Antenna Design Note

Rev. Antenna_Design_Note_V3.2

Date: 2020-02-13

Status: Released

www.quectel.com

Our aim is to provide customers with timely and comprehensive service. For any assistance, please contact our company headquarters:

Quectel Wireless Solutions Co., Ltd.
Building 5, Shanghai Business Park Phase III (Area B), No.1016 Tianlin Road, Minhang District, Shanghai, China 200233
Tel: +86 21 5108 6236
Email: info@quectel.com

Or our local office. For more information, please visit:
http://www.quectel.com/support/sales.htm

For technical support, or to report documentation errors, please visit:
http://www.quectel.com/support/technical.htm
Or email to: support@quectel.com

GENERAL NOTES

QUECTEL OFFERS THE INFORMATION AS A SERVICE TO ITS CUSTOMERS. THE INFORMATION PROVIDED IS BASED UPON CUSTOMERS' REQUIREMENTS. QUECTEL MAKES EVERY EFFORT TO ENSURE THE QUALITY OF THE INFORMATION IT MAKES AVAILABLE. QUECTEL DOES NOT MAKE ANY WARRANTY AS TO THE INFORMATION CONTAINED HEREIN, AND DOES NOT ACCEPT ANY LIABILITY FOR ANY INJURY, LOSS OR DAMAGE OF ANY KIND INCURRED BY USE OF OR RELIANCE UPON THE INFORMATION. ALL INFORMATION SUPPLIED HEREIN IS SUBJECT TO CHANGE WITHOUT PRIOR NOTICE.

About the Document

Revision History

Version	Date	Author	Description
1.0	2012-06-09	David WEI	Initial
1.1	2012-06-15	David WEI	Modified Figure 1
1.2	2012-08-01	David WEI	Added contact information for antenna manufacturers: Antenova and Pulse Electronics
1.3	2012-11-21	David WEI	Added contact information for GLONASS antenna manufacturer INPAQ
1.4	2013-07-10	David WEI	1. Added ceramic chip antennas 2. Updated contact information
1.5	2014-11-21	Jackie WANG	Added the antenna performance and LDS antenna
1.6	2015-04-11	Jackie WANG	Added applicable modules
1.7	2016-01-06	Mark ZHANG	1. Added external PCB antennas 2. Added contact information for antenna manufacturer SAINTENNA and JINGHONG
1.8	2016-06-01	Mark ZHANG	1. Updated the contact information of antenna manufacturer JESONCOM 2. Updated the address and contact information of antenna manufacturer Antenova
1.9	2017-07-14	Vick YANG	1. Added description of metal frame antennas in Chapter 3.8 2. Added description of internal Wi-Fi laminated antennas in Chapter 7 3. Updated antenna suppliers information in Chapter 8: ● Updated contact information of antenna manufacturers SAINTENNA and INPAQ ● Deleted information of antenna manufacturer JINGHONG ● Added information of antenna manufacturer

			SHEN XUN
2.0	2018-01-02	Vick YANG/ Beny ZHU	1. Optimized the description of EIRP (Effective Isotropic Radiated Power) in Chapter 2.1. 2. Updated the design note (item 3) for internal Wi-Fi laminated antenna. 3. Added Chapter 8: GNSS Antenna Isolation Design Requirements. 4. Updated the address and contact information (tel. number, email address and fax number) of antenna supplier Pulse. 5. Added Sunnyway and VLG as new antenna suppliers.
3.0	2019-06-21	Riley XU	Numerous changes have been made to this document and thus it is recommended to read in its entirety.
3.1	2019-07-04	Riley XU	Updated antenna suppliers information
3.2	2020-02-13	Riley XU	Added Quectel antenna design service information in Chapter 9.

Contents

Table Index

Figure Index

1 Recommended Component Placement on Main PCB

This document is applicable to all Quectel modules.

