



Design Review Checklist

Printed Circuit Board (PCB)

Last updated: 11/7/2019

- Sufficient clearances between high voltage traces.
- Trace widths are sufficient for traces carrying large currents.
- Low noise traces do not run too close to high-current traces or high-speed digital traces.
- For high-speed traces, ensure that line lengths are properly matched. The higher end layout packages have tools to do this.
- For high-speed traces also run simulations to ensure that there are no violations, including the effects of vias on signal propagation.
- Consider the effect of ground loops for designs without a proper ground plane.
- Be aware of the placement and orientation of magnetic components, if any, to avoid cross coupling.
- For very low-level signals, make sure that the appropriate guard rings are in place.
- Depending on the frequency of the signal, the typical FR4 substrate may not be suitable, and a low loss, but more expensive, substrate should be considered.
- Ensure that any RF lines have the proper complex impedance (typically 50 ohms). This includes confirming that a ground layer is underneath the trace (*microstrip*) and commonly on both sides as well (*coplanar waveguide*).
- On-board antennas have the proper ground clearances on all layers.
- Ground layers have sufficient stitching vias especially near any RF circuits.
- On-board chip antennas are placed according to the manufacturer's recommendations.

- If the design includes multiple on-board antennas be sure they are placed so as to maximize their distance to prevent cross interference.
- Switching power regulators are carefully layed out according to the manufacturer's datasheet.
- Power supply pins on any IC's have decoupling capacitors placed nearby.
- No footprint errors it's best to print out the layout at scale, and then physically lay all of the components on the printout to ensure the leads match up properly.
- Test points are included on all signal traces which aren't easily accessed (i.e. signals between leadless packages).
- No blind or buried vias are used unless absolutely required. Their use will significantly increase the board cost.
- Final board dimensions match correctly with the enclosure design.
- Any crystal connections are kept as short as possible.
- All components, connectors, jumpers, and test points are properly labeled in the silkscreen layer.
- No signal traces have 90-degree bends
- Signal traces don't unnecessarily jump between different board layers. This can also be a sign of auto-routing, which rarely produces a quality design.
- High-current traces are primarily routed on the outside layers.
- Silkscreen layer includes the proper board part number and the current revision.
- Design includes the necessary shields, mounting holes, and heat sinks.

- If required for assembly, confirm that the design includes the necessary fiducial.
- Polarities are marked for polarized components like some capacitors.
- Pin 1 is marked on all integrated circuits.
- Digital and analog sections are kept separated with grounds connected at a single point.
- Generate a 3D model of the PCB, and make sure that there is no interference with other parts of the device assembly.
- Design rules match the capabilities of the specific PCB manufacturer.