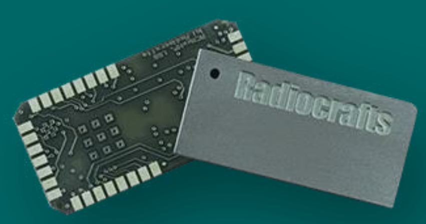
Radiocrafts
Embedded Wireless Solutions

Fast to Market. Proven Quality.

AN061: RF PCB layout recommendations

By: Thomas S. Knutsen

2022-10-20



RF PCB LAYOUT RECCOMENDATIONS

By Thomas S. Knutsen

Recommendations on RF PCB pattern for Radiocrafts modules

When designing the PCB layout for the RF modules, there are some measures that need to be taken to get the best performance from the radio module. In this application note, we will suggest some best practices for your product to succeed. You will also find examples of PCB layout in some of our reference designs posted on our website.

At Radiocrafts we are always available to do a board layout review if you need someone to look over your design before production.

A bit about PCB materials

When designing RF traces on a PCB, it's important to know the specifications of the PCB. These are:

- ***ε_r Relative Permittivity** usually between 2.2 and 5. This parameter changes with frequency and composition of the material. For FR4 based laminates this value is usually between 3.8 and 4.7, depending on weave pattern and thickness. Relative Permittivity may also be named Dielectric constant.*
- ***Tan δ Loss tangent**. Usually between 0.05 and 0.001. This parameter is a measure of how much loss there is in the material. Low number means low loss. For most sub 1GHz applications we do not compensate for Tan δ*
- ***PCB thickness** of the isolation (dielectric) part of the PCB between the two metal layers.*

All these parameters are available from your PCB manufacturer up on request.

PCB stack-up and ground planes

A PCB consists of one or more layers of copper, bonded to an isolation layer. This isolation layer can vary in thickness depending on the PCB thickness and type of material.

For the transmission lines outlined in this application note, it is important to have a good ground plane, bonded together with several vias to reduce the inductance between the ground layers.

The module should have a ground plane layer that all the ground pads are connected to, and this should be connected to the ground layer with several vias.

A common type of 4-layer PCB has the following stack-up:

As can be seen, removing layer 2 around the transmission line increases the distance to the ground plane, and makes the width of the transmission line longer. [4]

PCB transmission line types

RF modules and antennas are normally designed to have an impedance of 50Ω. To keep this impedance and connect the antenna to the module without loss, we need to use transmission lines with the same impedance. Any mismatch in the impedance, either in the transmission line or the antenna, will give a reflected wave and lead to less efficiency in the radio link.

There are 3 common types of transmission lines when designing a PCB:

- *Stripline*
  - *Requires the conductor to be embedded into the middle of the material, necessitating 3 layers or more in the board.*
  - *Used when putting transmission lines in the inner layers of multilayer PCB.*
  - *Gives low loss and high bandwidth.*
  - *Radiation is contained in the material of the PCB.*

- *Microstrip*
  - *Requires large areas without any traces to avoid loss.*
  - *Size is dependent on ε and PCB thickness.*
  - *Has a large field around the transmission line, giving higher loss.*

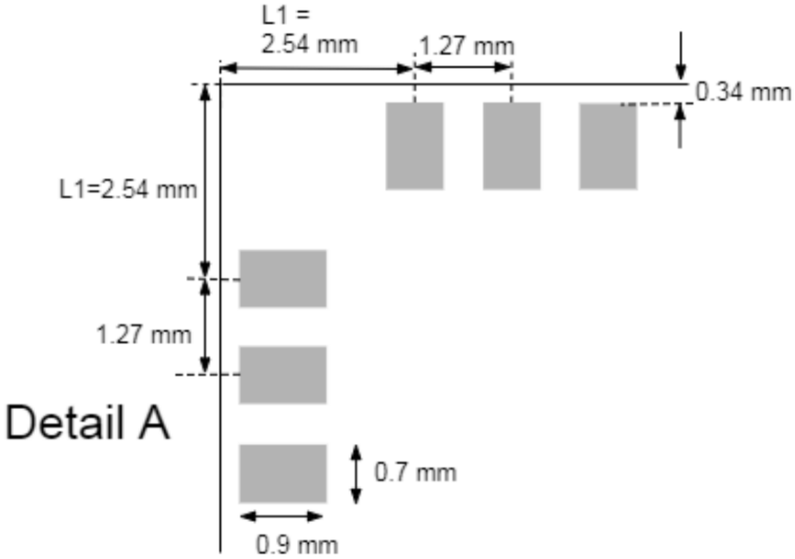
- *Coplanar waveguide (CPW) with ground*
  - *Controls the impedance with ground pour on both top side and bottom side of the PCB.*
  - *Requires control of processing tolerance at the PCB manufacturers*
  - *Distance between transmission line and ground pour is critical for impedance*
  - *Less critical on PCB thickness and ε_r.*

For these transmission lines, the width of the transmission line is a factor of the distance to the ground layer below it. If the width is too small, move the ground plane to a layer further down in the stack-up and remove the layer just below the transmission line.

