SIM68_RF Design_Application Note_V1.00

Document name：	RF Design Application Note
Version：	1.00
Author：	Xiaohan Jin
Date：	2012/05/25
Status：	Release
Document Control ID:	SIM68_RF Design_Application Note_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.

Version History

Version	Chapter	What is new
V1.00	New version	Created the doc

Preface

This document introduces SIM68 RF design application. It explains how to make a PCB layout, whether to add an external LNA or not basing on actual application, and introduces the jamming removing function.

Abbreviations

Terms	Explanation
TTFF	Time to first fix

Contents

1. PCB stack-up and RF transmission lines

1.1 Stack-up

There are some widely used types of PCB, such as two-layer PCB, four-layer PCB. Each type has corresponding ‘stack-up’ respectively.

Here is the ‘stack-up’ of two types of PCB (thickness = 1.0+/-0.1mm), customers may choose one in PCB design, as seen in figure1 and figure2. If the thickness is different, please adjust the ‘stack-up’ accordingly.

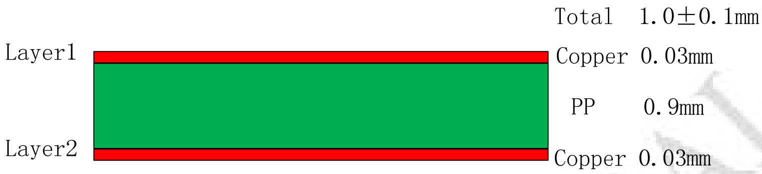


Figure 1: stack-up of Two-Layer PCB (thickness=1.0+/-0.1mm)

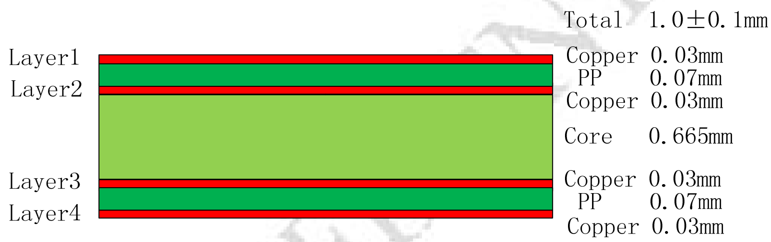


Figure 2: stack-up of Four-Layer PCB (thickness=1.0+/-0.1mm)

1.2 Transmission Line

In PCB layout consideration, a transmission line is a conductor that maintains special characteristic impedance. Since the impedance of the SIM68 module’s RF_ANT port is 50 ohms, the impedance of transmission lines between SIM68 and the antenna should be made to 50 ohms.

According to different types of trace geometry, there are many different forms of transmission lines. Some widely used forms are shown in following figure:

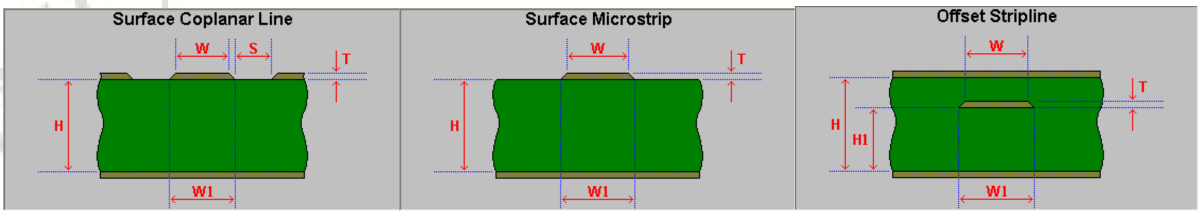


Figure3: Three Different RF transmission lines forms

Surface coplanar line, surface microstrip and offset stripline are three widely used types of transmission line. Customers may change impedance by adjusting RF trace width and separation depending on different 'stack up', as seen in following tables.

Table 1 is concerned about two-layer PCB shown in Figure1.

