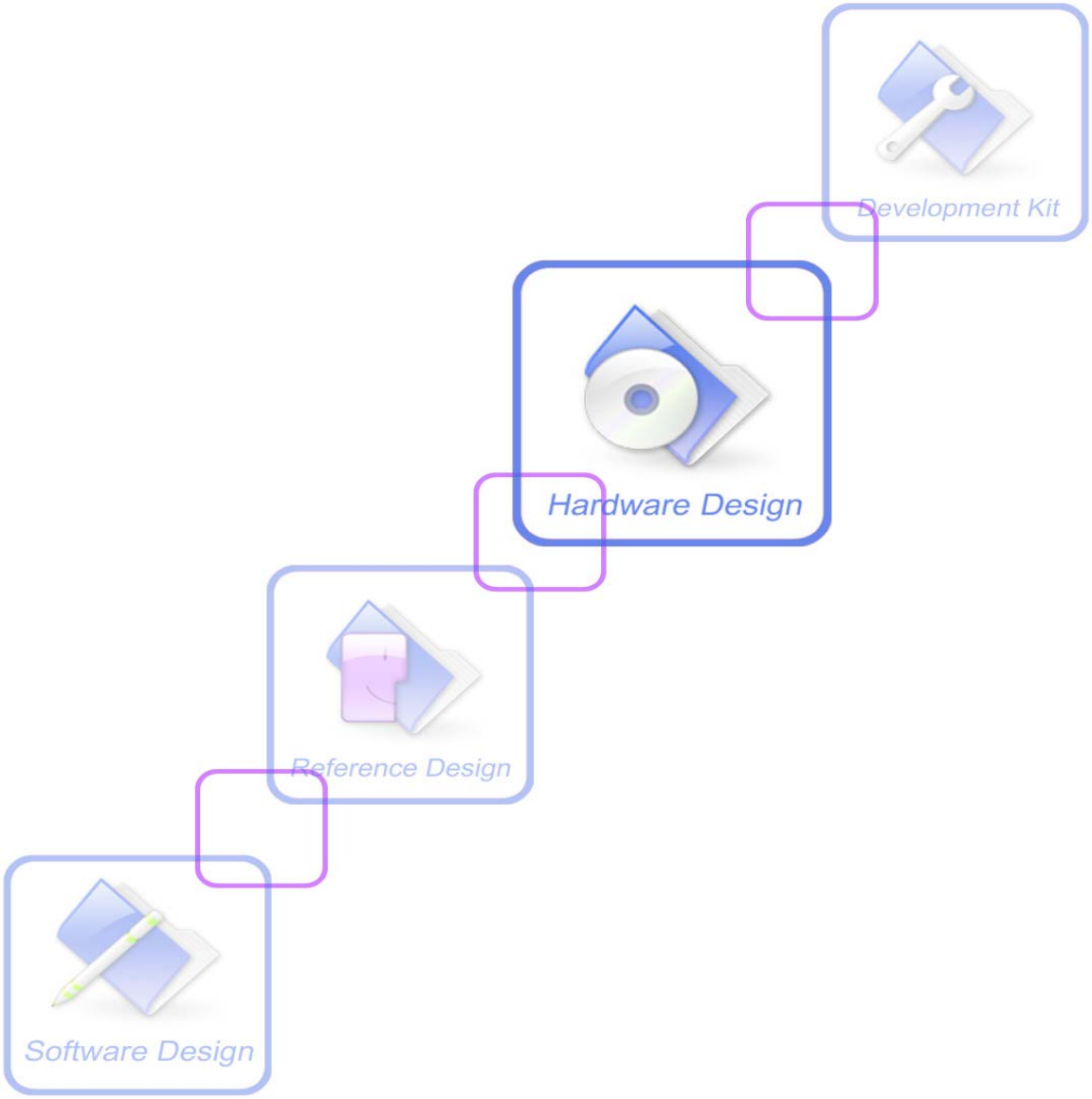
SIMCom
A company of SIM Tech

Application Notes for SIM33ELA _V1.00



Document Title	Application Notes for SIM33ELA
Version	1.00
Date	2014-03-03
Status:	Release
Document Control ID	Application Notes for SIM33ELA

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

Contents

1 Introduction

This document aims to support customers with antenna design for module SIM33ELA, external antenna and internal antenna can be applied. There are three ways to accomplish this.

