SIM900_Four-Layer PCB RF Hardware Design_Application Note_V1.01

Document Title:	SIM900 Four-layer PCB RF Hardware Design
Version:	1.01
Date:	2012-07-10
10Status:	Released
Document Control ID:	SIM900_Four-Layer PCB RF Hardware Design_Application Note_V1.01

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.

Content

Version history

Date	Version	Description of change	Author
2012-07-10	1.01	First Release	Jin Xiaohan

1、Introduction

This document describes the key points about four-layer PCB RF hardware design, which intended to give the guidance to improve the final product's RF performance.

The contents in this document include circuit design, component placement, PCB layout, and RF trace design.

2、Circuit design

The circuit design issue 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

The antenna circuit design can be divided into two types, according to the antenna port impedance, which one is 50 ohm and the other is non-50 ohm. Usually, the antenna connected with a coaxial cable is 50 ohm antennas; Inner antenna such as monopole or PIFA antenna is non-50 ohm antennas. But this classification is not absolutely, for detail, it is a necessary to consultant with the antenna vendor for the further antenna information.

2.1.1 50 ohm antenna

For 50 ohm antenna, the antenna could be connected to the SIM900 RF_IN pad directly via a coaxial cable pad or an antenna connector, which is shown as figure 1.

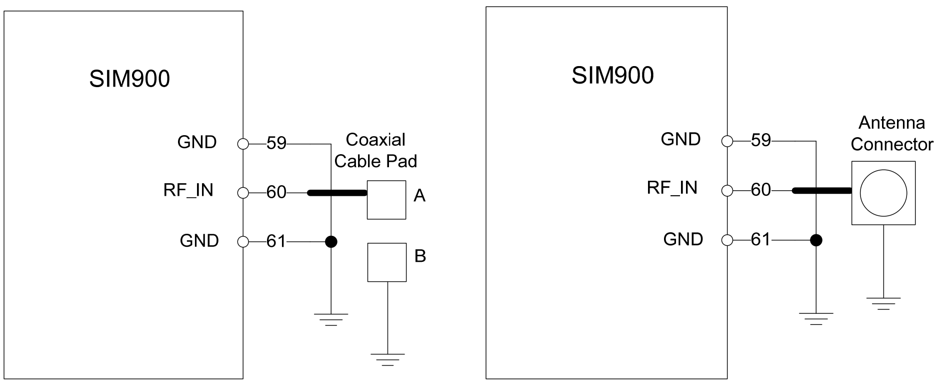


Figure1 50 ohm antenna circuit design

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, which is shown as figure2.

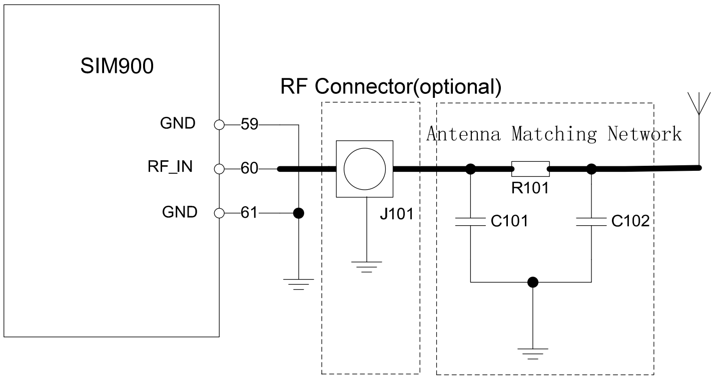


Figure2 Non-50 ohm antenna circuit design

In figure 2, a RF connector is recommended, which can be used for the conducted RF performance test and the product certification approval test.

The antenna matching network is used for antenna impedance tuning, it is a classical π type circuit topology. The component values are gotten from the antenna vendor depending on the antenna tuning result. For default, the component R101 is a 0 ohm resistor, and the component C101 and C102 are not mounted.

For the antenna vendor, here is a table list which has some business cooperation with SIMCom:

Table 1 Recommended antenna vendor

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)	The third floor of Buliding 1, Xixiang TaoHuaYuan, Science	0512-27656201

Company Ltd.	and Technology Innovation Park, sub-park one, Baoan district, Shenzhen city.	
VLG Communication Equipment (Shenzhen) Company Ltd. Shanghai R &D center	North Building, third floor, No.829 Yishan Road, Xuhui District, Shanghai.	021-54452321
Antenova Limited	Far Field House, Albert Road Stow-cum-Quy, Cambridge CB25 9AR, UK	+44(0)1223810600

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, which is shown as figure 3.

