SIM900_Two-layer PCB RF Design _Application Note_V1.02

Document Title:	SIM900 Two-layer PCB RF Design Application Note
Version:	1.02
Date:	2012-06-15
10Status:	Released
Document Control ID:	SIM900_Two-layer PCB RF Design_Application Note_V1.02

General Notes

SIMCom offers this information as a service to its customers, to support application and engineering efforts that use the products designed by SIMCom. The information provided is based upon requirements specifically provided to SIMCom by the customers. SIMCom has not undertaken any independent search for additional relevant information, including any information that may be in the customer's possession. Furthermore, system validation of this product designed by SIMCom within a larger electronic system remains the responsibility of the customer or the customer's system integrator. All specifications supplied herein are subject to change.

Copyright

Content

Version history

Date	Version	Description of change	Author
2012-04-17	1.01	First Release	Ye Haibing
2012-06-15	1.02	Modify the way to design RF trace	Ye Haibing

1、Introduction

This document describes the key points about two-layer PCB RF hardware design, including circuit design, components placement and PCB layout, which would help to improve RF performance.

The document includes antenna and power circuit design, PCB placement and layout, and RF trace design.

2、Circuit design

This section focuses on the circuit design which is divided into two parts. The first part is antenna circuit design, and the second is power supply circuit design.

2.1 Antenna circuit design

Antenna circuit design can be divided into two types according to the antenna port impedance, 50 ohm and non-50 ohm. Usually, vehicle antenna and coaxial cable antenna are 50 ohm antenna; inner antenna and FPC antenna are non-50 ohm antenna.

2.1.1 50 ohm antenna

For 50 ohm antenna, coaxial cable pad or antenna connector could be connected to the SIM900 RF_IN pad directly, shown as figure1.

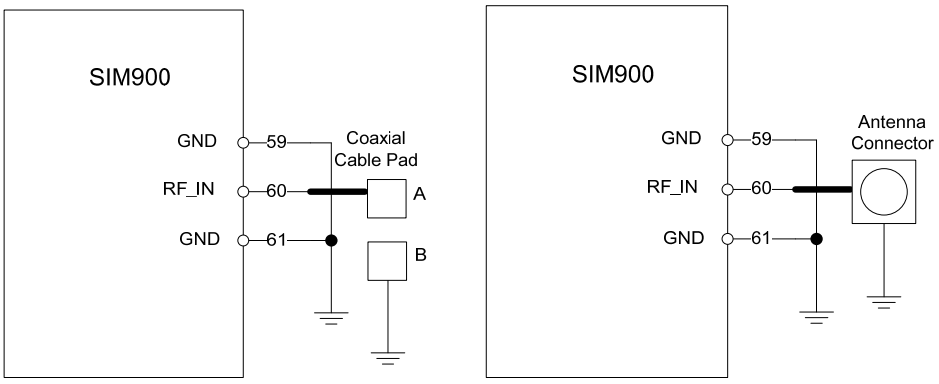


Figure1 50 ohm antenna circuit design

2.1.2 Non-50 ohm antenna

For non-50 ohm antenna, a matching network should be added between the antenna feed pad and SIM900 RF_IN pad, shown as figure2.

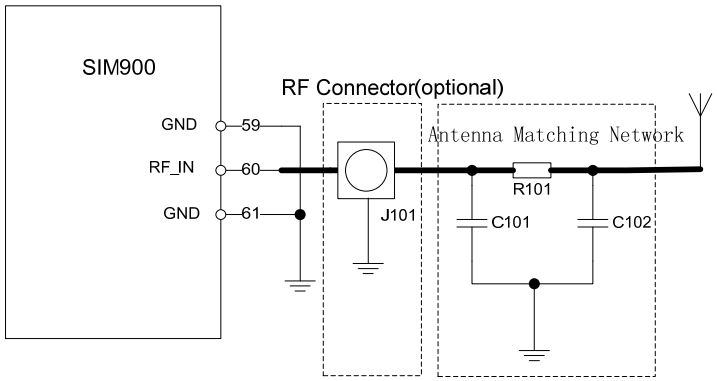


Figure2 Non-50 ohm antenna circuit design

In figure2, the RF connector is recommended for conducted RF performance test. Antenna matching network is used for antenna impedance tuning. In which, R101 is 0R, C101, C102 are not mounted as default, the final component value is decided by antenna tuning result. Recommender antenna supplier is shown as below:

