Perfect Wireless Experience
完美无线体验

FIBOCOM_RF Antenna Design Application Note

Version：V1.0.6

Date：2018-05-22

Applicability Type

No.	Type	Note
1	All modules	

Copyright

Attention

The document is subject to update from time to time owing to the product version upgrade or other reasons. Unless otherwise specified, the document only serves as the user guide. All the statements, information and suggestions contained in the document do not constitute any explicit or implicit guarantee.

Trademark

The trademark is registered and owned by Fibocom Wireless Inc.

Versions

Version	Date	Remarks
V1.0.0	2013-07-31	Initial Version
V1.0.1	2015-04-26	Update the description of copyright and attention.
V1.0.2	2015-08-24	Update the logo.
V1.0.3	2015-12-05	Add LTE information
V1.0.4	2017-12-13	Add RF bands
V1.0.5	2018-05-21	Add RF bands of CDMA/EVDO
V1.0.6	2018-05-22	Add description of GNSS Antenna (chapter 3.2) Update Figure 2-1

Contents

1 RF Interface

1.1 Introduction

There are two kinds of RF interface:

1) RF connector comes with module, you can connect to the antenna by a cable.

2) The module only has a RF antenna welding plate.

It also includes main antenna and diversity antenna:

1) Main antenna: GSM/WCDMA/TD-SCDMA/LTE/CDMA/EVDO Tx and Rx.

2) Diversity antenna: WCDMA diversity Rx.

1.2 Operation Band

Operating Band	Description	Mode	Tx (MHz)	Rx (MHz)
Band 1	2100MHz	LTE FDD/WCDMA	1920 - 1980	2110 - 2170
Band 2	1900MHz	LTE FDD/WCDMA /CDMA/EVDO	1850 - 1910	1930 - 1990
Band 3	1800MHz	LTE FDD	1710 - 1785	1805 - 1880
Band 4	1700MHz	LTE FDD/WCDMA	1710 - 1755	2110 - 2155
Band 5	850MHz	LTE FDD/WCDMA /CDMA/EVDO	824 - 849	869 - 894
Band 7	2600Mhz	LTE FDD	2500 - 2570	2620 - 2690
Band 8	900MHz	LTE FDD/WCDMA	880 - 915	925 - 960
Band 11	1500MHz	LTE FDD	1427.9 - 1447.9	1475.9 - 1495.9
Band 12	700MHz	LTE FDD	699 - 716	729 - 746
Band 13	700MHz	LTE FDD	777 - 787	746 - 756
Band 17	700MHz	LTE FDD	704 - 716	734 - 746
Band 18	800MHz	LTE FDD	815 - 830	860 - 875
Band 19	800MHz	LTE FDD	830 - 845	875 - 890
Band 20	800MHz	LTE FDD	832 - 862	791 - 821
Band 21	1500MHz	LTE FDD	1447.9 - 1462.9	1495.9 - 1510.9

Operating Band	Description	Mode	Tx (MHz)	Rx (MHz)
Band 26	850MHz	LTE FDD	814 - 849	859 - 894
Band 28	700MHz	LTE FDD	703 - 748	758 - 803
Band 29	700MHz	LTE FDD	N/A	716 - 728
Band 30	2300MHz	LTE FDD	2305 - 2315	2350 - 2360
Band 66	1700MHz	LTE FDD	1710 - 1780	2110 - 2200
Band 38	2600MHz	LTE TDD	2570 - 2620	
Band 39	1900MHZ	LTE TDD	1880 - 1920	
Band 40	2300MHz	LTE TDD	2300 - 2400	
Band 41	2500MHZ	LTE TDD	2496 - 2690	

1.3 Antenna Requirements

(1) Antenna Efficiency

Efficiency > 40%

(2) S11 or VSWR

S11 <-10dB, VSWR< 2:1

(3) Polarization

Linear polarization is recommended: it would be better if the polarization direction of diversity antenna is different with main antenna.

(4) Radiation Pattern

Omni-directional

(5) Gain

Recommended antenna gain ≤ 2.5dBi

(6) Bandwidth

Comply with the relevant standards

(7) TRP/TIS

TRP (Total Radiated Power)：

- W850/W900/W1900/W2100>19dBm
- GSM850>28dBm
- GSM900>28dBm
- DCS1800>25dBm
- PCS1900>25dBm
- LTE FDD/LTE TDD ALL Band >19dBm
- TD-SCDMA ALL Band >19dBm
- CDMA/EVDO >19dBm

TIS (Total Isotropic Sensitivity)：

- W850/W900<-102dBm
- W1700/W1900/W2100<-103dBm
- GSM850<-102dBm
- GSM900<-102dBm
- DCS1800/PCS1900<-102dBm
- LTE FDD/LTE TDD ALL Band <-95dBm (10MHz Bandwidth)
- TD-SCDMA ALL Band <-104dBm
- CDMA/EVDO<-102dBm

2 RF PCB Design

2.1 Layout Guideline

As Fibocom module does not have a RF connector, so customers need to trace a RF line to connect to antenna, or design a connector on the customer board. We recommend customers use a microstrip line, the shorter the better, insert loss should be less than 0.2dB; impedance should be less than 50ohm.