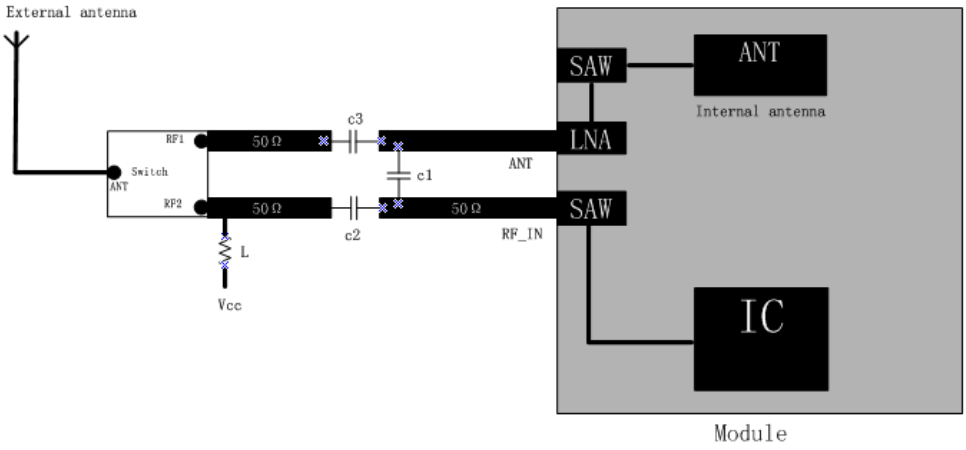


Figure1 Block diagram

- Only with internal antenna for SIM33ELA
- Only with external antenna for SIM33ELA
- With both external and internal antenna by the switch for SIM33ELA

2 Notes for Internal Antenna of SIM33ELA

2.1 Reference Design for Internal Antenna

If using the internal antenna, make sure the pad ANT be connected externally to RF_IN pad via a short trace with 50Ω impedance. The RF input signal path contains a first SAW band-pass filter before LNA, which provides excellent protection against out-of-band GNSS blocking caused by possible near-by wireless transmitters.

Figure 2 shows the reference design circuit for internal antenna. ANT pin and RF_IN could be connected with a short trace with 50Ω.