Table1 Recommended antenna supplier

Antenna Supplier	Address	Telephone
SkyCross Electronics (Shenzhen) Company Ltd.	Fiyta Building, Room 1105, Hi-Tech Industrial Park, South, Nanshan District, Shenzhen City	0755-33630829
SkyCross Electronics (Shenzhen) Company Ltd. (Shanghai Branch)	Building 6, No. 351, Chengjian Road, Minhang District, Shanghai.	021-64348850
The Huizhou Speed Wireless Technology Company Ltd	West Xinglong Street, Xiaojin Town, Huizhou City, Guangdong Province.	0752-2836239
The Huizhou Speed Wireless Technology Company Ltd.(Suzhou Branch)	Zhongke Intelligent No.1, No. 99, Weixin road, Weiting Town, Suzhou Industrial Park.	0512-85550782
VLG Communication Equipment (Shenzhen) Company Ltd.	The third floor of Buliding 1, Xixiang TaoHuaYuan, Science and Technology Innovation Park, sub-park one, Baoan district, Shenzhen city.	0512-27656201

VLG Communication Equipment (Shenzhen) Company Ltd. Shanghai R &D center	North Building, third floor, No.829 Yishan Road, Xuhui District, Shanghai.	021-54452321

2.2 Power supply circuit design

In order to suppress noise signal from power supply, a circuit for noise rejection should be added between the power supply and sim900 VBAT pad, shown as figure3.

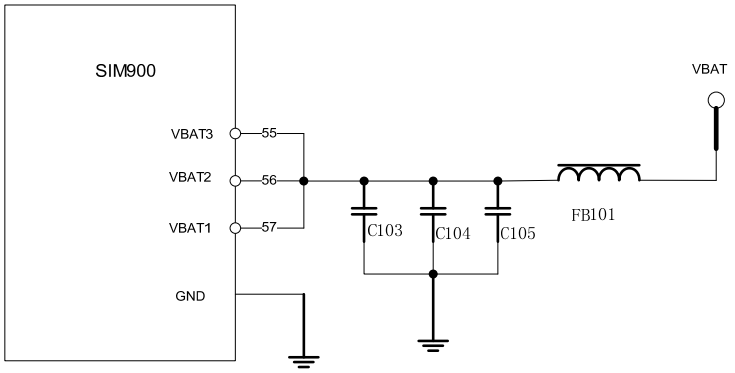


Figure3 power supply circuit design

In Figure3，the capacitors C103、C104、C105 are used for noise decoupling, the ferrite bead FB101 is used for filtering the noise from DC-DC. The values in the following table are gotten from an actual case, which maybe changed under different conditions.

Table2 the components' value from an actual case

Location	Description	Part Number	Supplier
FB101	0805，220ohm+/-25%@100MHz，DC 50mOhm,2A	FBMH2012HM221-T	Murata
C103	0402，22pF+/-5%，50V，C0G	GRM1555C1H220JA01D	Murata
C104	0402，47pF+/-5%，50V，C0G	GRM1555C1H470JA01D	Murata
C105	3528，100uF+/-20%，6.3V，Tantalum	TLJT107M006R0800	AVX

3、Components placement and PCB layout

3.1 Antenna part

In order to avoid interference and reduce RF loss, the components placement and PCB layout in antenna part should follow some key points as below:

（1） All RF conducted test interface should be placed closed to RF_IN pad, for example coaxial cable pad, antenna connector, the RF connector and so on.

（2） The antenna matching network should be placed close to the antenna feed pad.

（3） RF trace should be as short and direct as possible.

（4） Keep integrated ground plane under RF trace, shown as figure4.

