

RF Layout

Application Note

Rev. RF_Layout_Application_Note_2.2

Date: 2016-02-26



Our aim is to provide customers with timely and comprehensive service. For any assistance, please contact our company headquarters:

Quectel Wireless Solutions Co., Ltd.

Office 501, Building 13, No.99, Tianzhou Road, Shanghai, China, 200233

Tel: +86 21 5108 6236

Mail: info@quectel.com

Or our local office, for more information, please visit:

<http://www.quectel.com/support/salesupport.aspx>

For technical support, to report documentation errors, please visit:

<http://www.quectel.com/support/techsupport.aspx>

Or Email: Support@quectel.com

GENERAL NOTES

QUECTEL OFFERS THIS INFORMATION AS A SERVICE TO ITS CUSTOMERS. THE INFORMATION PROVIDED IS BASED UPON CUSTOMERS' REQUIREMENTS. QUECTEL MAKES EVERY EFFORT TO ENSURE THE QUALITY OF THE INFORMATION IT MAKES AVAILABLE. QUECTEL DOES NOT MAKE ANY WARRANTY AS TO THE INFORMATION CONTAINED HEREIN, AND DOES NOT ACCEPT ANY LIABILITY FOR ANY INJURY, LOSS OR DAMAGE OF ANY KIND INCURRED BY USE OF OR RELIANCE UPON THE INFORMATION. ALL INFORMATION SUPPLIED HEREIN IS SUBJECT TO CHANGE WITHOUT PRIOR NOTICE.

COPYRIGHT

THIS INFORMATION CONTAINED HERE IS PROPRIETARY TECHNICAL INFORMATION OF QUECTEL CO., LTD. TRANSMITTABLE, REPRODUCTION, DISSEMINATION AND EDITING OF THIS DOCUMENT AS WELL AS UTILIZATION OF THIS CONTENTS ARE FORBIDDEN WITHOUT PERMISSION. OFFENDERS WILL BE HELD LIABLE FOR PAYMENT OF DAMAGES. ALL RIGHTS ARE RESERVED IN THE EVENT OF A PATENT GRANT OR REGISTRATION OF A UTILITY MODEL OR DESIGN.

Copyright © Quectel Wireless Solutions Co., Ltd. 2016. All rights reserved.

About the Document

History

Revision	Date	Author	Description
1.0	2011-08-08	David WEI	Initial
2.0	2014-10-16	Frank WANG	1. Used the new technical document template 2. Updated RF reference schematic diagram
2.1	2015-04-11	Frank WANG	Added applicable modules
2.2	2016-02-26	Frank WANG	Updated the example of PCB layout in Figure 6

Contents

About the Document.....	2
Contents.....	3
Figure Index.....	4
1 Introduction.....	5
2 RF Reference Schematic Diagram.....	6
3 Coplanar Waveguide Structure Design.....	7
4 Coplanar WG PCB Layout Example and Guidelines.....	9

Quectel
Confidential

Figure Index

FIGURE 1: RF REFERENCE SCHEMATIC DIAGRAM	6
FIGURE 2: STRUCTURE OF COPLANAR WG.....	7
FIGURE 3: TWO LAYERS PCB LAYOUT	8
FIGURE 4: FOUR LAYERS PCB LAYOUT (THIRD LAYER AS REFERENCE GROUND)	8
FIGURE 5: FOUR LAYERS PCB LAYOUT (FOURTH LAYER AS REFERENCE GROUND)	8
FIGURE 6: AN EXAMPLE OF PCB LAYOUT.....	9

Quectel
Confidential

1 Introduction

The aim of this document is to offer some design rules for RF PCB layout, which will be helpful for achieving good RF performance and minimizing design period.

This document is applicable to all Quectel GSM, WCDMA and LTE modules.

