![SIMCom logo](A company of SIM Tech)

SIM868_RF_Design_Application Note_V1.00

Revision Note

FAQ

Application Note

Document Title:	SIM868_RF_Design_Application Note_V1.00
Version:	V1.00
Date:	2016-09-13
Status:	Released
Document Control ID:	SIM868_RF Design Guide_V1.00

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

This document contains proprietary technical information which is the property of SIMCom Limited., copying of this document and giving it to others and the using or communication of the contents thereof, are forbidden without express authority. Offenders are liable to the payment of damages. All rights reserved in the event of grant of a patent or the registration of a utility model or design. All specification supplied herein are subject to change without notice at any time.

Copyright © Shanghai SIMCom Wireless Solutions Ltd. 2008

Contents

Version history

Date	Version	Description of change	Author
2016-09-13	1.00	Initial version	Chen Xiaoxu

1   Introduction

This document describes the SIM868 module RF antenna interface and recommended antenna performance specifications. In order to get better RF performance, the customer should follow antenna design guides in the document.

2   SIM868 RF Interface overview

Designed for global market, SIM868 is integrated with a high performance GSM/GPRS engine , a BT engine and a GNSS engine.

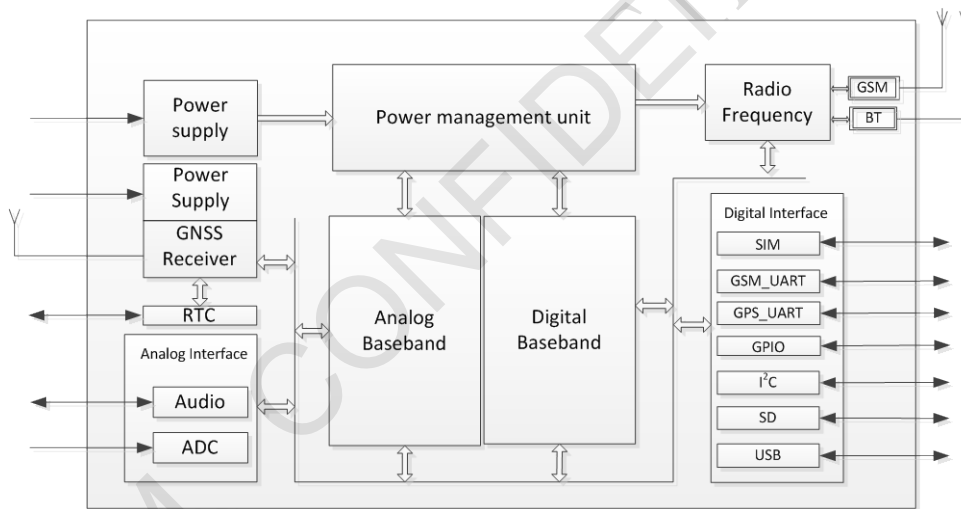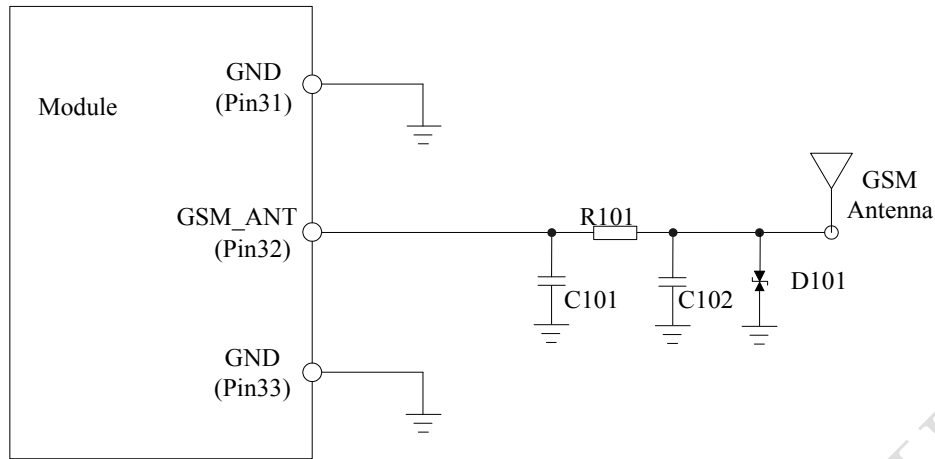The following figure shows a functional diagram of SIM868:

Figure 1: SIM868 functional diagram

3   Circuit Design

3.1   GSM antenna interface

The following is the recommended GSM antenna circuit design.

Figure 2: GSM antenna matching circuit without RF connector

In above figure, the components R101, C101 and C102 are used for antenna matching, the value of components can only be got after the antenna tuning, usually, they are provided by antenna vendor. By default, the R101 is 0Ω resistors, and the C101, C102 are reserved for tuning.

To get better ESD performance, a bidirectional TVS component D101 is placed near by the antenna port, which is used for ESD protection, the recommended part numbers of the TVS are listed in the following table:

Table 1: The recommended TVS part

Package Size	Part Type	Vendor
0201	LXES03AAA1-154	Murata
0402	LXES15AAA1-153	Murata

3.2 BT antenna interface

The following figure is the recommended BT antenna interface circuit design.
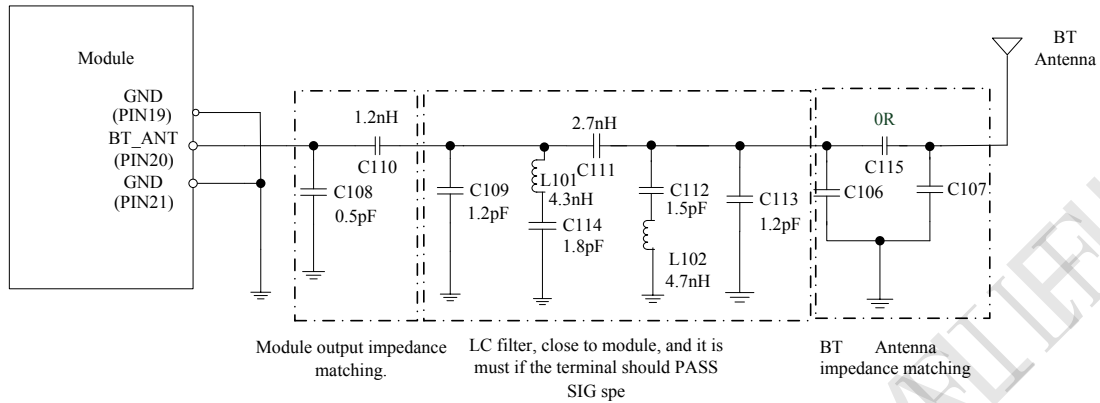
Figure 3: BT antenna matching circuit

The C108 and C110 are used for RF Matching，The default values of these two components are shown in Figure 3, but must be tuned on different PCB.

The LC filter is mandatory for SIG spec. and these components should be placed close to the module as can as possible. The default values of these components are shown in Figure 3, but must be tuned on different PCB.

The components C106, C115, C107 are for antenna impedance matching. These components should be placed as close as possible to the antenna port. By default, the C115 is 0Ω resistors, and the C106, C107 are reserved for tuning

3.3 GNSS antenna interface

For GNSS antenna interface design, there are two normal options: passive antenna and active antenna. The customer can choose passive antenna, if the path loss from the GNSS antenna to module's GPS_ANT pin is not much. If the GNSS antenna is far away from the module's GPS_ANT pin, then an active antenna should be a good choice.

