LIBRARY · REFERENCE GUIDE

ROUTING TRACES: WIDTH, CURRENT, AND VIAS

A trace is a wire in copper. Its width sets the current it carries (IPC-2221) and a via moves it between layers. Size power traces from current, keep signals short. With a live calculator.

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

A trace is a wire made of copper printed on the board. Its width sets how much current it can carry before it overheats, and a via is a plated hole that carries a trace to another layer. Size power traces from their current, keep signal traces short and direct, and change layers with a via only when you must.

TRACE WIDTH COMES FROM THE CURRENT

The current a trace can carry rises with its cross-section, which for a fixed copper thickness means its width. The IPC-2221 standard turns a target current and an allowed temperature rise into a minimum copper cross-section, from which the width follows once you know the copper weight. The calculator below implements it: give it the current and the temperature rise, and it returns the width.

$$A = (I / (k \times dT^{0.44}))^{(1 / 0.725)}$$

Here `A` is the copper cross-section, `I` the current, and `dT` the temperature rise you allow; `k` is a constant that is larger for an outer-layer trace than an inner one. Divide the cross-section by the copper thickness and you have the trace width.

CALCULATOR · PCB TRACE WIDTH CALCULATOR (IPC-2221)

Find the minimum trace width for a current at a chosen temperature rise (IPC-2221), for external or internal copper. Grounded in a real high-current board.

Interactive calculator: academy.onethousanddrones.com/tools/pcb-trace-width

FIND THE MINIMUM TRACE WIDTH FOR A CURRENT AT A CHOSEN TEMPERATURE RISE (IPC-2221).

COPPER WEIGHT AND VOLTAGE DROP

A trace's thickness is its copper weight, usually `1 oz` per square foot, about 35 micrometres of copper. Heavier copper carries more current in the same width. A long trace also has resistance, so a big current down a thin track drops voltage along the way and wastes it as heat. Widen a power trace and that drop falls.

VIAS MOVE A TRACE BETWEEN LAYERS

A via is a plated hole that connects copper on one layer to copper on another. Use one when a trace has to cross to the other side to get past an obstacle. A via has resistance and inductance of its own, and it carries limited current, so a high-current path uses several vias in parallel rather than one. Keep signal detours through vias few, because each one is a small discontinuity (KiCad).

- KiCad. PCB Editor (Pcbnew) documentation: the interactive router, track width, and vias. docs.kicad.org

WIDER COPPER CARRIES MORE CURRENT; A VIA CARRIES A TRACE TO ANOTHER LAYER.

DEEP DIVE · EXTERNAL VERSUS INTERNAL TRACES

IPC-2221 gives an external trace on an outer layer a larger current constant than an internal one, because an outer trace sheds heat to the air while an inner trace is buried in insulation. In the calculator that shows up as the external setting allowing a narrower trace for the same current than the internal setting. On a four-layer board, a power net routed on an inner layer has to be wider than the same net carried on the surface.

CHECKPOINT

1. What mainly sets how wide a power trace must be?

a. The color of the soldermask

b. The number of parts on the board

c. The current it carries

ANSWER · C

Wider copper carries more current before overheating; IPC-2221 sizes it from the current and the allowed temperature rise.

2. What is a via for?

a. Labeling the trace on silkscreen

b. Carrying a trace to another copper layer

c. Storing charge like a capacitor

ANSWER · B

A via is a plated hole connecting copper between layers.

3. To carry a large current through vias, you should do what?

a. Use several vias in parallel

b. Use one very small via

c. Avoid vias entirely on the power net

ANSWER · A

Each via carries limited current, so parallel them on a high-current path.

- Prerequisite: power and heat
- Calculate it: the PCB trace width calculator
- Next: ground and power planes on a real layout