（5） The ground sides of RF components must connect as directly as possible to the nearest ground reference.

（6） Avoid crossing or parallel trace under RF trace.

（7） Keep adequate ground vias surrounding the RF trace.

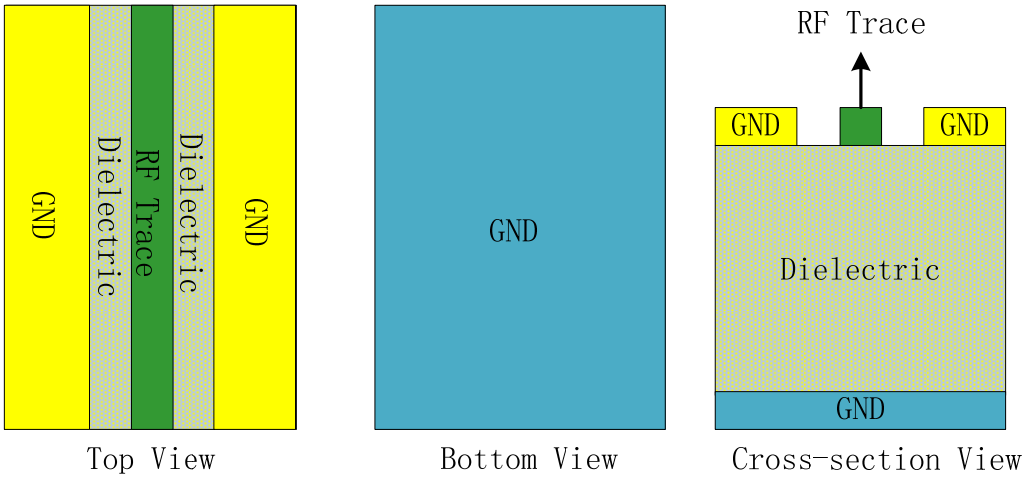


Figure4 integrated ground plane under RF trace