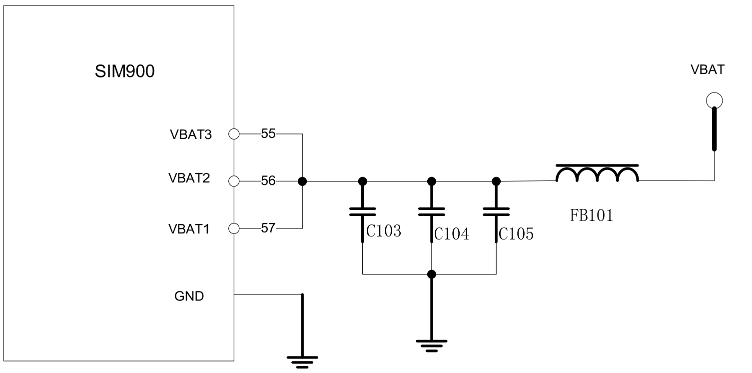


Figure3 Power supply circuit design

In Figure 3，the capacitors C103、C104、C105 are used for noise decoupling, the ferrite bead FB101 is used for filtering the noise from DC-DC.

The table 2 is a typical component value list, which is extracted from an actual design case, and maybe changed depending on the actual product design.

Table 2 Typical component value list

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 guarantee good performance, the components placement and PCB layout in antenna part should be considered carefully, and following some general rules as below:

（1） The RF interface used for RF conducting test such as coaxial cable pad, RF connector should be placed close to RF_IN pad. Make the RF trace as short as possible to reduce the impact of impedance mismatch and loss.

（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 figure 4.

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

（6） Avoid any other signal crossing or parallel directly under the RF trace.

（7） Keep adequate ground vias surrounding the RF trace, shown as figure 5.