Table 1: Impedance control of Two Layer PCB (thickness=1.0+/-0.1mm)

SIGNAL layer	GND layer	Target Impedance	Expected Width	Expected Separation
L1(surface coplanar line)	L2	50 Ω	1.0MM (39MIL)	0.23MM (9MIL)
L2(surface coplanar line)	L1	50 Ω	1.0MM (39MIL)	0.23MM (9MIL)
L1(surface microstrip)	L2	50 Ω	1.7MM（67MIL）	
L2(surface microstrip)	L1	50 Ω	1.7MM（67MIL）	

Table 2 is concerned about four-layer PCB shown in Figure 2.

Table 2: Impedance control of Four Layer PCB (thickness=1.0+/-0.1mm)

SIGNAL layer	GND layer	Target Impedance	Expected Width
L1(surface microstrip)	L2	50 Ω	0.114MM（4.5MIL）
L4(surface microstrip)	L3	50 Ω	0.114MM（4.5MIL）
L2(offset stripline)	L1,L3	50 Ω	0.099MM（3.9MIL）
L3(offset stripline)	L2,L4	50 Ω	0.099MM（3.9MIL）

1.3 Impedance Calculation Tool

When calculating the proper trace width, a professional EDA tool, such as CITS25 is helpful. Here is an example for calculating parameters in the form of surface coplanar line.

The interface of the software is shown in the following figure 4.

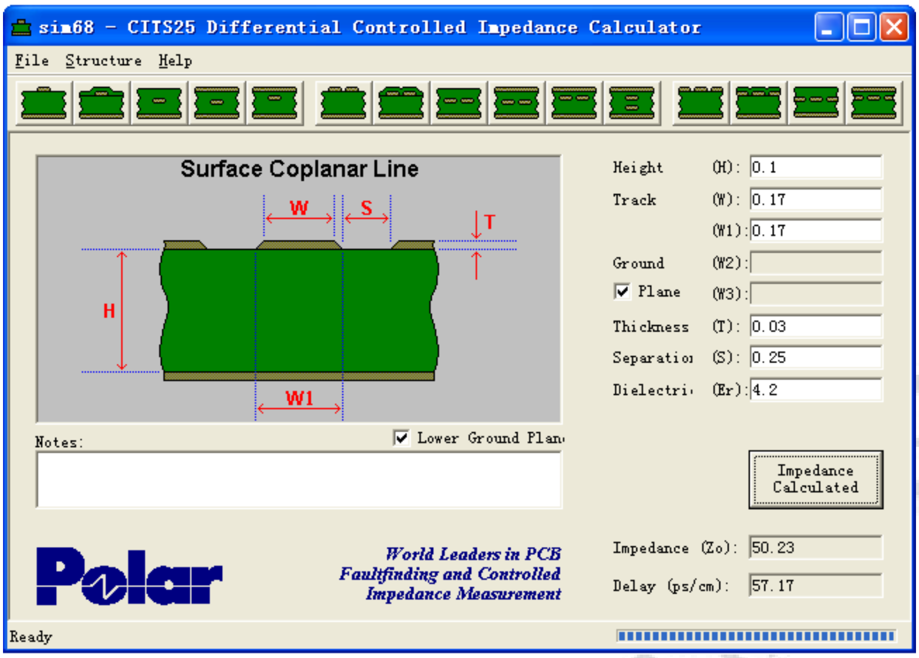


Figure 4: CITS25 tool for calculating proper trace width

All the number filling in the blank should be in the same unit, such as mils, mm etc. Choosing the PCB stack-up in a series of types showed above, Surface Coplanar Line, Surface Microstrip, Offset Stripline etc.

Fill in the blank according the following explanation:

Height (H): 0.1	the thickness of PP between RF trace and reference ground.
Track (W): 0.17 (W1): 0.17	the width of RF trace, W for the top and W1 for the bottom.
Thickness (T): 0.03	the thickness of copper trace.
Separation (S): 0.25	the width between RF trace and ground in the same layer.
Dielectric (Er): 4.2	the dielectric of the pp material.

