

PCB STACKUPS: LAYERS, MATERIALS, IMPEDANCE

A stackup is the sandwich of copper and insulator the board is built from. Two layers is the cheap default; four adds dedicated power and ground planes. The fab sets the thicknesses.

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

A stackup is the sandwich of copper and insulation the board is built from. Two layers is the cheap default. Four or more lets you dedicate internal layers to power and ground, which makes a quieter board. The fabricator sets the exact thicknesses, and you design to their stackup.

COPPER, PREPREG, AND CORE

A board is layers of copper foil bonded to an insulating substrate. The rigid inner insulator is the core; the sheets that bond layers together under heat and pressure are prepreg. The insulator itself is usually **FR-4**, a woven glass-epoxy laminate that is cheap, strong, and stable enough for almost everything you will build.

TWO LAYERS VERSUS FOUR

A two-layer stackup is copper, insulator, copper: signals and ground share both sides. A four-layer stackup is signal, ground, power, signal, so the two inner planes give every outer signal a return plane a fraction of a millimetre away. Tighter spacing between a signal and its return plane means lower loop inductance, which is the whole reason to add layers.

A FOUR-LAYER STACKUP: COPPER FOILS SEPARATED BY PREPREG AND A CORE, WITH DEDICATED INNER PLANES.

CONTROLLED IMPEDANCE, AND WHEN IT MATTERS

For fast signals like USB or high-speed data, a trace and its reference plane behave like a transmission line with a characteristic impedance set by the trace width, the distance to the plane, and the insulator. When a signal is fast enough that this matters, you ask the fab for a controlled-impedance stackup and route those traces to the width they specify. For low-speed hobby work it rarely comes up, and knowing the term tells you when to reach for it.

THE FAB SETS THE STACKUP

You do not invent the thicknesses. Each fabricator publishes a stackup table, the exact copper weights and insulator heights they build, and you design to it. A four-layer order from a low-cost fab comes with a standard stackup you can look up before you route, so your impedance and via choices match what they will actually make.

DEEP DIVE · WHY TIGHTER PLANE SPACING WINS

The loop inductance of a signal and its return falls as the two get closer, because inductance grows with the area of the loop between them. In a four-layer stackup the fabricator can put a signal layer only a few tenths of a millimetre from its reference plane, far closer than the full board thickness that separates the two sides of a two-layer board. Less loop area means less inductance, less ringing, and less radiated noise, which is why dense or fast designs justify the extra layers and cost (Altium).

- [Altium. The right way to use power planes in a 4-layer PCB stackup \(signal, ground, power, signal and prepreg spacing\).-resources.altium.com](#)

CHECKPOINT**1. What does a four-layer board add over a two-layer board?**

- A second silkscreen color
- Dedicated internal power and ground planes**
- A larger board outline

ANSWER · B

The inner planes give signals a nearby return, lowering loop inductance.

2. Who sets the exact layer thicknesses in a stackup?

- The soldering iron temperature
- The schematic editor's defaults
- The fabricator, in their published stackup table**

ANSWER · C

You design to the fab's stackup; they build fixed copper and insulator heights.

3. Controlled impedance mainly matters for what?

- Fast signals like USB and high-speed data**
- Slow power rails
- The silkscreen legend

ANSWER · A

Fast traces behave like transmission lines; their impedance depends on the stackup.

- Prerequisite: grounds and power rails
- Next: design rules and DRC