28 Rules for PCB Design

Publicerades 15 januari 2016

Lisa Chu
RAYMING PCB & ASSEMBLY MANAGER

+ Följ

These 28 Basic PCB design guidelines set out best practic e to reduce the cost of your boards and to minimize the ri sk of errors arising during manufacture.

High power boards have different rules especially in term s of spacing, traces size and power isolation. Manufacture rs have different requirements; make sure you read their o wn guidelines before sending your design. Naming and fil e formats also vary depending on the manufacturers.

PCB Layout

1. Create your board frame on a 0.05” grid. Make the lower left corner start at 0,0.

2. Usually the board frame is rectangular. For specific reasons you could also do other ty pes of shapes such as polygons.

3. Stick parts on a 0.05” grid. You should not break this rule unless you have a very good reason.

4. Any LED should be labeled with its purpose (power, status, D4, Lock, etc).

5. Idem for connectors: e.g Vin, Port1, Batt, 5V, etc.

6. Idem for pins where applicable: e.g TX, Power, +, -, Charge, etc.

7. Idem for switches and switch states: eg. On, Off, USB etc.

8. When applicable, it is better to avoid having vias go through the silkscreen when addi ng labels.

9. Group components together. For example the resistors surrounding a transistor in your schematic will also be grouped together on the PCB.

10. Minimum drill size should be 15 mil.

11. Minimum annular ring size should be 7 mil.

12. 7 mil is the minimum size for traces. 8mil is acceptable. When possible try to keep the traces size to 10mil.

13. Use thicker traces for power lines. 12mil=100mA max, 16mil=500mA max etc.

14. 7mil between traces and space is reasonable.

15. Avoid 90 degree corners. Straight lines with 45 degree corners are preferable.

16. Where applicable use a ground pour on top/bottom layers.

17. To prevent pours from shorting to traces make sure you use a 10mil isolation setting o n any of the ground pour.

Schematic Layout

1. Use a GND symbol for all the GND connections.

2. Use appropriate power symbols for All VCC, 5V, 3.3V etc.

3. Add color notes to separate a complex design into various smaller bits (for example,c harge circuit, accelerometer, etc).

Footprints

1. All footprints need a reference designator {{refdes}}. If you come across a part on a b oard that doesn’t have this, you should change it and save the library. For parts requiri ng it a pin one marker should be defined.

2. All footprints need silkscreen indicators showing mechanical sizes, dimensions, or an ything wired about the part.

3. To prevent it from flaking off easily silkscreen within a footprint or board should not go over pads or metal that will be exposed.

4. Top component layer should be marked by a red centre cross.

5. Package outline layers should outline the actual package size.

6. The Top Courtyard layer should include all of the pins.

7. When adding a footprint make sure you add a solder mask.

8. Every new footprint and part will have a human readable des

9. More detail, contact me;) Lisa Chu skype: raypcb09