

RF LAYOUT

Application Guide

Version: RF LAYOUT_Application Guide_V2.2

Date: 2016-02-26

Table of contents

Document History2 Table of
Contents 3 Picture
Index...4
1 introduction.....5
2 RF Interface Schematic Recommendations.....6
3 50 Ohm characteristic impedance coplanar waveguide structure design..... 7
4 Coplanar Waveguide PCB LAYOUT Examples and Precautions9

image index

Figure 1: Module RF Reference Schematic 6

Figure 2: Schematic diagram of coplanar waveguide structure 7

Figure 3: Schematic diagram of two-layer board LAYOUT..... 8

Figure 4: Schematic diagram of four-layer board LAYOUT (the reference ground is the third layer)..... 8

Figure 5: Schematic diagram of four-layer board LAYOUT (the reference ground is the fourth layer)..... 8

Figure 6: Example diagram of PCB LAYOUT..... 9

1 Introduction

This document mainly introduces the PCB routing precautions of the module's peripheral RF circuit to help customers to correctly carry out when using the Quectel module. RF part PCB layout design to ensure RF

performance, reducing customer design cycles. This document applies to all Quectels

LTE modules.

Support Bands : LTE Band 2/4/5/12

2 RF Interface Schematic Recommendations

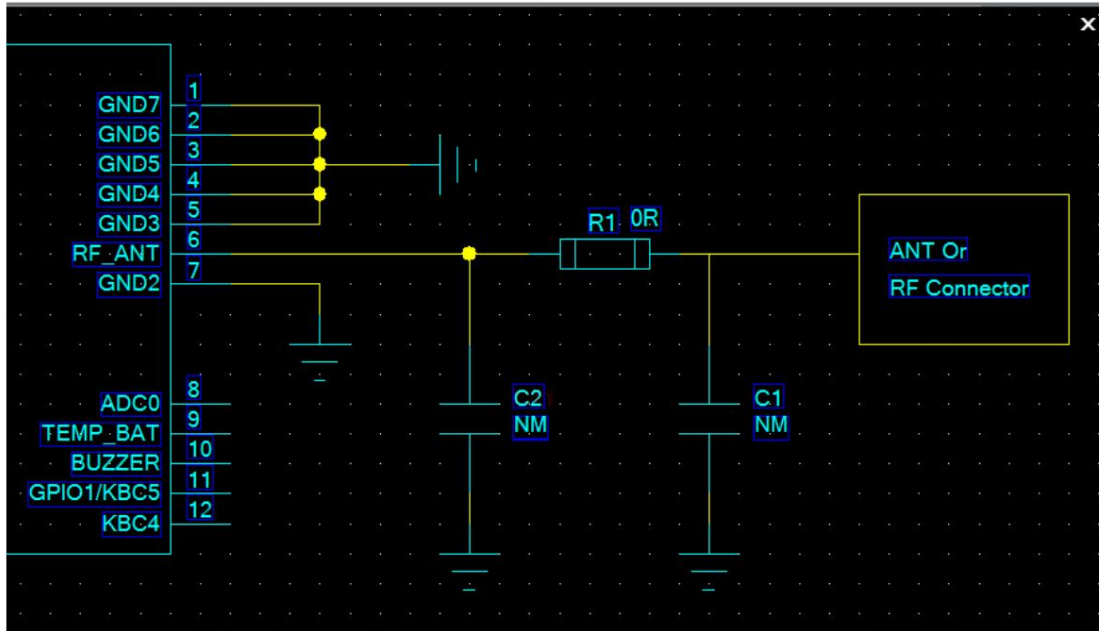


Figure 1: Module RF Reference Schematic

Among them, C1, R1, and C2 are reserved matching circuits to optimize the performance of the antenna port. The default values are: R1 is pasted with 0R resistor; C1 is not pasted; C2 is not pasted.

3 50 Ohm Characteristic Impedance Coplanar Waveguide Structure Design

It is recommended to use the coplanar waveguide microwave transmission form, and the characteristic impedance is 50 ohms, as shown in Figure 2.