After filling in all blanks, press "Impedance Calculated", and the Impedance is calculated. All the number can be adjusted in practical use, and the closer to 50ohms the impedance is the better.

2. PCB Layout Consideration

When designing the PCB layout of SIM68, some common rules in RF systems should be followed:

- Keep the trace between antenna pad and RF input straight and as short as possible.
- Control the RF trace in 50 ohm impedance.

- Stitch a row of ground vias with 4mm spacing on either side of the RF input trace.
- Do not run any digital signals directly under GPS module, in any layer.
- Keep all other digital traces away from the GPS module's RF input circuit.
- Add ground vias to reference ground throughout the board.
- Provide a solid, low inductance connection from Vcc to the RF input circuit ground plane.

It is recommended that different antennas should be put as far as possible, basing on the following two reasons:

- First, every antenna contains different kinds of metal, which will reflect the satellite signal, thus reducing the signal strength received by SIM68.
- Second, high power transmission signal from other antenna will raise the noise floor in SIM68 and reduce the CNo.

3. External LNA

Sometimes an External LNA can make the module work better. When making a choice, please consider that SIM68 already has a fairly good LNA (NF=0.9) integrated in the module.

The followings are three typical applications of SIM68:

A. Using an active antenna
In this condition, the following form is recommended:

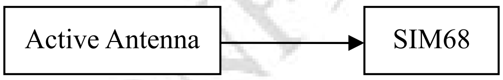


Figure 5 Active antenna application

In this kind of application, the characteristic of LNA in the antenna decides the performance of the system. The path loss on PCB trace is not important. Choosing a high performance active antenna with a low noise figure is most important.

B. Using a passive antenna, with short PCB trace between antenna and SIM68
In this condition, the following form is recommended:

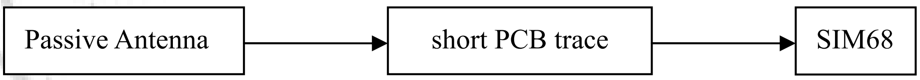


Figure 6 Passive antenna application with short PCB trace

In this kind of application, the short PCB trace contributes little path loss to the system. Considering about the high performance of integrated LNA, if the passive antenna is good enough, no extra LNA is recommended.

C. Using a passive antenna, with long PCB trace or additional RF chip (such as an additional SAW for an environment full of noise) between antenna and SIM68

In this condition, the following form is recommended:

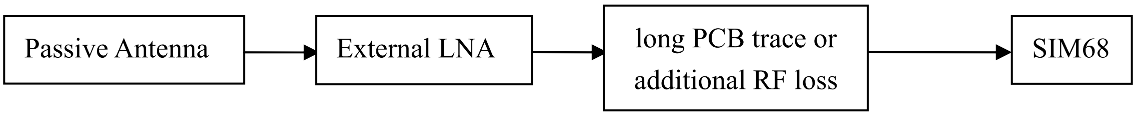


Figure 7 Passive antenna application with long PCB trace or additional RF chip

In this kind of application, long PCB trace or additional RF chips will add several dB loss to the system, which cannot be compensated by the LNA in SIM68. In such condition, an external LNA with a low noise figure (below one) can make the module perform better.

4. Jamming Removing Function

GPS and Glonass signals received on the ground from satellites outer space are low amplitude signals, so they are easily to be interfered by different kinds of jamming signals. The CNo will become lower and the TTFF will become longer.

Sim68 has a special jamming removing function to keep it working well in such condition, and the following table shows its performance under the continuous wave interference.