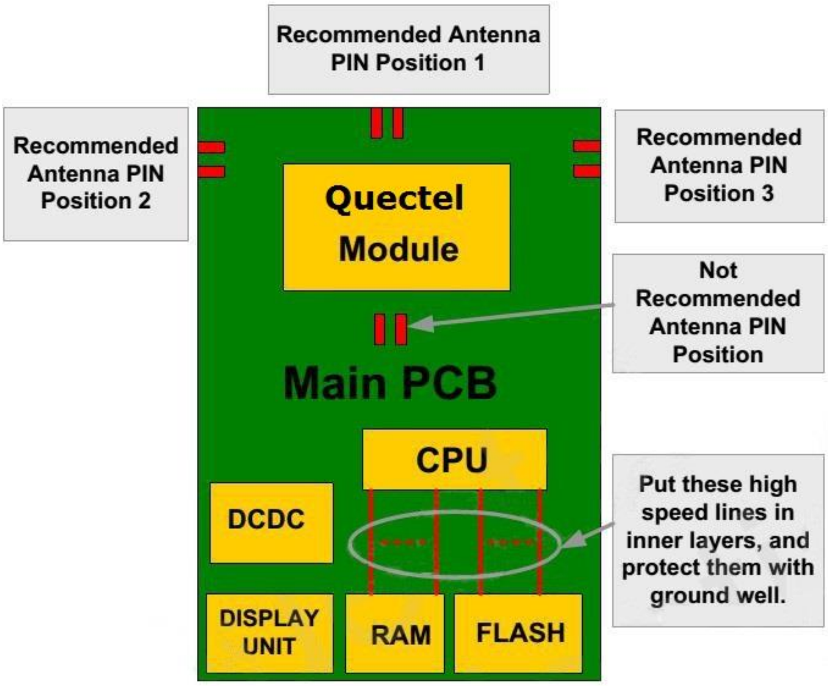


Figure 1: Recommended Component Placement

Notes:
1. It is suggested to keep the RF ports at the edge of the PCB, and keep them away from the other circuits. Avoid vias and layer changes when routing RF traces. And RF traces should be designed to have an impedance of 50Ω. Please keep the distance between RF ports and antennas as short as possible. If there is a long distance and it cannot be shortened anymore, it is recommended to use antenna cables for antenna connection.
2. Please put antenna feed points at the edge of the main PCB, rather than in the center. For two antennas working at a similar frequency range, the distance between them should be more than the quarter-wavelength at the lowest frequency. When the antennas are too close, they are recommended to be placed orthogonally to each other.
3. Generally speaking, the isolation between the antennas should be at least 10dB to avoid interference.

4.	Please keep the antenna as far away from CPU/SDRAM/Flash/DC-DC converters/Display FPC connectors as possible. The antenna should be placed on opposite side of PCB on which these components are mounted.
5.	Please keep the high-speed lines between CPU and SDRAM/Flash/Display FPC connectors as short as possible, and put these lines in inner layers with ground shielding on not only upper and lower layers but also right and left sides. Add an EMI filter on high speed lines between CPU and display FPC if necessary.
6.	Please put CPU/SDRAM/Flash/DC-DC converters/Display FPC connectors into the shielding case, and copper-nickel-zinc alloy shielding cases are preferred.

2 Basic Parameters and Requirements of Antennas

2.1. Basic Parameters of Antennas

Gain (dBi): The ratio of “power of antenna” and “power of isotropic radiation from an ideal current source” in maximum transmitting direction with the same input power. “dBi” is widely used as the unit of antenna gain.

Gain (dBd): The ratio of “power of antenna” and “power of half wave dipole antenna” in maximum transmitting direction with the same input power. When it represents the same gain, one formula indicating relationship between dBi and dBd is given as below: dBi=dBd+2.15.

Directivity: The ratio of “power of antenna” and “power of isotropic radiation from an ideal current source” in maximum transmitting direction with the same radiated power.

Efficiency: The ratio of the antenna radiation power and antenna input power.

Gain=Directivity × Efficiency
Efficiency=Output Power/Input Power

APIP (Antenna Port Input Power): The input power of antenna.

EIRP (Effective Isotropic Radiated Power): EIRP (Effective Isotropic Radiated Power) is the amount of power that a theoretical isotropic antenna (which evenly distributes power in all directions) would emit to produce the peak power density observed in the direction of maximum antenna gain. It is also called Equivalent Isotropic Radiated Power. EIRP can take into account the losses in transmission line and connectors and include the gain of the antenna. The EIRP is often stated in terms of decibels over a reference power emitted by an isotropic radiator with an equivalent signal strength. The EIRP allows comparisons between different emitters regardless of type, size or form. From the EIRP, and with knowledge of a real antenna's gain, it is possible to calculate real power and field strength values.

EIRP=Pt × Gt
Pt: the output power of the transmitter (unit: dBm)
Gt: the antenna gain of the transmitting antenna (unit: dBi)

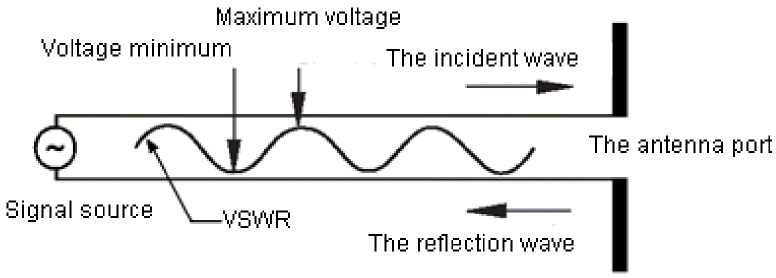
Logarithmic (dB) formula:
EIRP=P - Loss + G
P: output power of transmitter (unit: dBm)
Loss: feeder loss between transmitter output terminal and antenna feed source (unit: dB)
G: antenna transmission gain (unit: dBi)

PEIRP (Peak Effective Isotropic Radiated Power): The peak value of EIRP.