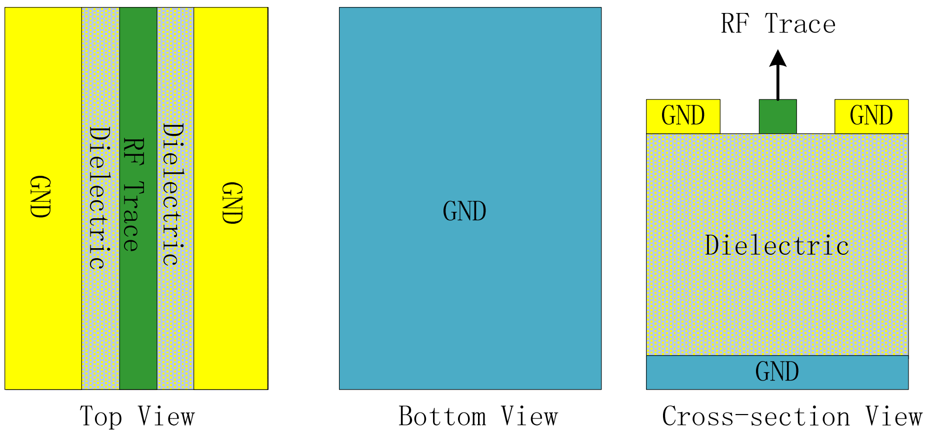


Figure4 Integrated ground plane under RF trace

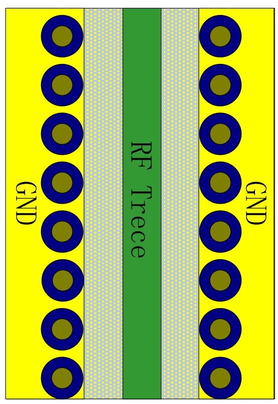
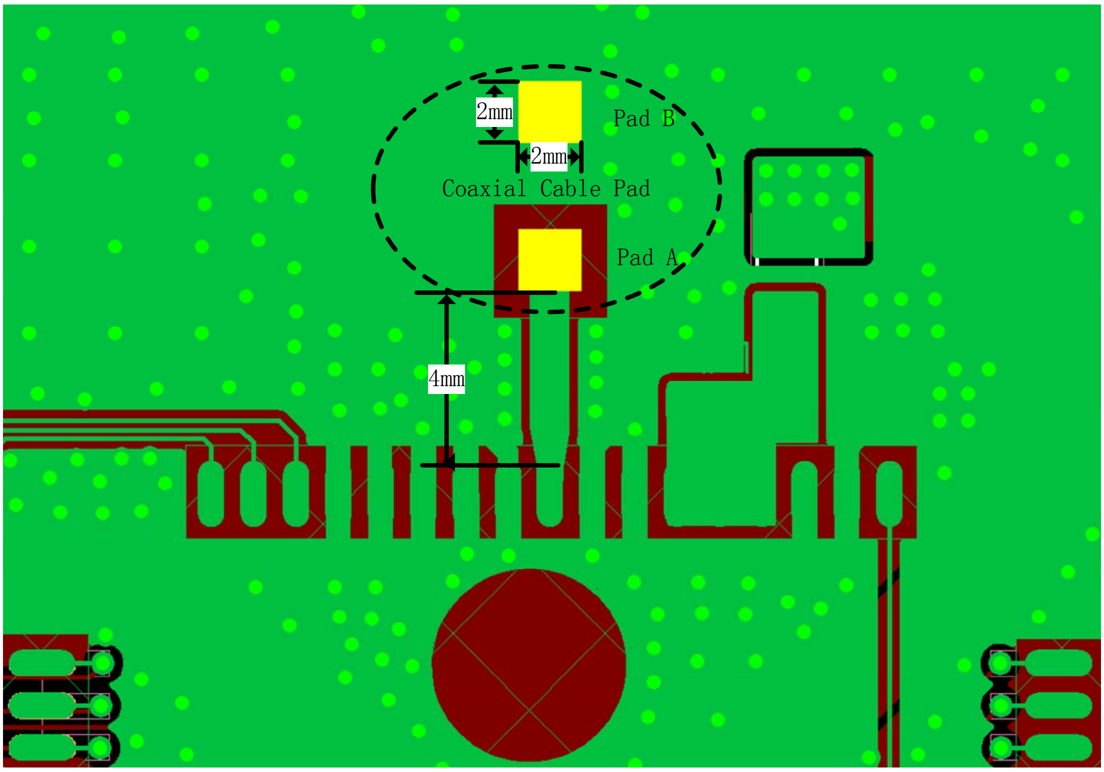
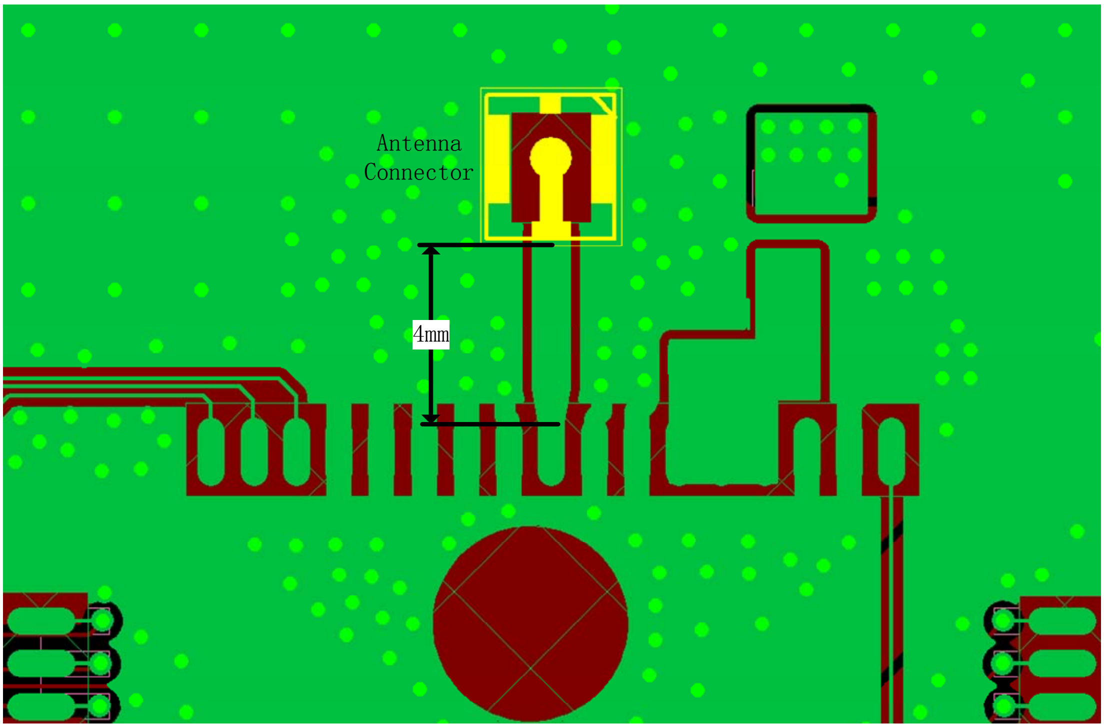


Figure5 Ground vias surrounding the RF trace

As described in section 2.1, for antenna with different port impedance, the illustrated component placement and PCB layout are shown as figure 6 (for 50 ohm antenna) and figure 7 (for non-50 ohm antenna) respectively.



(a) Coaxial cable pad



(b) Antenna connector

Figure6 Components placement and PCB layout (50 ohm antenna)

In figure 6(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 figure 6(b), the distance between SIM900 RF_IN pad and antenna connector should be less than 4mm. Other than the GSC type antenna connector used in the figure, some alternatively type antenna connector also can be used, such as SMA, TNC, etc. Some frequently used GSC type antenna connector part number and vendor are shown as table 3.

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 table 3, U.FL-R-SMT(10) and 20279-001E-01 are two compatible parts, but MM9329-2700RA is a little bigger. For the detailed part information, please refer to the part specification sheet respectively.