For both passive and active antenna, the recommended circuits are shown as below.
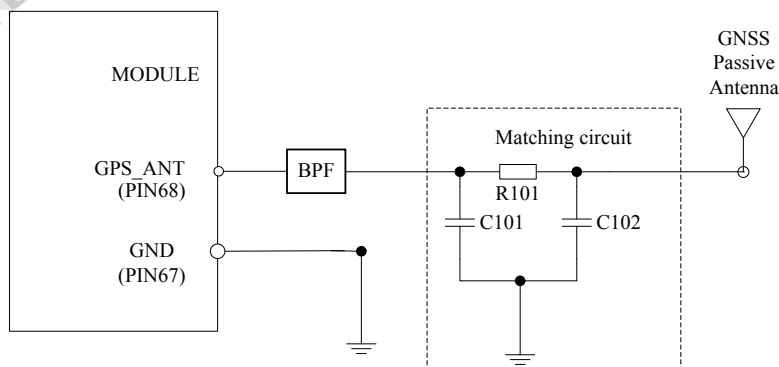
Figure 1: GNSS passive antenna matching circuit
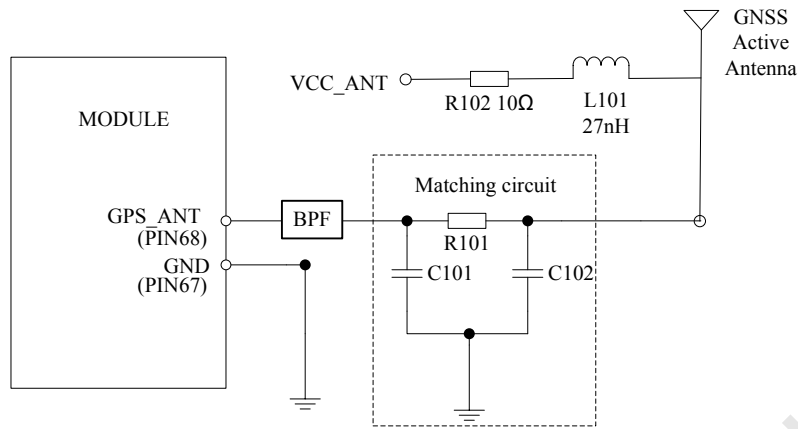
Figure 5: GNSS active antenna matching circuit

In above figure, the components R101, C101 and C102 are used for antenna matching, the components' value only can be got after the antenna tuning. For default, R101 is 0Ω, C101 and C102 are not mounted.

The component R102 is used for current limiting under the situation that the active antenna was shorted. The component L101 is a RF choke inductor, to avoid the RF signal loss, and usually, the value of L101 is no less than 27nH.

The component BPF is a SAW filter, which is used for out of band noise signal suppression. The recommended part numbers of the BPF are listed in Table 2:

Table 2: The recommended GNSS BPF parts

Package Size	Part Type	Vendor
1.35*1.05	SAFEB1G57KE0F00	Murata
1.4*1.1	B8313	Epcos
1.1*0.9	SAFFB1G56KB0F0A	Murata
1.1*0.9	B8813	Epcos

3.4 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 SIM868 GPS_VBAT/VBAT pad, shown as Figure 6.
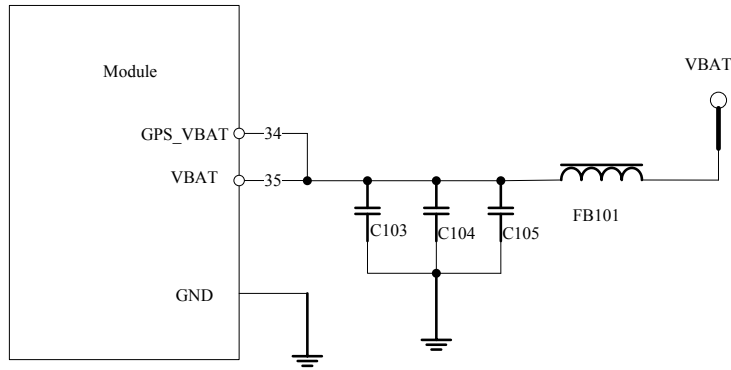
Figure 6: Power supply circuit design

In Figure 6，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.

