FOUR-LAYER PCB

Four-layer pcbs are very commonly used type of pcb especially when more routing space is needed in the pcb or signal integrity is required in the board. And mostly used in low-density digital and RF PCBs. In this report will discuss the common types of 4-layers pcbs and the use of each of them and the advantages and disadvantages for each type.

1.types of four layer pcb :

- **Two Internal GND Planes**

In this type the inner two layers of the pcb board is used as ground layers and power can be placed in the surface layers using large copper pours. These is very efficient stack up when routing is needed on both sides of the pcbs.

- **Two External GND Planes**

In this type the two external layers or the pcb board is used as ground and the inner layers is used for signals . this stackup is not effective for high speed routing because the problem of cross talk between the internal layers but it is effective for low-noise systems like analog system that requires low noise to operate well.

- **Signal-ground-power-signal**

in this type there is a dedicated power layer and dedicated signal layer and it is useful when we have a high power and many signals in the pcb design and this is required in power electronics applications for example.

- **Signal-grond-grond-power**

This stack up is used when the design needs large rails of power and the ground place is needed to support some signals. This is used when the design have low signal count so the signals can just fit in one layer , and the dedicated power layer can provide high current.

2.applications of four-layer pcb arrangements:

For the first type of stack up (INTERNAL GND PLANES) the most common application is high speed digital circuits like

- SDRAM
- DDR
- USB
- HDMI
- LVDS
- Ethernet

For the second type of stack up (EXTERNAL GND PLANES) the most common usage is shielding and improving signal integrity like :

- FPGA
- IOT-DEVICES
- DC-DC CONVERTERS
- MICROPROCESSIORS BOARDS

For the third type of stack up (**Signal-ground-power-signal**) the most common usage is creating balance between performance, electromagnetic interference and signal integrity in applications like :

- **Radio frequency**
- **Analog & digital signals**

3.Advantages and disadvantages for each type :

(1).**Signal-Ground-Signal-Power**

Advantages:

- good noise shielding and less interference
- simple routing design
- good signal integrity

Disadvantages:

- higher noise in power plane

(2). **Signal-Power-Ground-Signal**

Advantages:

- good ground isolation
- efficient power distribution
- good signal integrity

Disadvantages:

- higher noise in power plane
- Complex Design

(3). **Internal Ground Planes**

Advantages:

- Controlled Impedance
- Better Crosstalk Prevention
- Enhanced Power Integrity

Disadvantages:

- Difficulty in Manufacturing
- Limited Heat Dissipation