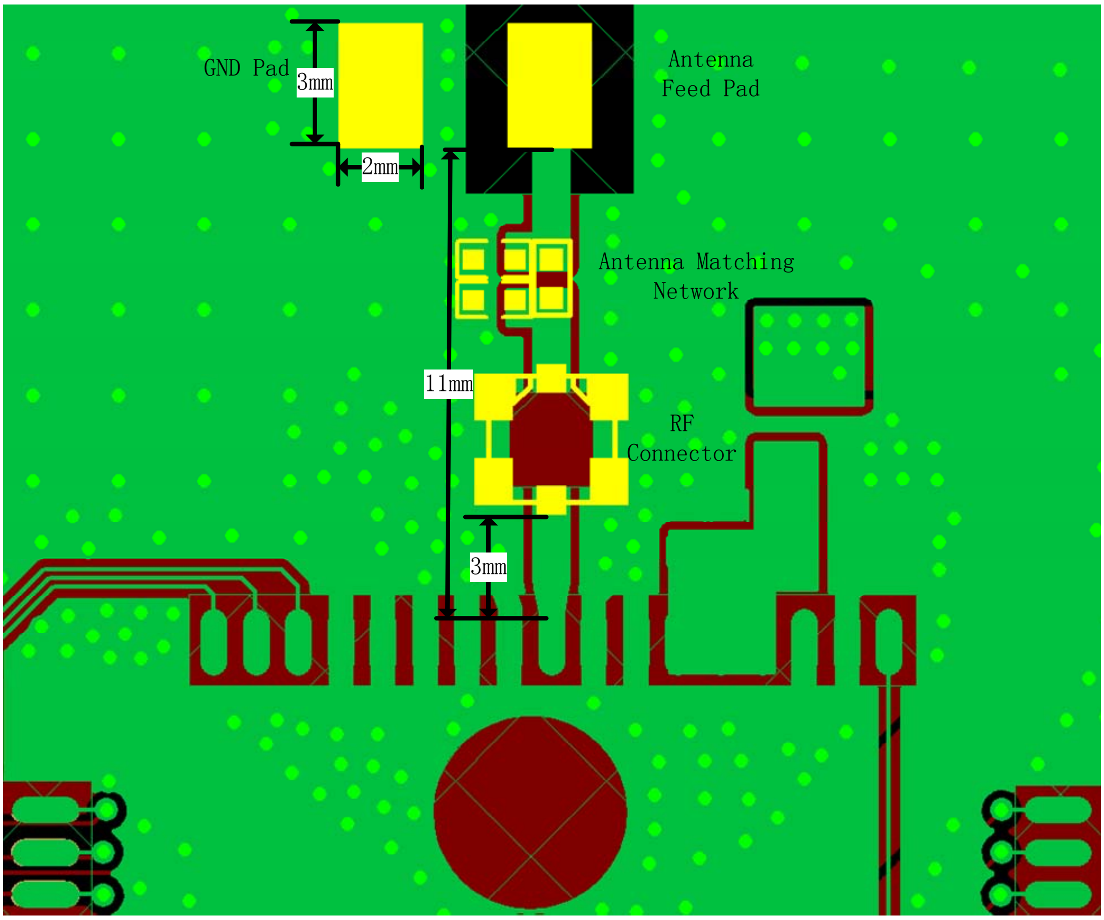


Figure7 Components placement and PCB layout (non-50 ohm antenna)

In figure 7, 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 table 4.

Table 4 RF connector vendor and part number

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 table 4, MM8430-2610RB3, ECT818000251 and C90-101-0004 are compatible each other. For details, please refer to the part specification sheet.

3.2 Power supply part

For the same purpose as section 3.1, the component placement and PCB layout of power supply part also should follow some general rules:

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

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

（3）The decoupling capacitors should be close to the module VBAT pad, the sequence is shown as figure 8.

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

（5）Make use of the advantage of four-layer PCB, keep the power trace and other signal which carry with noise signal be layout in the inner layer.

