Design Guide

RF Antenna

V1.5

Disclaimer

Customers must design and develop their products referring to the information provided in the document. The Company shall not be liable for any damage caused by failure to comply with relevant operation or specifications or rules. Due to product version upgrade or other reasons, the Company reserves the right to modify any information in this document at any time without prior notice and any responsibility. Unless otherwise agreed, all statements, information and suggestions in this document do not constitute any express or implied guarantee.

Copyright Notice

Unless specially authorized by the Company, the recipient of the documents shall keep the documents and information received confidential, and shall not use them for any purpose other than the implementation and development of this project. Without the written permission of the Company, no unit or individual shall extract or copy part or all of the contents of this document without authorization, or transmit them in any form. The Company has the right to investigate legal liabilities for any offense and tort in connection with violation of confidentiality obligations, or unauthorized use or malicious use of the said documents and information in other illegal forms.

Trademark Statement

Fibocom The trademark is registered and owned by Fibocom Wireless Inc.

Other trademarks, product names, service names and company names appearing in this document are owned by their respective owners.

Contact Information

Website: https://www.fibocom.com

Address: 10/F-14/F, Block A, Building 6, Shenzhen International Innovation Valley, Dashi First Road, Xili Community, Xili Subdistrict, Nanshan District, Shenzhen

Tel: 0755-26733555

Safety Instructions

Do not operate wireless communication products in areas where the use of radio is not recommended without proper equipment certification. These areas include environments that may generate radio interference, such as flammable and explosive environments, medical devices, aircraft or any other equipment that may be subject to any form of radio interference.

The driver of any vehicle shall not operate wireless communication products while controlling the vehicle, otherwise will be reduced the driver's control of the vehicle, resulting in safety risks.

Wireless communication devices do not guarantee effective connection under any circumstances, such as when the (U) SIM card is invalid or the device is in arrears. In an emergency, please use the emergency call function when the device is turned on, and ensure that the device is located in an area with sufficient signal strength.

Contents

Applicable Model

index	Applicable Model	Description
1	Applicable to all FIBOCOM modules and solutions	NA

Change History

V1.5 (2023-12-21)	Update GNSS L2/L5 frequency band information
V1.4 (2023-05-25)	Update frequency band information
V1.3 (2022-07-12)	Add Figure 1. GNSS Active Antenna
V1.2 (2021-12-02)	Changed the format to the new style and the version number to two-digits.
V1.1.0 (2020-12-30)	Standardized the document format, corrected errors found in the document, including grammar, spelling, etc., and optimized the sentence structure.
V1.0.9 (2020-06-13)	Standardized the document format.
V1.0.8 (2020-02-19)	Added new models.
V1.0.7 (2020-02-05)	Added NR and MIMO antenna information.
V1.0.6 (2018-05-22)	Updated description of section 3.2.2. Updated Figure 2. π-type circuit Standardized the document format
V1.0.5 (2018-03-16)	Added RF bands of CDMA/EVDO and related indexes.
V1.0.4 (2018-01-31)	Added BeiDou RF bands, GNSS circuit, and CDMA/EVDO related indexes.
V1.0.3 (2015-12-05)	Added LTE information.
V1.0.2 (2015-08-24)	Updated the logo.
V1.0.1 (2014-12-26)	Changed the company name to Fibocom Wireless Inc.

V1.0.0 (2013-07-31)	Initial version.

1 Antenna Interface

1.1 Introduction

Fibocom modules have two types of antenna structures:

1. The module has its own RF connector, such as M.2 module, which allows RF cable to directly connect to the antenna.
2. The module only has RF antenna pad, such as LGA type module. The RF cable can be connected to the antenna after PCB design of the RF signal line.

Antennas are categorized into main antenna, diversity antenna and MIMO antenna according to functions.

1. Main antenna: GSM/WCDMA/TD-SCDMA/LTE/CDMA/EVDO/NR transceiver antenna.
2. Diversity antenna and MIMO antenna: WCDMA/LTE diversity receiving antenna, which has only receiving function and does not have the transmitting function. If the module supports NR SRS switch function, the diversity antenna and MIMO antenna will also have the transmitting function. Please refer to the hardware manual of each module for details.

1.2 Operating Bands

Operating Band	Description	Mode	Tx (MHz)	Rx (MHz)
Band 1	IMT 2100MHz	NR/LTE FDD/WCDMA	1920–1980	2110–2170
Band 2	PCS 1900MHz	NR/LTE FDD/WCDMA/ GSM/CDMA/EVDO/	1850–1910	1930–1990
Band 3	DCS 1800MHz	LTE FDD/GSM	1710–1785	1805–1880
Band 4	AWS 1700MHz	NR/LTE FDD/WCDMA	1710–1755	2110–2155

Operating Band	Description	Mode	Tx (MHz)	Rx (MHz)
Band 5	CLR 850MHz	NR/LTE FDD/WCDMA/ GSM/CDMA/EVDO	824–849	869–894
Band 6	UMTS 800MHz	WCDMA	830–840	875–885
Band 7	IMT-E 2600Mhz	NR/LTE FDD	2500–2570	2620–2690
Band 8	E-GSM 900MHz	NR/LTE FDD/WCDMA/ GSM	880–915	925–960
Band 9	UMTS 1700MHz	LTE FDD	1749.9–1784.9	1844.9–1879.9
Band 12	700MHz	NR/LTE FDD	698–716	728–746
Band 13	USMH Block C	LTE FDD	777–787	746–756
Band 14	700MHz	NR/LTE FDD	788–798	758–768
Band 17	LSMH Blocks B/C	NR/LTE FDD	704–716	734–746
Band 18	Japan Lower 800Mhz	NR/LTE FDD	815–830	860–875
Band 19	Japan Upper 800Mhz	NR/LTE FDD	830–845	875–890
Band 20	EUDD 800MHz	NR/LTE FDD	832–862	791–821
Band 25	1900MHz	NR/LTE FDD	1850–1915	1930–1995
Band 26	ECLR 850MHz	NR/LTE FDD	814–849	859–894
Band 27	ECLR 850MHz	LTE FDD	807–824	852–869
Band 28	700MHz	NR/LTE FDD	703–748	758–803
Band 29	LSMH blocks D/E	LTE FDD	N/A	717–728
Band 30	2300MHz	NR/LTE FDD	2305-2315	2350-2360