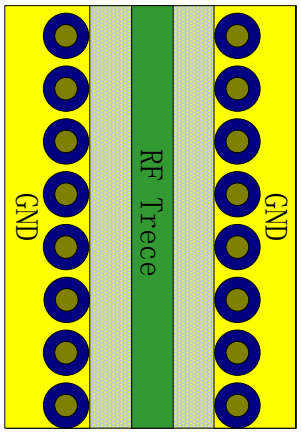
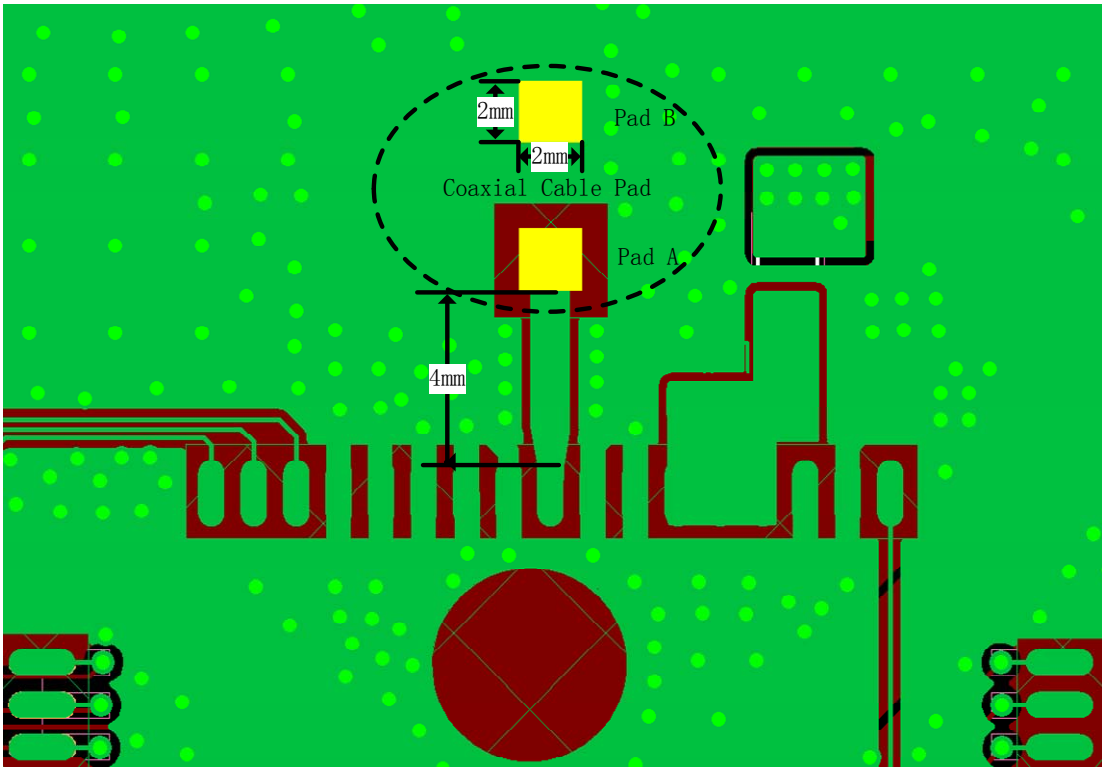
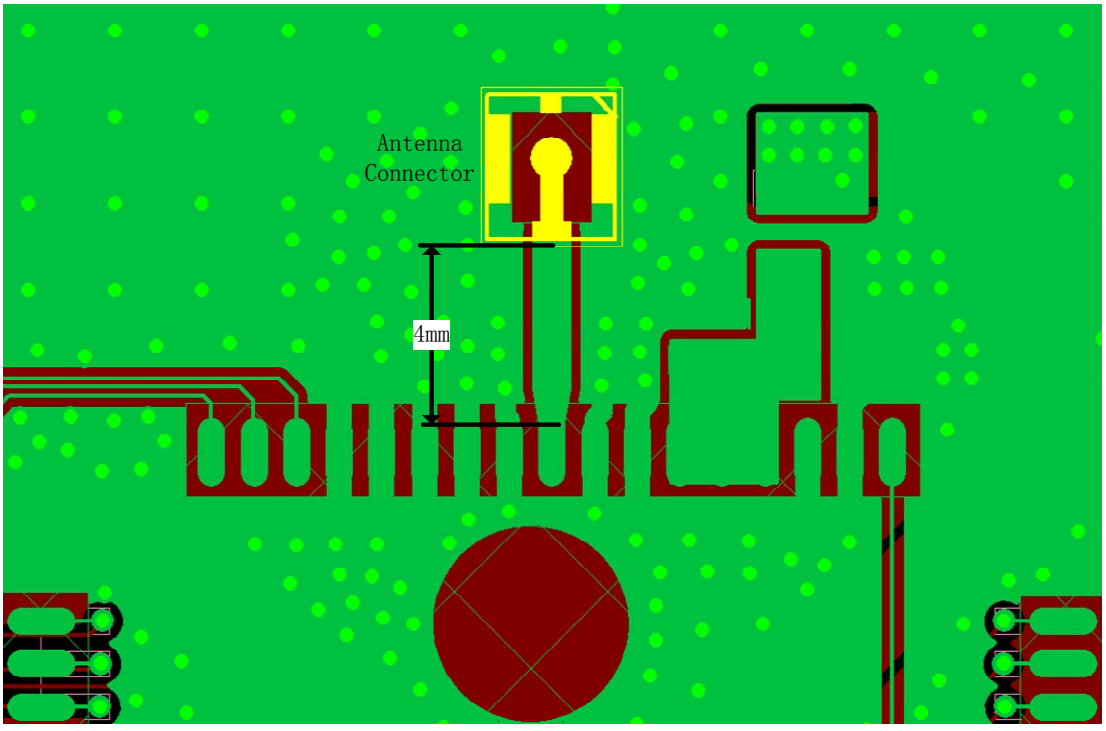


Figure5 ground vias surrounding the RF trace

According to different port impedance of antenna, the components placement and PCB layout are shown as figure6 (50 ohm antenna), figure7 (non-50 ohm antenna).