Table 3: Difference Performance under CW interference (GPS part)

The signal strength is -130dBm in this test. The CW interference frequency is 1575.42MHz.										
Satellite Number	1	6	8	10	13	17	20	24	33	34
CNo(without interference)	42	42	42	42	42	42	42	42	42	42
CNo(-100dBm CW interference)	30	35	36	32	35	34	34	40	33	27
CNo(-90dBm CW interference)	17	25	27	21	25	23	24	33	23	26
CNo (-100dBm CW interference with jamming removing function on)	40	40	40	40	40	40	40	40	40	40
CNo（-90dBm CW interference with jamming removing function on)	36	36	36	36	36	36	36	36	36	36

Table 4: Difference in CNo under CW interference(GLONASS part)

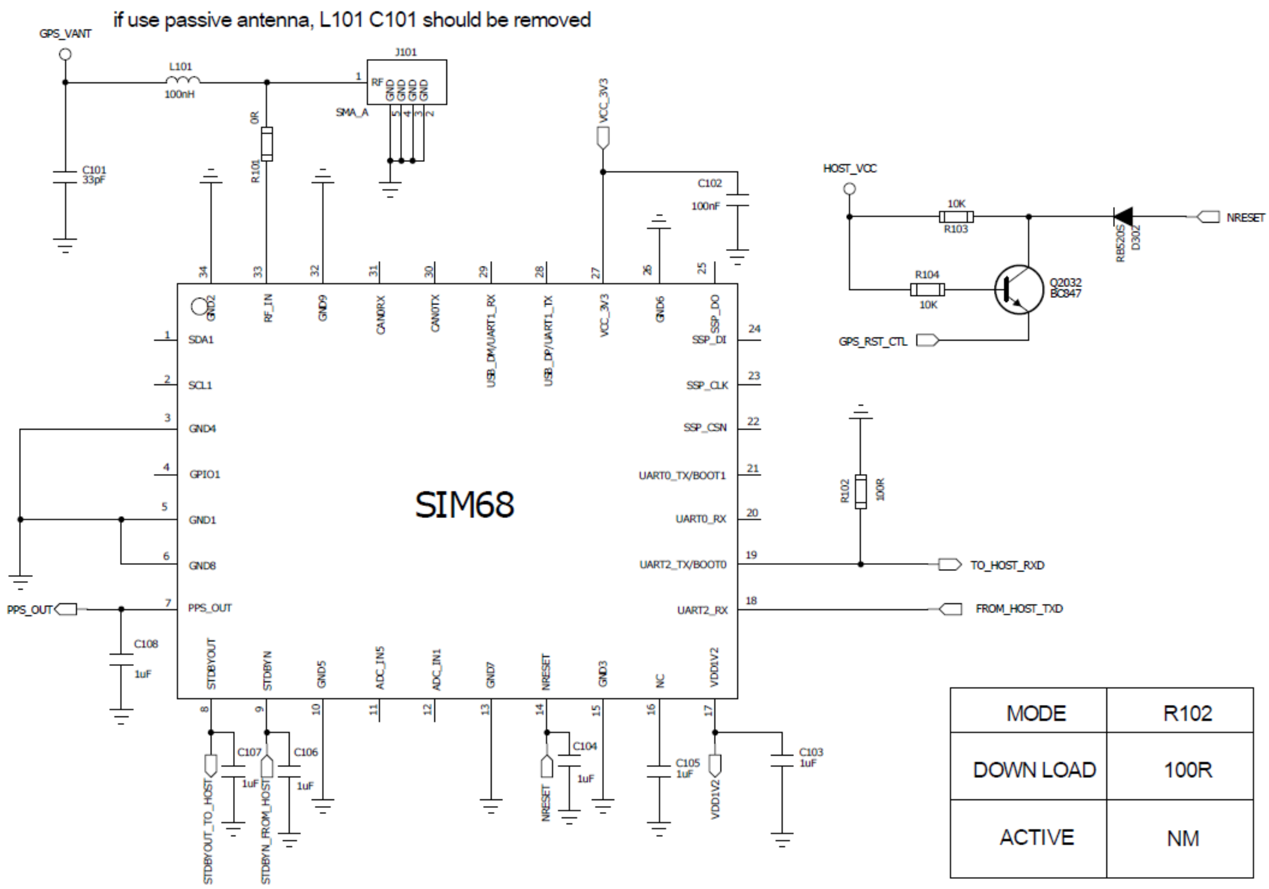
The signal strength is -130dBm in this test. The CW interference frequency is 1603MHz.					
Satellite Number	69	71	72	75	76
CNo(without interference)	40	40	40	40	40
CNo(-100dBm CW interference)	40	40	40	38	33
CNo(-80dBm CW interference)	31	30	30	30	16
CNo (-100dBm CW interference with jamming removing function on)	40	40	40	40	40
CNo (-80dBm CW interference with jamming removing function on)	31	30	30	30	30