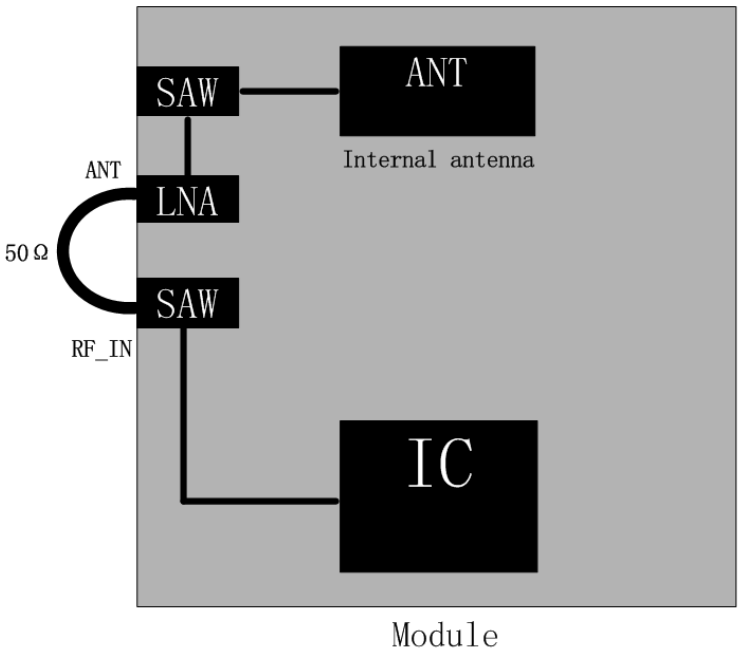


Figure2 Reference design circuit for internal antenna

2.2 Layout suggestion

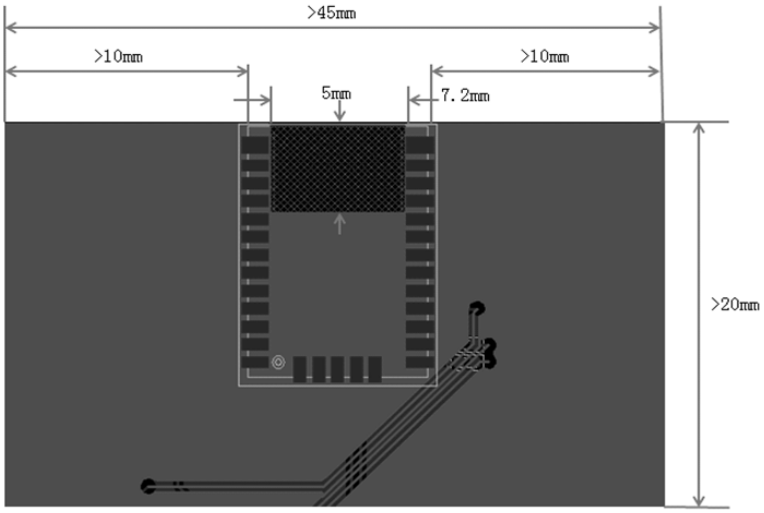


Figure3 Mother board ground plane and SIM33ELA placement

Under the internal antenna there should be keep out 5x7.2mm. Any trace should not be layout under the module directly. This area should be dedicated to keep-out to both traces and assigned to ground plane (copper plane), except for via holes, which can be placed close to the pad under the module. If possible, the amount of via holes underneath the module should be minimized. Note that the internal GNSS antenna requires a small ground plane clearance area (keep out 5x7.2 mm) without copper plane & trace for all layers under the antenna. Placement of other components is not allowed under the keep

out on opposite side.

2.3 Location Suggestion

Modules are installed at various locations of circuit board as shown below and their radiation performances are measured in 3D chamber. The gray segments are ground clearance areas and brown area is ground plane.