Operating Band	Description	Mode	Tx (MHz)	Rx (MHz)
Band 32	1470MHz	LTE FDD		1452-1496
Band 66	1700MHz	NR/LTE FDD	1710-1780	2110-2200
Band 71	680MHz	NR/LTE FDD	663-698	617-652
Band 34	IMT 2100MHz	NR/LTE/TD-SCDMA	2010–2025	
Band 38	IMT-E 2600MHz	NR/LTE TDD	2570–2620	
Band 39	TDD 1900MHz	NR/LTE TDD/TD-SCDMA	1880–1920	
Band 40	IMT 2300MHz	NR/LTE TDD	2300–2400	
Band 41	BRS/EBS 2500MHz	NR/LTE TDD	2496–2690	
Band 42	3500MHz	LTE TDD	3400-3600	
Band 43	3700MHz	LTE TDD	3600-3800	
Band 48	3600MHz	NR/LTE TDD	3550-3700	
Band 46	5500MHz	LTE TDD		5150-5925
n77	3700MHz	NR	3300–4200	
n78	3500MHz	NR	3300–3800	
n79	4700MHz	NR	4400–5000	
GPS L1				1575.42±1.023
GPS L2				1227.60±1.023
GPS L5				1176.45±1.023
GLONASS L1				1602.5625±4

Operating Band	Description	Mode	Tx (MHz)	Rx (MHz)
Beidou				1561.098±2.046
Galileo				1559-1592
QZSS				1575.42±1.023

For Operation bands, please refer to the hardware guide of each module for details.

1.3 Antenna Requirements

1. Antenna Efficiency

Antenna efficiency: > 40% (below 1GHz); > 50% (over 1GHz)

2. S11 or VSWR

The S11 and VSWR value for the antenna of the module are recommended to be smaller than –10dB and 2:1, respectively.

3. Polarization

Linear polarization is recommended: It is recommended that the polarization direction varies with the antennas.

4. Radiation Pattern

Omni-directional

5. Gain

Antenna gain ≤ 2dBi

6. Antenna Isolation

Antenna isolation > 25dB

7. Antenna Correlation Coefficient

Antenna correlation coefficient < 0.5

8. Bandwidth

Comply with the relevant standards of China or international standards

9. TRP/TIS

TRP (Total Radiated Power):

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

TIS (Total Isotropic Sensitivity):

- W850/W900 < –102dBm
- W1700/W1900/W2100 < –103dBm

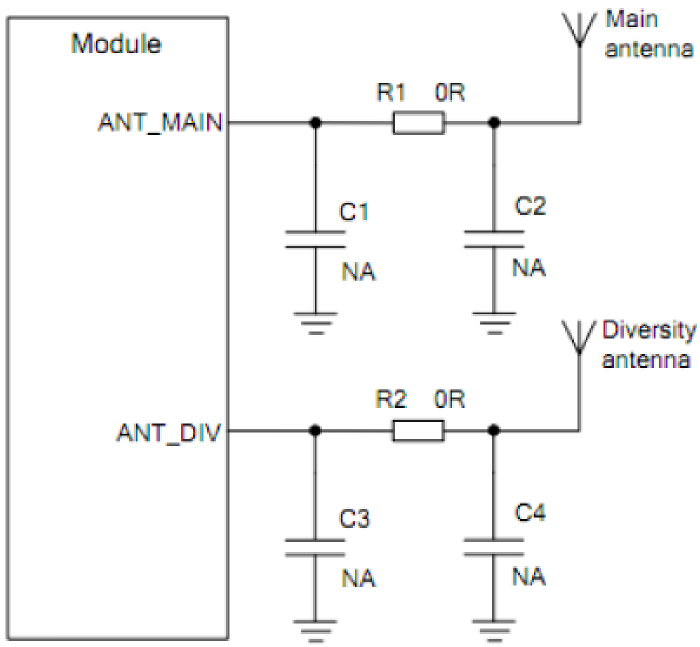
- GSM850 < –102dBm
- GSM900 < –102dBm
- DCS1800/PCS1900 < –102dBm
- TD-SCDMA ALL Band < –104dBm
- CDMA/EVDO < –102dBm
- LTE FDD/LTE TDD ALL Band < –93dBm (10MHz Bandwidth)
- NR TBD

2 RF PCB Design

2.1 Cabling Guideline

For modules that don't have a RF connector, customers need to route a RF cable to connect to the antenna feeding point or connector. We recommend customers use a microstrip line. The shorter the better. The insertion loss should be controlled less than 0.2dB; and impedance should be controlled within 50Ω.

Add a π-type circuit (two parallel-component- grounded pins are connected directly to the main GND) between the module and antenna connector (or feeding point) for antenna debugging.



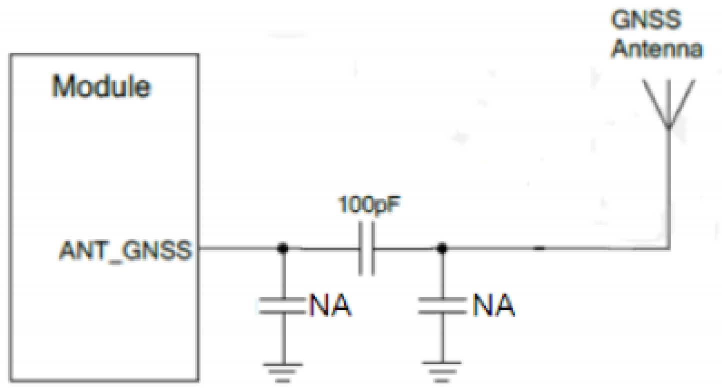


Figure 2. π-type circuit

This signal line impedance is controlled within 50Ω during PCB cabling, and the RF performance is closely related to this cabling. PCB parameters that will affect the cabling impedance include:

- Cabling width and thickness
- Dielectric constant and thickness of media
- Thickness of pad
- Distance from ground line
- Nearby cabling

2.2 Impedance Design

The RF impedance of the two antennas' interface should to be controlled within 50Ω.