**Table 3: 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

3.5 Noise Suppression Consideration

SIM868 is a multi-antenna system, in order to get better GNSS receive performance, the noise suppression should be considered.

For in band noise, SIM868 provide the Anti-Jamming function to remove the noise signal from the other antenna or environment. This function is disabled by default, the customer can enable this feature via PMTK command. The PMTK command "$PMTK286,1*23" is used for enable AIC function, The PMTK command "$PMTK286,0*22" is used for disable AIC function;

The jamming suppression performance is shown in Figure 7. From the picture, we can see that the noise added to SIM868 is larger 24dB than other module when the C/No decrease 3dB.
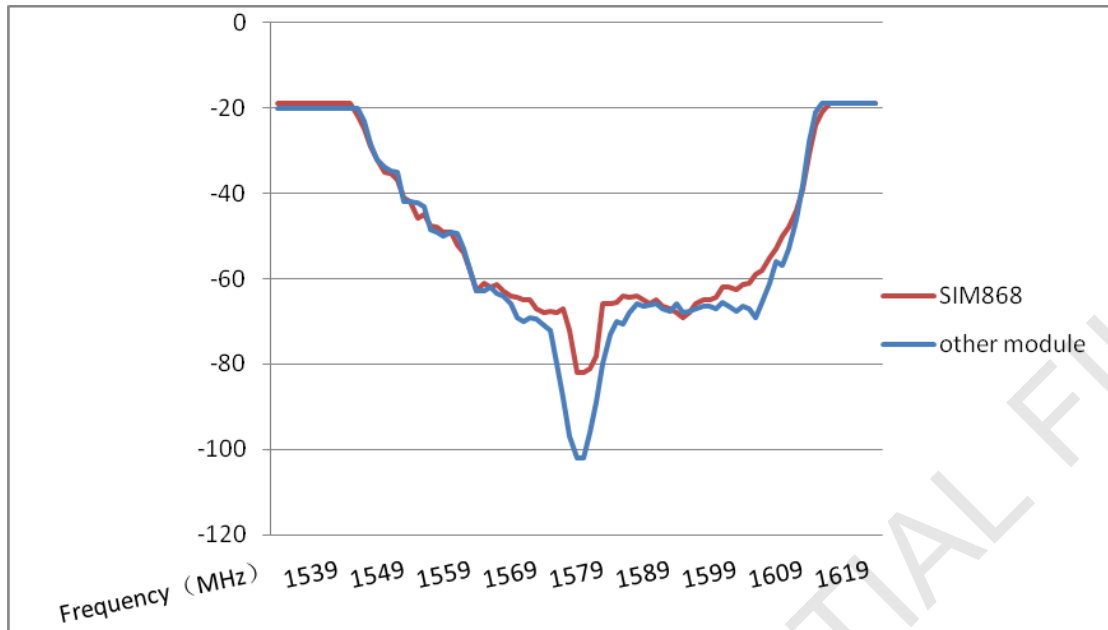
Figure 7: In-band jamming immunity—3dB desensitization curve

For out band noise, there are two methods to avoid the impact of noise.

a. Increase antenna isolation.

It is important to keep enough isolation between all the antennas. As per experience, the isolation about 15dB is OK. It can avoid the most of problems about antenna co-existence. There are many methods can be used to increase the antenna isolation, such as keep enough distance between each antenna, choose proper antenna polarization, etc.

b. Add a GNSS BPF filter.

The customer can add a GNSS BPF filter to filter the interference from other antennas and the environment. The specification of a typical BPF filter is shown in Table 4. From the table, we can see the BPF can provide about minimum 34dB attenuation at GSM and BT frequency band. It can remove the most noise signal from GSM/BT antenna.