Add a π-type circuit (two parallel device grounded pin connect directly to the main land) between the module and antenna connector for antenna matching.

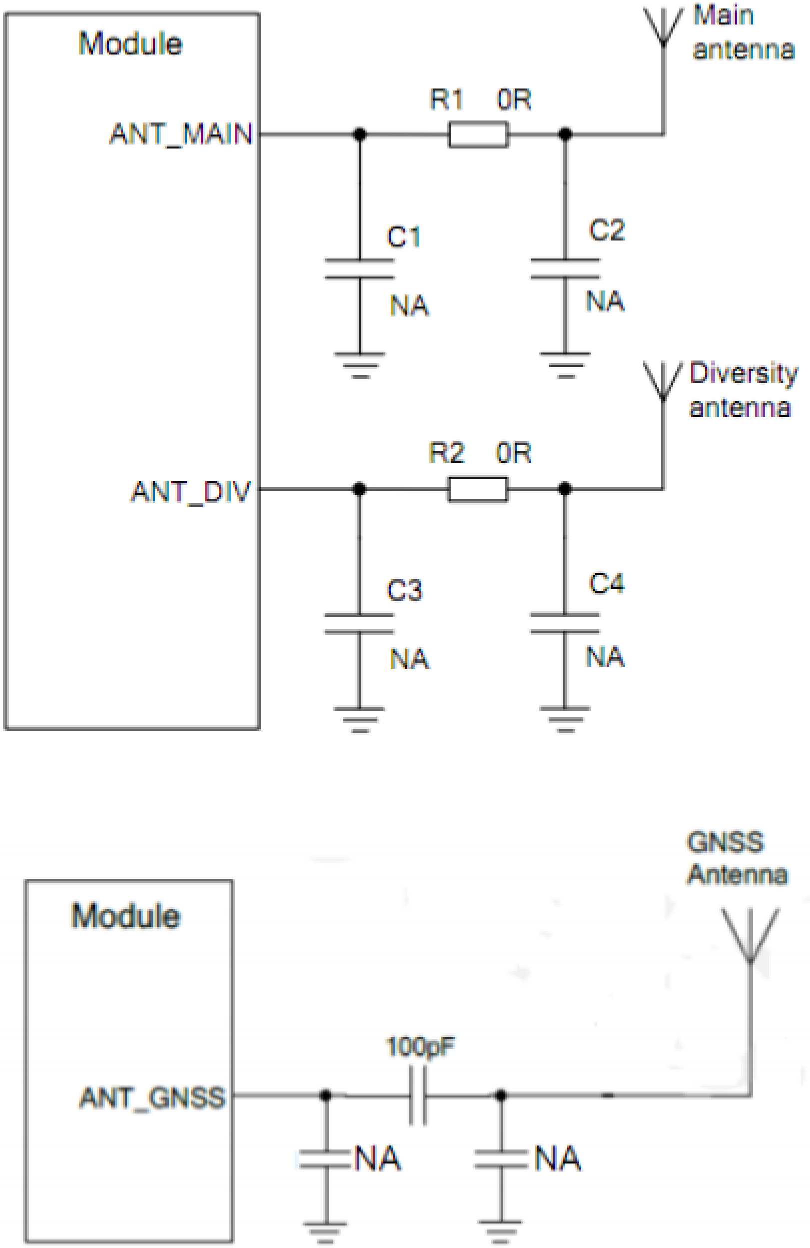

Figure 2-1π-type circuit

This trace controls 50Ω, and the RF performance is closely related to this trace. Here are some PCB parameters which will affect impedance:

- Trace width and trace thickness
- Dielectric constant and thickness
- The thickness of pad
- Distance with ground wire
- Nearby trace

2.2 Impedance Design

The RF impedance of the two antennas' interface should not exceed 50ohm.

In practice, the other PCB parameters like thickness, number of layers and laminate can affect the RF trace, reference GND layer is different under different situation, and the trace would be different.

2.2.1 3W Principle

When antenna RF signal traces on PCB, the first thing you need to consider is to follow "3W principle".

In order to reduce crosstalk between the lines, please ensure that line spacing is large enough, if the line spacing is at least 3 times of the width, 70% of the electric field between the lines does not interfere with each other, and this is called "3W principle".

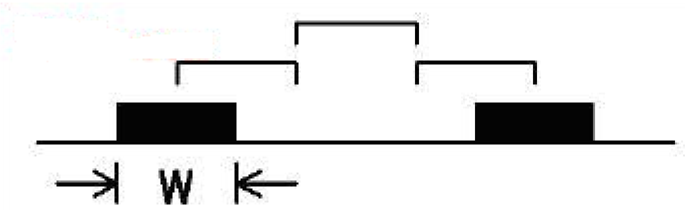

Figure 2-2 3W principle

2.2.2 Impedance Design for Double Layers

As most customers use double panel, here are the examples for 1.6mm PCB and 1.0mm PCB.

Case 1: 1.6mm PCB

In the following picture, the white area shows 50ohm impedance control line. Considering the thickness of the board, it is hard to follow 3W principle, so we suggest the customers do not place any component near the antenna, and the trace on PCB should be far from the RF part.