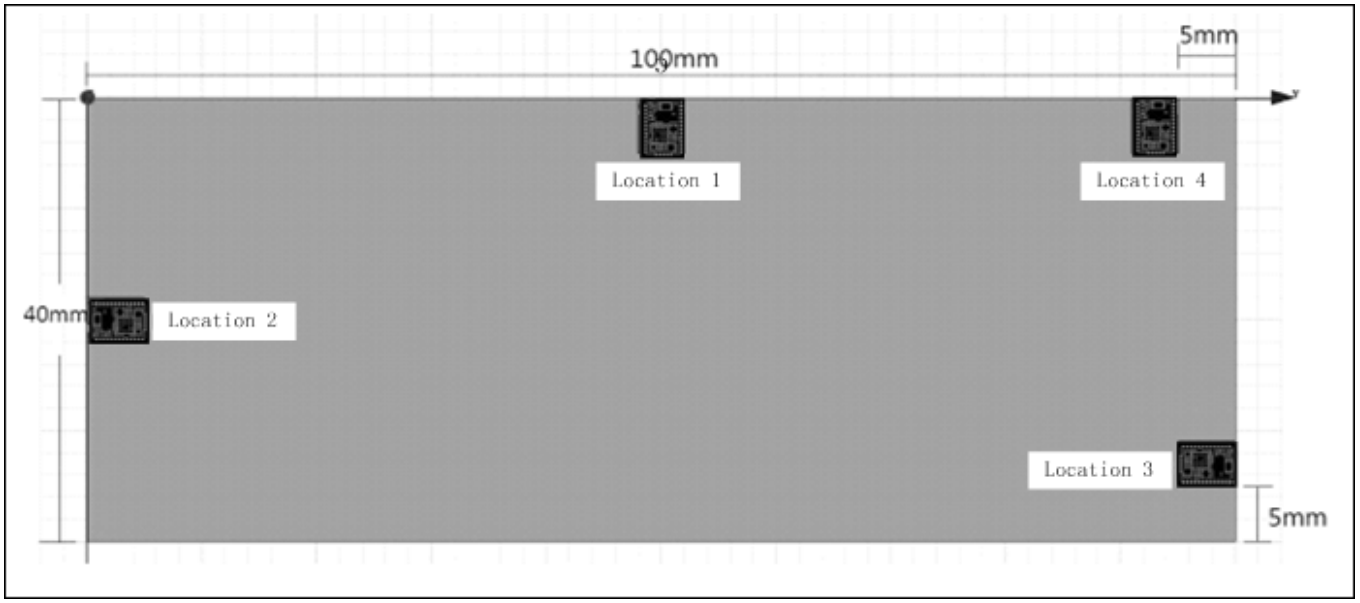


Figure4 various location of module

Table1 Radiation efficiency vs. Frequency at Various Locations

Unit: %

Frequency(GHz)	1.570	1.571	1.572	1.573	1.574	1.575	1.576	1.577	1.578	1.579	1.580
Location 1	77.50	78.31	78.72	79.74	79.95	80.16	79.95	79.95	79.95	79.95	79.94
Location 2	36.84	37.06	37.58	37.96	38.19	38.50	38.42	38.04	37.81	37.43	36.99
Location 3	24.55	24.95	25.09	25.22	25.22	25.27	25.22	25.04	24.69	24.38	23.99
Location 4	42.50	43.11	43.73	43.99	44.26	44.36	44.08	43.73	43.19	42.58	41.48

Choices of module locations and the dimensions of circuit board, Location Priority:

Location 1 ⟶ Location 4 ⟶ Location 2 ⟶ Location 3

The recommended minimum dimensions of circuit board are 40*7mm.The minimum distance recommended between module and corner is 5mm.

2.4 Metallic components placement suggestion

A, B and C directions should not have any metallic components. If metallic components are presented below module (B direction), at least 3mm distance is required.

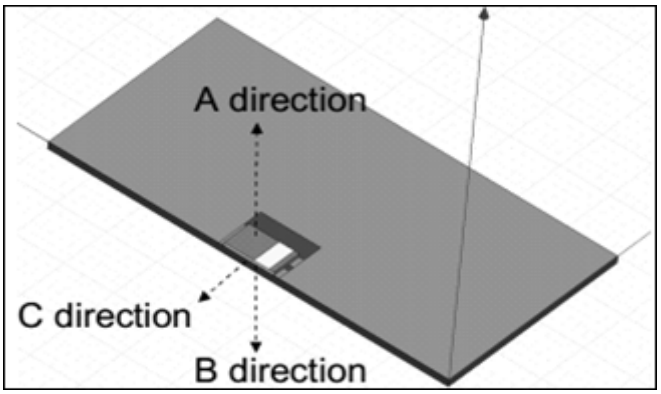


Figure 5

The distance t should be greater than 2mm when the height of metallic component is under 2mm. The distance t should be greater than 5mm when the height of metallic components is above 3mm.

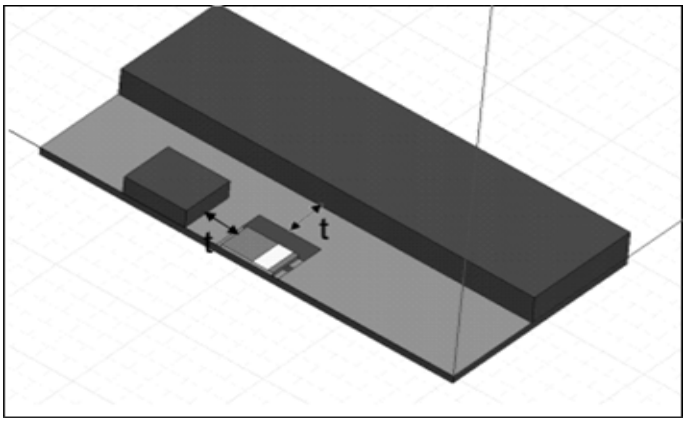


Figure 6

Figure 5, 6 Distance between module and metallic components

The SIM33ELA module is intended to be assembled at the top edge of the mother board. The internal antenna performance relies on the ground plane of mother board, the recommended size is 80x40mm, but larger or smaller ground plane can be used.， minimum ground plane size is 45x20mm, it could affect antenna performance if in smaller size. Optimum placement is at the center of the top edge but offset placement is allowed by keeping at least 10mm distance to nearest ground plane edge.

