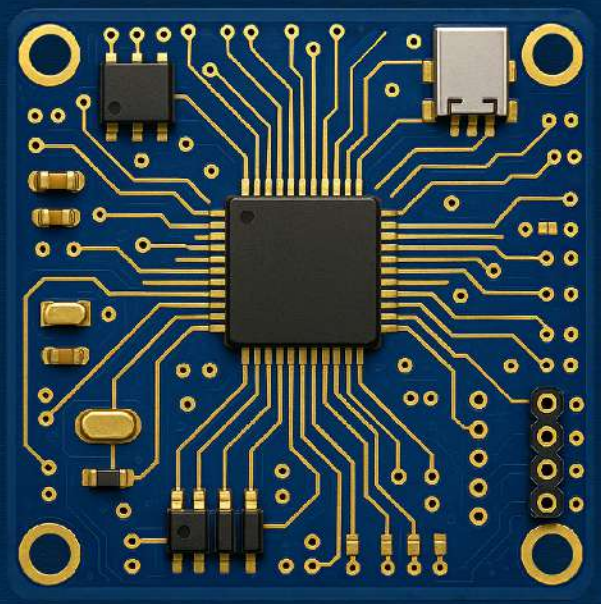
PREDICTABLE DESIGNS

DESIGN REVIEW CHECKLIST

PRINTED CIRCUIT BOARD (PCB) LAYOUT



- [ ] 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.

For Entrepreneurs and Startups:

Develop and launch your electronic product FASTER

without costly mistakes!

Doing it alone is slow and risky. Small missteps now can become big problems later. Skip the mistakes and launch faster.

Frustrated things aren't moving as fast as you want?

Feeling unsure what to do next?

Meet your guide: John Teel

Hey there, I'm a former microchip design engineer at Texas Instruments and founder of a hardware startup that sold products in hundreds of retail stores. My chip designs are in devices from Apple, Intel, and more.

Now, my full-time focus is helping people like you bring new electronic products to life, without wasting time, money, or risking everything.

CLICK HERE TO LEARN MORE!