ERP (Effective Radiated Power): Comparing to half wave dipole antenna, it is the power obtained in maximum transmitting direction.

VSWR (Voltage Standing Wave Ratio):

$$\mathrm{VSWR} = \frac{\mathrm{Vmax}}{\mathrm{Vmin}} = \frac{1+|\Gamma|}{1-|\Gamma|}$$



VSWR is commonly represented in Return Loss (RL) (indicated as S11) in engineering:

$$\mathrm{RL} = -20\lg\frac{V+1}{V-1}\ (\mathrm{dB})$$

The corresponding relationship between RL and VSWR is shown in the table below:

Table 1: VSWR and Return Loss

VSWR	1.20	1.25	1.30	1.35	1.40	1.50	2.00
Return Loss (dB)	-20.83	-19.09	-17.69	-16.54	-15.56	-13.98	-9.54

2.2. Basic Requirements of Antennas

Table 2: Basic Requirements of Antennas

Items	Requirements
Frequency Band	Determined by the supported operating bands of devices
VSWR	≤ 3
Gain (dBi)	≥ 1
Max Input Power (W)	50
Input Impedance (Ω)	50
Polarization Type	● Linear Polarization: Horizontal Linear Polarization Vertical Linear Polarization ● Circular Polarization: Right Hand Circular polarization (RHCP) Left Hand Circular polarization (LHCP)

3 Embedded 2G/3G/4G/5G Antennas

3.1. Flexible PCB Antennas (FPC Antennas)

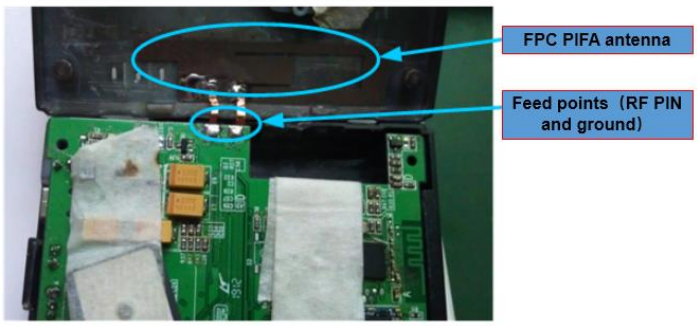


Figure 2: FPC Antenna (PIFA Antenna, Fed from Soldering Pads)

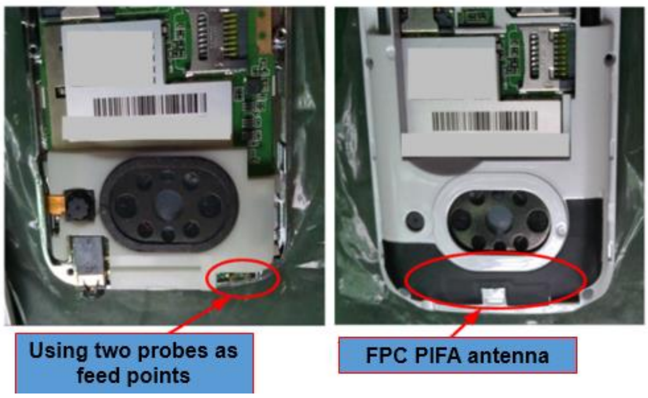


Figure 3: FPC Antenna (PIFA Antenna, Fed from Spring Probes)

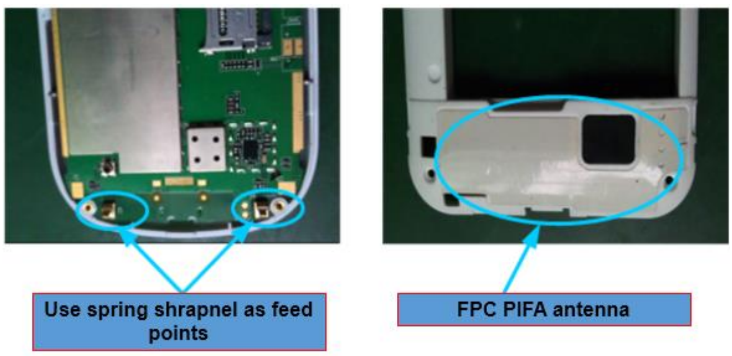


Figure 4: FPC Antenna (PIFA Antenna, Fed from Metal Shrapnel)