It's been proved that the following design can ensure that RF impedance does not exceed 50ohm, and least affected.

The RF line width is 43 mil, and the distance with GND (blue) is 8 mil. The reference layer is below the RF trace; normally which is large ground (red). Beside the RF trace, there should be large areas of paved copper, punch a lot of holes along the trace direction. Do not put any other traces or component near the RF trace and reference ground, as shown in the figure 2-3:

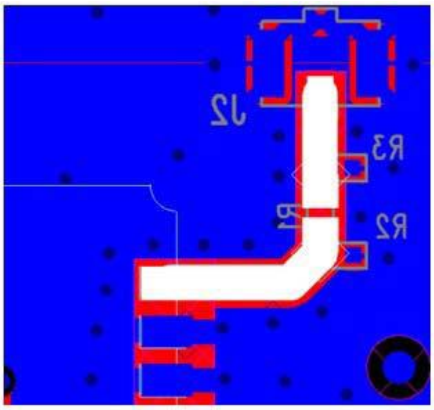

Figure 2-3 50ohm impedance line

Impedance calculation:

Figure 2-4 Impedance Calculation

Unit（mil）

The factory confirms the Er1, T1, C1, C2, and CEr parameters generally, sometimes you need to confirm with the board factory due to different process.

H1 represents PCB thickness, W1 represents trace width and D1 represents the distance from trace to GND. Please notice error in practice.

Case 2: 1.0mm PCB

When the thickness of PCB is 1.0mm, the RF line width is 35mil, and the distance with GND (blue) is 8 mil.

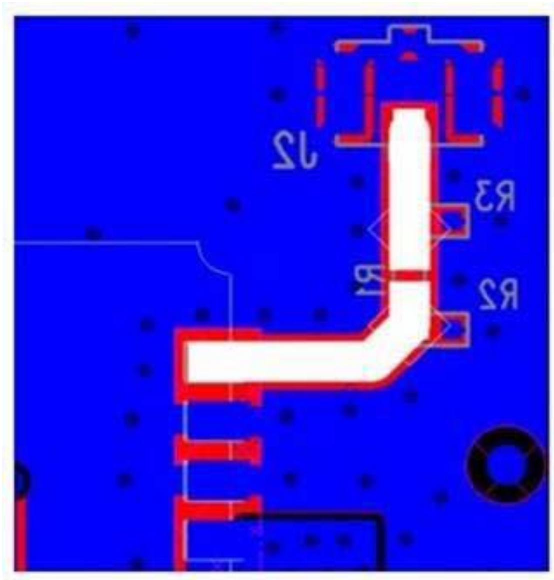

Figure 2-5 RF Trace

Impedance calculation:

Figure 2-6 Impedance Calculation

2.2.3 Impedance Design for Four Layers

Four layers, the thickness of the PCB is 1.0mm, RF line traces in Lay1, reference layer is Lay2 (GND Layer). The laminates are different from different factories; you can refer to the following figure:

4 Layers (1+2+1)

	Layer	Finished Thicknes		Material
	S/M			
1	Cu	25	um	0.33OZ + Plating
	Prepreg	65	um	PP 1080
2	Cu	25	um	0.5OZ + Plating
	Core	508	um	0.540mm(H/H OZ)
3	Cu	25	um	0.5OZ + Plating
	Prepreg	65	um	PP 1080
4	Cu	25	um	0.33OZ + Plating
	S/M			
	Total	800	um	
	Tolerance	±100	um	

Figure 2-7 4 Layers Lamination

The thickness from Lay1 to Lay2 is 65um, RF traces 4mil, the distance from RF to GND should be higher than 3 times of RF width.

The blue area is Lay2 and the red area is Lay1, the highlighted part is RF line.

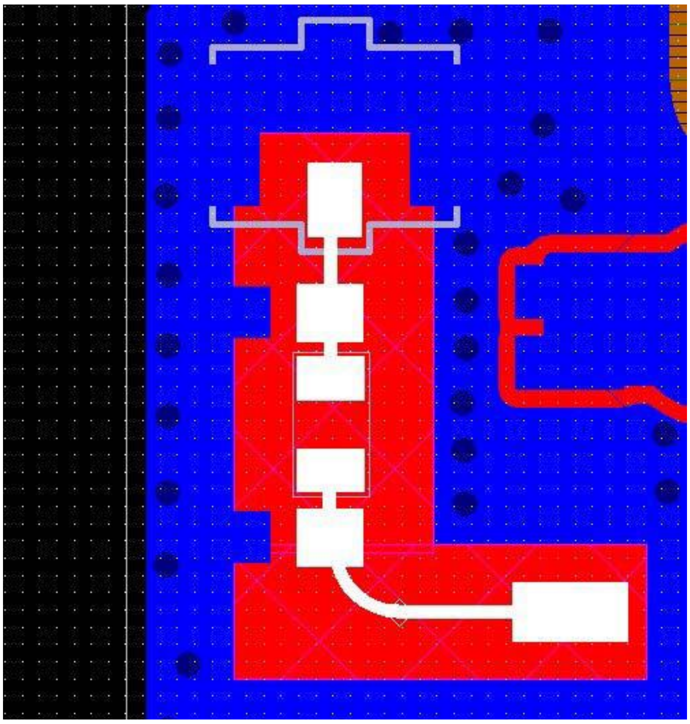

Figure 2-8 RF Trace

Impedance calculation:

If the value of D1 exceeds 3 times of W1, it has weak effect on impedance.

