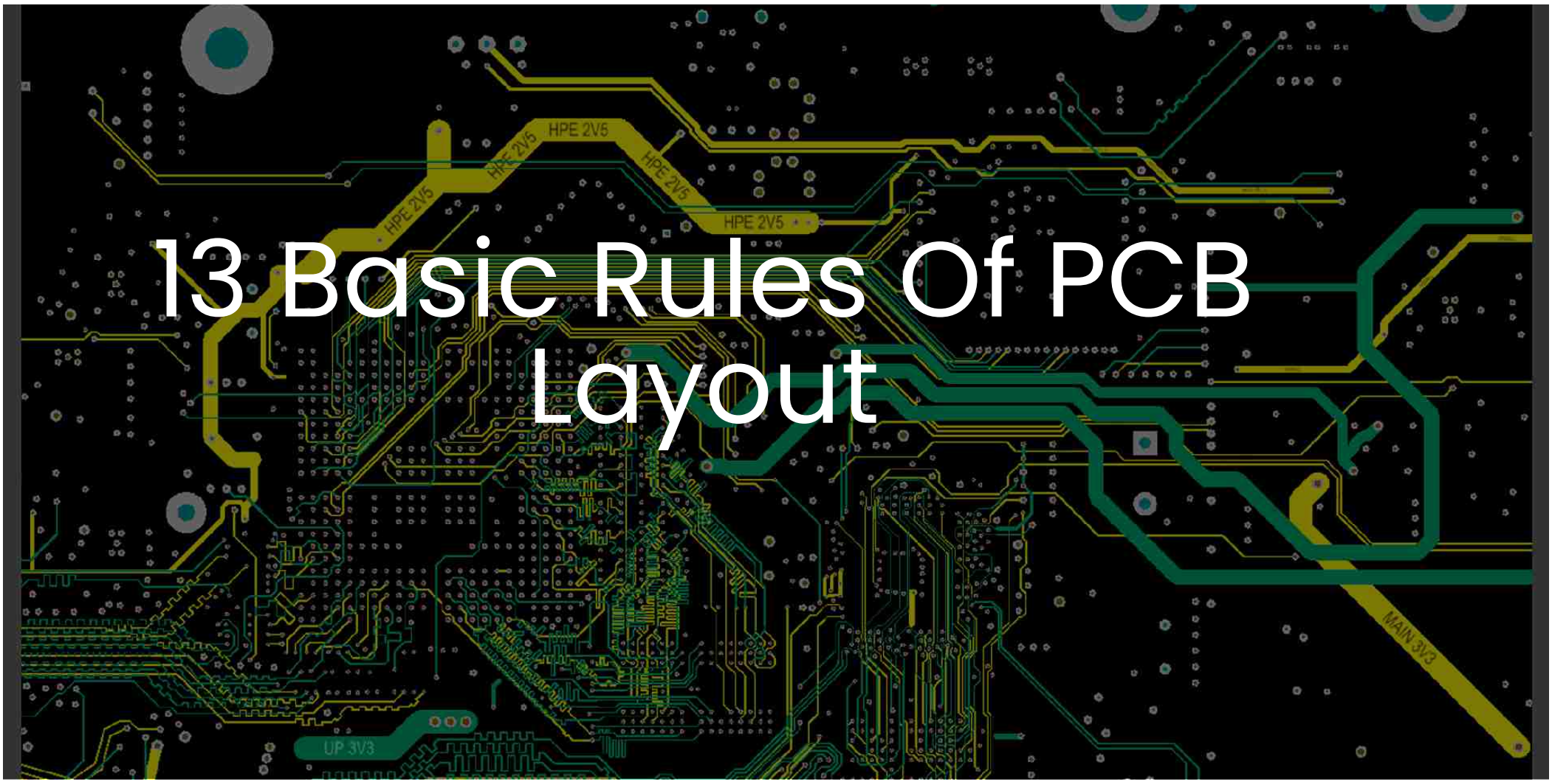