Notes:
1. The PIFA antenna fabricated on FPC can be fixed within a plastic device case, thus especially suitable for PDA and automotive devices which have a demanding requirement on space. Heat staking is recommended to be used to reinforce the antenna. The antenna is typically fed from an intermediate point and a ground point, and when it is found that the bandwidth of a high-frequency band is not enough during antenna tuning, another ground point will be used to increase the bandwidth.
2. Keep the distance between antenna and the main PCB more than 5mm. (The actual distance will be determined by the consulting results from antenna supplier.)
3. For the device case with a curved shape, relief hole and relief groove should be added when fixing the antenna to avoid warpage and unstable fixation. As the effectiveness of 3M adhesive will be weakened in high ambient temperature, heat staking or other mounting methods should be used to fix the FPC antenna.
4. Feed points can be designed as soldering pads, probes or shrapnel. Customers can choose according to actual usage.

3.2. Embedded Antennas with Plastic Bracket

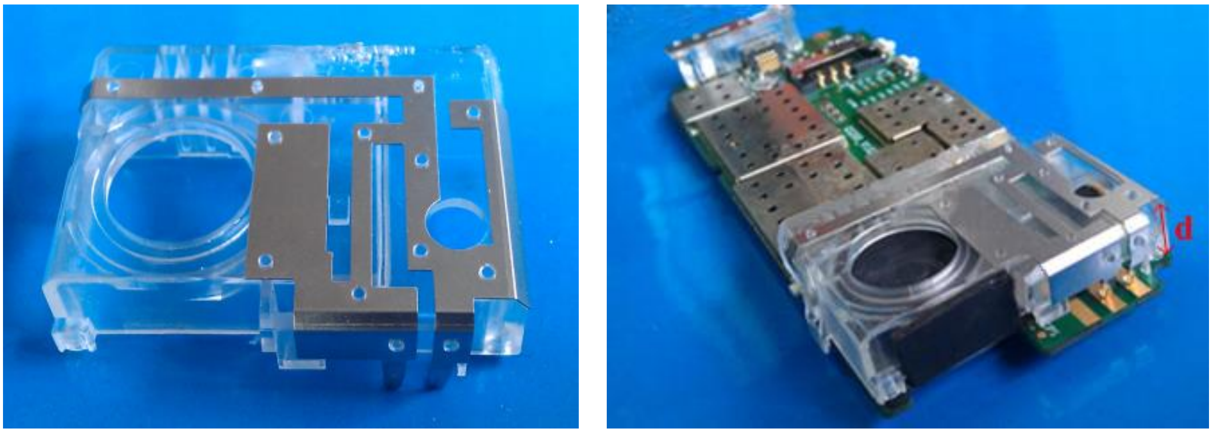


Figure 5: Example of a PIFA Antenna with Plastic Bracket

Notes:
1. The antenna is typically fed from an intermediate point and a ground point, and when it is found that the bandwidth of a high-frequency band is not enough during antenna tuning, another ground point will be used to increase the bandwidth. (RF engineers and structural engineers should both be involved during antenna selection and initial design stages. And the bracket should be designed according to the requirements of antenna supplier.)
2. The material of this kind of antenna is hard. The antenna trace routing is simple and inflexible. And there is usually a large antenna trace area, therefore it has more requirements on environment conditions.

3.3. Laser Direct Structuring (LDS) Antennas

Figure 6: Example of an LDS Antenna

Notes:	
1.	The antenna features high compatibility and precision, stable performance as well as simple and environment-friendly manufacturing process.
2.	Trace can be routed on device case or bracket, thus saving space within the device.
3.	The LDS production process and the material of bracket make the price of LDS antenna much expensive than other antennas.
4.	As the antenna is attached to a plastic bracket, deformation of the bracket will cause fracture of the antenna.

3.4. Metal-Frame Antennas

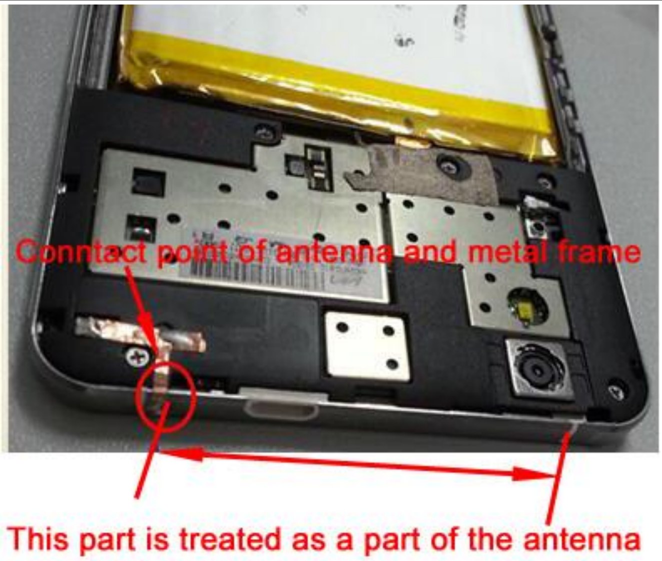


Figure 7: Example of a Metal-Frame Antenna

Notes:

1. This type of antenna is the mainstream 4G antenna. In a 4G device, the metal frame of the device acts as a part of the antenna. The integration of antenna into device facilitates device appearance design.
2. The antenna design employs tuner and switch for performance improvement.
3. This kind of antenna has a demanding requirement on R&D effort and the R&D cost is high.

3.5. Chip Antennas

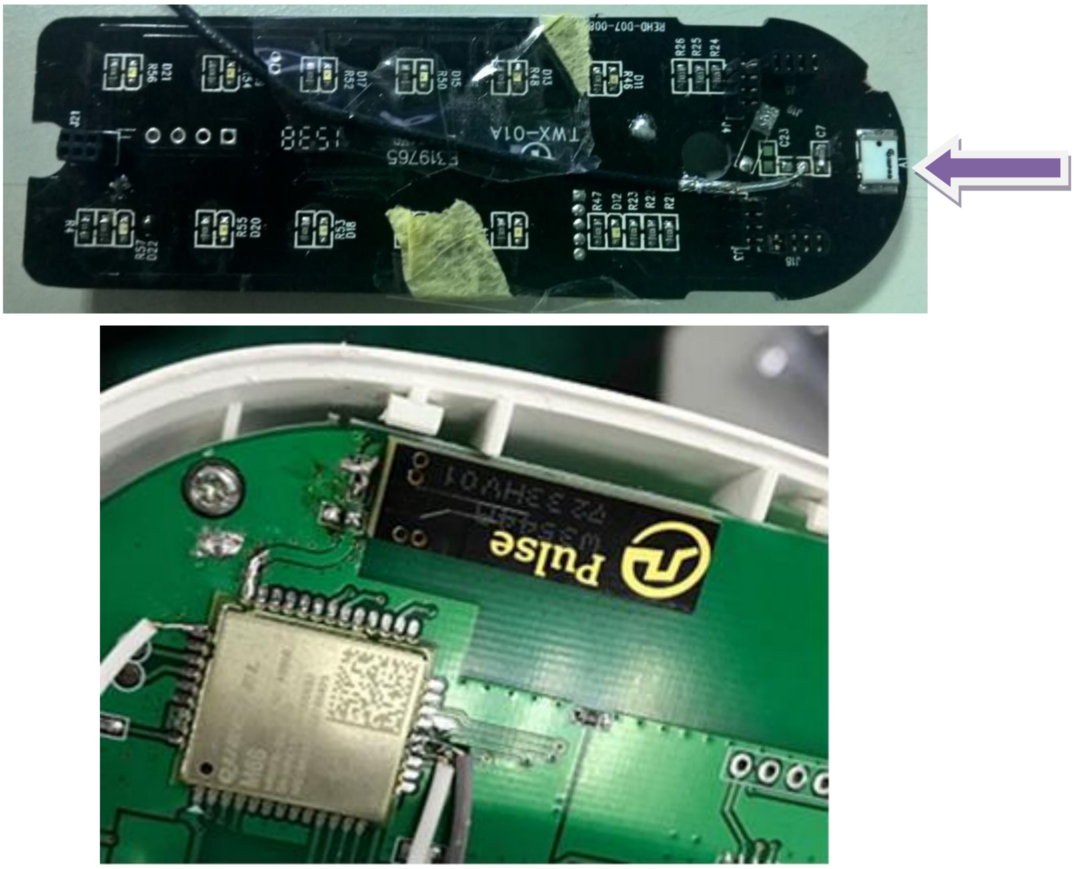


Figure 8: Example of a Chip Antenna

Notes:
1. This type of antenna requires large clearance area and its cost is high.
2. The requirements on reserved space and antenna placement proposed in antenna supplier's datasheet should be strictly followed.
3. The antenna can be surface mounted and it should be placed at the edge of PCB.
4. The environment and antenna matching have a notable influence on antenna performance.

