DISK91.COM – TECHNOLOGY BLOG

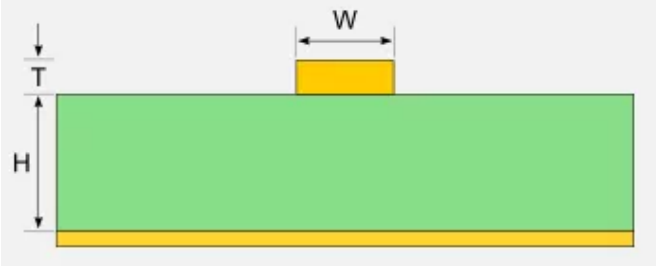
TECHNOLOGY, HARDWARE, SECURITY, HACKING (DIY), IOT

Design a 50 ohm impedance microstrip line for RF signals

For RF signal you must draw a 50 ohm net between chip and antenna to get the best performance.This post is the result of my web research on this topic …

There are two ways to design your emitter to antenna solution:

- A microstrip line : basically you have a net on the top driving the signal and a ground plane on the PCB bottom



MICROSTRIP LINE

- A coplanar wave guide : your signal is drive by a PCB net on top with two ground plane area on its left & right. If you also have a ground plane on the bottom it's a grounded coplanar wave guide

The way to get a 50 ohm communication way is different depending on your choice.

COMMON CONSIDERATION

The impedance of the line is mainly depending on your PCB characteristics. The most important are:

- PCB Material

The PCB material are standardized with different norms. One of the most common is FR4. It defines range for most of the parameter we need to know for the wire size calculation. The problem is range are large and the calculation will vary depends on where your PCB is in the given range.

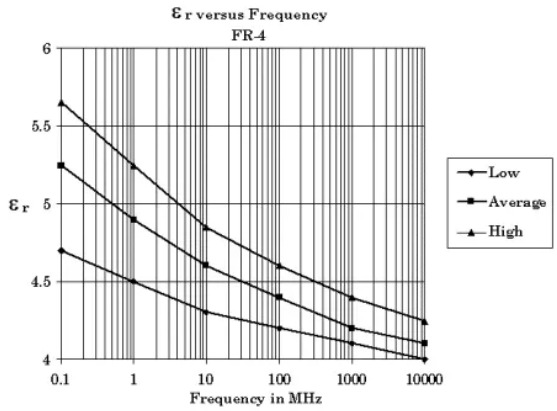
- The Thickness of dielectric = the PCB height.

  This is why we found RF pcb : they are particularly thin as this is allowing to make smaller RF wires. This can be a variable parameter depending on your PCB provider sourcing and these variation may impact you radio quality.

- Relative Dielectric Constant (ε_r)

  It determines how much electrostatic energy can be stored per unit of volume when unit voltage is applied and basically how the PCB act as a capacitor (as this definition is really simplified you can learn more here)

  This constant depends on the material and the frequency of the communication. For a FR4 dielectric.



SOURCE : HTTP://WWW.CADESIGN.NET/ARTICLES/RELATIVE_PERMITTIVIT

For a 433MHz line it will be between 4.2 and 4.5 (assume 4,35) ; for 868MHz 4.15 and 4.4 (assume 4,30). (I did not found more precise information on Eurocircuit website for non RF pool circuit)

MICROSTIP LINES

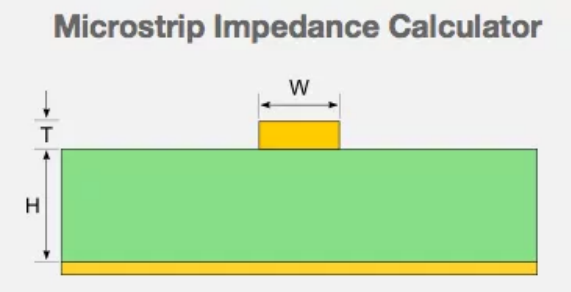
The impedance depends on the net width, the net height, the board type and so one. There are multiple tools on Internet to calculate the net width to use. Two of them is the following one :

- Calculate your microstrip transmission line
- Calculate your microstrip transmission line

Its gives you the impedance based on different parameters

- Height (H)
- Width (W)
- Thickness (T)
- Substrate Dielectric (Er)

These elements refer to the following diagram :



On this list, many are constants and depends on the PCB making process and components. So you will find the elements on your PCB provider website.

