

# RF Layout

# Application Note

Rev. RF\_Layout\_Application\_Note\_2.2

Date: 2017-07-21

# About the Document

## History

Revision	Date	Author	Description
1.0	2017-07-01	Chunmao Li	Initial
2.0	2017-07-10	Chunmao Li	1. Used the new technical document template 2. Updated RF reference schematic diagram
2.1	2017-07-15	Chunmao Li	Added applicable modules
2.2	2017-07-21	Chunmao Li	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

## 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

# 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 ZF GSM, WCDMA and LTE modules.

## 2 RF Reference Schematic Diagram

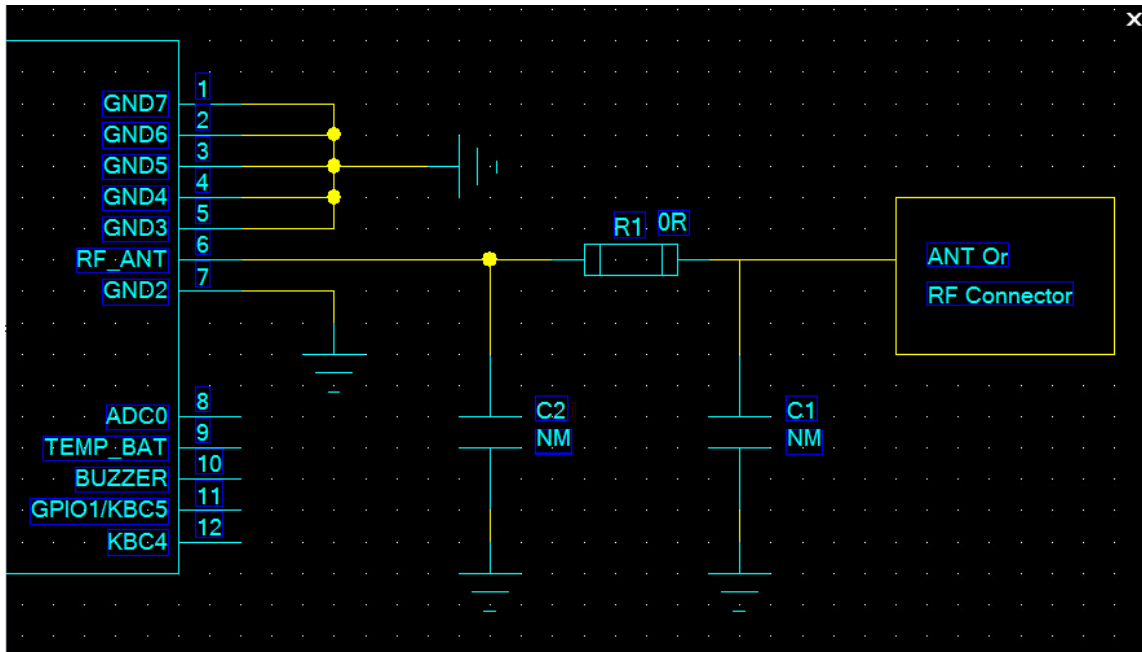


Figure 1: RF Reference Schematic Diagram

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.