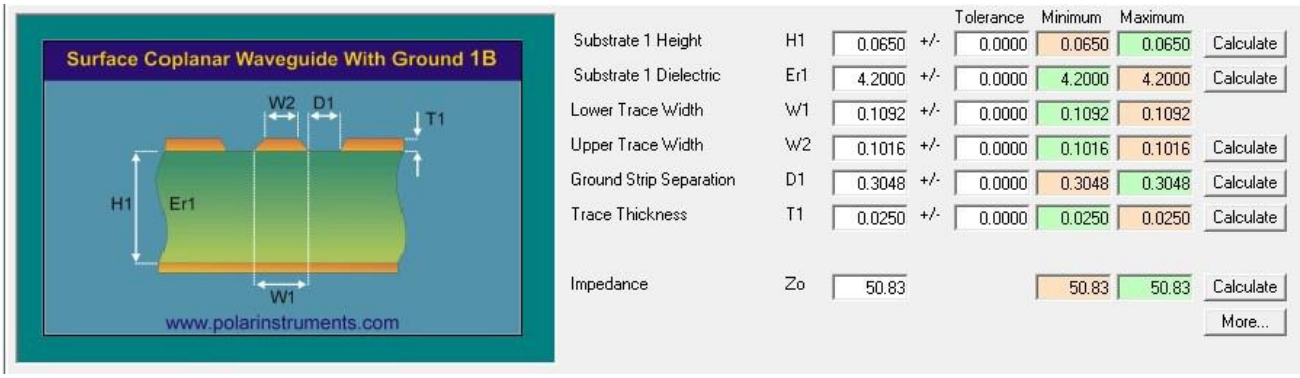

Figure 2-9 Impedance Calculation

3 Main Antenna Design

3.1 External Antenna

External antenna is easy to design; there is enough space to ensure the performance. Normally, it uses SMA connectors.

Figure 3-1 External Antennas

3.2 GNSS Antenna

3.2.1 Design principle of passive antenna

It is necessary to avoid interference sources around the antenna, and put them in four corners of the board as far as possible, away from GSM, WIFI and BT antennas, and do not need DC bias voltage.

3.2.2 Design principle of active antenna

The active antenna is integrated with a LNA (Low Noise Amplifier), and LNA needs power supply. Some modules support direct power supply and power LNA through GNSS_ANT. Some module models do not support the power supply to LNA, so the external power supply is needed. The active antenna needs be far away from any noise source to gets good performance, the active antenna recommends placed on the four corners of the board, and as far as possible, away from GSM, WIFI and BT antennas.
The RF signal needs be connected to an external power source through a 100nH inductor and not be connected directly, thus preventing power noise from entering the antenna.

3.3 Internal Antenna

3.3.1 Design Guideline

1) Layout

Avoid installing any scattered unshielded components around the RF module, the shielding box should be regular, at the same time, do not punch too much ventilation holes.

Do not use hole slot which is long shape.

The component which contains metal structure like speaker, vibrator, and camera substrate should be grounded.

For fold and slide machine, the FPC cannot be too long, please add grounding shield on both sides.

2) Wiring

Please use arc in the bend when tracing the signal line which is related to RF, notice the isolation and characteristic impedance.

RF ground should be well-designed; the antenna leads from RF module should be microstrip line.

The antenna RF feeder welding plate use rounded rectangle plate; the size is 2mm x 3mm, don’t cover copper on all the PCB layers in pads and the around area ≥0.8mm.

When there are two feed points, the center distance between RF and ground pad should in the range of 4mm~5mm.

3.3.2 Internal Antenna Types

There are three types: PIFA antenna, IFA and Monopole. As internal antenna may cause some interference problems to the product, so it has more requirements.

Here are the comparisons between PIFA and Monopole:

Type	Below the Antenna Projection	Antenna Feed	Antenna Volume	Electric Properties	SAR
PIFA	Grounded	2	Big	Excellent	Low
Monopole	Not grounded	1	Small	Good	High

3.3.2.1 PIFA Antenna

1) Antenna structure

There are two feeding points between the antenna and main board, one is module output, the other is RF ground. The antenna is on the top of the device. The distance between the signal point and earth point should be at least 4-5mm.

Put signal point and earth point in different place, place more earth points would be help when design the antenna.