Quectel
Confidential

2 RF Reference Schematic Diagram

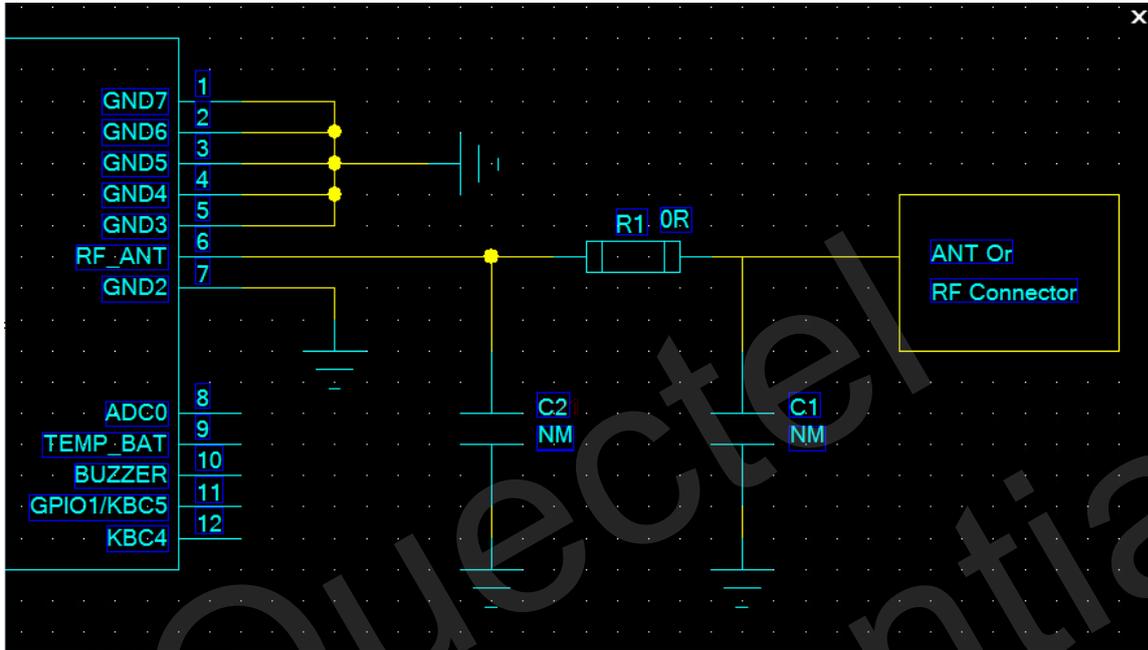


Figure 1: RF Reference Schematic Diagram

C1, R1 and C2 form a “PI” type matching circuit which is reserved for antenna optimization. By default, R1 is 0ohm while C1 and C2 are both Not Mounted (NM).

3 Coplanar Waveguide Structure Design

The recommended coplanar waveguide structure is shown as Figure 2.