3.6. PCB Trace Antennas

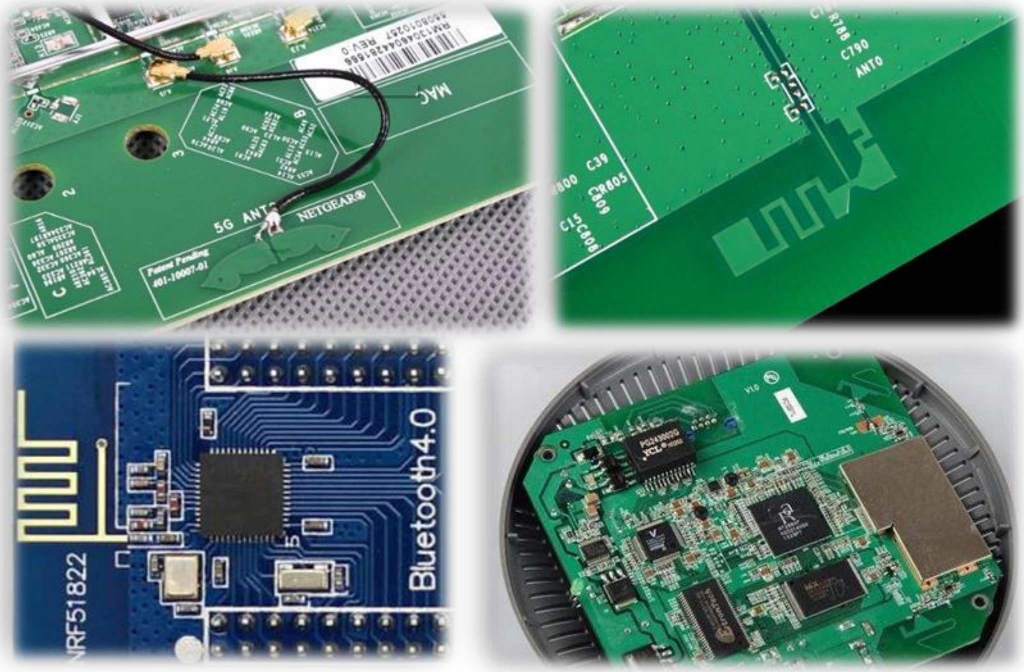


Figure 9: Example of PCB trace Antennas

Notes:

1. Antenna design assessment should be carried out at the initial stage of PCB layout so as to reserve an appropriately sized area for antenna.
2. The environment will have an obvious impact on the antenna performance. And the material, placement and layout will all affect the antenna performance.
3. The antenna is mainly used for devices with limited operation bands. Besides the antenna trace area, the length of the ground plane of PCB should not be less than the quarter-wavelength at the lowest frequency.

3.7. Other Antennas

Figure 10: Example of Other Antennas

Notes:
1. The antennas listed above are typically fixed into a plastic device case or mounted in a suitable notch, and thus they do not occupy any space on the main PCB.
2. The antenna takes large space and metal components cannot be placed in close proximity. The length of the antenna usually falls between the half-wavelength and one-quarter wavelength at the frequency of operation.
3. The antenna can be connected by RF connector or soldered onto the RF output port on the main PCB.
4. During antenna tuning, it may be necessary to make use of the device/PCB's GND.

4 External 2G/3G/4G/5G Antennas

4.1. Dipole Antennas

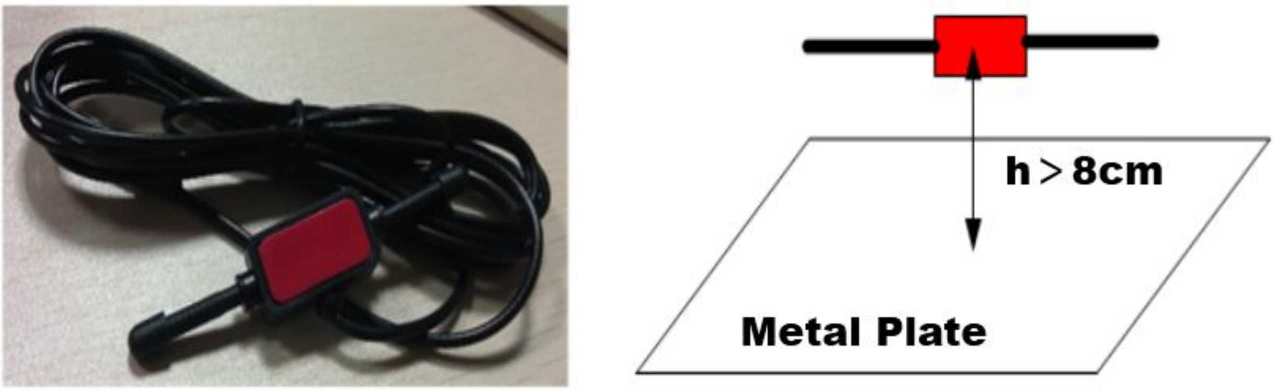


Figure 11: Example of a Dipole Antenna

Notes:

1. The height between antenna and metal plate should be at least 8cm. The dipole antennas are mainly used for 2G applications due to limited frequency range.
2. As the antenna cable length will impact the antenna performance, low-loss cables are recommended.