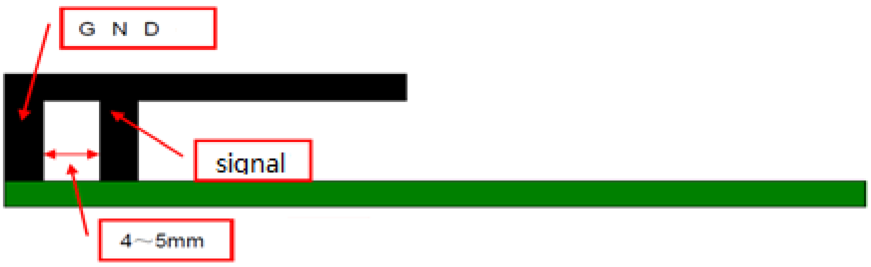

Figure 3-2 Location of signal point and earth point for PIFA antenna

2) Main Board

There are complete paving inside the antenna projection area, do not place any component in the antenna area, the length of PCB board should be 90-110mm, when the length of the antenna is 105mm, its performance is better.

3) Structure Style

a. scaffolding

The antenna consist of plastic bracket and metal (radiator). Plastic bracket and metal are fixed by hot melt method. The plastic are BS or PC material, the metal are beryllium copper, phosphor copper, stainless steel. If you want to use FPC, please add two pins in the main board, the cost is a little high.

b. attached

Attach the metal (radiator) to the back cover of the machine.

4) Feed point of PIFA antenna

The feed point must be higher than 2mm * 3mm, try to place it at the edge of the PCB board, round is the best, square with rounded corners is better, the distance between feed point pad and ground should be higher than

1mm.

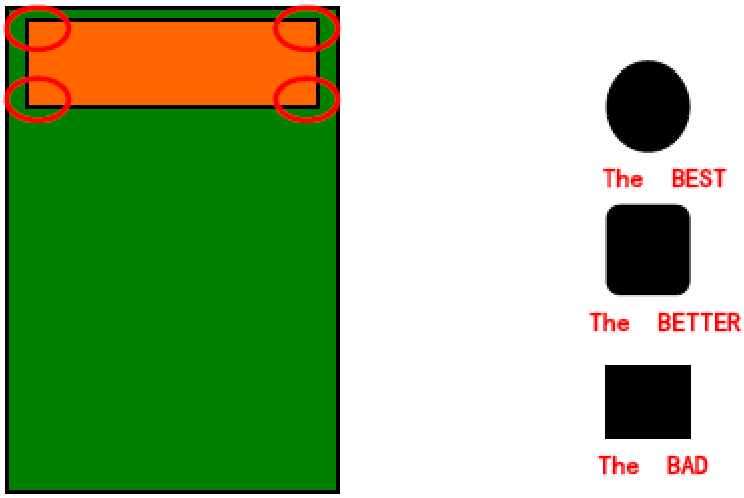

Figure 3-3 Pad Design Requirement

5) Requirements on height and square

Type	Height	Square
GSM/DCS	>6mm	>15mm x 40mm
GSM/DCS/PCS	>6.5mm	>17mm x 40mm
GSM850/GSM900/DCS1800/PCS 1900	>8mm	>20mm x 45mm

Note:

WCDMA/TD-SCDMA/LTE/CDMA/EVDO antenna design can refer to GSM antenna.

3.3.2.2 MONOPOLE Antenna

1) Antenna structure

There are one feeding point between the antenna and main board, which is module output. The antenna is on the top of the device.

2) The following figure shows the monopole antenna reference design:

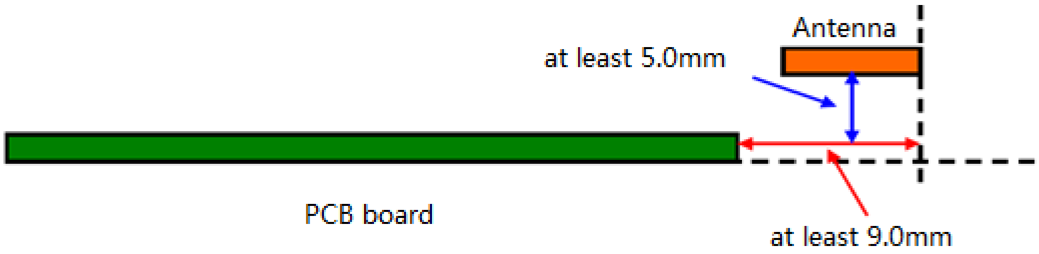

Figure 3-4 Antenna Location

3) There is no paving or PCB inside the antenna projection area, do not place any component in the antenna area.

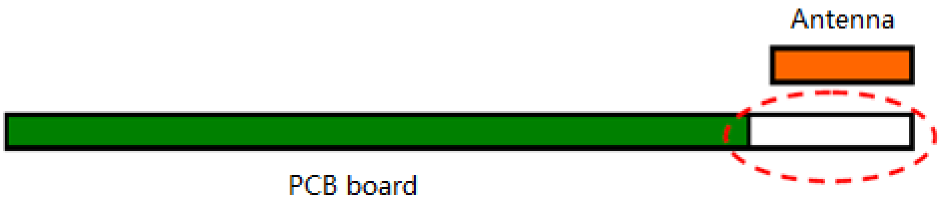

Figure 3-5 Antenna Projection Area Requirements

The length of PCB board should be 80-100mm, when the length of the antenna is 95mm, its performance is better.

4) Structure Style

Like PIFA antenna, it has scaffolding and attached style.

5) Feed point of Monopole antenna

It is the same with PIFA antenna.