- For Eurocircuit these information are given in the price calculator interface. The standard Height is 1.55mm ; the Thickness once finished is 35um.
- For Seeed you can choose from 0,6 to 3mm. Thickness is from 1oz (0,035mm) to 3oz (0,1 mm) – See this calculator for oz to mm conversion.

Here is a table of W depending on the other standard parameters (I choose the one acceptable in term of price)

@868MHz – Er 4,3	H(mm)	T(mm)	W (mm)
	0,8	0,035	1,52
	1,0	0,035	1,90
	1,2	0,035	2,29
	1,55	0,035	2,97
	1,6	0,035	3,06
RF Pool – Er 3,45	0,5	0,035	1,10

The main difficulty is to be able to create a trace up to 3 mm large when you have as a source and destination pads around 1mm and in the middle you have to integrate capacitors & inductor CMS with width about 0.8mm. We also have to take into account that a CMS will make the Thickness to change.

As a complement you can read this paper with larger theory.

COPLANAR WAVE GUIDE

Grounded CoPlanar Waveguide have an advantage to not take the wire Thickness as a parameter. The other parameter are like defined previously.

@ 868 Mhz – Er 4,3	H(mm)	S(mm)	W(mm)
	0,8	0,254	1,08
	1,0	0,254	1,23
	1,2	0,254	1,35
	1,55	0,254	1,52
	1,6	0,254	1,54
RF Pool – Er 3,45	0,5	0,254	0,94

As we can see with Grounded coplanar waveguide the width of the track is really thinner than with Microstrip and the gap between min and max have a lower range. As a consequence the tolerance to small variation is better.

You can calculate your wire size with this calculator :

- (Grounded) coplananar wave guide calculator
- coplanar waveguide calculator

As a complement you can read this website with larger theory

SHARE :

Twitter Facebook 1 Google LinkedIn Pinterest Tumblr Print

RELATED POSTS:

1. **miniVNA Tiny Plus review** I've got my miniVNA Tiny+ this summer and start making test with it. A VNA is a Vector Network Analyzer....
2. **Take a look at SigFox radio signals with airspy usb software defined radio** I recently bought an AirSpy mini SDR (Software defined radio) dongle. Understand, it is a radio scanner you connect on...
3. **Choose your smd (MLCC) capacitor** There are different type of smd capacitor ; the one I mainly use are mlcc – multi layer ceramic capacitor....
4. **Design a raspberry Pi HAT shield** When you want to design a shield for Raspberry Pi, you have to follow the HAT standard. This standards describes...

5 RESPONSES TO *DESIGN A 50 OHM IMPEDANCE MICROSTRIP LINE FOR RF SIGNALS*

Vijendra N *says:*
24 JANUARY 2018 AT 13:00

I Am using 2.4GHz Antenna, they have recommended 50ohm impedance, so can anyone tel me that how do I calculate the following Height (H),Width (W), Thickness (T) and Substrate Dielectric (Er)

REPLY

Paul *says:*
31 JANUARY 2018 AT 20:03

I propose you clic on one of the two link in the blog post ...

REPLY

Vertul Verma *says:*
4 JUNE 2018 AT 12:16

How is it that the length of the microstrip does not play a role in the impedance? Considering the trace to have some minimal resistance too, doesn't the length have any impact?

REPLY

Paul *says:*
5 JUNE 2018 AT 14:16

I'm not an expert but I think the following : a network cable is 50 Ohm whatever the length is.

REPLY

luke *says:*

19 JUNE 2018 AT 20:00

The calculators are for impedance not resistance, resistance does effect the rf output but it only attenuates the signal.

REPLY

This site uses Akismet to reduce spam. Learn how your comment data is processed.