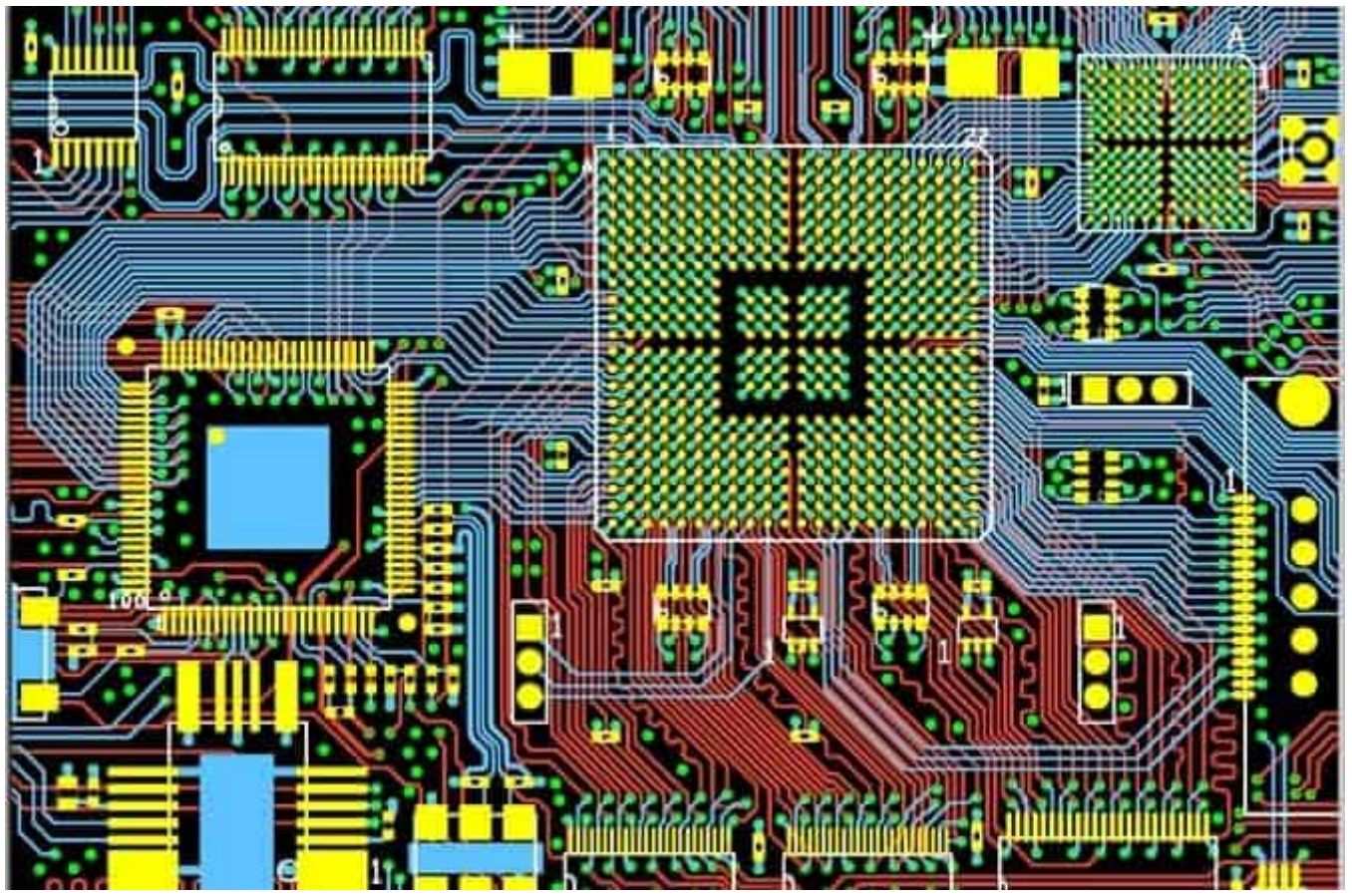