6) Requirements on height and square

Type	Height	Square
GSM/DCS	>5mm	>35mm x 7mm
GSM/DCS/PCS	>6mm	>35mm x 8mm
GSM850/GSM900/DCS1800/PCS 1900	>6mm	>40mm x 10mm

Note:

WCDMA/TD-SCDMA/LTE/CDMA/EVDO antenna design can refer to GSM antenna.

3.3.2.3 IFA Antenna

IFA antenna is similar with Monopole antenna, but it also has PIFA antenna characteristics, it has two feeder branches, ground can exists below the antenna, it has the advantage that covers the instability of Monopole antenna, and the antenna space is larger than Monopole antenna.

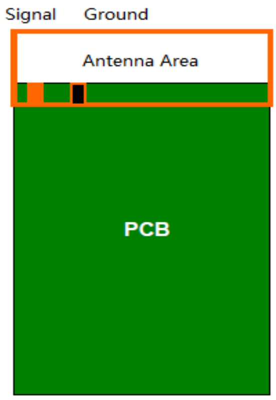

Figure 3-6 Location of signal point and earth point

Antenna space requirement: MONOPOLE<IFA< PIFA.

3.3.3 Other Design

3.3.3.1 Speaker

Put beads in serials on speaker can reduce the impact on RF.

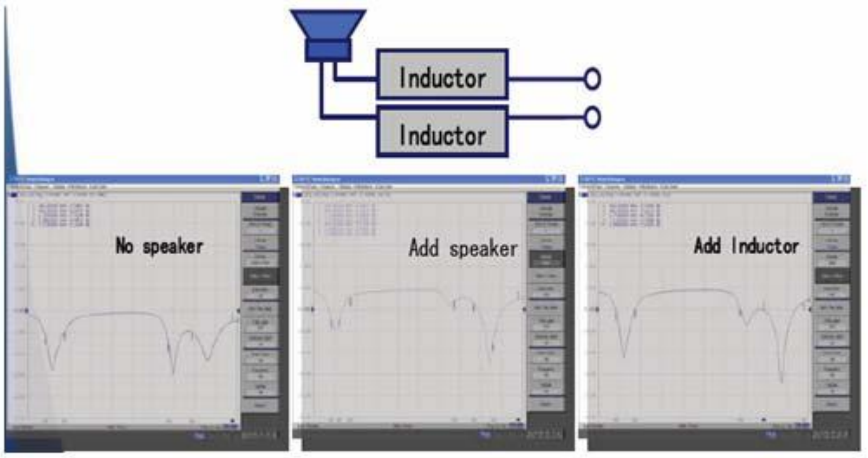

Figure 3-7 VSWR Improvement

3.3.3.2 Metal Structural Parts

All the metal structural parts must be grounded correctly; try to shield the circuit part.

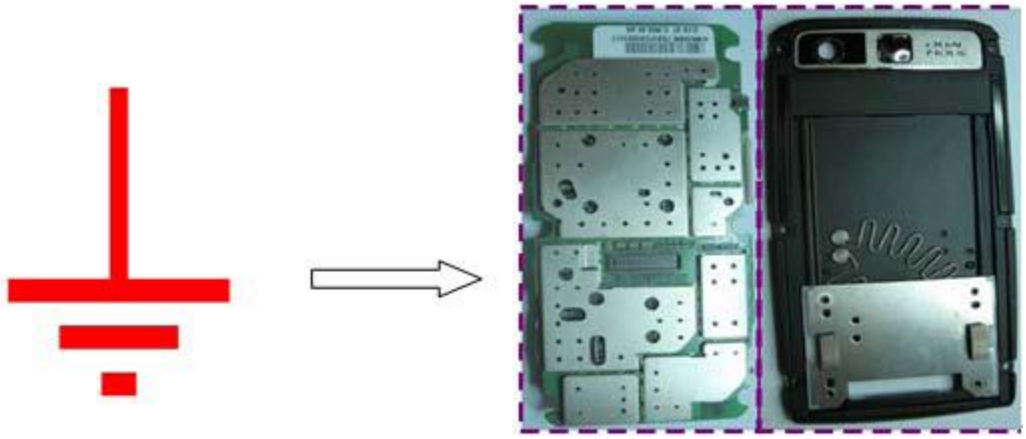

Figure 3-8 Metal Device Grounded

3.3.3.3 Battery

The battery should be far away from antenna.

Monopole antenna: the distance between battery and antenna ≥5mm

PIFA antenna: the distance between battery and antenna ≥3mm

Do not put the battery connector right before antenna.