Table 4: A typical specification of GNSS BPF filter

Item		Specification		
		-30 to +85°C	25+2/2°C	typ.
Nominal Center Frequency(fc)		1561.10/1575.42/1602 MHz		
Insertion Loss				
1)	1559.05 to 1563.15 MHz	1.8 dB max.	1.6 dB max.	1.3 dB
2)	1573.42 to 1577.42 MHz	1.2 dB max.	1.1 dB max.	0.9 dB
3)	1597.55 to 1605.89 MHz	1.9 dB max	1.6 dB max.	1.4 dB
Absolute Attenuation				
1)	777 to 798 MHz	34 dB min	34 dB min	38 dB
2)	824 to 915 MHz	34 dB min	34 dB min	37 dB
3)	10 to 925 MHz	34 dB min	34 dB min	37 dB
4)	925 to 960 MHz	34 dB min	34 dB min	37 dB
5)	1427 to 1463 MHz	37 dB min	37 dB min	41 dB
6)	1710 to 1785 MHz	35 dB min.	35 dB min	38 dB
7)	1850 to 1980 MHz	35 dB min	35 dB min	39 dB
8)	2400 to 2570 MHz	35 dB min	35 dB min	42 dB
9)	2570 to 3000 MHz	30 dB min	30 dB min	44 dB

4   Components placement and PCB layout

4.1   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 Figure 9.

（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.

Top View   Bottom View   Cross-section View
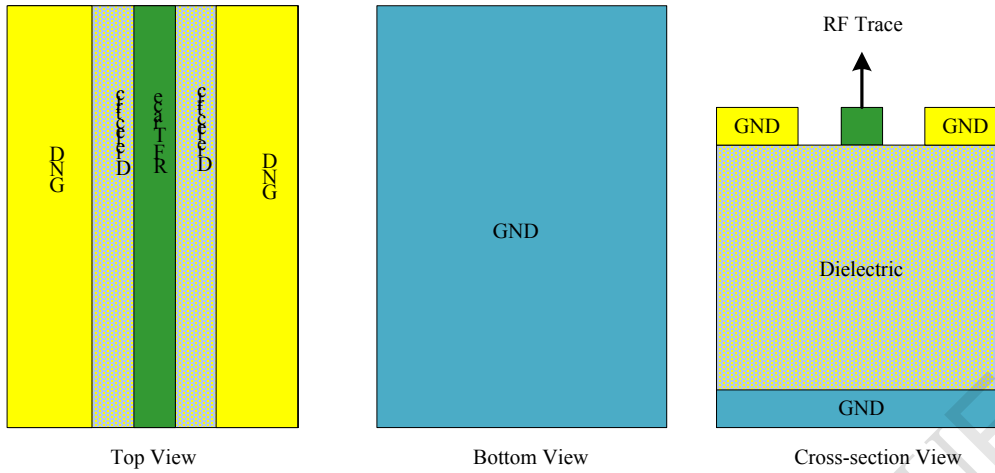
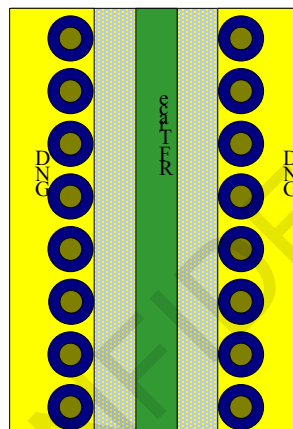**Figure 8: Integrated ground plane under RF trace**


Figure 9: Ground vias surrounding the RF trace

4.2 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.

4.2.1 1.0mm Two-layer PCB

The way to design RF trace in 1.0mm Two-Layer PCB is shown as Figure 10.