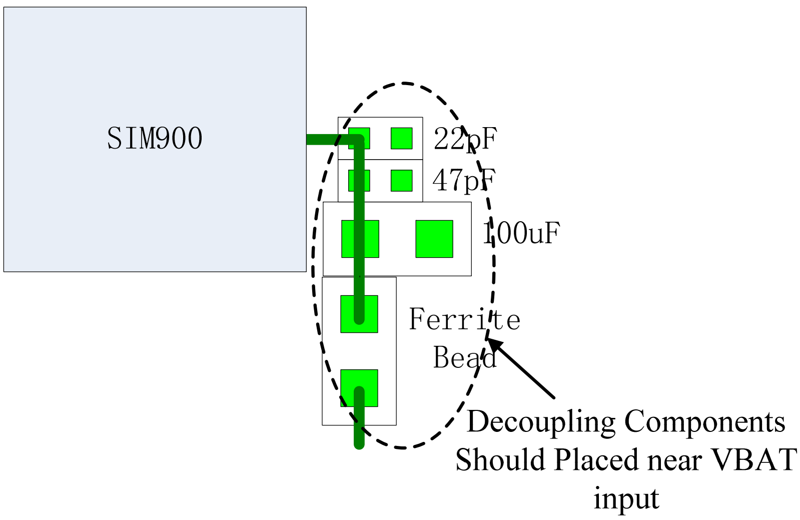
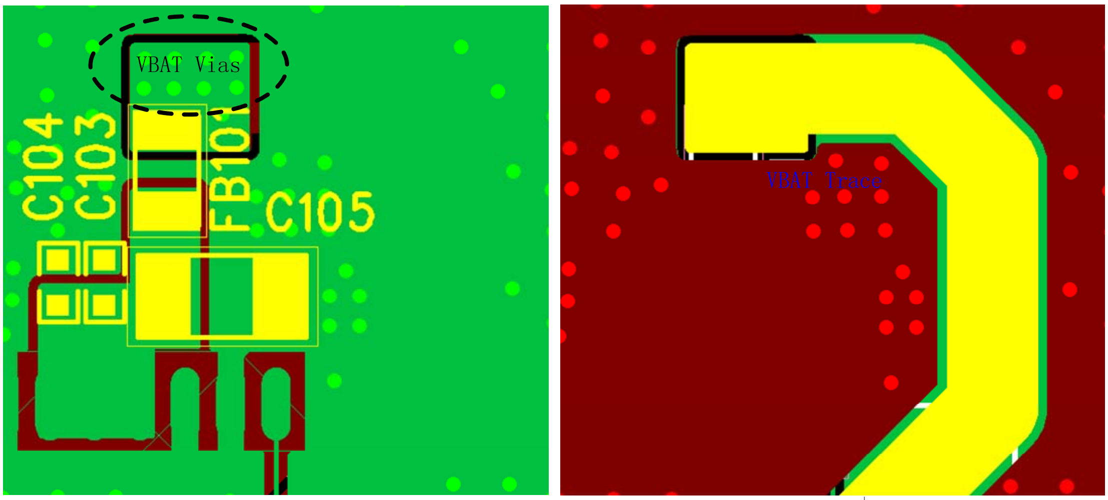


Figure8 Decoupling components placement

An illustration is shown as figure 9.



Top Layer Inner Layer

Figure9 Components placement and PCB layout in power supply part

In figure 9, 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 inner 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、Four-layer PCB RF trace design

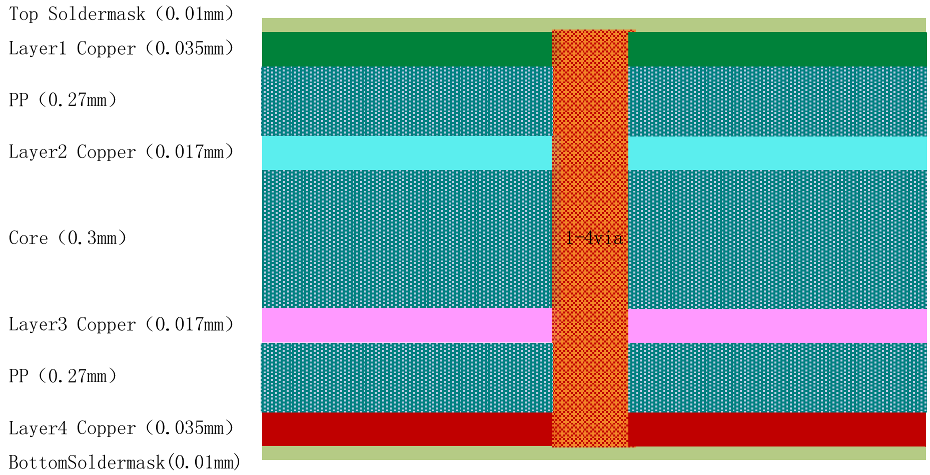
The impedance of RF trace is decided by PCB stack up, trace width, separation to ground and dielectric constant. Among these parameters, the PCB stack up plays a main and important role.

For four-layer PCB, the PCB stack up can be divided into two types, the full through hole PCB and HDI PCB which use blind and buried via technology. Figure 10 and figure 12 are illustrations of these two type PCB stack up.

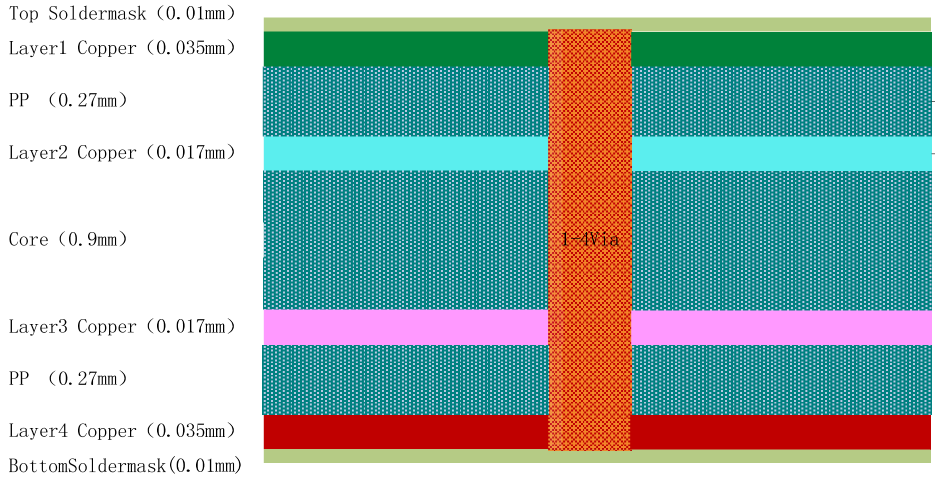
In the latest section 4.1 and 4.2, the recommended RF trace design will be given, which is based on the PCB stack up shown as figure 10 and figure 12 respectively.