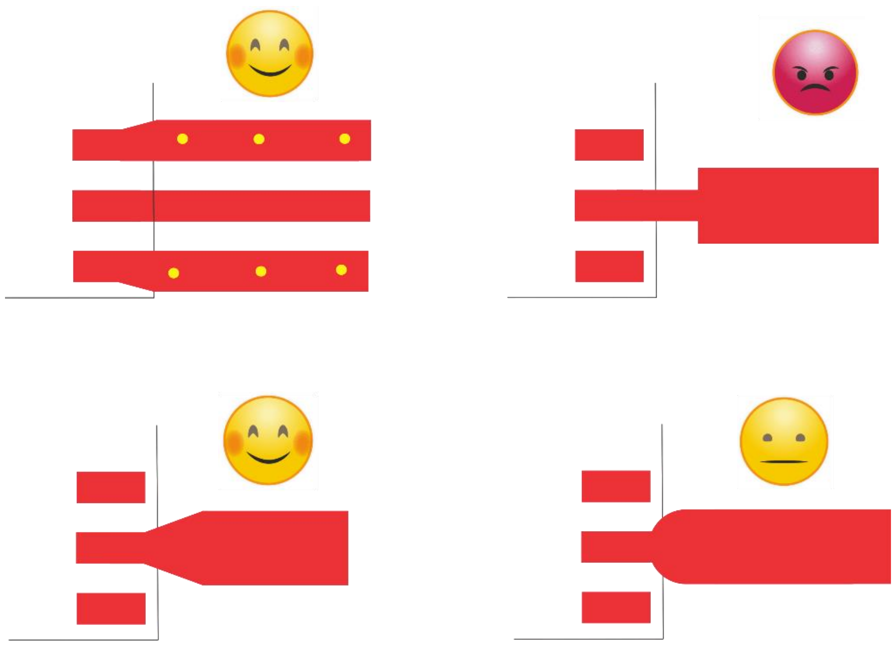
We would recommend using Coplanar waveguide and using an online calculator to calculate the parameters. Examples of this are in the references section [1].

Interface module – Transmission line

The interface between the module and the transmission line has the potential to make an impedance mismatch. The recommended PCB pattern for modules is found in the datasheet. The example here has the RF output on a 0.7mm X 0.9mm pad, on a 1.27mm grid. This lends itself well to using 0.7mm Coplanar waveguide.

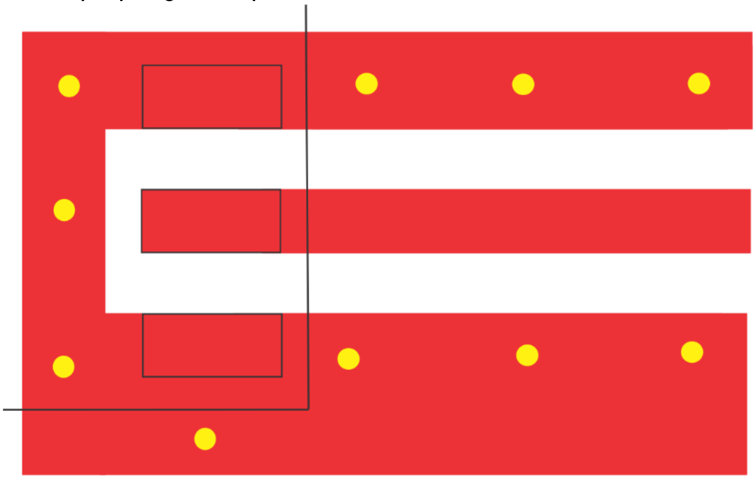


If there is a need to use wider CPW then this interface should be done with a taper or curved taper onto the module. A sudden change in width means a sudden change in transmission line impedance. This will give reflections, lower output power and less than optimal performance. [2]



Module grounding

It is important that the module is properly grounded with several vias to the ground layer around the transmission line. A proper ground pour should be used around the module.



Vias inside the pad may lead to reliability issues when soldering the module in place. The recommended approach is to have the vias outside the solder pad.

Corners and vias

When laying out the transmission line, the corners should be mitered (cut off the corner to remove some capacitance) to avoid reflections.