Zoom in of Surface Coplanar Line
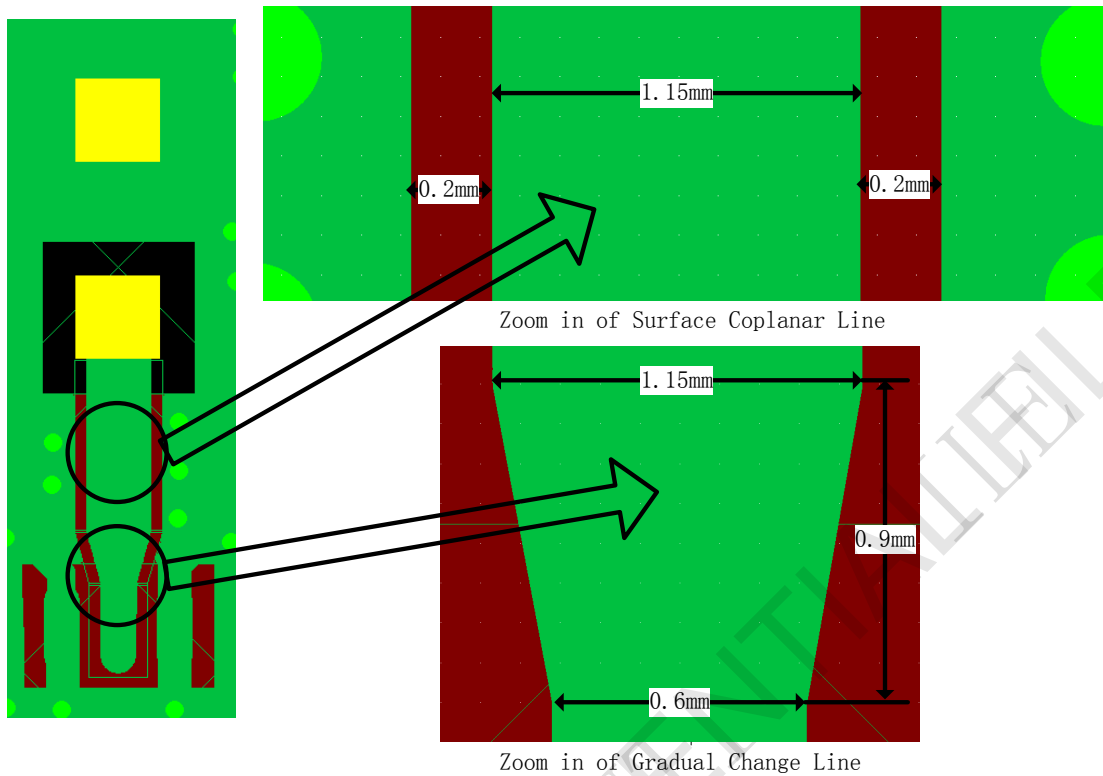
Zoom in of Gradual Change Line
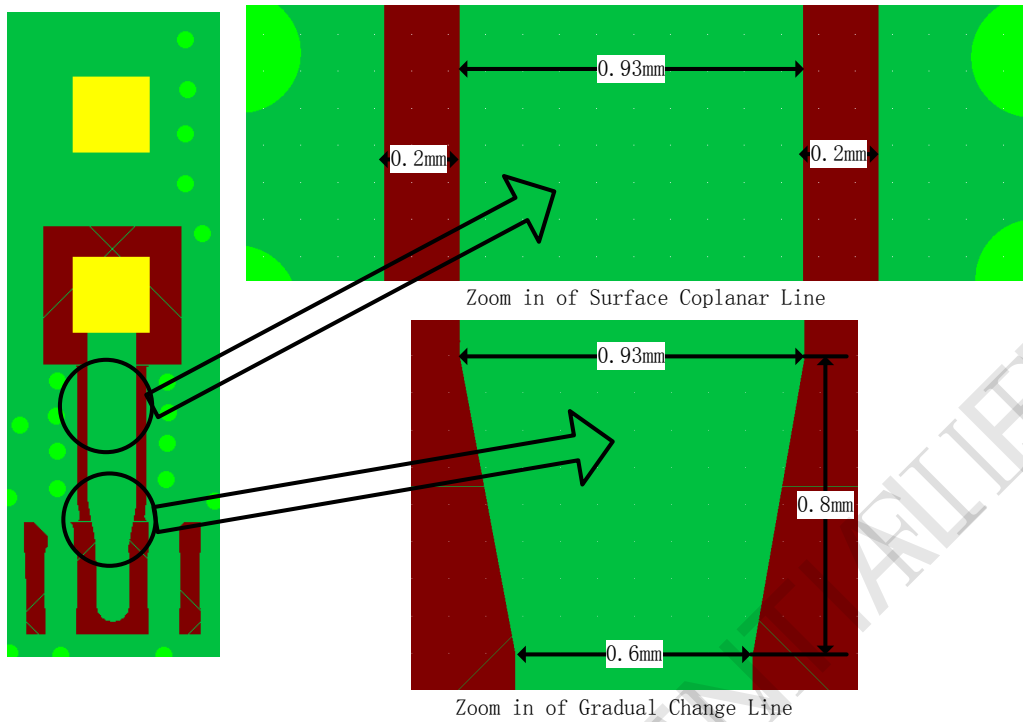
Figure 10: The way to design RF trace in 1.0mm Two-Layer PCB