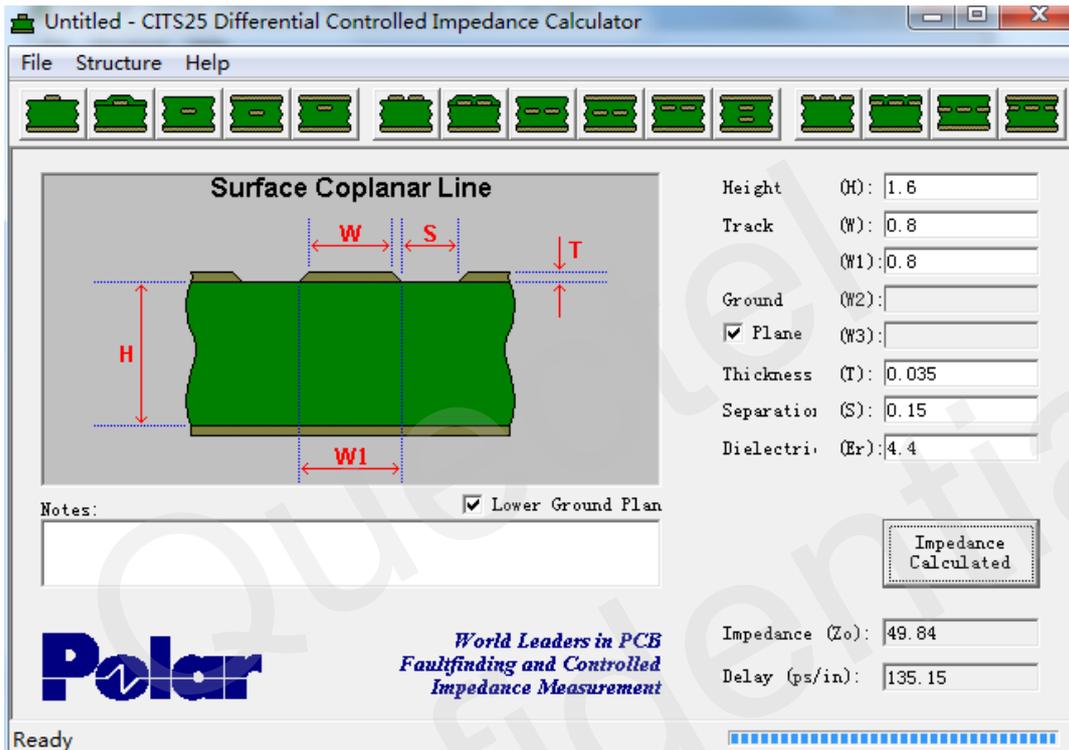


Figure 2: Structure of Coplanar WG

The factors which influence impedance include dielectric constant (usually 4.2~4.6, here is 4.4), dielectric height (H), RF trace width (W), the space between RF trace, the ground (S) and copper thickness (T). When T=0.035mm, the recommended value of W and S for 50 ohm coplanar WG under different PCB structure is listed in Table 1.

Table 1: Recommended Value of W and S for 50 ohm Coplanar WG under Different PCB Structure

Dielectric Height (H)	RF Trace Width (W)	Space between RF Trace and the Ground (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 there are two layers, the TOP layer is the signal layer, and the BOTTOM layer is the reference ground, as shown in Figure 3. If there are 4 layers, the reference ground could be the second layer, the third layer or the fourth layer. If third layer is chosen, the second layer should be kept out and the width of keepout area should be at least five times of the trace width, as shown in Figure 4. If the fourth layer is chosen, both the second and third layer should be kept out and the width of keepout area should be at least five times of the trace width, as shown in Figure 5. Same as 6 or more layers.

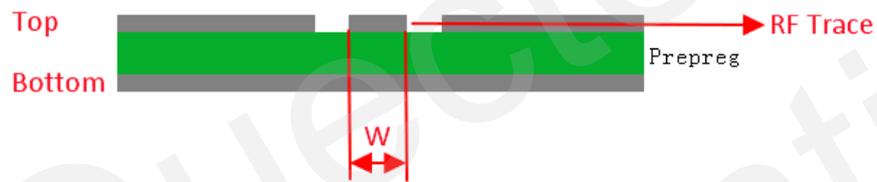


Figure 3: Two Layers PCB Layout

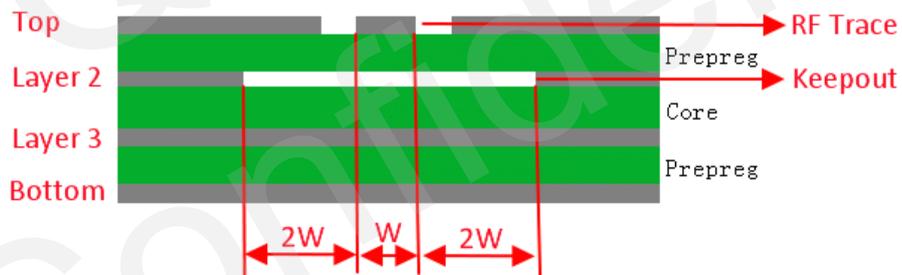


Figure 4: Four Layers PCB Layout (Third Layer as Reference Ground)

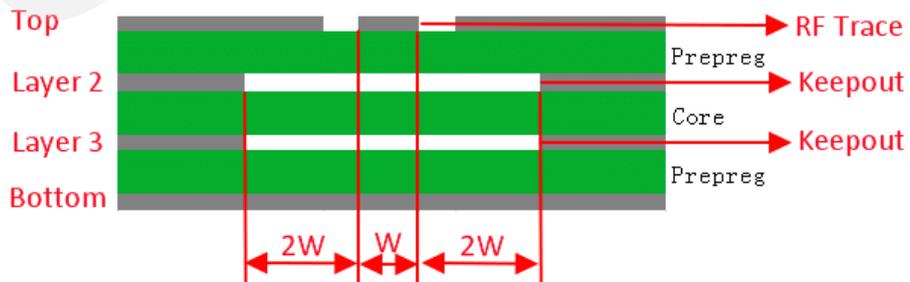


Figure 5: Four Layers PCB Layout (Fourth Layer as Reference Ground)

