PCB Via Calculator

March 12, 2006

This Javascript web calculator calculates the resistance, voltage drop, and power loss of printed circuit board vias. Note that vias are made out of plated copper which typically has a resistivity of 1.7E-6 to 2.2E-6 Ohm-cm. The calculator has an input box for the resistivity which defaults to 1.9E-6 Ohm-cm.

Updates:
May 22, 2006 - Added thermal resistance calculation.
January 19, 2007 - Minor Clarifications.
March 28, 2007 - Updated resistivity. See comment 12.
June 21, 2007 - Added estimated ampacity. See comment 17.

Inputs:

Finished Hole Dia	10	mil
Plating Thickness	1.4	mil
Via Length	60	mil

Optional Inputs:

Applied Current	1	Amps
Plating Resistivity	1.9E-6	Ohm-cm

Electrical Results:

Resistance	0.000895	Ohms
Voltage Drop	0.000895	Volts
Power Loss	0.000895	Watts
Estimated Ampacity	2.26	Amps

Thermal Results:

Thermal Resistance	117	Deg. C/Watt

Notes:
Resistance = Resistivity*Length/Area

Recent Posts

Area = pi*(Inner_dia + Plating_thk)*Plating_thk
Resistivity = 1.9E-6 Ohm-cm (plated copper)
(plated copper is much more resistive than pure copper)
Copper Thermal_Resistivity = 0.249 cm-K/W (at 300K)
Est_Ampacity [Amps] = k*(Temp_Rise [deg C])^b*(Area [mils^2])^c
For IPC-2221 external layers: k = 0.048, b = 0.44, c = 0.725

References:
[1] "Constructing Your Power Supply - Layout Considerations", by Robert Kollman
http://focus.ti.com/lit/ml/slup230/slup230.pdf
[2] "Current Carrying Capacity of Vias", by Doug Brooks and Dave Graves
http://www.ultracad.com/articles/viacurrents.pdf

Posted in : Calculators, PCB Author : Brad

Comments

1. The CircuitCalculator.com Blog » PCB Trace Width Calculator - March 21, 2006

[…] PCB Trace Width Calculator January 31, 2006 This Javascript web calculator calculates the trace width for printed circuit boards based on a curve fit to IPC-D-275. Also see the via calculator. New features: […]

2. The CircuitCalculator.com Blog » PCB Via Thermal Resistance - May 22, 2006

[…] By popular demand, I've also added thermal resistance calculation to the PCB Via Calculator. Thanks for the good suggestions! […]

3. Jay Wang - June 25, 2006

I found an article describing the via vs. current from the following link. It seems helpful. What is your opinion ?

www.ultracad.com/articles/viacurrents.pdf

4. Administrator - June 26, 2006

Jay,

It is an excellent reference. It basically recommends making the cross sectional area of the via equal to the trace. This seems like a sound principal. ~~One thing that it does not mention is that the resistivity of the plated copper used in the vias is higher than copper foil used in the traces due to porosity and impurities in the plated copper. This is taken into account in the via resistance calculator above.~~ (See comment 12 below regarding the strikeout.) However, since the via is generally small compare to the trace to which it is connected, the trace will act as a good heat sink for the via. It therefore still seems reasonable to follow the

- Save Your Back by Using a High Desk
- Schematic and PCB Design Process
- 3-Phase Calculations
- Wire Parameter Calculator
- Hex, Decimal, and Binary Converter
- Skin Effect Calculator
- Four-Layer High-Speed PCB Design
- PCB Stack-Up Design And Impedance Calculator
- Effects Of Corners In PCB Traces
- Transmission Line Calculator

Categories

- Uncategorized
- Announcements
- Calculators
  - Electrical Engineering
  - Health
  - PCB
- Excel
- VBA
- Simulation & Modeling
  - SPICE
- Reviews

Links

- ChipSight
- ExcelVBA
- LTspice
- OhmArt
- Power Supply Design

FEEDS FULL COMMENTS

recommendation in the UltraCAD article.