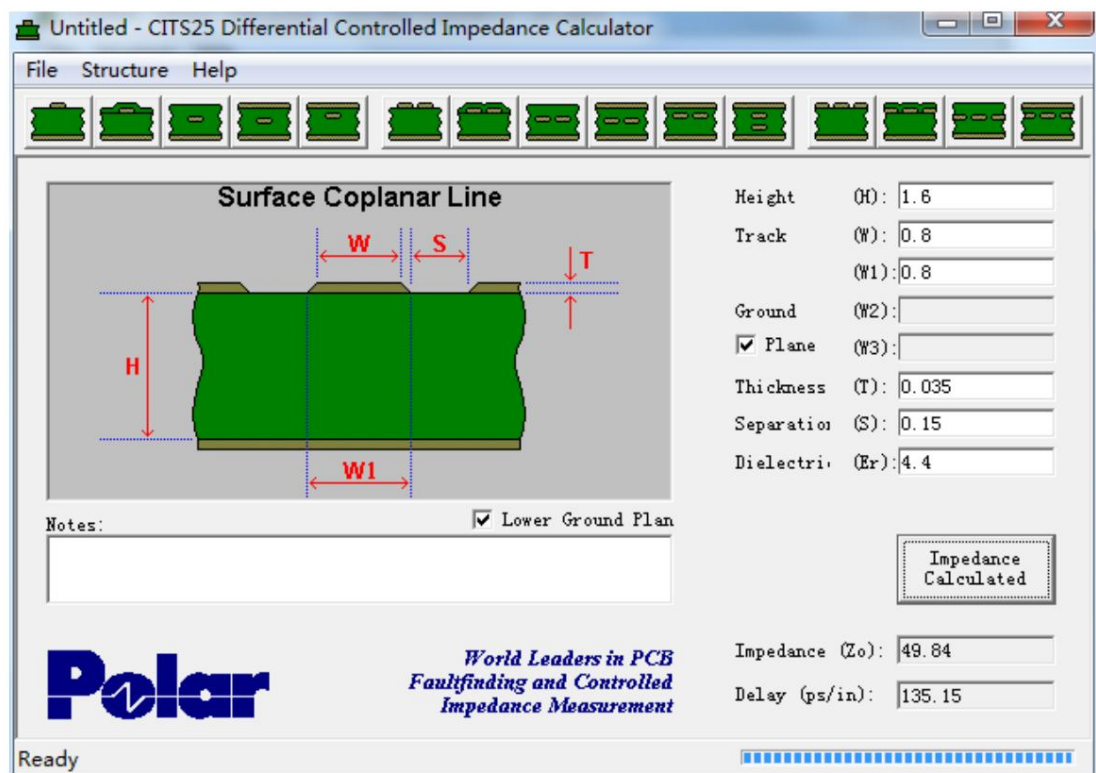


Figure 2: Schematic diagram of the coplanar waveguide structure

The main factors affecting the characteristic impedance of coplanar waveguides are the dielectric constant of the substrate (usually 4.4), the distance between the signal layer and the reference ground H, the line width W, the ground clearance S, and the copper thickness T. Table 1 lists the distance H between different signal layers and the reference ground. When the thickness of copper skin is T=0.035mm, the line width W corresponding to the characteristic impedance of 50 ohms and the recommended value of the ground clearance S:

Table 1: 50 ohm coplanar waveguide line width and recommended distance to ground corresponding to the distance between different signal layers and reference ground

Distance H between signal layer and reference ground	Line width W	Ground clearance S
0.076mm	0.1188mm	0.15mm
0.1mm	0.1623mm	0.2mm
0.15mm	0.24mm	0.2mm

0.8mm	0.8mm	0.18mm
1.0mm	0.8mm	0.17mm
1.2mm	0.8mm	0.16mm
1.6mm	0.8mm	0.15mm
2mm	0.8mm	0.14mm

If it is a 2-layer board, the signal layer is the Top layer, and the reference ground is the Bottom layer, as shown in Figure 3 below. If it is a 4-layer board, the reference ground can be the 2nd Layer, Layer 3 or Layer 4. If the reference ground is the third layer, the second layer directly under the signal layer should be forbidden, and the width of the forbidden area should be at least the signal line. 5 times the width, as shown in Figure 4 below. If the reference ground is the 4th layer, the 2nd and 3rd layers directly under the signal layer must be forbidden to be laid, and the width of the forbidden area shall be at least 5 times the width of the signal line, as shown in Figure 5 below. If it is more than 6-layer board and so on.