3 Notes for external Antenna of SIM33ELA

External antenna can be directly connected to RF_IN pin. Care also should be taken for external antenna in detail. If using active external antenna, Vcc is needed for power supply. What calls for special attention is that the antenna must be decoupled from DC voltage with a inductance of L(33nH). C2(22pF) is necessary for being connected to eliminate the DC. And if using passive external antenna, VCC and C2 are not necessary.

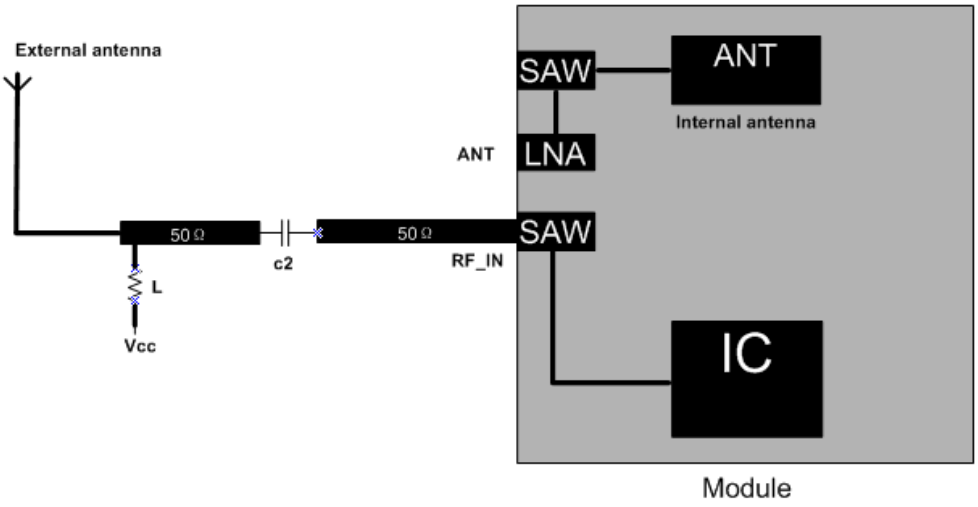


Figure7 Reference design circuit for external antenna

When using external antenna to obtain excellent receiving performance, a good choice will always be required. The antenna is the most critical item for successful GNSS receiving in a weak signal environment. Proper choice and placement of the antenna will ensure that satellites at all elevations can be seen, and therefore, accurate fix measurements are obtained.

Sometimes it also contains a passive matching network to match the electrical conn ection to 50 Ohms impedance.

The most common antenna type for GNSS applications is the patch antenna. Patch antennas are flat, generally have a ceramic and metal body and are mounted on a metal base plate.

The GNSS Patch antenna consists of a radiating patch on one side of a dielectric material substrate backed by a ground plane on the other side, as shown in figure 7:

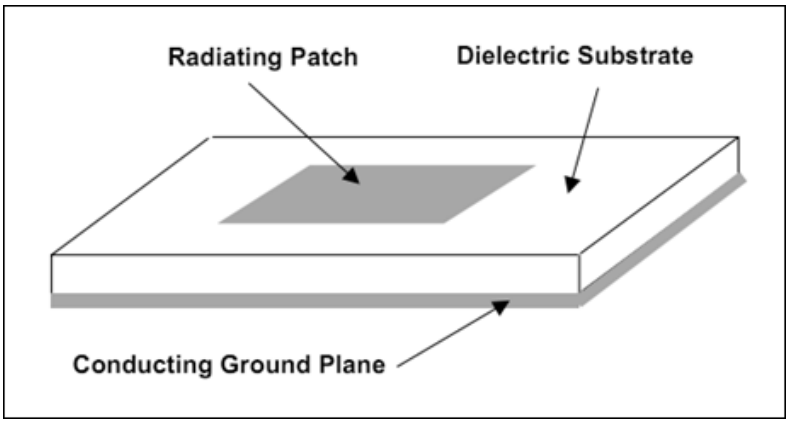


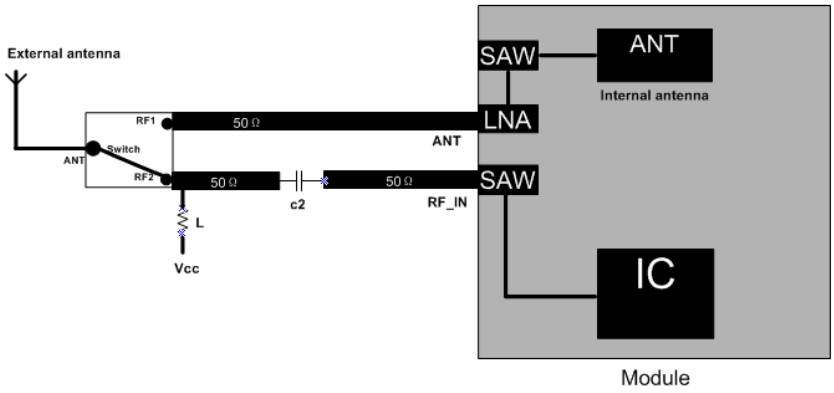
Figure 8 Basic features of GPS pa3tch antenna

When the external antenna is integrated into the customer's product, the following rules should be followed strictly It includes:

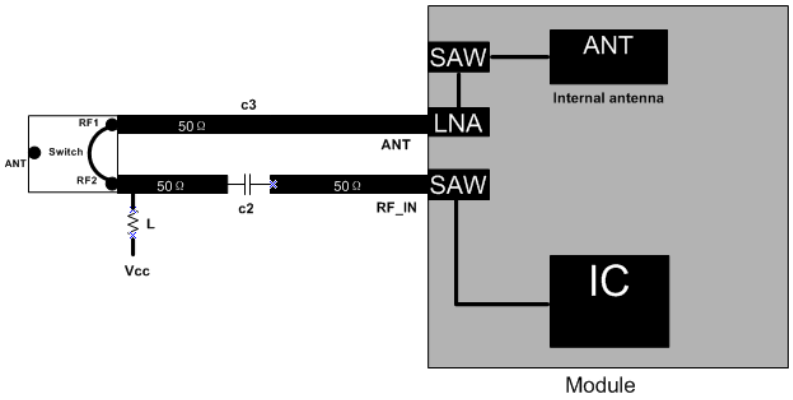
1) The most important rule is to ensure the antenna towards the sky.
2) The antenna should not be covered by any metal device or metalized enclosure. It is because the metal device will block the most GNSS signal reach to the antenna
3) The height of the device around the module should not exceed the antenna at least. Otherwise, the antenna performance will be affected more or less.
4) The device carry large amounts of interfering signals around the module, should be placed far away from the module, or should be shielded by a shielding can.

4 Notes for external and internal Antenna of SIM33ELA

A switch is necessary for using both external and internal antenna on SIM33ELA. Reference design is shown in Figure 9. As shown in Figure 9(a) if external antenna is connected with the switch ANT port, the switch works on the mode of ANT connecting RF2 and RF1 open. Also VCC and C2 are necessary if active external antenna is used.. As shown in Figure 9(b) if external antenna is not used, switch works on the mode of RF1 connecting RF2 and ANT open. The object of the switch is shown in Figure10. More Information about the switch could be gotten from relevant company's website http://www.aliner.com.tw



(a)



(b)

Figure9 reference design circuit for both external and internal antenna by the switch

Figure10 object of switch (Aliner P/N 12-301AA-T)

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 3328

Fax: +86 21 3252 2030

URL: www.sim.com/wm