4.1 RF trace design for through hole PCB

Figure 10(a) and 10(b) are through hole type PCB stack up with different PCB thickness, and figure 10(a) is for 1.0mm thickness PCB, figure 10(b) is for 1.6mm thickness PCB.



(a) Total PCB thickness is 1mm



(b) Total PCB thickness is 1.6mm

Figure10 The through hole type PCB stack up

Though the total PCB thickness is different, the thickness of dielectric material between the Top layer and the Second layer is the same, which is 0.27mm. So the way to design RF trace also is the same, which is shown as figure 11.

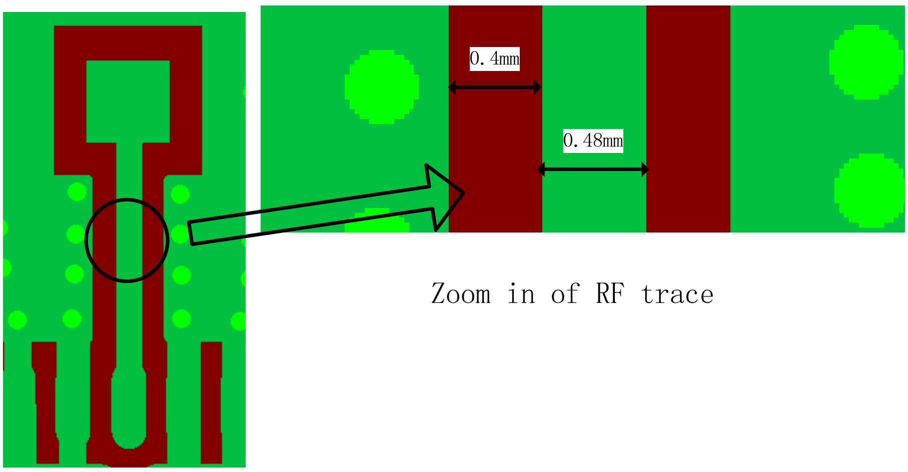


Figure11 The way to design RF trace in through hole type four-layer PCB

In figure11, the width of RF trace is 0.48mm, the distance between RF trace and ground on each side is 0.4mm.

4.2 RF trace design for HDI PCB

The figure 12 is the illustration of a typical 1.0mm total thickness HDI type PCB stack up, the thickness between each layer, is shown as the left side of the picture.

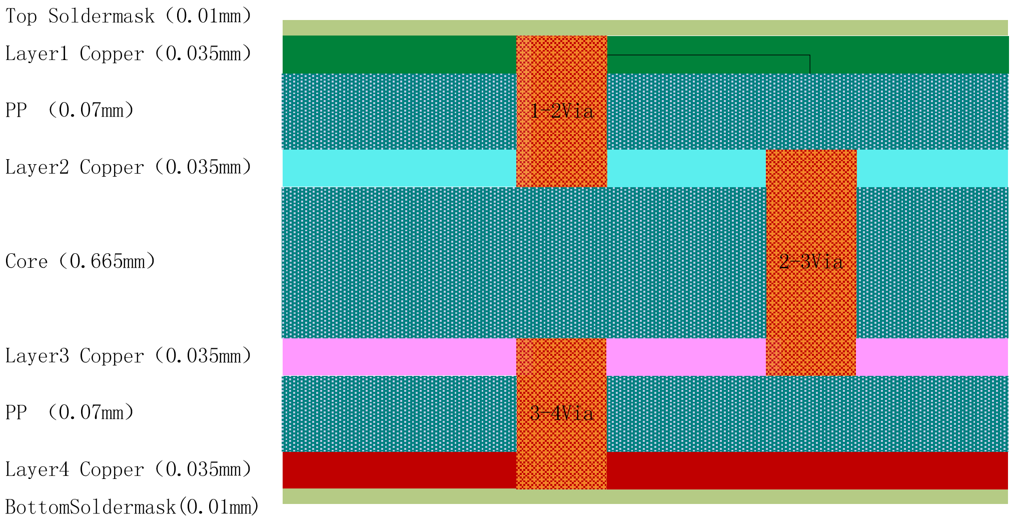


Figure12 HDI PCB with 1.0mm thickness

The way to design RF trace in 1mm four-layer buried/blind via PCB, which stack-up is according to figure6, is shown as figure 13.

