

Ten Rules for Components Placement on PCB Design

Terms of using this article

This article is primarily for internal use in Quick-teck. Now we decided to open it up publicly. We try to ensure the information in this are as accurate as possible, but please be aware we don't take any responsibility for anything that results from this article. You're using this at your own risk.

Details

1. Start placement of big/main/critical components(this could be MCU,DSP,FPGA,DDR, clock devices) and then place support component like resistors, capacitors around them.
2. Use 50 or 100mil as component grid to place these big components (QFP, BGA, SOP, SOIC, or through hole connectors). Use 25mil as component grid for these SMT resistors/capacitors or other small passive components.
3. Isolate the analog, digital and power supply section.
4. Clock driver/synchronizer should be located close to clock oscillator.
5. Arrange components in rows and columns, and oriented uniformly for ease of installation, inspection and rework. When possible, all polarized components such as tantalum capacitor and diodes in the same direction. Polarity of these components should be indicated on the silkscreen.
6. Keep at least 40mil space between components and 100mil space from component to board edge. Locate connectors on one edge or one corner of the board.
7. Try to place all components on the top side only. If not possible, only these low-profile, small size and low thermal dissipation component (like SMD passive parts) can be on bottom side.
8. Place decoupling capacitors close as much as possible to VCC pin on active components.
9. Keep at least 200mil clearance from transformer to electrolytic capacitor.
10. Locate indication LEDs, test points, switches, jumpers and adjustable components for easy access.

One more rule, it's always good to double check analog, high frequency, RF, high voltage, high profile, heavy or high heat components before starting trace routing.