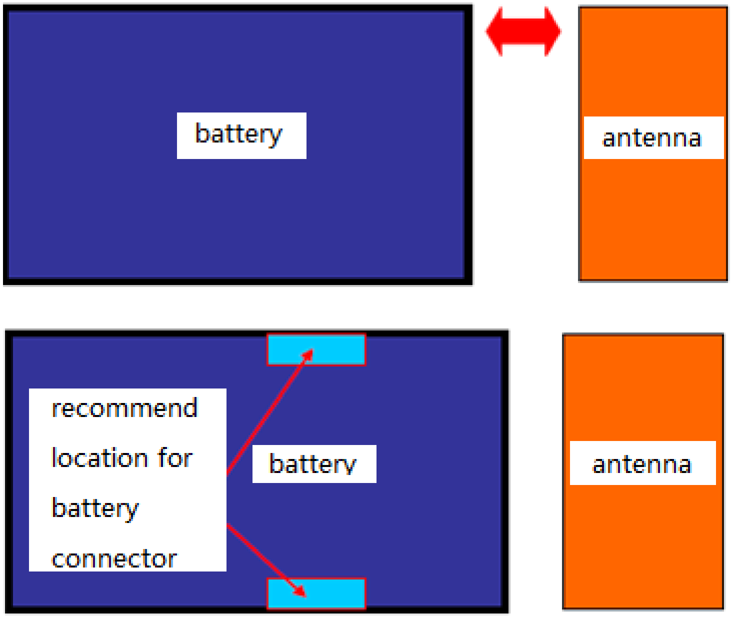

Figure 3-9 Battery Design

3.3.3.4 Large Components

Do not place large metal components such as oscillator, speaker, and receiver around the antenna; it may greatly affect the electrical performance of antenna. Do not spray the cover of the antenna with conductive paint, be caution when you use plating.

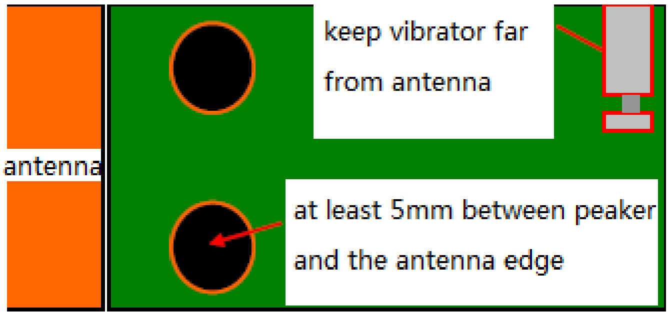

Figure 3-10 Location of large components

4 Common Problems for Internal Antenna

Factors that would affect emission performance:

1) Low antenna sensitivity If the module can meet the test requirements, and the radiant power of the antenna can meet the requirements, it may caused by the board design.

2) Chassis As the internal antenna is sensitive to the nearby medium, so the design of chassis is closely related to antenna performance.

3) Poor Speaker layout

4) Poor battery layout

Factors that would affect receptivity:

1) The circuit design of LCD, LDO, DC/DCD is poor, which effect coupling sensitivity.

2) Harmonic Impact caused by VCXO or TXVCO of 19.2MHZ, 26MHZ, and 38.4MHZ.

3) Poor coupling sensitivity caused by SIM clock.

4) Poor FPC layout.

Factors that would affect EMC:

1) Poor FPC layout.

2) Board antenna space radiation is absorbed by the metal element coupling, produces a certain amount of secondary radiation, frequency associated with the size of metal parts. So this kind of component should have a good grounding to eliminate or reduce secondary radiation.

5 Diversity Antenna Design

The diversity reception technique is an effective technique on combating channel fading. It can greatly increase the transmission reliability of multipath fading channel. Its essence is to use two or more different ways to receive the same signal to overcome fading and improve reception performance of the system.

Some Fibocom products support diversity reception function. If you want to use this function, please add a diversity antenna.

The design methods of diversity antenna are the same with main antenna, the efficiency can reduce 3dB.

The isolation between main antenna and diversity antenna should be higher than 12dB.

The polarization direction of diversity antenna and main antenna should be different.

6 Antenna Test

6.1 Passive Test

You will need: network analyzer, anechoic chamber.

The passive test evaluates the radiation performance of the device by focusing on the radiation parameters like gain, efficiency and antenna pattern. Although it considers the environment (such as the device around the antenna, open and close lid) impact on the antenna performance, it cannot tell the final radiated transmit power and receive sensitivity.

Figure 6-1 Passive Test Environment

6.2 Active Test

You will need: universal radio communication tester, spectrum analyzer, anechoic chamber, SAR tester.

Test item: TRP, TIS, SAR, directivity.

Antenna system is determined by the whole device, the antenna is just an important part. The antenna performance of the whole device must be concluded by the active test results.

The active test evaluates the radiation performance of the device by focusing on the transmit power and receive sensitivity. It tests the transmit power and receive sensitivity of all directions in three-dimensional space in specific anechoic chamber, it can directly reflect the radiation of the whole device.

CTIA (Cellular Telecommunication and Internet Association) formulated relevant standards of OTA (Over the Air).

6.2.1 TRP

TRP (Total Radiated Power): the average value of the transmit power of the entire radiation sphere. It reflects the transmit power of the whole device, and it is related to the transmit power and antenna radiation performance when the whole device is under conduction.