13 Basic Rules Of PCB Layout



1. According to the circuit module layout, the relevant circuit for realizing the same function is called one module, and the components in the circuit module should adopt the principle of near concentration, the digital circuit and the analog circuit are separated at the same time.

2. Do not mount the electronic components within 1.27mm around the non-mounting holes,such as tooling hole and standard holes.

Do not mount components within 3.5mm (for M2.5) and 4mm (for M3) around the mounting holes such as screws;

3.Avoid placing holes under the components such as the horizontally mounted resistors, inductors (insers), electrolytic capacitors, etc., so as to avoid short-circuiting the vias and component housings after wave soldering;

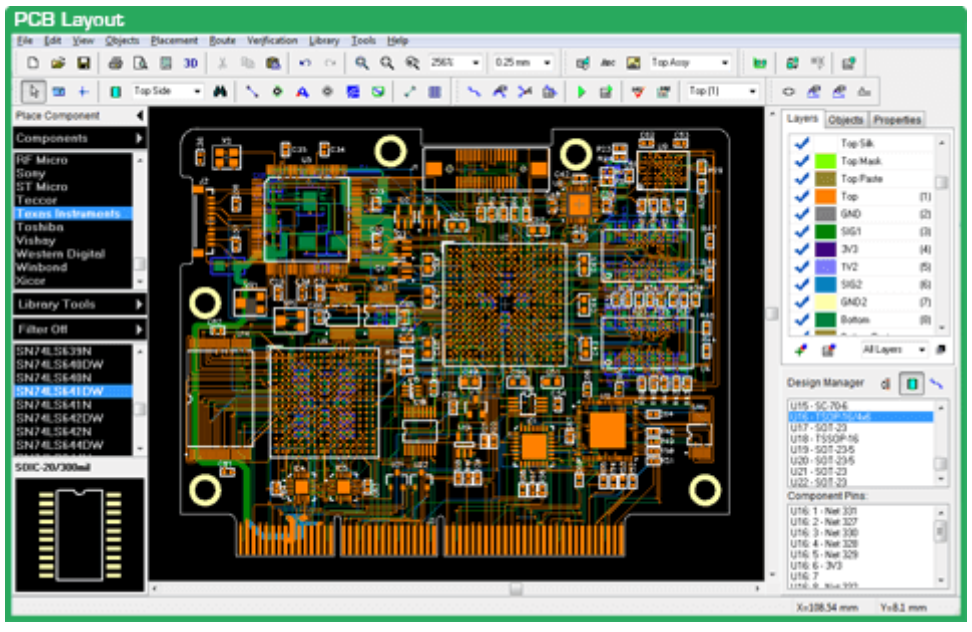
4.The distance between the outer side of the component and the edge of the board is 5mm;

5. The outer side of the mounting component pad and the outer side of the adjacent interposing component are greater than 2 mm

6.Metal shell components and metal parts (shield boxes, etc.) can not touch other components, can not be close to the printed lines, pads, the spacing should be greater than 2mm. Positioning holes, fastener mounting holes, oval holes And the other side of the square hole outside the plate is larger than 3mm.

7.Heating componentscannot be in close proximity to the wire and the heating-sensitive components; the high-heating components should be evenly distributed;

8.The power socket should be placed around the printed circuit board as much as possible. The power socket and the bus bar terminal connected to it should be placed on the same side. Special care should be taken not to place the power socket and other solder connectors between the connectors to facilitate the connection. These sockets, connector soldering and power cable design and cable tie. The spacing between the power socket and the solder connector should be considered to facilitate the plugging and unplugging of the power plug (RayMing www.raypcb.com)



9. Other components:

All IC components are unilaterally aligned, and the polarity of the polar components is clearly marked. The polarity on the same printed board shall not be more than two directions. When two directions are present, the two directions are perpendicular to each other

10. the board surface wiring should be properly dense, when the difference between the density is too large, it should be filled with mesh copper foil, the grid is larger than 8mil (or 0.2mm);

11. There should be no through holes on the chip pads,avoid the solder joints from being lost due to solder paste loss. Important signal lines are not allowed to pass through the socket pins

12. the patch is unilaterally aligned, the characters are in the same direction, and the package direction is consistent

13. Polarized componentsshould be as consistent as possible in the direction marked by the polarity on the same board.

II.Rules of components wiring

1. Draw wiring in the area where the wiring area is ≤1 mm from the edge of the PCB board and within 1 mm around the mounting hole.

2. The power cord should be as wide as possible, should not be lower than 18 mil; signal line width should not be lower than 12mil; cpu input and output should not be lower than 10mil (or 8mil); line spacing is not less than 10mil;

3. The normal through hole should not less than 30mil;

4. Double in-line: Pad in 60mil, Aperture in 40mil;

1/4W resistor: 51*55mil (0805 surface mount);60mil when in-line of the PAD, aperture is 42mil;

Infinite capacitance: 51*55mil (0805 surface mount); 50mil in line of the PAD, aperture is 28mil;

5. The power line and ground should be as radial as possible, and the signal line can not appear loopback.

Related Posts:

1. **Basic Rules of Layout and Components Wiring**
2. **Antenna Design and RF Layout Rules (Part I)**
3. **5 Golden Rules for PCB Design**
4. **4 Rules of Designing Circuit Width and Spacing**