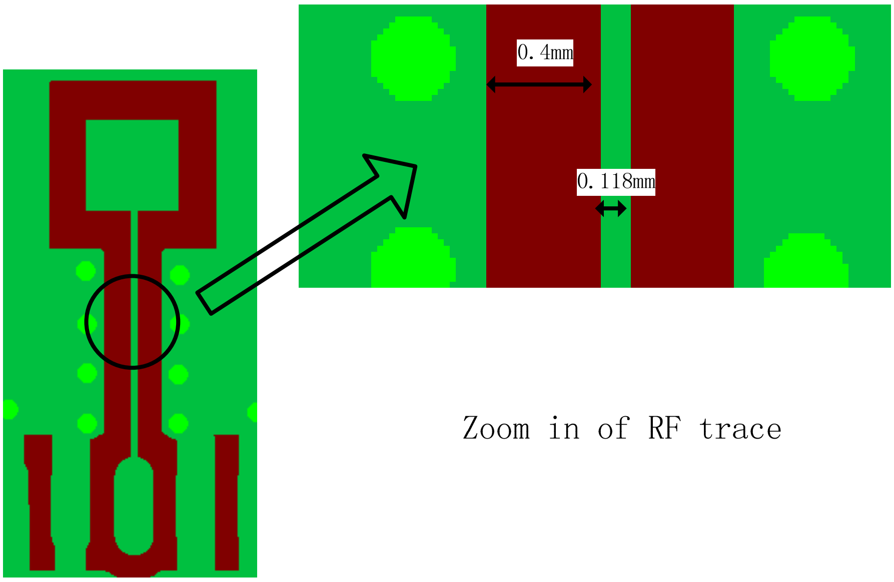


Figure13 The way to design RF trace in four-layer HDI PCB

In figure13, the width of RF trace is 0.118mm, the distance between RF trace with ground on each side is 0.4mm.

Appendix: Case study with improper RF trace design

A、RF trace not direct

In actual application, this always happens when the designers neglect the importance of straight RF trace. Sometimes it is caused by keeping away from other PCB trace or mechanical structure.

The following is a comparison of SIM900 performance on two PCB with different trace:

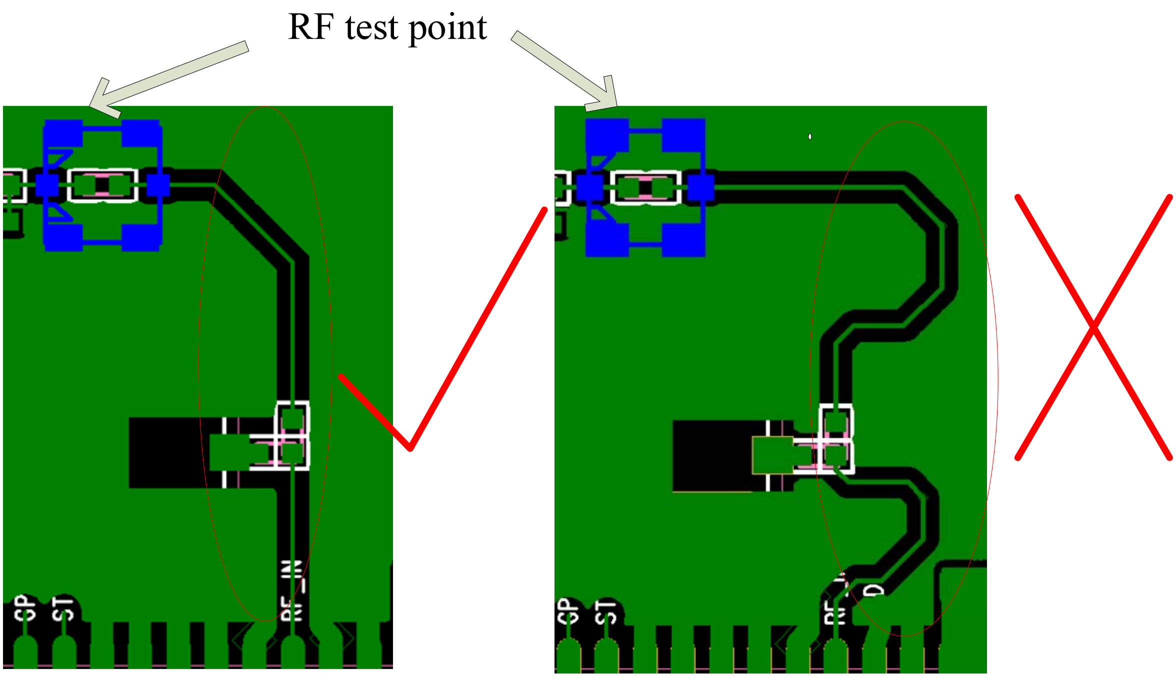


Figure14 Comparison between two PCB about direct trace

In figure 14, the one on the left is correct and the right is bad.

