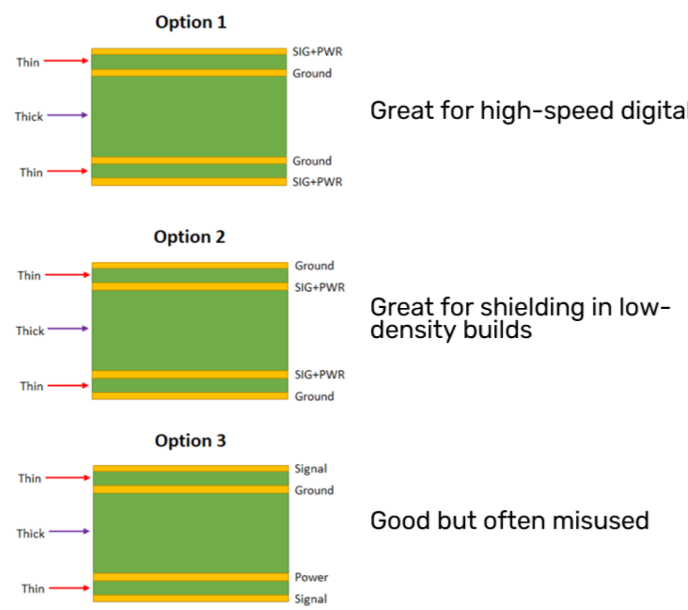
Multi-layer board

1. Introduction:

Among multi-layer stackups, four-layer stackups are probably the most common type of PCB stackup. Four-layer stackups allow for high-speed routing, integration of high power, and multiple signal layers where needed. In instances where routing islands need to be carved out above different ground or power regions in the PCB, four-layer stackups give you enough space with ground planes to implement such strategies

2. Types of Four-Layer PCB Stackups:

a. **SIG+PWR/GND type stackups**
b. **SIG/GND/PWR/SIG stackups**
c. **SIG/GND/SIG + PWR/GND stackups**



These stackups all have some common characteristics, which are related to how four-layer stackups are produced by manufacturers:

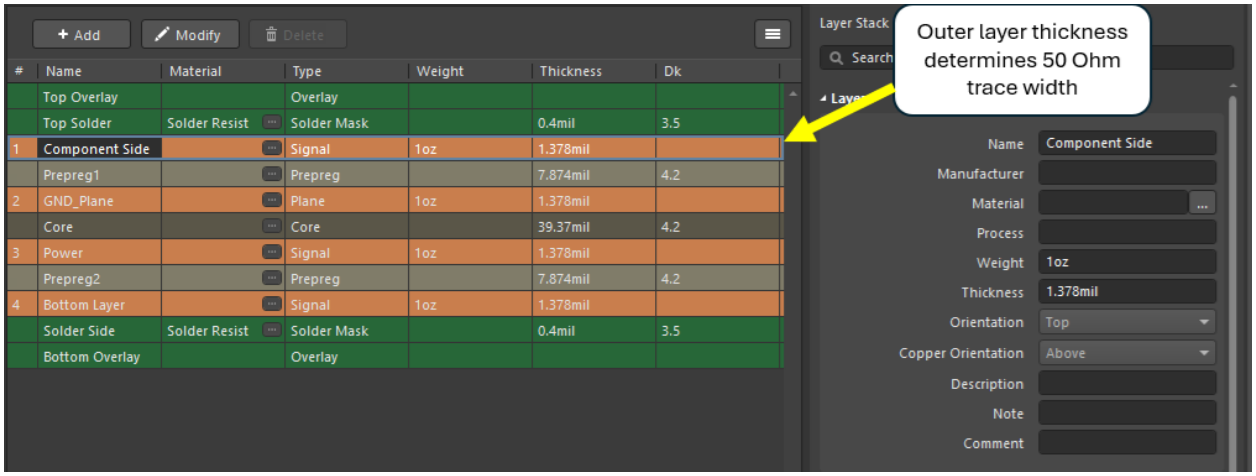
- They have a thick core layer and thin outer prepreg layers

- The core layer is normally copper clad so that it can be easily used to form planes
- The outer layers are almost never HDI build-up layers
- The core and prepreg materials could have any Tg value
- The materials could be low-loss FR4 or PTFE laminates

Using Each Layer in the Common Four-Layer Stackups:

Signal and ground as they are placed together in the four-layer stackup shown above allow for these PCBs to be used for many high-speed digital systems. When a ground plane is placed next to a signal layer, it is possible to design traces to have specific impedance values, which is based on the trace width and the layer thickness. This is a requirement for most high-speed digital interfaces which are faster than SPI.

Digital interfaces that can be easily supported on a four-layer PCB stackup include SDRAM, DDR, USB, HDMI, LVDS, and Ethernet. These interfaces require traces to have specific impedance (either single-ended or differential or both). To size the traces appropriately, the dielectric thickness is adjusted; thinner layers give a narrower trace in order to hit the impedance requirement.



Applications:

Smartphones and Tablets: The compact size and high functionality of these devices often require a four-layer PCB to manage power distribution and maintain signal integrity for high-speed data transmission.

Wearable Devices: Devices like smartwatches and fitness trackers use four-layer PCBs to accommodate their small form factor while ensuring reliable performance.

Infotainment Systems: The complex circuits in car infotainment systems, which handle audio, video, and connectivity features, benefit from the enhanced performance of a four-layer PCB.

Advanced Driver Assistance Systems (ADAS): These systems rely on four-layer PCBs for reliable signal processing and power management in safety-critical applications.

Advantages:

Improved Signal Integrity:

The additional layers in a 4-layer PCB, particularly the dedicated ground and power planes, provide better control over impedance and reduce signal noise and crosstalk. This is essential for high-speed circuits where signal integrity is critical.

Enhanced EMI Performance:

The presence of ground planes close to the signal layers helps in shielding and reduces electromagnetic interference (EMI). This is especially important in applications where minimizing noise and interference is crucial, such as in RF circuits or sensitive analog designs.

Better Power Distribution:

A dedicated power plane ensures stable voltage distribution across the board, reducing voltage drops and improving the overall reliability of the circuit. This is particularly beneficial in designs with high power demands.

Thermal Management:

The internal layers of a 4-layer PCB can also help in heat dissipation, improving the thermal management of the board. This is useful in applications where heat buildup is a concern, such as in power electronics.

Disadvantages:

Higher Cost:

The manufacturing process for a 4-layer PCB is more complex and costly compared to 2-layer boards. This can be a significant factor in cost-sensitive projects, particularly for low-volume production.

Complexity in Design and Manufacturing:

Designing a 4-layer PCB requires more careful planning, especially in terms of layer arrangement, impedance control, and via placement. This complexity can lead to longer design cycles and potentially higher chances of errors if not managed properly.

Limited Flexibility in Layer Arrangement:

While a 4-layer PCB offers more routing space than a 2-layer board, it still has limitations compared to higher-layer boards (e.g., 6-layer or 8-layer). Designers must carefully consider how to allocate signal, ground, and power layers to optimize performance, which can be restrictive depending on the design requirements.