Brad

5. Michael Howell – July 27, 2006

Is there a Z-axis Via thermal strain or failure calculator? That would be useful, too.

6. The CircuitCalculator.com Blog » PCB Trace Resistance Calculator – August 24, 2006

[…] PCB Trace Resistance Calculator January 24, 2006 This Javascript web calculator calculates the trace width for printed circuit boards based on a curve fit to IPC-2221 (formerly IPC-D-275). Also see the via calculator. New features: […]

7. Arthur Matteson – January 23, 2007

I am skeptical of the 6e-6 ohm-cm value for resistivity of plated copper. I have been doing some copper plating experiments of my own, including thickening 1oz to 15oz and making large vias (by writing inside the hole with a pencil to make a seed for copper). My measurements indicate a value of 2.2e-6 ohm-cm for unannealed or 1.9e-6 ohm-cm for annealed (400F, 45 minutes). Even with a +/-30% error, this is much closer to what would be expected. Electroplating is often said to be used to purify copper, so at least with the use of the proper chemicals, the resistivity should be around 2e-6 ohm-cm.

http://www.thinktink.com/stack/volumes/volvi/copplate.htm

http://www.reed-electronics.com/semiconductor/article/CA6329028

http://www.copper.org/innovations/1997/Dec1997/wiremetallurgy.html

8. Brad – January 23, 2007

Arthur,

Thanks for the good information. The source of the 6e-6 ohm-cm figure was the Robert Kollman article referenced above. I will look into this some more and probably will end up at least making the resistivity a variable in the via calculator.

Brad

9. Arthur Matteson – January 23, 2007

Well here's my TI application note:

http://www.egr.msu.edu/~mattes12/appnote3.pdf

:-)

10. Brad – January 24, 2007

Nice app note!

11. Jet lew – March 25, 2007

thank you from china

12. Brad – March 28, 2007

It was verified that the stated resistivity for plated copper in the above referenced article “Constructing Your Power Supply – Layout Considerations” was too high. Therefore the resistivity used by the calculator was changed from 6E-6 to 1.9E-6 Ohm-cm to reflect more realistic values seen in PCBs today. The resistivity was also made into an input box so it can be changed by the user if desired.

Brad

13. Ron B – May 2, 2007

Any further thoughts about adding inductance and capacitance to the via calculator?????

14. Brad – May 2, 2007

Ron B,

Adding inductance and capacitance to the via calculator is a good idea. However, the inductance and capacitance are determined as much by the space between the via and surrounding ground planes as by the via’s geometry. So, in order to calculate the inductance and capacitance, one has to provide a user interface to describe the complete geometry of all other planes that come close to the via. I will see if I can come up with something for vias similar to my Transmission Line Impedance Calculator.

Brad

15. Ron B – May 15, 2007

Brad,
re: via L and C
Have you seen the via editing portion within Hyperlynx? If not, let me know and I can send you a few screenshots?

16. Brad – May 16, 2007

Ron,

Sure, the send screen shots if you can.
Brad

17. Brad – June 21, 2007

I added an **estimate** of the via's ampacity based on the article in reference [2] above and IPC-2221 guidelines. **It should be stressed that no detailed studied were done and this is only an estimate so please read all the following disclaimers**. My approach is to assume that the via is equivalent to a trace of the same cross-sectional area, and then apply the ampacity guidelines of IPC-2221. In calculating the ampacity, I chose to use the equation for external traces and a conservative temperature rise allowance of 10 degree C. For these assumptions to hold, the board should have at least one full copper plane and the via should be connected to a trace or plane that can also handle the current. Also be aware that temperature rises stack up for all heat sources in close proximity. So, if you put a bunch a vias close together, each one may contribute a 10 degree C rise if running at the estimated ampacity.

18. Rob – January 3, 2008

I don't think the formula for the cross sectional area of the via is correct. Wouldn't it be the area given by the outer diameter (inner diameter + plating thickness) minus the area given by the inner diameter?

19. Brad – January 3, 2008

Hi Rob,

I double checked, and with a little algebra, my equation for the via's cross sectional area is equivalent to what you are saying.

Brad