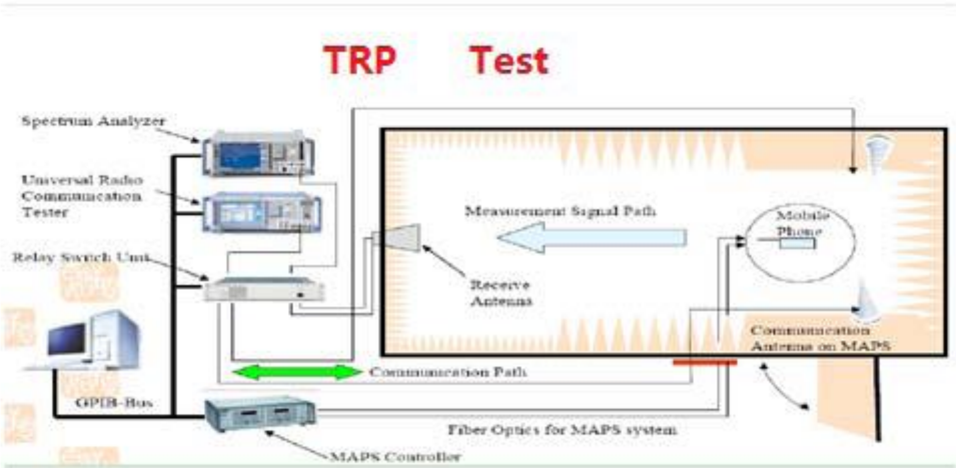

Figure 6-2 TRP Test Environment

6.2.2 TIS

TIS (Total Isotropic Sensitivity): reflects the receive sensitivity of the entire radiation sphere. It reflects the reception of the whole device; it is related to the conduction sensitivity and radiation performance of the antenna.

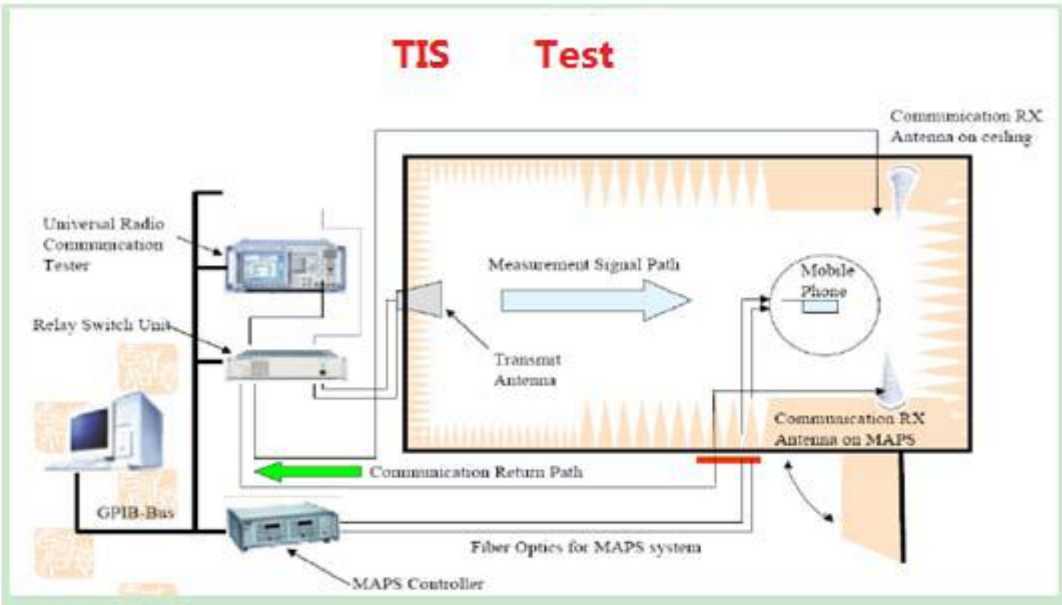

Figure 6-3 TIS Test Environment

6.2.3 SAR

SAR (Specific Absorption Rate) is a measure of how transmitted RF energy is absorbed by human tissue.

Note:

SAR is mainly for portable and mobile communications equipment.

Figure 6-4 SAR Test Environment

6.2.4 TRP and SAR

TRP reflects the far- field radiation performance of the antenna, while SAR reflects the near- field of the antenna. TRP value is expected to be high, so that the power from power amplifier absorbed by antenna is radiated, the connectivity of the antenna interface can be assured. However, TRP value is expected to be low in SAR test, then the power absorbed by human brain is relatively small and it can pass SAR test. So TRP and SAR is a pair of contradictory parameters, how to balance these two parameters level must be taken into consideration in the beginning.

Here are some solutions:

1) Choose the appropriate antenna type is the most important part.

For example, monopole is high efficiency and high SAR, which means the coupling effect is stronger between the monopole and human brain. The overall performance of PIFA antenna is better, as the ground of PCB blocks the side which is close to human brain, compares to maximum radiation direction, the high-frequency band attenuate 5-6dB in the direction of human brain direction, so the SAR value is low in PIFA antenna, it is an ideal antenna in internal antenna.

2) Notice SAR problem when design, choose appropriate antenna considering the structure of the whole device, ensure the performance of the antenna and also pass SAR, for example, you can place the antenna at the bottom of the PCB. For external type like helical antenna, please notice the distance between the antenna and the human brain to meet the SAR test requirements.

3) If it failed to pass SAR test, you can try to reduce the antenna performance, such as use material with high loss, this need corporation from the antenna provider.

4) Change antenna trace or adjust the antenna pattern.

5) Reduce the output power of the power amplify on the condition that it comply with the standards.