(a) Coaxial cable pad



(b) Antenna connector

Figure6 Components placement and PCB layout (50 ohm antenna)

In figure6 (a), Pad A、Pad B are used to solder RF coaxial cable. The pad dimension is 2mm*2mm. The distance between SIM900 RF_IN pad and Pad A should be less than 4mm.

In figure6 (b), the antenna connector is GSC type, other type connector also can be used, for example SMA, TNC. The distance between SIM900 RF_IN pad and antenna connector should be less than 4mm. The frequently used GSC type antenna connectors are shown as below.

Table3 GSC type antenna connector

Vendor	Part Number	Web Site
MURATA	MM9329-2700RA1	http://www.murata.com
HRS	U.FL-R-SMT(10)	http://www.hirose-connectors.com
I-PEX	20279-001E-01	http://www.i-pex.com/cn

In table3, U.FL-R-SMT(10) and 20279-001E-01 are compatible, but MM9329-2700RA is a little bigger. For details, please refer to the part specification

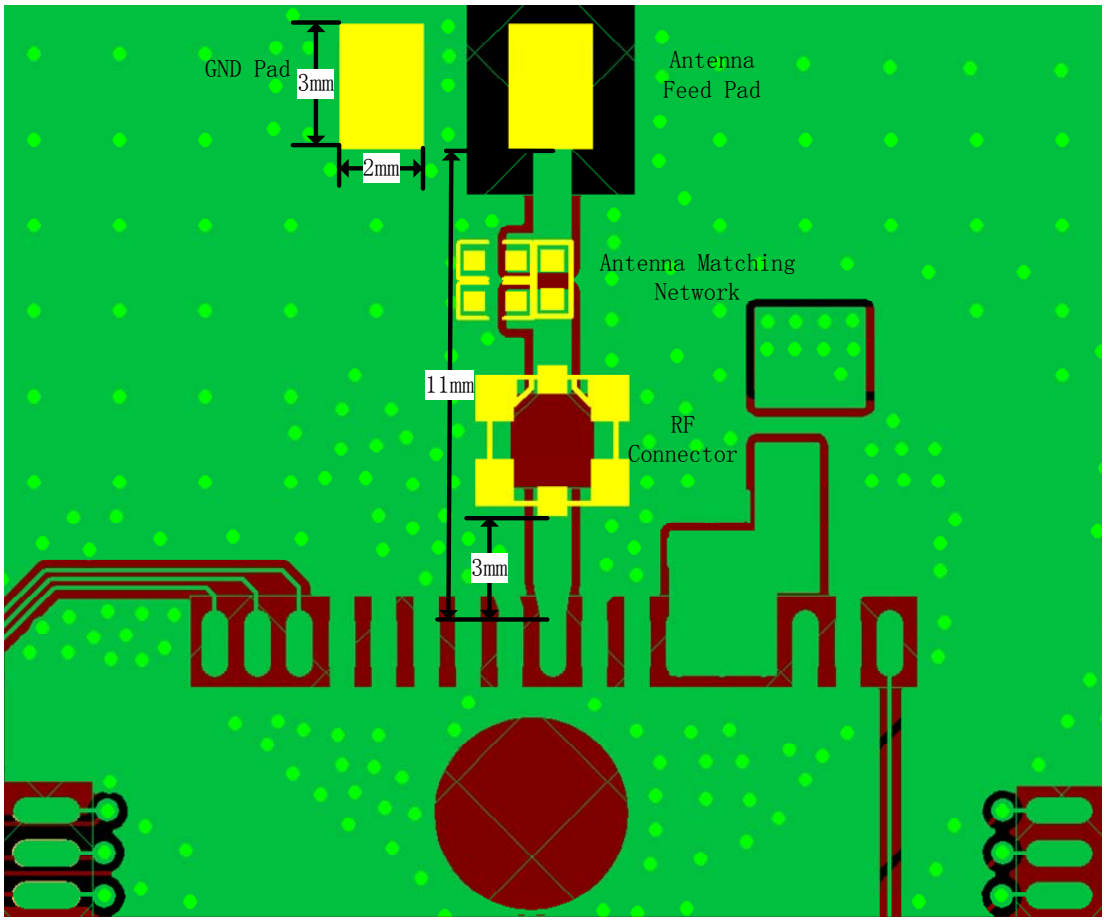


Figure7 Components placement and PCB layout (non-50 ohm antenna)