In practical application, RF cabling mode is designed according to other parameters of PCB, such as reference layer thickness, number of layers and stacking. Different reference GND layer will lead to different cabling design.

2.2.1 3W Principle

During antenna RF signal cabling design 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 line width, 70% of the electric field between the lines will not interfere with each other, and this is called "3W principle".

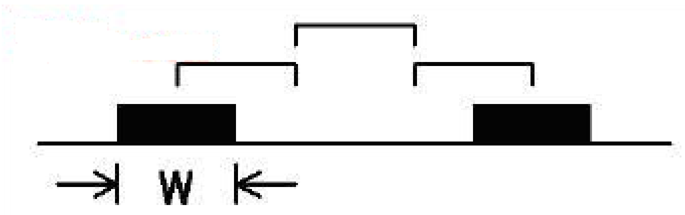


Figure 3. 3W principle

2.2.2 Impedance Design for Double Layers

The following examples show PCB design for 1.6mm board thickness and 1.0mm board thickness.

Case 1: 1.6mm PCB

In the following picture, the white area shows 50Ω impedance control line. Considering the thickness of the board, it is hard to follow 3W principle (ensure 50Ω impedance and avoid 50Ω impedance affecting surrounding components), so we suggest the customers do not place any component near the antenna, and keep the PCB cabling away from the RF part.

It’s been proved that the following design can ensure 50Ω RF impedance and minimize impacts.

The RF line width is 43 mils, and the distance from the line to the GND (blue) is 8 mil. The reference layer is below the RF cabling; which is a complete ground (red) normally. Beside the RF cabling, install complete ground protection and punch a lot of holes along the cabling direction. Do not put any other cable or component near the RF cabling and reference ground, as shown in the Figure 4. 50Ω impedance line

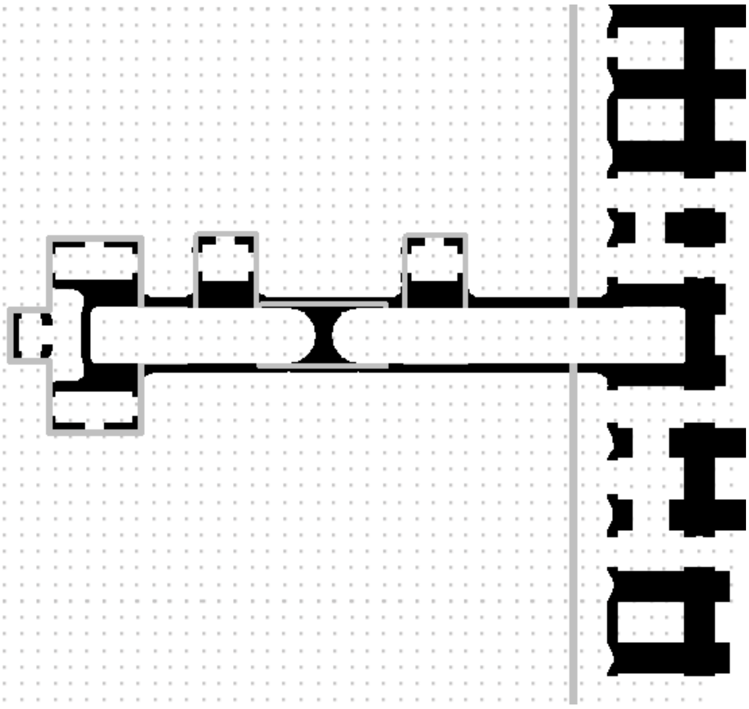


Figure 4. 50Ω impedance line

50Ω impedance calculation:

Figure 5. Impedance calculation (unit: mil)

The PCB vendor confirms the Er1, T1, C1, C2, and CEr parameters generally, sometimes you need to confirm with the PCB vendor due to different process. H1 represents PCB thickness, W1 represents trace width and D1 represents the distance from line to GND. Please notice variation in practice.

Case 2: 1.0mm PCB

When the thickness of PCB is 1.0mm, the RF line width on the board is 35mil, and the distance from line to GND (blue) is 8 mils.

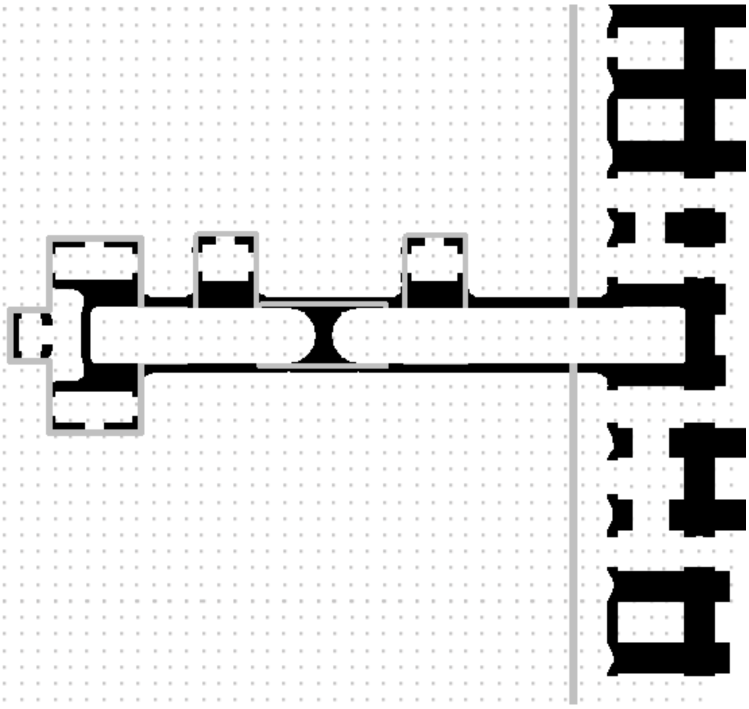


Figure 6. RF cabling

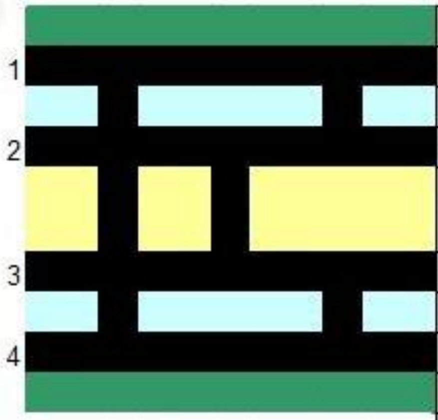
50Ω impedance calculation:

Figure 7. Impedance calculation

2.2.3 Impedance Design for Four Layers

Four layers boasts the PCB board thickness 1.0mm. RF line is on Lay1, and reference layer is on Lay2 (GND Layer). The stacking varies with PCB vendor; you can refer to the following figure:

4 Layers (1+2+1)



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

Figure 8. Four-layer lamination

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

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

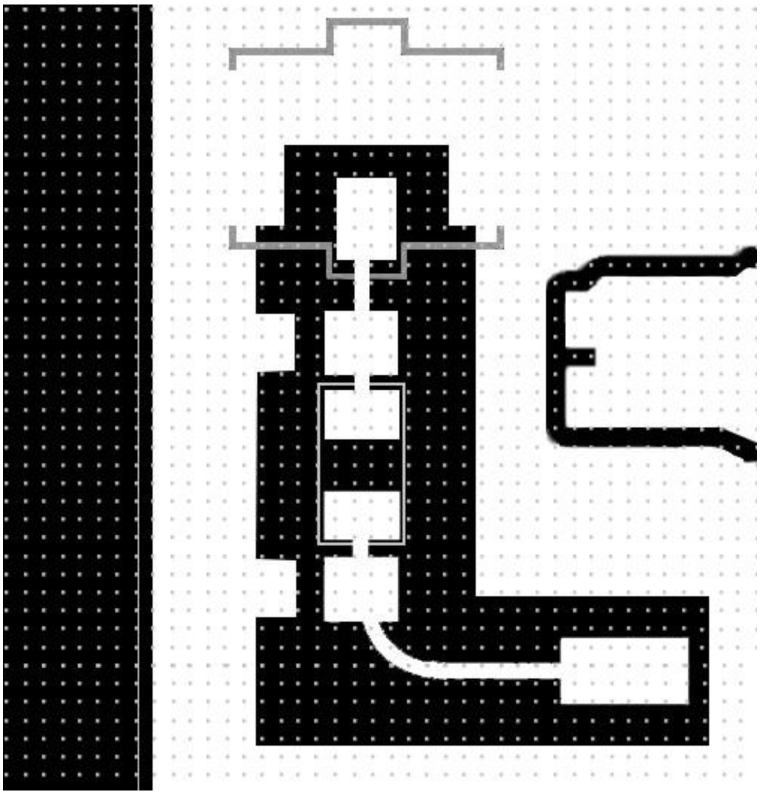


Figure 9. RF cabling

50Ω impedance calculation:

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

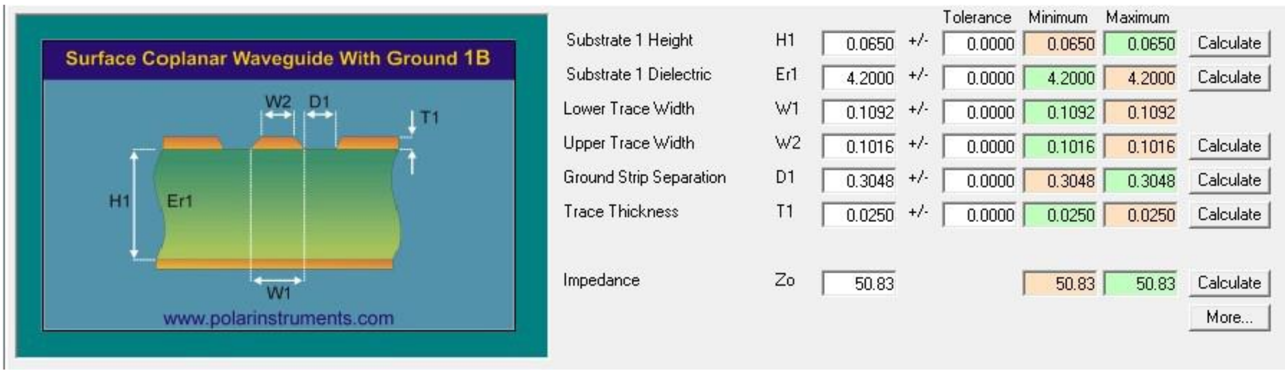


Figure 10. Impedance calculation for four-layer board top layer cabling

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 11. 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 design them in four corners of the board to keep away from GSM, WIFI and BT antennas. DC bias voltage is not required.

3.2.2 Design Principle of Active Antenna

The active antenna is integrated with a LNA (Low Noise Amplifier), and such an antenna requires a power supply. Some modules support direct power supply and power the active antenna through GNSS_ANT. Some modules do not support direct power supply, so an external power supply is needed. The active antenna must be kept away from any noise source to ensure good performance. Design the active antenna in the four corners of the board to keep away from GSM, WIFI and BT antennas.

The RF signal must be connected to an external power source through a 100nH inductor without direct connection to prevent power noise from entering the antenna.100pF capacitor for DC blocking.

