

GROUND AND POWER PLANES ON A REAL LAYOUT

On a real board, ground is a filled copper plane rather than a trace. A plane gives every return a low-inductance path under its signal. Pour it, keep it unbroken, and stitch it with vias.

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

On a real board, ground is a filled sheet of copper, a plane that covers a whole layer, rather than a single thin trace. A plane gives every signal's return current a low-inductance path directly beneath it, which is what keeps a board quiet. Pour a ground plane, keep it unbroken, and stitch it with vias.

WHY A PLANE BEATS A TRACE

Return current follows the path of least impedance, and at any real speed that path is the copper directly under the signal trace, not a far-off thin wire. A continuous plane offers that path everywhere, so the loop between a signal and its return stays tiny and radiates almost nothing. This is the return-path idea from the fundamentals, now poured in copper (All About Circuits).

- [All About Circuits. How to use return paths for better PCB design \(why a plane gives a lower-impedance return than a trace\).](#) allaboutcircuits.com

KEEP THE PLANE WHOLE

A plane only works while it is continuous. Route a trace across the ground layer and you cut a slot in it, and every return current that needed to cross that slot has to detour around the end, making a large loop that radiates and picks up noise. So keep signals off the ground layer, and where one must cross the plane, give its return a way across.

STITCH IT WITH VIAS

On a two-layer board, ground exists on both sides, joined by stitching vias: plated holes that tie the two ground areas into one. Stitching keeps the whole ground at one potential and shortens the path a return takes to hop layers. Sprinkle stitching vias across the board, and especially wherever a signal changes layers.

POWER POURS AND THERMAL RELIEF

A power rail can be poured as a plane too, giving it a low-resistance spread to every part that taps it. Where a plane connects to a pad it uses a thermal relief, a few short spokes instead of solid copper, so the pad can still be soldered without the whole plane sinking the heat away from the iron.

THE RETURN CURRENT MIRRORS THE SIGNAL IN THE PLANE RIGHT BENEATH IT; STITCHING VIAS TIE THE GROUND TOGETHER.

DEEP DIVE · GOING TO FOUR LAYERS

A two-layer board shares its ground plane with signals. A quieter board uses four or more layers so the fabricator can dedicate whole internal layers to ground and power, sandwiched close to the signal layers. That tight spacing drops the loop inductance further, and it frees the outer layers for routing. It is the same principle as the ground plane, built into the structure of the board (Altium).

- [Altium. The right way to use power planes in a 4-layer PCB stackup \(dedicated internal ground and power planes\).-resources.altium.com](https://resources.altium.com)

CHECKPOINT**1. Why pour a ground plane instead of running a thin ground trace?**

- a. It gives return current a low-inductance path right under the signal**
- b. It uses less copper overall
- c. It makes the board cheaper to fabricate

ANSWER · A

A continuous plane keeps every return loop tiny, which keeps the board quiet.

2. What happens if you route a signal trace across the ground plane?

- a. Nothing, the plane is unaffected
- b. It cuts a slot that forces returns on a long detour**
- c. The plane starts carrying more current

ANSWER · B

A slot in the plane breaks the return path; currents detour around it and radiate.

3. Stitching vias do what?

- a. Add decoration to the silkscreen
- b. Increase the board thickness
- c. Tie ground areas together into one low-impedance plane**

ANSWER · C

They join ground copper across layers so the whole ground sits at one potential.

- [Prerequisite: grounds and power rails](#)
- [See it on a real board: the L1.01 build](#)
- [Next: PCB stackups, layers, materials, impedance](#)