In figure7, the antenna feed pad dimension is 2mm*3mm. The distance between RF connector and SIM900 RF_IN pad should be less than 3mm. The distance between antenna feed pad and SIM900 RF_IN pad should be less than 11mm. The frequently used RF connectors are shown as below.

Table4 RF connector

Vendor	Part Number	Web site
MURATA	MM8430-2610RB3	http://www.murata.com
ECT	ECT818000251	http://www.ectsz.com
SPEED	C90-101-0004	http://www.speedtech.com.tw

In table4, MM8430-2610RB3, ECT818000251 and C90-101-0004 are compatible. For details, please refer to the part specification.

3.2 Power supply part

Power supply part should follow some key points as below:

（1）Power supply trace should not cross RF area.

（2）The width of power trace should be more than 1.6mm.

（3）The decoupling capacitor should be close to the module VBAT pad, the sequence is shown as figure8.

（4）The power supply trace should be surrounded by ground to get better noise decoupling.

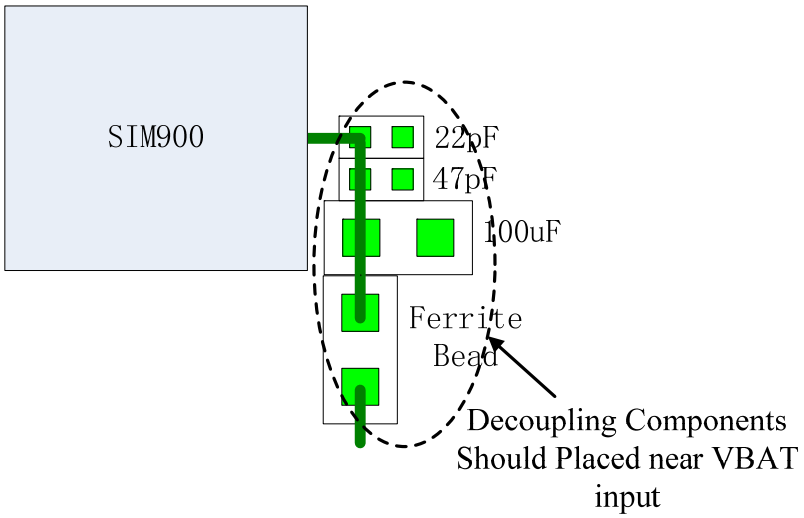
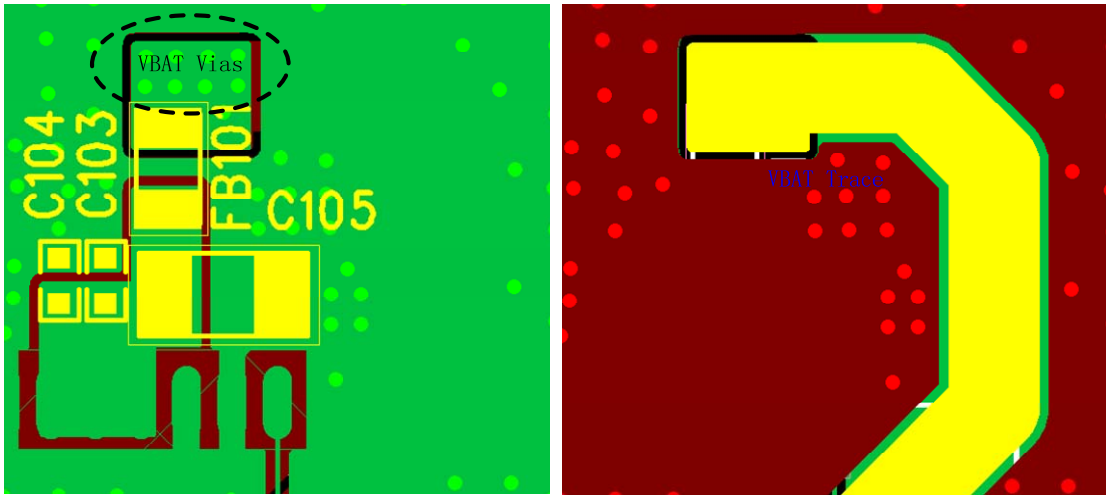