4.2. Sucker Antennas

Figure 12: Examples of Sucker Antennas

Notes:
1. Sucker antennas are easy to build and install. The choice of antenna types depends on different working environments.
2. Please place the antennas on metal components to achieve the best performance.
3. As the length of an antenna cable negatively affects the antenna performance, it is recommended to use low-loss cables when long trace is required.

4.3. Sleeve Antennas

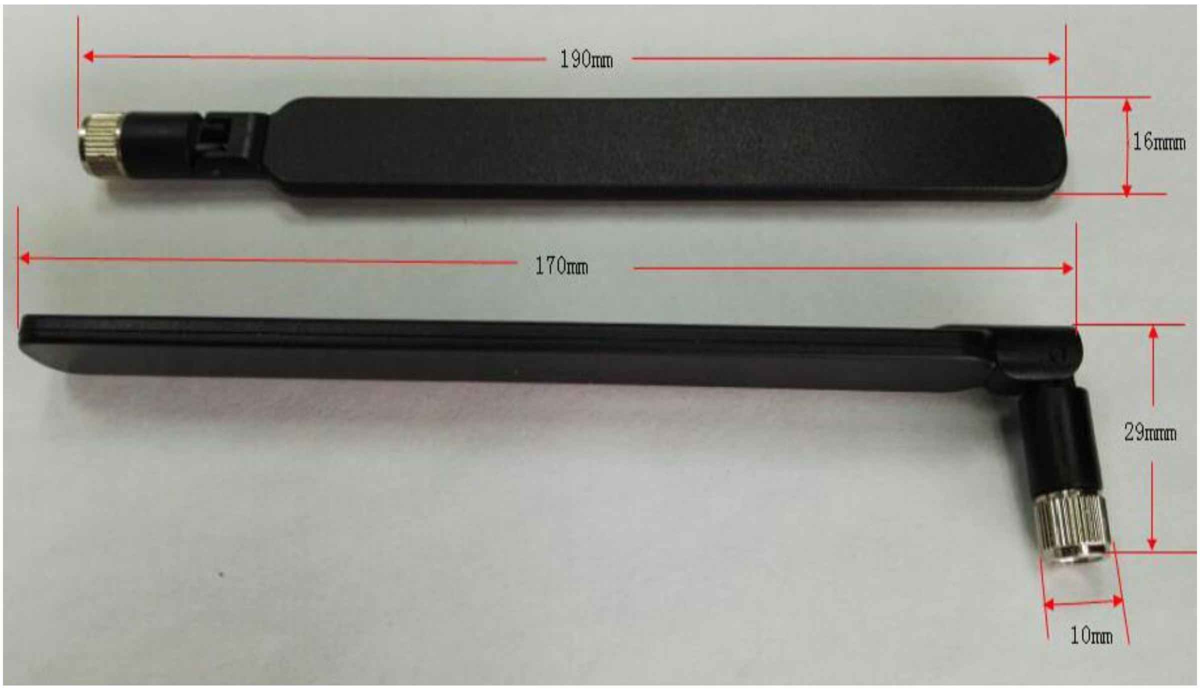


Figure 13: Example of Sleeve Antennas

Notes:
1. Sleeve antennas are available in a great variety of types and they typically support wide operation frequency range. They are easy for replacement.
2. They can be fabricated in the form of PCB, copper pipe or FPC.
3. The choice of antenna types depends on different working environments.
4. With proper stands, sleeve antennas can be turned into sucker antennas.

5 Embedded GNSS Antennas

5.1. Embedded GNSS Active Antennas

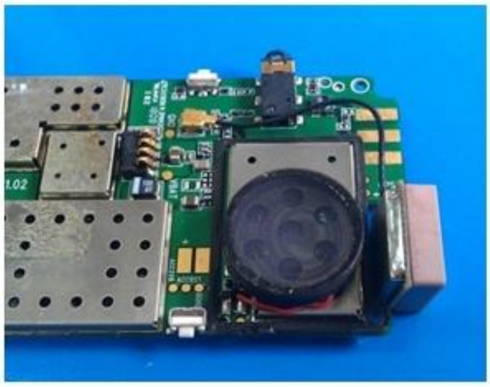


Figure 14: Example of Embedded GNSS Active Antennas with a Welding Needle

Notes:
1. The active antenna has a built-in LNA to improve signal strength. Please keep the antenna radiation surface towards open sky.
2. Please make sure the heights of metal components nearby are lower than the antenna.
3. Square-shaped antenna is right hand circular polarized, while rectangle-shaped antenna is linear polarized. And the former is preferred for better satellite signal receiving.
4. Please keep the antenna cable as short as possible, and low-loss cables are recommended.
5. It is necessary to reserve LNA power supply circuit on the motherboard and a blocking capacitor should be reserved to block DC. Inductors above 56nH should be connected in series between the power supply and the impedance line.