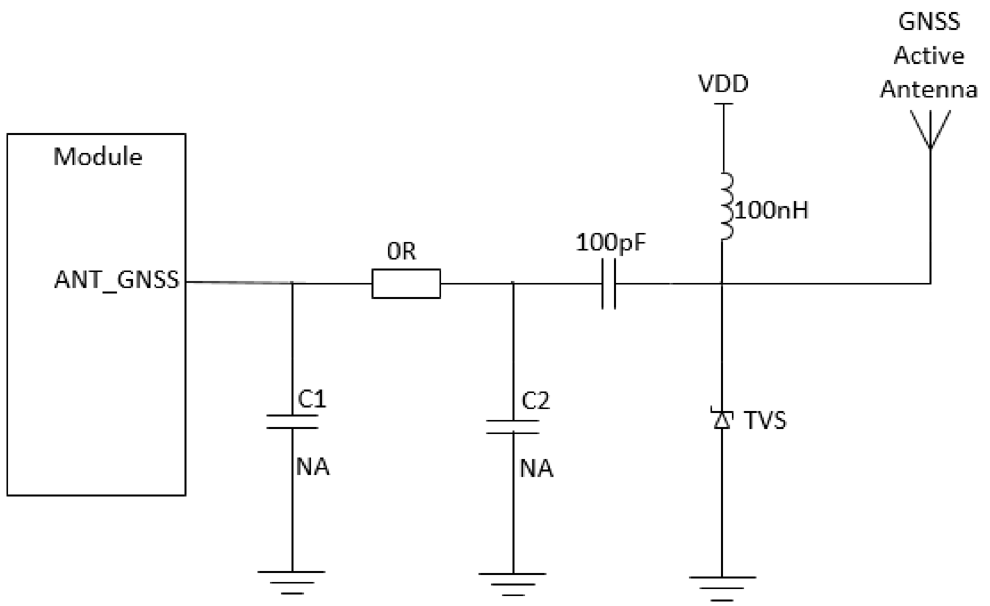


Figure 12. GNSS Active Antenna

3.3 Internal Antenna

3.3.1 Design Guideline

1. Placement

- The antenna shall be arranged in the corners of the module.
- Avoid placing metal elements near the antenna.
- The shielding parts shall be as neat as possible. Do not use long strip shaped hole slots.
- Components with metal structure, such as horn, vibrator, and camera base plate shall be

grounded.

- Avoid using long FPC. If a long FPC is required, add grounding shields on both sides.

2. Cabling

- When connecting RF cabling, apply circular arc treatment at the turning, take grounding and pay attention to characteristic impedance.
- RF ground shall be designed properly, PCB board and edge of ground shall be provided with "ground wall", and antenna led from RF module shall be made into microstrip line.
- The antenna RF feeding point welding pad is a round rectangular pad with the size of 2mm × 3mm. All layers of PCB that include the pad and surrounding and that are equal to and greater than 0.8mm are not covered with copper.
- The center distance between RF and ground pad shall be between 4 mm and 5 mm.

3.3.2 Internal Antenna Types

There are three kinds of built-in antennas: PIFA, IFA and monopole. The built-in antennas may form interference and other potential problems in the product, so there are more requirements in the design.

Here are the comparisons between these three types of antennas:

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
IFA	Grounded	2	Mid	Great	Mid

3.3.2.1 PIFA Antenna

1. Antenna structure

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

The signal point and GND point can be put in different places, and more GND points mean more choices during antenna design.

Figure 13. Location of signal point and GND point for PIFA antenna

2. Main board

There is complete paving in the antenna projection area. Do not place any component in the antenna area. The recommended length of PCB board should be 90mm–110mm. The antenna performance is improved if the board length is 105mm.

3. Structure types

a. Bracket

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

b. Attached

Attach the metal sheet (radiator) to the back cover of the module.

4. Feeding point of PIFA antenna

The feeding point must be greater than 2mm × 3mm. Try to place it at the edge of the PCB board, and adopt round shape. Square with rounded corners is also preferred. The distance between feeding point pad and ground should be equal to or greater than 1mm.

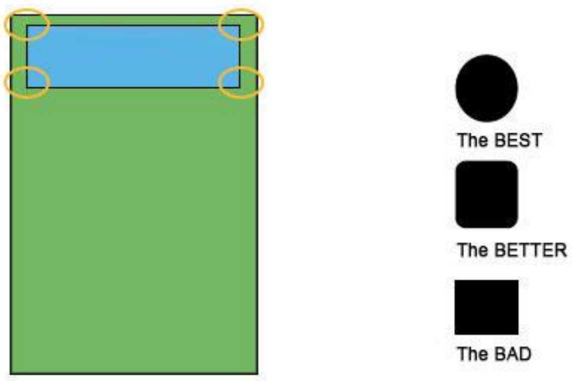


Figure 14. Pad design requirement

5. Requirements on height and area

Type	Height	Area
GSM/DCS	> 6mm	> 15mm × 40mm
GSM/DCS/PCS	> 6.5mm	> 17mm × 40mm
GSM850/GSM900/DCS1800/PCS1900	> 8mm	> 20mm × 45mm

For details about WCDMA/TD-SCDMA/LTE/CDMA/EVDO/NR antenna design, refer to GSM antenna.

3.3.2.2 Monopole Antenna

1. Antenna structure

   There is one feeding point between the antenna and main board, which is module output. It is recommended to place the antenna on the top of the module.

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

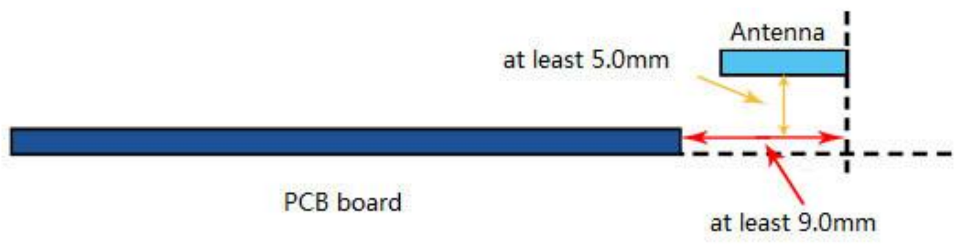


Figure 15. Antenna location

3. There should be no paving or PCB in the antenna projection area. Do not place any component in the antenna area.

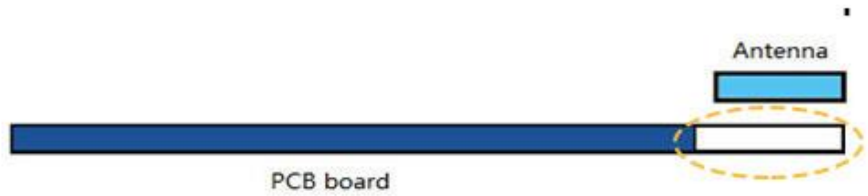


Figure 16. Antenna projection area requirements

   The recommended length of PCB board should be 80mm–100mm. The antenna performance is improved if the board length is 95mm.