Figure8 decoupling components placement

Components placement and PCB layout are shown as figure9.



Top Layer Bottom Layer

Figure9 Components placement and PCB layout in power supply part

In figure9, ferrite bead FB101，capacitor C103，C104，C105 are mounted on the top layer, near to SIM900 VBAT Pad. The VBAT trace is routed from the bottom layer to the top layer, keeping away from RF area. The number of VBAT vias should be more than six. The VBAT trace should be routed through ferrite bead, decoupling capacitor C105, C104, C103 as sequence.

4、Two-layer PCB RF trace design

In two-layer PCB, the distance between top-layer and bottom-layer is very large. The width of 50 ohm impedance controlled RF trace will be too wide, which is different from the width of RF_IN pad. In order to reduce impedance sudden change of RF trace, gradual change line should be used.

Especially, in order to reduce path loss, RF trace should make as shorter as possible.

4.1 1.0mm Two-layer PCB

The way to design RF trace in 1.0mm Two-Layer PCB is shown as figure10.

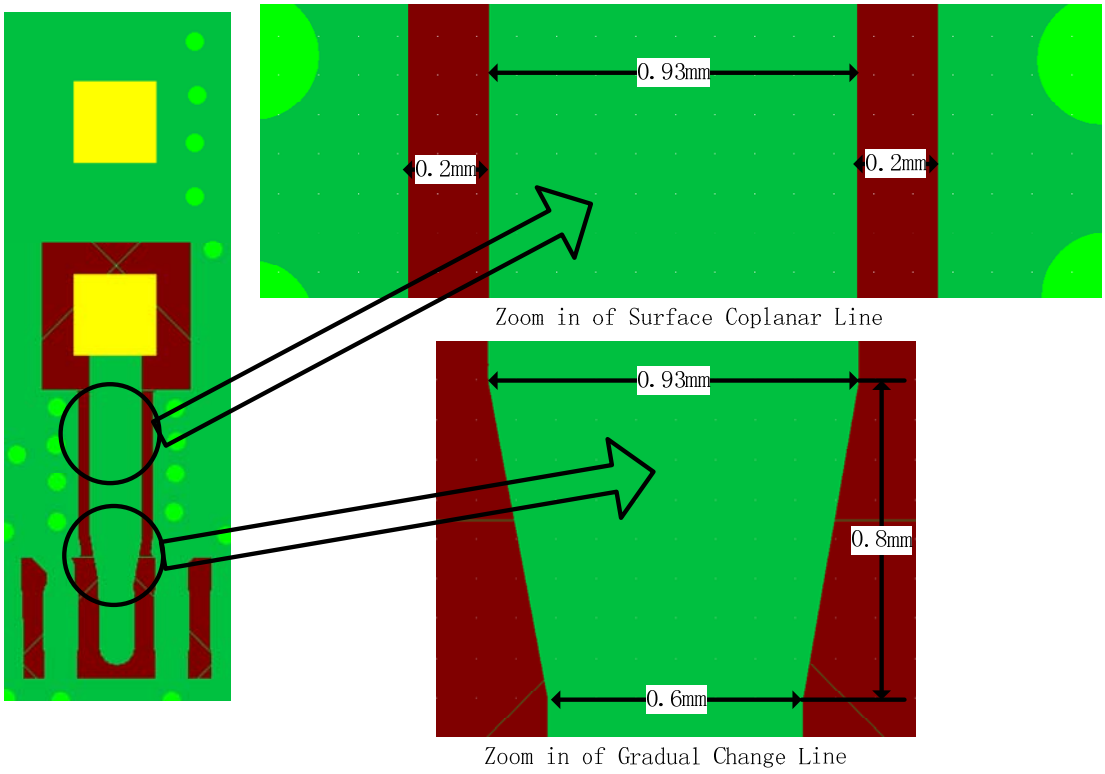


Figure10 the way to design RF trace in 1.0mm Two-Layer PCB

In figure10, the width of RF trace is 0.93mm, the distance between RF trace with ground on each side is 0.2mm.