In Figure 10, the width of RF trace is 0.93mm, the distance between RF trace with ground on each side is 0.2mm.

4.2.2 1.6mm Two-layer PCB

The way to design RF trace in 1.6mm Two-Layer PCB is shown as Figure 11.

Zoom in of Surface Coplanar Line

Zoom in of Gradual Change Line

Figure 11: The way to design RF trace in 1.6mm Two-Layer PCB

In Figure 11, the width of RF trace is 1.15mm, the distance between RF trace with ground on each side is 0.2mm.

4.3 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 Figure 12.
（4）The power supply trace should be surrounded by ground to get better noise decoupling.
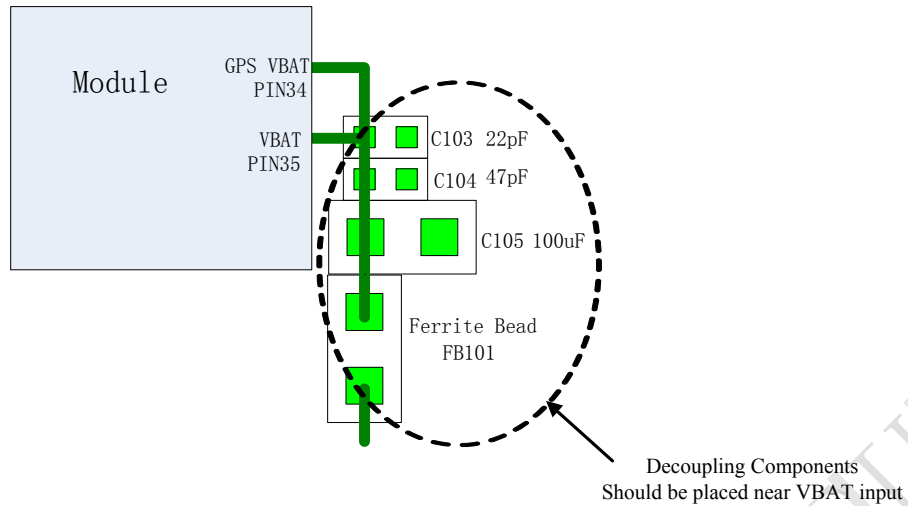
Figure 12:   Decoupling components placement

In Figure 12, ferrite bead FB101，capacitor C103，C104，C105 are mounted on the top layer, near to SIM868 VBAT Pad. 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.

Contact us:
Shanghai SIMCom Wireless Solutions Ltd
Add: SIM Technology Building A,
No. 633, Jinzhong Road, Shanghai, P. R. China 200335
Tel: +86 21 3252 3300
Fax: +86 21 3252 3020
URL: www.simcomm2m.com