4. Structure types

   Like PIFA antenna, the monopole antennas fall into bracket type and attached style.

5. Feeding point of Monopole antenna

It is the same with PIFA antenna.

6. Requirements on height and area

Type	Height	Area
GSM/DCS	> 5mm	> 35mm × 7mm
GSM/DCS/PCS	> 6mm	> 35mm × 8mm
GSM850/GSM900/DCS1800/PCS1900	> 6mm	> 40mm × 10mm

For details about CDMA/EVDO/WCDMA/LTE/TD-SCDMA antenna design, refer to GSM antenna.

3.3.2.3 IFA Antenna

IFA antenna shares similarity with Monopole antenna and PIFA antenna. IFA antenna has two feeding branches, and allows ground under the antenna. The antenna has better stability than Monopole antenna, and the antenna space requirement is between Monopole antenna and PIFA antenna.

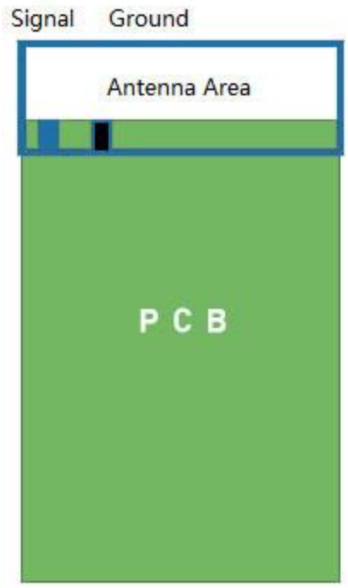


Figure 17. Location of signal point and GND point

Antenna space requirement: monopole < IFA < PIFA. For other requirements, refer to the PIFA and monopole requirements.

3.3.3 Design of Internal Antenna Surroundings

3.3.3.1 Speaker

Connecting beads or inductors on speaker can reduce the impact on RF.

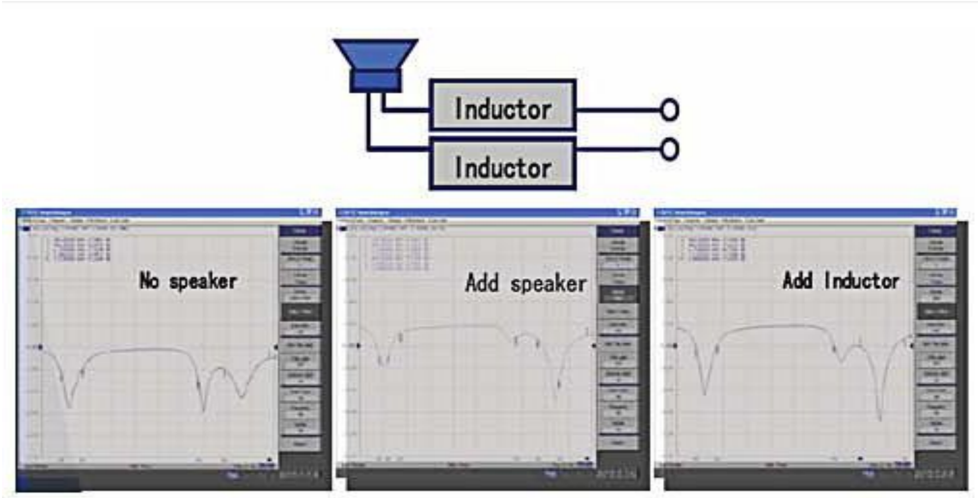


Figure 18. VSWR improvement

3.3.3.2 Metal Structural Parts

All the metal structural parts must be grounded correctly and reliably, and the circuit part must be shielded.