4.2 1.6mm Two-layer PCB

The way to design RF trace in 1.6mm Two-Layer PCB is shown as figure11.

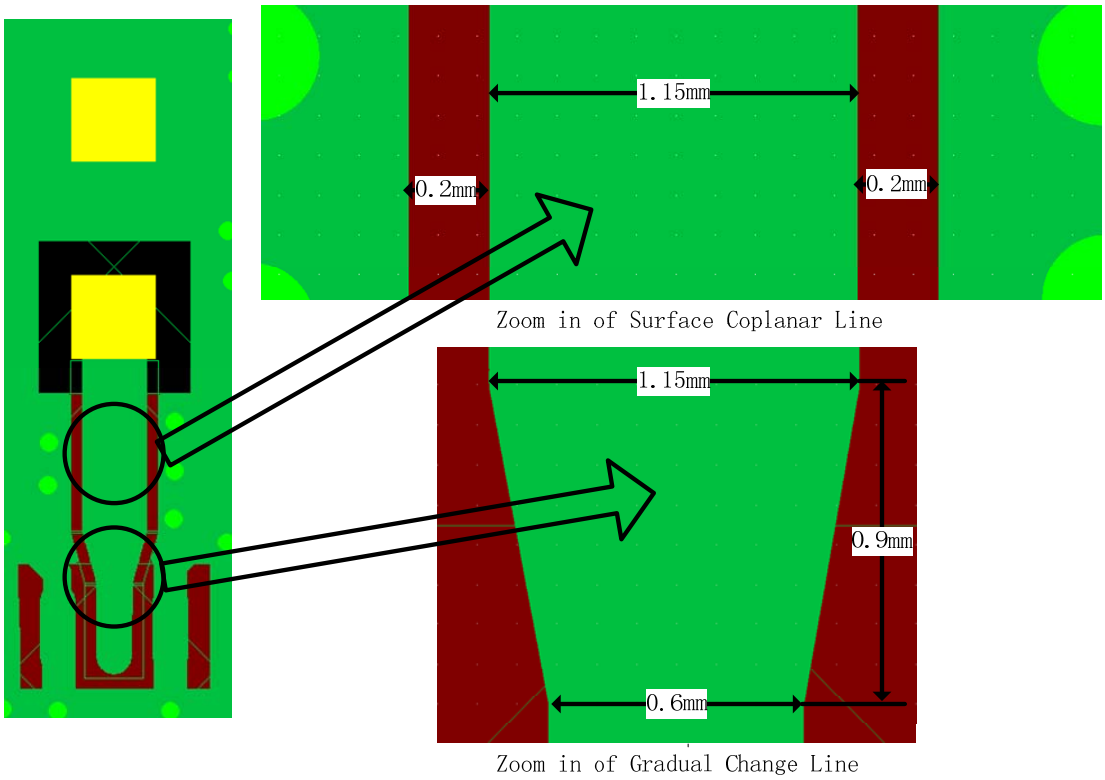


Figure11 the way to design RF trace in 1.6mm Two-Layer PCB

In figure11, the width of RF trace is 1.15mm, the distance between RF trace with ground on each side is 0.2mm.

Contact us:

Shanghai SIMCom Wireless Solutions Ltd.
Add: Building A, SIM Technology Building, No.633 Jinzhong Road, Changning District, Shanghai, P. R. China 200335
Tel: +86 21 3252 3300
Fax: +86 21 3252 3301
URL: www.sim.com/wm