As an alternative, it is possible to use a curved line, with a radius larger than 3 times the line width. In practice this will work just as well as a mitered bend. [3]

Vias in the RF line should be avoided as these introduce inductance and lead to uncontrolled impedance variations in the feedline. An approximation of via inductance can be calculated by:

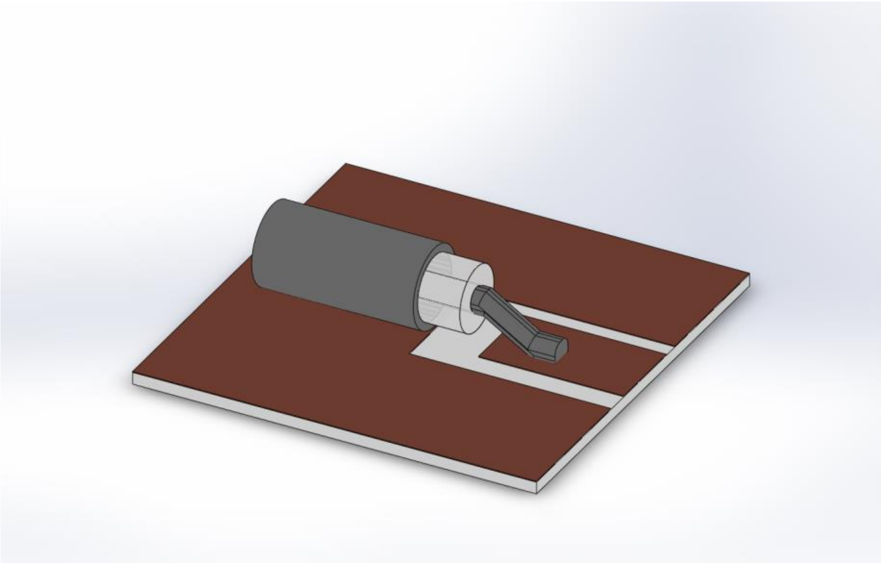
$$L_{Via} = 5h\left(\ln\left[\frac{2h}{r}\right] + 1\right) \quad L_{via} in\, nH$$

Where h = PCB thickness and r = via diameter.¨

Several vias will reduce the inductance and improve preformance if a via is needed.

Coax interface

When interfacing to a coaxial cable, a connector is recommended. SMA and UFL connectors are commonly used. If a connector needs to be avoided for cost reasons, the coax should be soldered in such a way that the transmission line continues into the coax. For a coax, the shield should be soldered to the ground for a distance of at least 2 times the diameter of the coax to contain the ground currents. The wave in a Coaxial cable travels on the inside of the braid and the termination needs to be done properly to avoid reflections.



References

[1] Coplanar Waveguide with ground calculator: : [https://chemandy.com/calculators/coplanar-waveguide-with-ground- calculator.htm](https://chemandy.com/calculators/coplanar-waveguide-with-ground-calculator.htm)
[2] Matthaei, Young and Jones, Microwave filters, Impedance-Matching networks, and Coupling Structures, Artech 1963, section 5.07 page 199.
[3] Microwaves101.com on mitered bends: https://www.microwaves101.com/encyclopedias/mitered-bends
[4] On the drawings Red are copper traces, Yellow is vias, Grey is PCB isolation layer. For the 4-layer boards, base material is drawn with a darker grey than prepreg, to show the difference.

Document Revision History

Document Revision	Changes
1.0	First release

Disclaimer

Radiocrafts AS believes the information contained herein is correct and accurate at the time of this printing. However, Radiocrafts AS reserves the right to make changes to this product without notice. Radiocrafts AS does not assume any responsibility for the use of the described product; neither does it convey any license under its patent rights, or the rights of others. The latest updates are available at the Radiocrafts website or by contacting Radiocrafts directly.

As far as possible, major changes of product specifications and functionality, will be stated in product specific Errata Notes published at the Radiocrafts website. Customers are encouraged to check regularly for the most recent updates on products and support tools.

Trademarks

RC232™ is a trademark of Radiocrafts AS. The RC232™ Embedded RF Protocol is used in a range of products from Radiocrafts. The protocol handles host communication, data buffering, error check, addressing and broadcasting. It supports point-to-point, point-to-multipoint and peer-to-peer network topologies.

All other trademarks, registered trademarks and product names are the sole property of their respective owners.

Life Support Policy

This Radiocrafts product is not designed for use in life support appliances, devices, or other systems where malfunction can reasonably be expected to result in significant personal injury to the user, or as a critical component in any life support device or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness. Radiocrafts AS customers using or selling these products for use in such applications do so at their own risk and agree to fully indemnify Radiocrafts AS for any damages resulting from any improper use or sale.

Radiocrafts Support:

Knowledge base: https://radiocrafts.com/knowledge-base/
Application notes library: https://radiocrafts.com/resources/application-notes/
Whitepapers: https://radiocrafts.com/resources/articles-white-papers/
Technology overview: https://radiocrafts.com/technologies/
RF Wireless Expert Training: https://radiocrafts.com/resources/rf-wireless-expert-training/

Contact Radiocrafts

Sales requests: https://radiocrafts.com/contact/