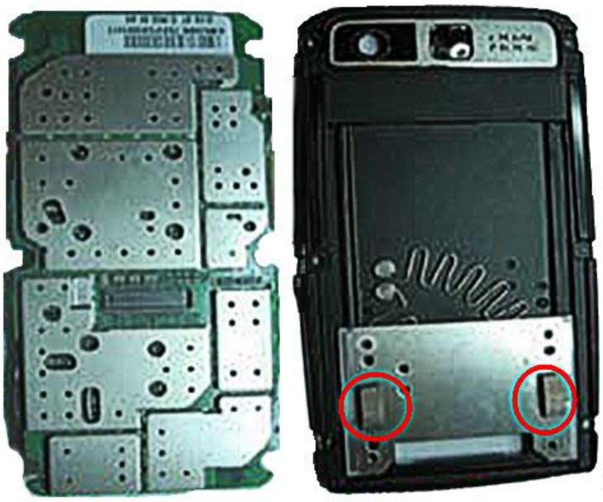


Figure 19. Metal Device Grounded

3.3.3.3 Battery

1. 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

2. Do not put the battery connector right beside the antenna.

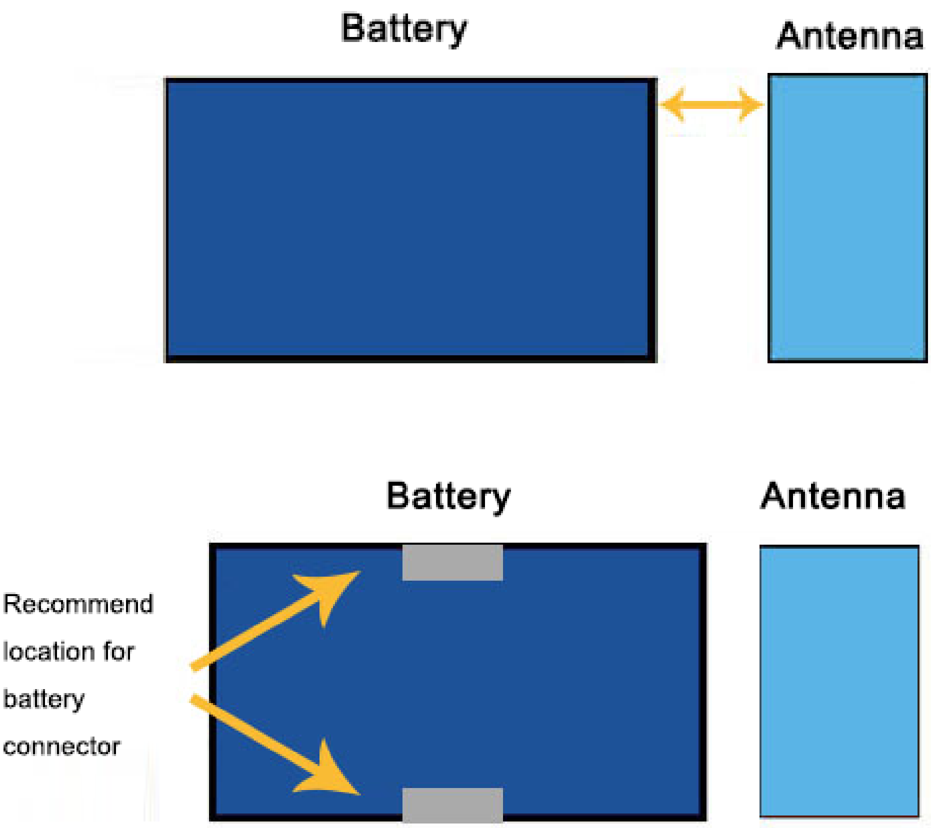


Figure 20. Battery design

3.3.3.4 Large Components

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

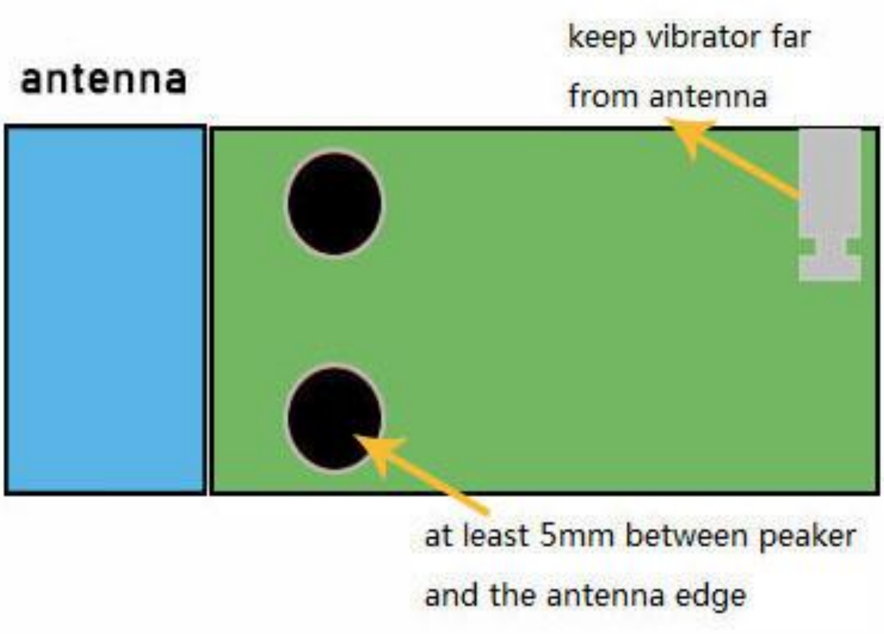


Figure 20. Location of large components

3.3.4 Common Problems for Internal Antenna

- Factors that would affect TRP (Total Radiated Power) performance:
    - Shell: As the internal antenna is sensitive to the nearby medium, so the design of shell is closely related to antenna performance.
    - Poor speaker layout will affect antenna performance.
    - Poor battery layout will affect antenna performance.
- Factors that would affect TIS (Total Isotropic Sensitivity) performance:
    - If both the conductive Performance of module and the radiated power of antenna meet requirement, then low RX sensitivity may be caused by main board design issue
    - Poor coupling sensitivity is caused by poor circuit design of LCD, LDO, and DC/DC.
    - Device receiving performance is affected by VCXO or TXVCO harmonic of 19.2MHZ, 26MHZ, and 38.4MHZ systems.

  - Poor coupling sensitivity is caused by SIM card clock.
  - Poor FPC layout affects the receiving performance of the device.
- Factors that would affect EMC (electromagnetic compatibility):
  - Poor FPC layout affect EMC performance of the device.
  - The metal element may absorb the antenna radiated power and produce a certain amount of secondary radiation, and coupling frequency is associated with the size of metal parts. Therefore, this kind of component should have a good grounding to eliminate or reduce spectrum emission.

4 Diversity/MIMO Antenna Design

- Diversity receiving technology is a main anti fading technology, which can greatly improve the transmission reliability in multipath fading channels. Its essence is to use two or more different methods to receive the same signal to overcome the fading and improve the receiving performance of the system.
- Diversity antenna can also multiplex different transmission paths in space using division multiplexing technology and receive data from the multiple different paths in parallel to improve the receiving throughput.
- The function of MIMO antenna is similar to that of diversity antenna, and they both can resist against fading and improve throughput.
- The customer is recommended to design the corresponding antenna according to the antenna requirements of each module antenna port.
- The design method of diversity antenna and MIMO antenna is consistent with that of main antenna. It is recommended to control the difference of the efficiency of diversity antenna and MIMO antenna from that of main antenna by no more than 3dB.
- The isolation of each antenna shall be greater than 25dB, and the antenna correlation coefficient shall be less than 0.5. High isolation does not mean good correlation coefficient. Customers need to evaluate two indexes separately. The isolation and correlation coefficient of antenna generally depend on:
  - Distance between antennas
  - Antenna type
  - Antenna directionality

5 Antenna Test

5.1 Passive Test

You will need: network analyzer and 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 the test 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 transmitting power and receiving sensitivity.