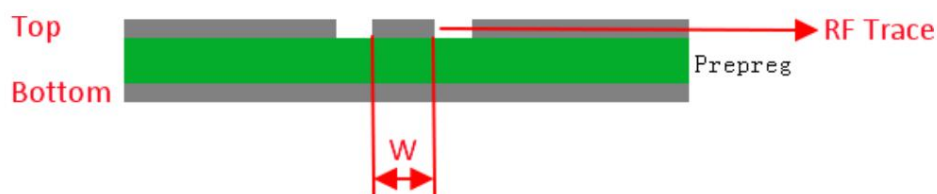


Figure 3: Schematic diagram of two-layer board LAYOUT

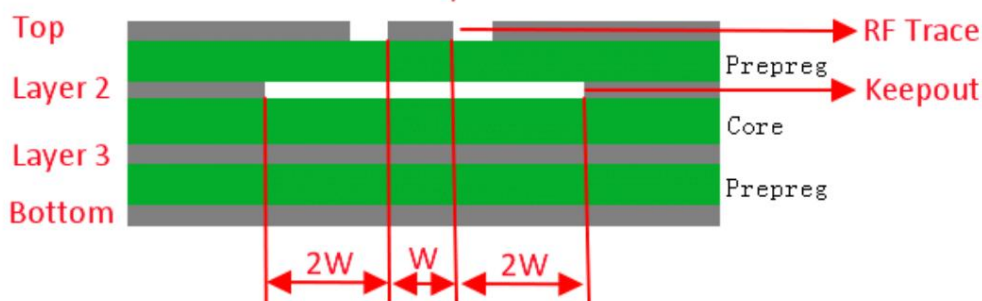


Figure 4: Schematic diagram of the four-layer board LAYOUT (the reference ground is the third layer)

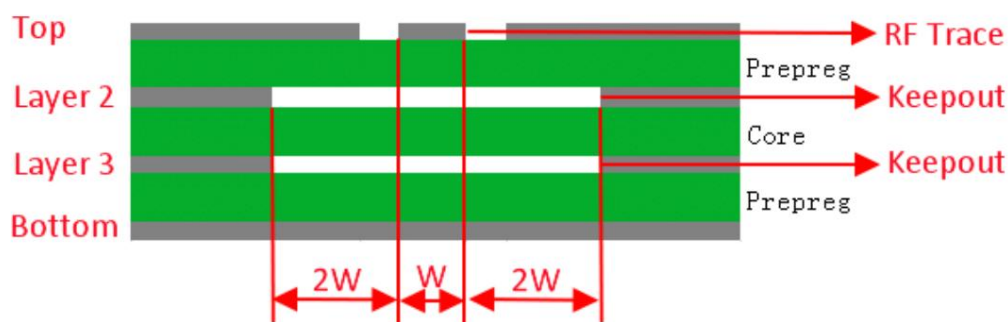


Figure 5: Schematic diagram of the four-layer board LAYOUT (the reference ground is the fourth layer)

