MT-101.pdf