From table 3 and table 4, we can see that in-band CW noise can impact CNo seriously without special data processing method. After switching on SIM68's jamming removing function, the CNo recovers a lot, which shows its ability in stable working in environment full of noise.

An advantage of the Glonass system is also shown in the test. Since it has a wider working band, the CW interference can only affect several satellite signals, not the whole, and the other can still work well in this condition.

Appendix

A. SIM68 test board circuit and layout



MODE	R102
DOWN LOAD	100R
ACTIVE	NM

Figure 8: Test board Circuit

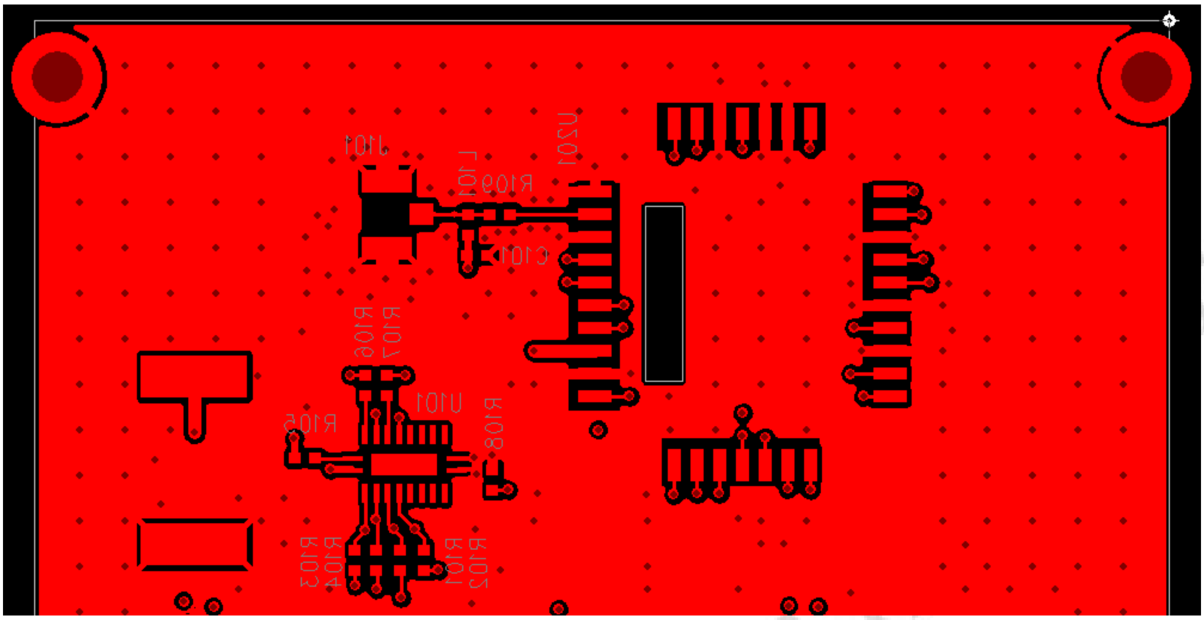


Figure 9: Test Board Layer1

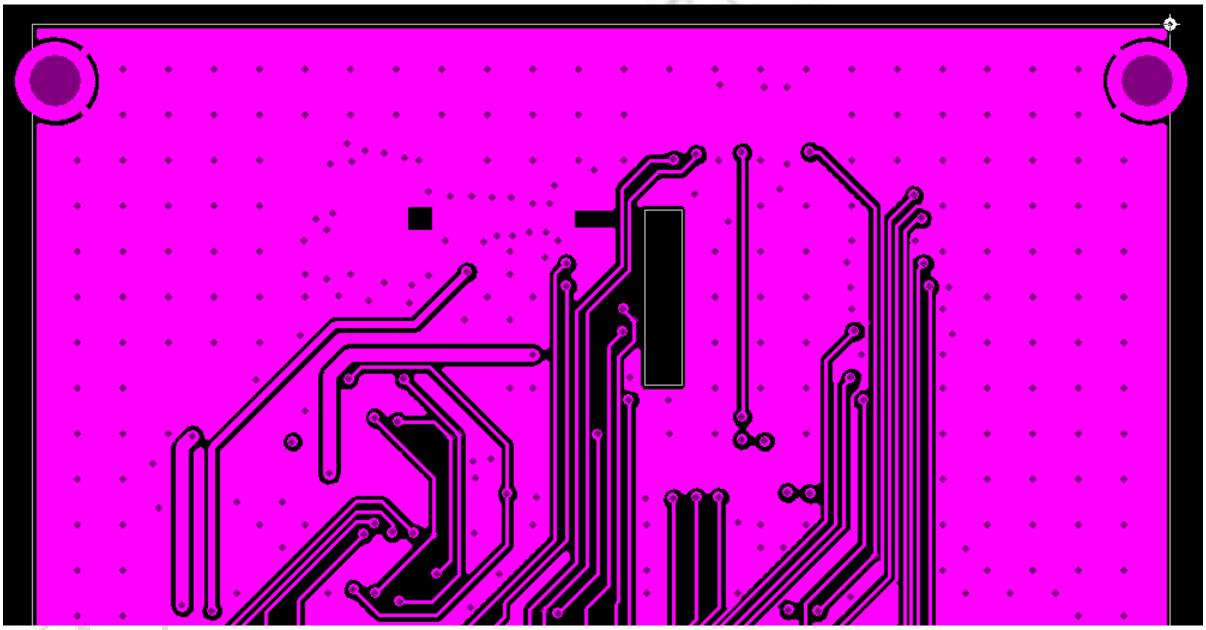