Figure 21. Passive test environment

5.2 Active Test

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

Test item: TRP, TIS, SAR, and directionality.

Antenna system is determined by the whole device, and the antenna is just an important part of the whole device. 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 transmitting power and receiving sensitivity. The test measures the transmitting power and receiving sensitivity of the device in all directions in 3D space in specific anechoic chamber, which can directly reflect the radiation performance of the whole device.

Please refer to 3GPP TS 34.114, OTA (over the air) standard developed by CTIA (cellular telecommunications and Internet Association).

5.2.1 TRP

TRP (Total Radiated Power): the average value of the transmitting power of the entire radiation sphere. It reflects the transmitting power of the whole device, and it is related to the transmitting power and antenna radiation performance of the device in conductive state.

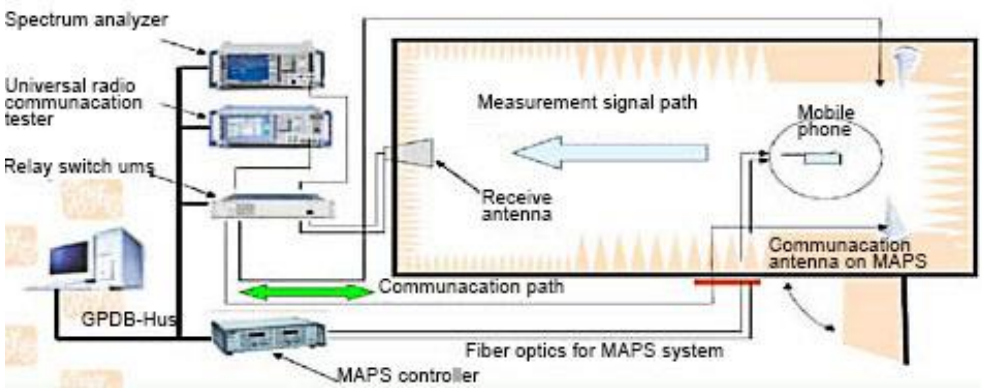


Figure 22. TRP test environment

5.2.2 TIS

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

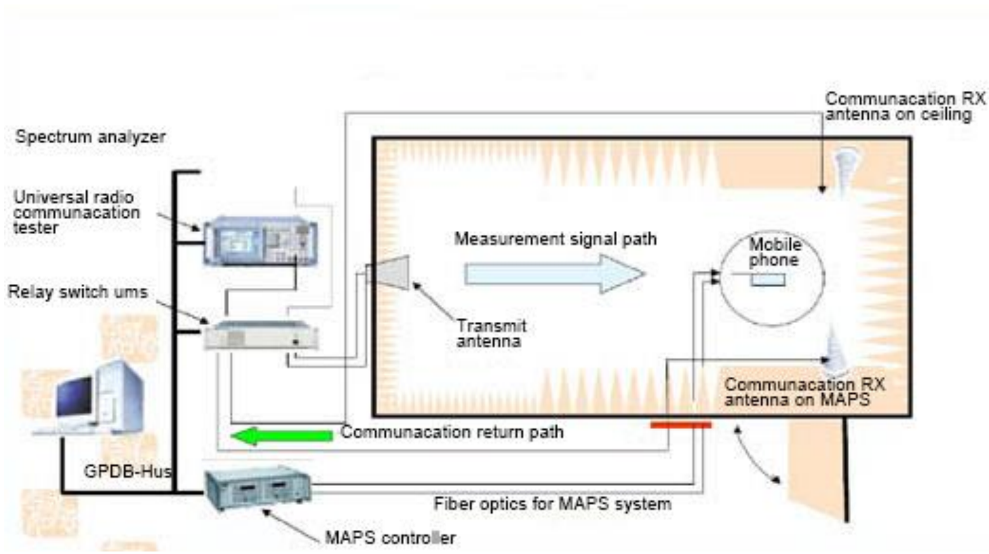


Figure 23. TIS test environment

5.2.3 SAR

SAR (Specific Absorption Rate) is a measurement of how much electromagnetic wave energy of wireless products is absorbed. Under an external Electromagnetic field, human body will generate an induced Electromagnetic wave field.

SAR is designed mainly for handheld and wearable communications devices.

Figure 24. SAR test environment

5.2.4 TRP and SAR

TRP reflects the far-field radiation performance of the antenna, while SAR reflects the near-field of the antenna. TRP value in OTA is expected to be high, so that the power from power amplifier to antenna can be effectively radiated, and 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 are a pair of contradictory parameters. How to balance these two parameters level must be taken into consideration in the beginning of antenna design?

Here are some solutions:

1. Choose the appropriate antenna type. For example, the monopole antenna boasts high efficiency and high SAR, which means the coupling effect is stronger between the monopole and human brain. The PIFA antenna boasts better overall performance. The part of the PIFA antenna that is close to human brain is partially shielded by PCB ground, the high-frequency band in the direction of human brain is attenuated by 5dB to 6dB compared to that in the maximum radiation direction. Therefore, the SAR value is low in

PIFA antenna, which is an ideal antenna in internal antennas.

2. At the beginning of antenna design, we need to consider the SAR problem, mainly in the structure design. An appropriate form of antenna is selected in combination with the structure of the device to ensure the antenna performance and meet the SAR index requirements. For example, place the antenna at the bottom of PCB. For the external spiral antenna, the distance between the antenna and human brain should be controlled in order to meet the needs of SAR test.

3. If the antenna design failed to pass SAR test, you can try to reduce the antenna performance, for example, using material with high loss. This needs corporation from the antenna provider.

4. Change antenna cabling or adjust the antenna pattern.

5. Reduce the output power of the power amplifier on the condition that the operation complies with the standards.

The preceding solutions are a result of balance between the SAR test requirements and TRP test requirements.