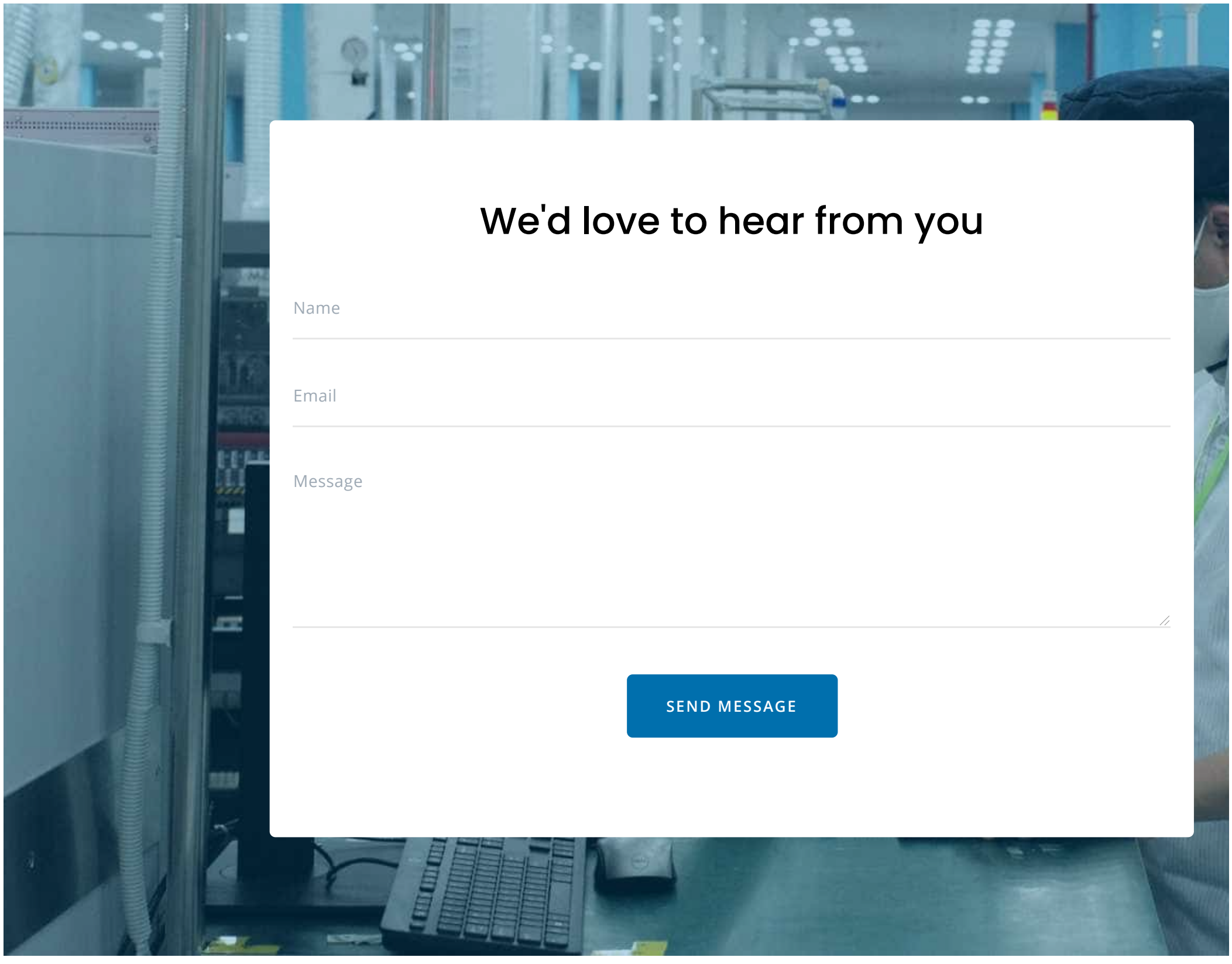
Recommended Posts

14 Layer PCB

Basic Skills about PCB etch

What is Nan Ya PCB?

Two types Soldering Dip and SMD



We'd love to hear from you

Name

Email

Message

SEND MESSAGE

Capabilities

Rigid PCB Capacity

Prototype PCB Assembly

Multilayer PCB

Rigid flex PCB

PCBA Test

Flexible PCB

Contact

12#,2nd Fu'an Industrial city,Dayangtian Zone Fuyong Street,Bao'an Shenzhen, 518103 China

sales@raypcb.com

RayMingPCB

+86-755-27348087

Connect with us

Certifications

Delivery