Figure 10: Test Board Layer2

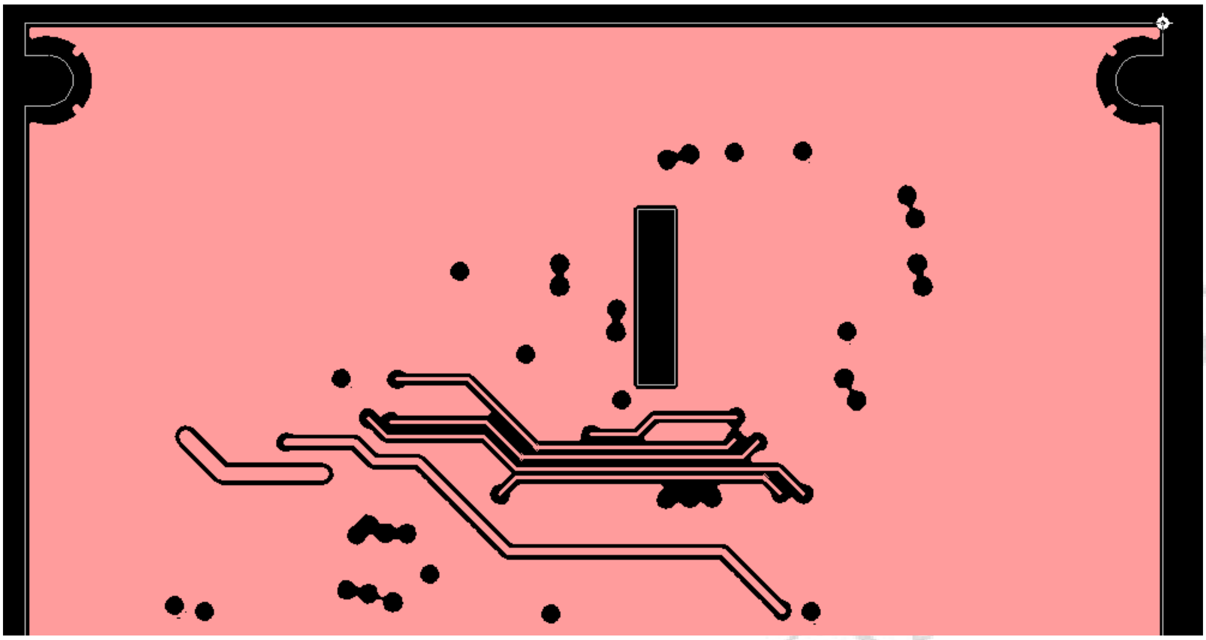


Figure 11: Test Board Layer3

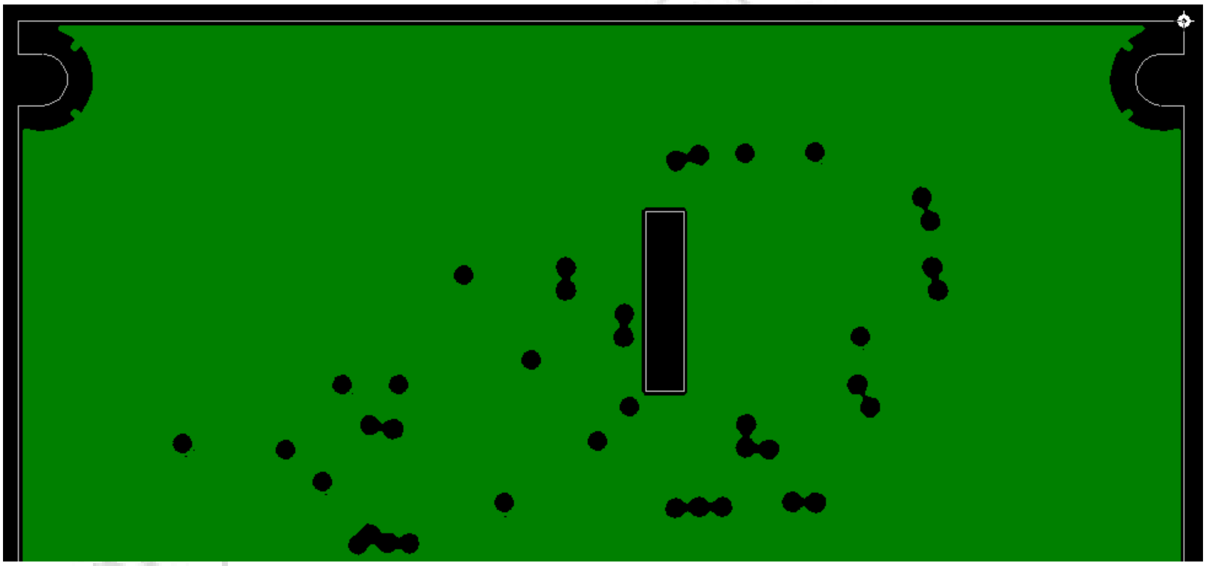


Figure 12: Test Board Layer4

Contact us:
Shanghai SIMCom wireless solutions Ltd.
Address: Building A, SIM Technology Building, No. 633 Jinzhong Road, Shanghai, P. R. China 200335
Tel: +86 21 3252 3300
Fax: +86 21 3252 3301
URL: www.SIM.com/wm