4 Coplanar WG PCB Layout Example and Guidelines

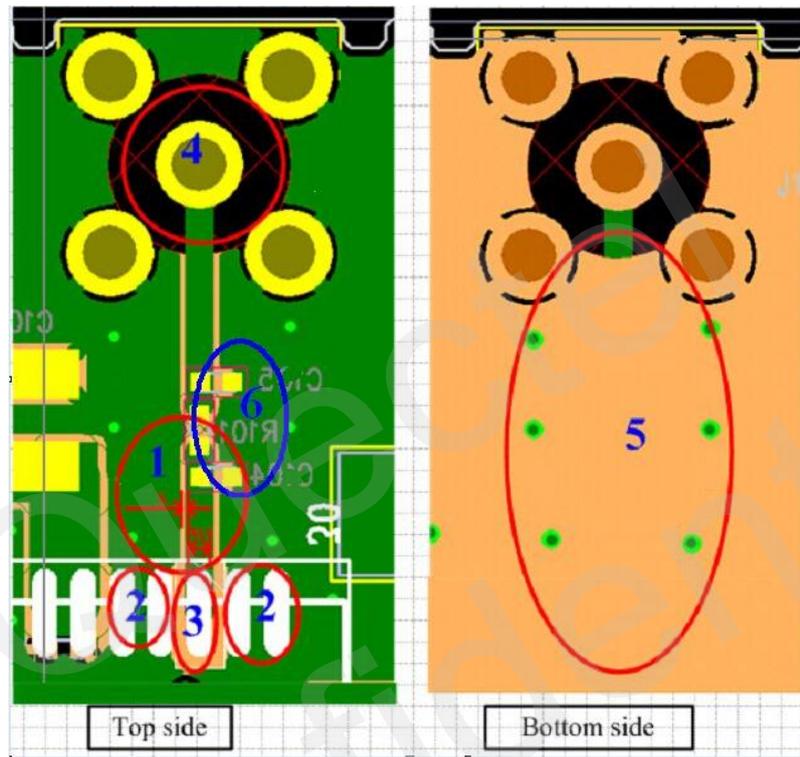


Figure 6: An example of PCB layout

There are 6 guidelines should be taken into account, as marked in the above figure:

1. Control corresponding W and S of 50 ohm coplanar waveguide. Use the common PCB as FR4 medium (dielectric constant is 4.2) and take copper clad of 35 um thickness as an example. Values of W and S for 50 ohm coplanar WG under different PCB structure is shown as Table 1. Keep in mind to remind PCB manufacturers to keep the accuracy of W and S.
2. Do not hot sealing the PIN in this position and make it contacted with the ground closely enough.
3. Keep out pouring copper in the surface layer and reduce parasitic effect. The RF trace line should be as short as possible. It will be better for RF trace line to avoid vertical angle layout. The RF trace line should be kept 135 degree angle around the corner.

4. Keep a certain distance between signal pad and ground when packaging the device. Refer to Figure 6. If the signal pad is in SMD type, pouring copper on the corresponding signal pad.
5. Ensure the corresponding reference ground of RF trace line is integrated and do not forget to add more ground via to help RF reflow. The ground and RF trace should be kept at least two times of the trace width. Guarantee the contact area which is in the same layer with RF trace is as large as possible and its corresponding reference ground in the opposite layer is as integrated as possible, meanwhile ensure the two layer ground is connected by amount of ground hole.
6. Three components consist of PI type matching circuit shown as Figure 6. Place the pad to antenna as close as possible, as shown in Figure 6. If the distance between SMA and RF PIN is too short to place the three pin of PI type matching circuit, PI type matching circuit can be changed into L matching circuit.

Quectel
Confidential