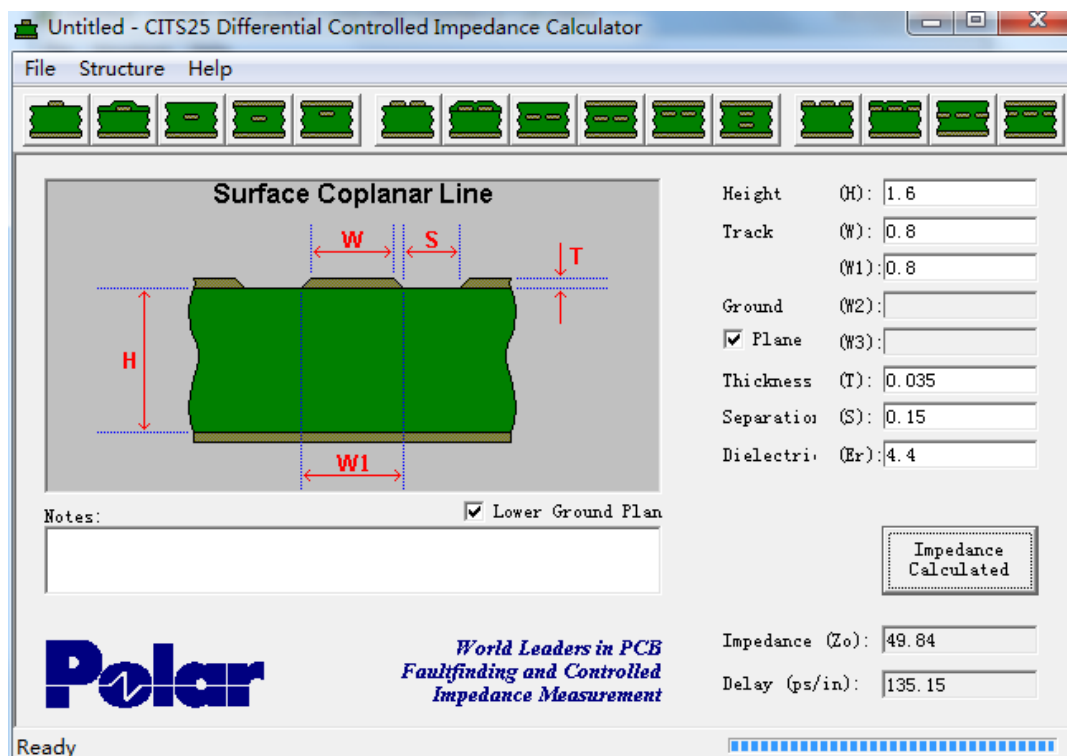


Figure 2: Structure of Coplanar WG

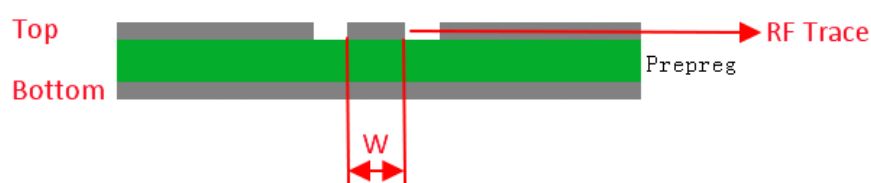
The factors which influence impedance include dielectric constant (usually 4.2~4.6, here is 4.4), dielectric height (H), RF trace width (W), the space between RF trace, the ground (S) and copper thickness (T). When T=0.035mm, the recommended value of W and S for 50 ohm coplanar WG under different PCB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

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

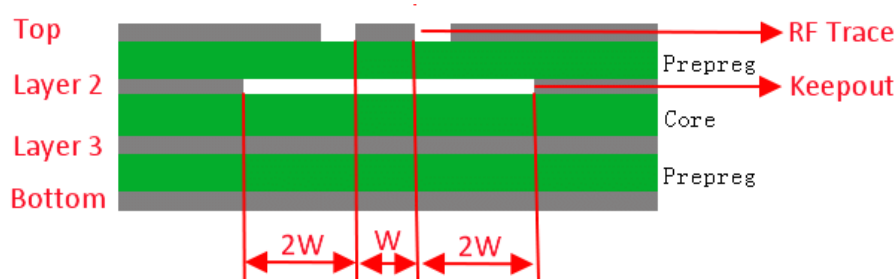
Dielectric Height (H)	RF Trace Width (W)	Space between RF Trace and the Ground (S)
0.076mm	0.1188mm	0.15mm
0.1mm	0.1623mm	0.2mm
0.15mm	0.24mm	0.2mm

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

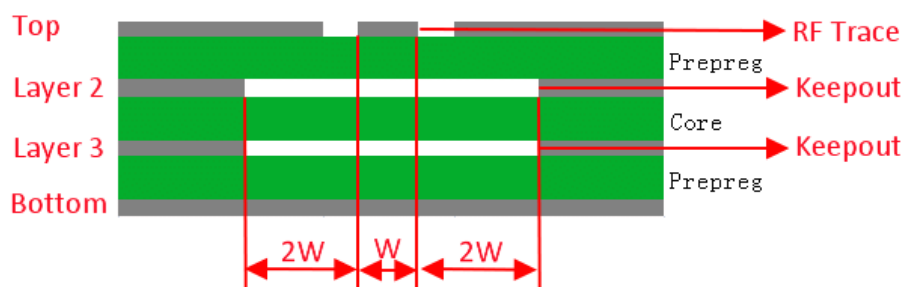
If there are two layers, the TOP layer is the signal layer, and the BOTTOM layer is the reference ground, as shown in Figure 3. If there are 4 layers, the reference ground could be the second layer, the third layer or the fourth layer. If third layer is chosen, the second layer should be kept out and 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

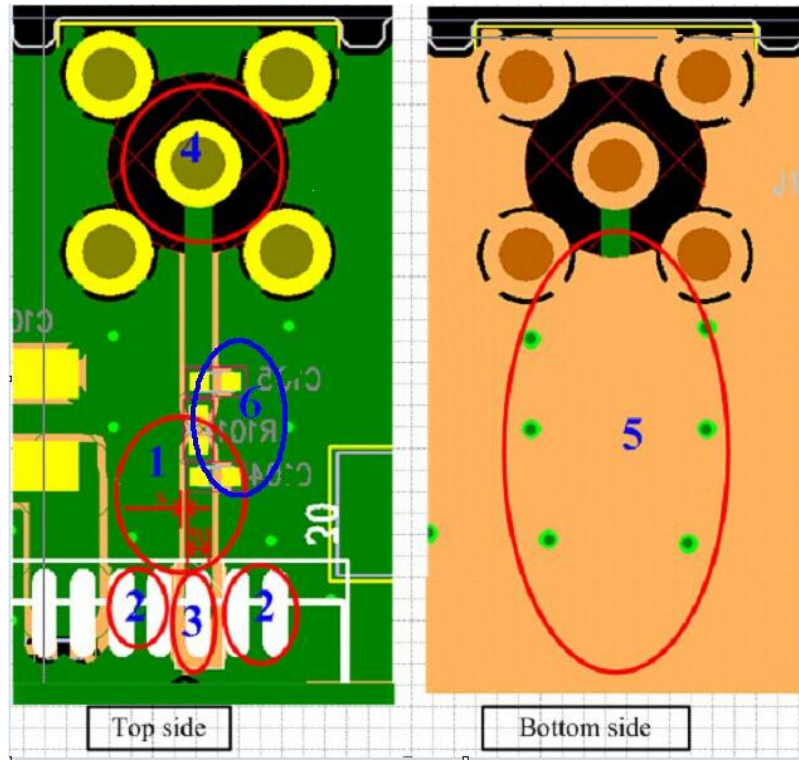


Figure 6: An example of PCB layout

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.