JLCPCB Help Center

Search

Get Started

Ordering

EDA Tool Help

Technical Capabilities

• Technical Guidelines

PCB Assembly

Shipping & Delivery

Payment & Invoice

Certificates

API Solution

Privacy & Policies

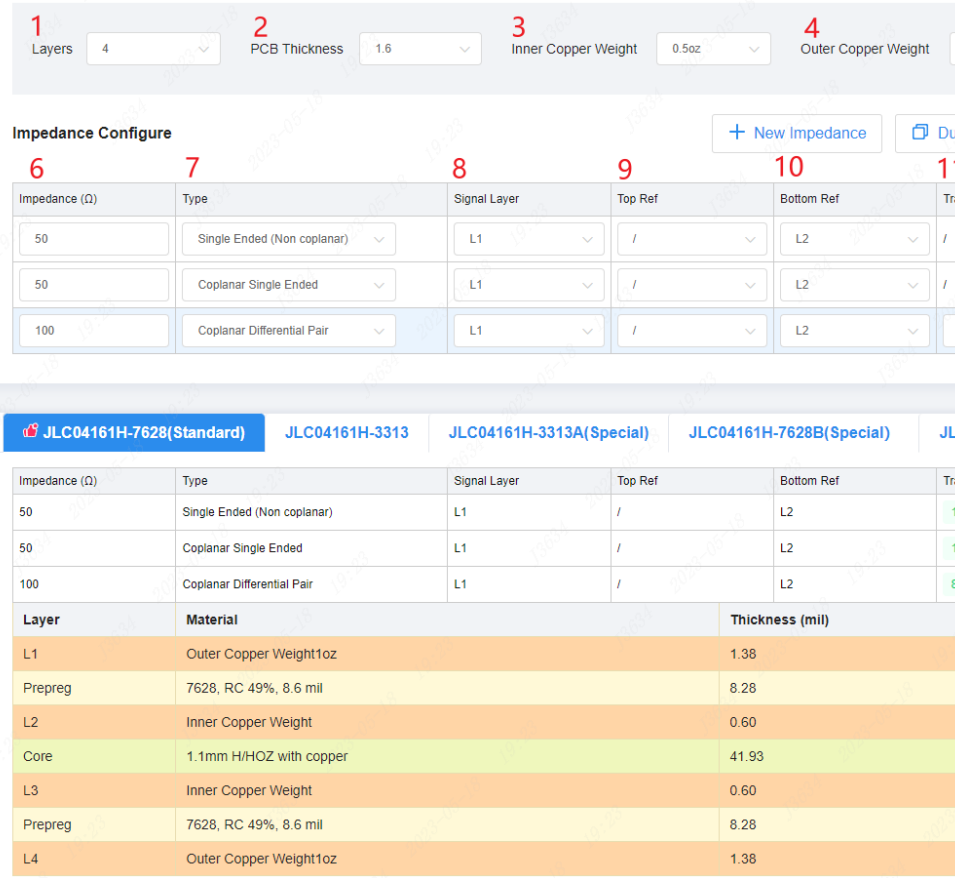
Activity&Bonus

Help Center / Technical Guidelines / User Guide to the JLC…

User Guide to the JLCPCB Impedance Calculator

The JLCPCB Impedance Calculator computes track width values and recommended stack-ups from user-input values target impedance, conductor spacing (for edge-coupled pairs), and conductor-to-ground gap (coplanar waveguides).

User Interface



1. **Layers**: Total number of copper layers in the board.
2. **Finished Thickness**: Thickness of the finished PCB, including soldermask.
3. **Internal Copper Thickness**: Copper weight for internal layers. The calculator only supports 0.5 oz and 1 oz; for ca contact customer support.
4. **External Copper Thickness**: Copper weight for the top and bottom layers. Although JLCPCB can produce boards supports 1 oz because (1) the wider tolerance of 2 oz traces hinders impedance control, and (2) most signal traces or
5. **Unit**: Options are mm, mil, µm, and inch.
6. **Impedance (Ω)**: The desired impedance. The range of accepted values is 20 to 90 Ω for single-ended and 50 to 15
7. **Type**: Options are microstrip (single-ended), coplanar (single-ended), edge-coupled (differential), and dual coplana selected before changing this option.
8. **Layer**: Which layer the signal should be on.
9. **Reference Layer Above**: Only needed if the signal is on an internal layer.
10. **Reference Layer Below**: Does not need to be one layer below the signal; for example for a signal on L3, this opti ground plane associated with the signal on the reference layer selected.
11. **Conductor Spacing**: The gap between the two conductors in a differential trace. Increasing conductor spacing in
12. **Conductor-to-Ground Gap**: The gap between the signal conductor and the surrounding ground in a coplanar wa

Notes

· The first stack-up recommended by the calculator usually has the lowest cost and quickest turn-around.
· The 3313 prepreg replaces the previously available 2313. Their thickness and dielectric constant (ε_r) are the same.
· For the same nominal copper weight, internal layers have a slightly smaller thickness than external layers due to a s during deoxidation etc. For example, 0.5 oz copper on internal layers is 15.2 µm thick 4- to 8-layinstead of the nomina

· Results for 4- to 8-layer boards are calculated assuming Nan Ya Plastics NP-155F core material, and SYTECH S10(correct material should be selected when ordering so that the calculated parameters give accurate impedances.

Calculation Parameters Used

External copper thickness (1 oz)	1.6 mil
Internal copper thickness (0.5 oz)	0.6 mil
Internal copper thickness (1 oz)	1.2 mil
Base soldermask thickness	1.2 mil
Copper-surface soldermask thickness	0.6 mil
Soldermask thickness in between traces	1.2 mil
Soldermask dielectric constant (εr)	3.8
Trace top width	Trace base width – 1 mil

Nan Ya Plastics NP-155F (4 to 8 layers)

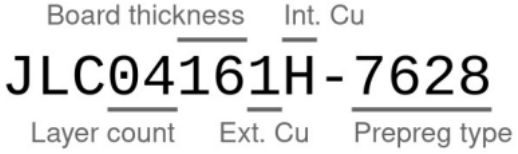
Core Thickness	εr	Prepreg Type	Resin Content	Nominal Thickness	εr
0.08 mm	3.99	7628	49%	8.6 mil	4.4
0.10 mm	4.36	3313 (2313)	57%	4.2 mil	4.1
0.13 mm	4.17	1080	67%	3.3 mil	3.91
0.15 mm	4.36	2116	54%	4.9 mil	4.16
0.20 mm	4.36				
0.25 mm	4.23				
0.30 mm	4.41				
0.35 mm	4.36				
0.40 mm	4.36				
0.45 mm	4.36				
0.50 mm	4.48				
0.55 mm	4.41				
0.60 mm	4.36				
0.65 mm	4.36				
0.70 mm	4.53				
> 0.70 mm	4.43				

SYTECH (Shengyi) S1000-2M (10+ layers)

Core Thickness	εr	Prepreg Type	Resin Content	Nominal Thickness	εr
0.075 mm	4.14	106	72%	1.97 mil	3.92
0.10 mm	4.11	1080	69%	3.31 mil	3.99
0.13 mm	4.03	2313	58%	4.09 mil	4.31
0.15 mm	4.35	2116	57%	5.00 mil	4.29
0.20 mm	4.42				
0.25 mm	4.29				
0.30 mm	4.56				

The values provided above are for reference only. Some of the values have been calculated from test results by JLCF

Stack-Up Naming Scheme



Last updated on May 18, 2023

Was this article helpful? Yes No