4 Coplanar Waveguide PCB LAYOUT Examples and Precautions

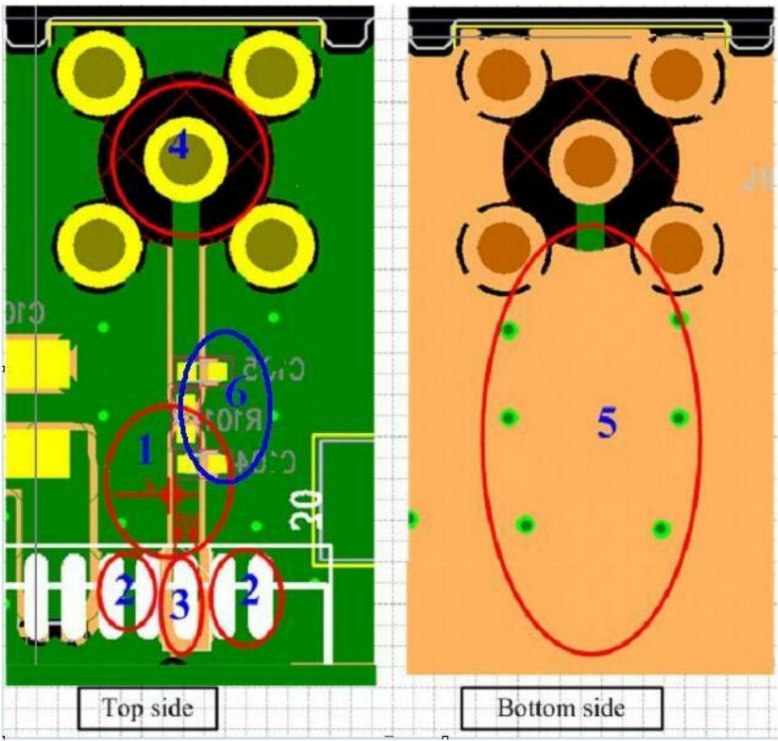


Figure 6: PCB LAYOUT

The example diagram combined with the numbers in the figure above, there are 6 points to be noted:

1. Control the line width W and spacing S corresponding to the 50 ohm coplanar waveguide. Take the common PCB board as FR4 medium (dielectric constant is 4.2) and the copper thickness is 35um as an example, the line width and ground clearance corresponding to the thickness between different signal layers and reference ground are shown in Table 1. It is especially reminded that the PCB board factory needs to control the precision of the line width W and the ground clearance S on both sides. 2. The ground PIN is not used as a thermal pad, and should be in full contact with the ground. 3. The surface layer is slightly forbidden to reduce parasitic effects. The RF line should be as short as possible, and it is best to avoid right-angle routing. It is recommended to go 135 degrees if there is a corner. 4. When connecting the device package, pay attention to keep the signal pins at a certain distance from the ground, please refer to Figure 6. If it is a mounted pad, dig the ground under the corresponding signal pin pad. 5. Ensure that the reference ground corresponding to the RF line is complete, add ground holes to help RF return, and keep at least 2 ground holes and signal lines.

The distance to multiply the line width. Ensure that the grounding area of the same layer of the RF line is as large as possible, and the reference ground on the other side is as complete as possible, and ensure that a certain amount of ground holes are connected to the two layers of ground.

6. Three components make up the type matching

Pi

circuit shown in Figure 1. When placing, the pad should be placed close to the antenna, as shown in the figure above. If the distance between the antenna connection end (SMA) and the RF PIN pin of the module is too short to fit the three pieces of the y-shaped circuit, it can be changed to an L-shaped matching circuit.

Warning & Statement:

This module meets the requirements of Part 15 Subpart B, FCC CFR Title 47 Part 2.

Integration is strictly limited to fixed categorized end-products where a separation distance of at least 20 cm between the radiating part and any human body can be assured during normal operating conditions.

This module only allows connection antenna in the instruction manual. If other antennas are used, re-evaluation is required.

This module is test stand-alone, if more another modules work together with this module, please evaluation the multiple RF exposure.

This device complies with Part 15 of the FCC Rules. Operation is subject to the following two conditions: (1) this device may not cause harmful interference, and (2) this device must accept any interference received, including interference that may cause undesired operation.

Changes or modifications not expressly approved by the party responsible for compliance could void the user's authority to operate the equipment.

NOTE: This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates, uses instructions, may cause harmful interference to radio communications. However, there is no guarantee that interference will not occur in a particular installation. If this equipment does cause harmful interference to radio or television reception, which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:

- Reorient or relocate the receiving antenna.
- Increase the separation between the equipment and receiver.
- Connect the equipment into an outlet on a circuit different from that to which the receiver is connected.
- Consult the dealer or an experienced radio/TV technician for help.

If the FCC identification number is not visible when the module is installed inside another device, then the outside of the device into which the module is installed must also display a label referring to the enclosed module. This exterior label can use wording such as the following: "Contains Transmitter Module FCC ID: 2A8PN-EC25"

This equipment complies with FCC radiation exposure limits set forth for an uncontrolled environment.

This equipment must be installed and operated with minimum distance 20cm between radiator & your body