The transmitted power test result of SIM900 on these two boards is shown as table 5.

Table 5 Transmitted power comparison between direct and indirect trace

	channel	direct	indirect		channel	direct	indirect
GSM850	128	32.5	32.4	GSM900	1	32.6	32.5
	189	32.6	32.4		62	32.7	32.5
	251	32.6	32.4		124	32.7	32.6
DCS1800	channel	direct	indirect	PCS1900	channel	direct	indirect

	512	29.7	29.3		512	30.2	29.5
	698	29.8	29.4		661	30.2	29.5
	885	30	29.4		810	30.2	29.6

The result shows that on the PCB with indirect trace, in each band, especially the high band, the transmitted power is lower. The SIM900 module does not work well.

Solution:

Pay attention to the importance of direct trace. When designing the PCB layout, consider the RF transmission way in advance. Do not place too many irrelevant electronic components and mechanical structures beside the RF trace.

B、RF trace impedance mismatch

In actual application, this always happens when designers neglect the importance of RF trace impedance and do not deal with stack-up, trace width and separation carefully.

The following test will show the impact on terminal output power from different RF trace impedances on PCB.

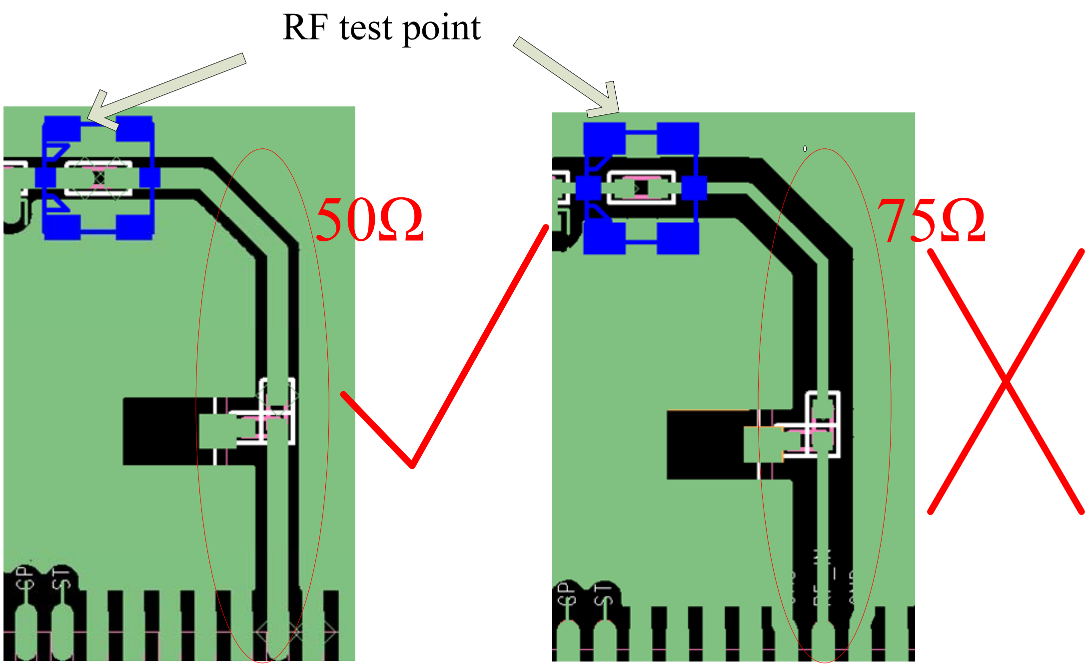


Figure15 Comparison between two PCB about RF trace impedance

The left is 50Ω RF trace, and the right is inappropriate RF trace (75Ω in this example).

The result of a test about SIM900 transmitted power on these two PCBs is shown as table 6.

Table 6 Transmitted power comparison between impedance match and mismatch

GSM850	channel	50Ω	75Ω	GSM900	channel	50Ω	75Ω
	128	32.7	32.7		1	32.7	32.5
	189	32.8	32.7		62	32.7	32.5
	251	32.7	32.6		124	32.7	32.5
DCS1800	channel	50Ω	75Ω	PCS1900	channel	50Ω	75Ω
	512	29.3	27.8		512	30.1	28.6
	698	29.4	27.9		661	30.1	28.8
	885	29.7	28.2		810	30.1	28.9

On the PCB with impedance mismatch (75Ω), in each band (especially the high band), the transmitted power is much lower. The SIM900 module does not work well.

Solution:

First, design the RF trace according to 50Ω impedance rules, and calculate trace width and separation basing on PCB stack-up carefully.

Second, it is recommended to add a RF test interface adjacent to the RF_IN pin of SIM900 for RF conducting test.

Last, make the RF trace as short as possible to reduce the impact of mismatch and loss, and keep products working well.

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