This Javascript web calculator calculates the trace width for printed circuit board conductors for a given current using formulas from IPC-2221 (formerly IPC-D-275).

Inputs:

Current	1	Amps
Thickness	4	oz/ft^2

Optional Inputs:

Temperature Rise	10	Deg C
Ambient Temperature	25	Deg C
Trace Length	1	inch

Results for Internal Layers:

Required Trace Width	0.195	mm
Resistance	0.0164	Ohms
Voltage Drop	0.0164	Volts
Power Loss	0.0164	Watts

Results for External Layers in Air:

Required Trace Width	2.96	mil
Resistance	0.0427	Ohms
Voltage Drop	0.0427	Volts
Power Loss	0.0427	Watts

Notes:

The trace width is calculated as follows:

First, the Area is calculated:

Area[mils^2] = (Current[Amps]/(k*(Temp_Rise[deg. C])^b))^(1/c)

Then, the Width is calculated:

Width[mils] = Area[mils^2]/(Thickness[oz]*1.378[mils/oz])

For IPC-2221 internal layers: k = 0.024, b = 0.44, c = 0.725

For IPC-2221 external layers: k = 0.048, b = 0.44, c = 0.725

where k, b, and c are constants resulting from curve fitting to the IPC-2221 curves

Disclaimer:
These calculations are believed to be correct, but not guaranteed. Use at your own risk!

Trace Width Calculator FAQs

QUESTION: Very cool PCB width tool! I would like to know its limits though. I entered a 65 amp current requirement and it returned a track width that must be incorrect.

ANSWER: The original graphs that this tool is based on (published in IPC-2221) only cover up to 35 Amps, up to 0.4 inches of trace width, from 10 to 100 degrees C of temperature rise, and copper of 0.5 to 3 ounces per square foot. The formulas used here will simply extrapolate when the values are outside of these ranges.

QUESTION: I used your PCB trace width calculator. Intuitively I would say the required internal trace width would be less than the external case since the external trace can peal off; the opposite is true according to the calculator???? Why?

ANSWER: In air, the external layers have better heat transfer due to convection. A good heat insulator blankets the internal layers, so they get hotter for a given width and current. Since the Trace Width Calculator tries to control the *temperature rise* of the traces, it makes the internal traces wider. In vacuum, or in a potted assembly, you should use the *internal* layer guidelines even for the *external* layers.

QUESTION: What does *temperature rise* mean and how does it apply?

ANSWER: *Temperature rise* means how much hotter the trace will get with current flowing in it compared to without. You have to decide how much temperature rise your board can handle based on the operating environment and the type of PWB material used. Ten degrees is a very safe number to use for just about any application. If you can live with the trace width required for a ten-degree rise, you are good to go. If you want to try to skinny up the traces, ask for 20 degrees of temperature or more.

QUESTION: I use "wagon wheels" or "spokes" when connecting to a ground plane to make it easier to solder to. The trace width calculator is telling me to make the "spokes" so wide that it defeats the purpose. What should I do?

ANSWER: The wagon wheels spokes are very short length traces and are heat sunk to the plane. The trace width calculator uses empirical formulas based on long traces with no special heat sinking. Generally, the wagon wheel spokes do not have to be anywhere near as wide as long traces. However, at this time, I don't know of a good way to do calculations for wagon wheel spokes.

QUESTION: What are Mils?

ANSWER: A Mil is 1/1000 of an inch.

This calculator is hosted for 4pcb.com by circuitcalculator.com
Email questions or comments about this calculator to:

info@circuitcalculator.com