

Bron: <http://www.electronic-products-design.com/geek-area/electronics/pcb-design/general-pcb-design/a-good-pcb-design-checklist#more-499>

Good PCB Design Checklist

A good list of checks to make before committing the design of a new printed circuit board:

- EMI & Signal Integrity checks:

Are there decoupling capacitors in all areas where there are connectors or via's, to minimise EMI loop sizes of signals and their return ground path? If not do you need to add some? A power plane will act like the ground plane to high speed signals as long as the return signal can jump to the real ground plane through a nearby decoupling capacitor.

Ensure all high speed signal traces run over their own ground / power planes. Do not allow say a digital signal to travel over the analog plane unless it is going to a device in that area and in which case follow the devices digital ground trace to minimise the loop and therefore noise.

If there are any slots or gaps in the Gnd / Power planes ensure no high speed signals run over them (to avoid the return path having to loop round, creating EMI problems).

For high speed signals minimise track stubs (to below the critical length – ideally < 6.5mm, no more than 12mm for a 1ns rise time signal).

Ideally high speed connectors should have the ground plane getting through between pins to avoid signal return paths having to go round the connector to a ground pin on it.

Differential pair tracks are as close together as possible, or spaced based on impedance calculation?

Do any high speed signals need termination? This can be to VCC or Gnd and ideally should be at end of bus after the last receiving device. A good value is typically 50ohm, but anywhere from 30 – 100ohms is often fine. Should ideally match the impedance of the track.

- Add test points for important buses and connections to tight SMD chips so its easy to attach a wire or scope probe.
- Are power tracks big enough and do they have big enough via's?
- Are there any very big through hole components such as elect capacitors? If so don't connect to internal layers on multilayer designs in case they get knocked and the internal connection is damaged.
- Check connections into power planes on >2 layer boards are big enough (enough via's and big enough tracks).
- Check all important IC power pins have good decoupling capacitor connections
- Check crystal connections are short
- Do you need to add manufacturer test points (pads) for voltage rails etc?
- Are there components that need copper plane heatsinking (voltage regulators, mosfets etc)?
- Do you need to separate earth tracks from other tracks to avoid static problems?
- Are all jumpers, connectors etc labelled on the silkscreen?
- Check all surface mount pads have tracks that come out of the end, not the side (i.e. no links between adjacent IC pads that will look like a short during inspection).
- Do you need to add fiducial marks for the pick and place machine?

Bron: <http://www.electronic-products-design.com/geek-area/electronics/pcb-design/general-pcb-design/a-good-pcb-design-checklist#more-499>

- Are there nets that need their length equalised (e.g. fast differential connections such as Ethernet)?
- Add a hatch fill done on both sides connected to GND to improve EMC? These are a good general setting for the fill:- Track width: 0.3mm, Grid size: 0.6mm
- Create board outline on a mechanical layer
- Do a final design rule check
- Double check mounting positions will be OK in the equipment the PCB is fitted in.