5.2. Embedded GNSS Passive Antennas

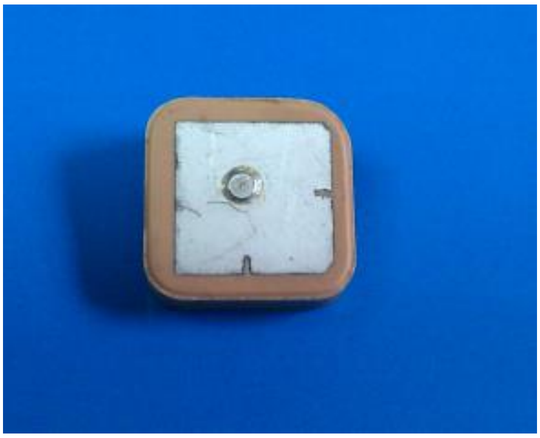


Figure 15: Example of an Embedded GNSS Passive Antenna with a Welding Needle

Figure 16: Application of GNSS Passive Antennas with a Welding Needle

Notes:
1. An LNA should be mounted on the motherboard to improve signal strength of the passive antenna. Please keep the antenna radiation surface towards open sky.
2. Please make sure the heights of metal components nearby are lower than the antenna.
3. Square-shaped antenna is right hand circular polarized, while rectangle-shape antenna is linear polarized. And the former is preferred for better satellite signal receiving.

6 External GNSS Antennas

6.1. External GNSS Antennas

Figure 17: Example of an External GNSS Antenna

Notes:
1. Please keep the antenna radiation surface towards open sky.
2. The antenna should be placed away from metal components and provided with sufficient clearance.
3. Customers will decide the length of antenna cable according to different applications. Please keep the cable as short as possible.
4. The LNA gain may vary with module types.
5. It is necessary to reserve LNA power supply circuit on the motherboard and a blocking capacitor should be reserved to block DC. Inductors above 56nH should be connected in series between the power supply and the impedance line.

7 Internal Wi-Fi Laminated Antennas

7.1. Internal Wi-Fi Laminated Antennas

Figure 18: Example of an Internal Wi-Fi Laminated Antenna

Notes:
1. Wi-Fi antennas typically work at 2.4GHz and/or 5GHz operating bands. As a type of directional antenna, they can realize maximized performance. The distance between the antenna and motherboard is preferred to be 20mm.
2. The antenna should be placed away from high metal components and provided with sufficient clearance.
3. The top patch antenna in green color works at 5GHz, and the ceramic antenna in white color works at 2.4GHz. The polarization design for the two antennas should be the same as that for the transmitting antenna.
4. The antenna cable length should be designed according to application demands. Please keep the cable as short as possible and low-loss cables are recommended.

8 Antenna Isolation Design Requirements

8.1. Antenna Isolation

For two or more antennas with overlapping frequency bands in space, antenna isolation is defined as the ratio of a signal received by one antenna to the signal transmitted by another antenna. The isolation depends on the radiation pattern of the antenna, the spatial distance of the antennas, etc. Antenna isolation improvement is to take measures to minimize the impact of various interference on the receiver.

The isolation between antennas can be increased by:

- Increasing the physical separation between the antennas
- Placing antennas working in adjacent bands orthogonally to each other
- Having the antenna's peak radiation in different or opposite directions
- Changing the antenna trace implementation method (such as length, width)

Additionally, adding a filter in the RF signal path will help to improve the desired signals' suppression effect on those unwanted signals, thus abating the interference caused by insufficient antenna isolation.

8.2. Isolation between 3G/4G/5G Antenna and GNSS Antenna

- The isolation between a 3G/4G/5G antenna and a GNSS active antenna should be at least 10dB.
- The isolation between a 3G/4G/5G antenna and a GNSS passive antenna should be at least 15dB.

It is recommended to add a filter between a GNSS antenna and a module to suppress interference from other antennas.

8.3. Isolation Design of Other Antennas

- The isolation of other antennas should be at least 15dB.
- Isolation design is vital to antennas working at similar frequencies. When two antennas are used synchronously, one of them is recommended to be designed to support only part of the bands to ensure better performance.
  For example, for a device supporting GSM B1/B3/B5/B8 & LTE B1/B3/B7, the main antenna should support all the bands while the diversity antenna only needs to support LTE bands.

9 Antenna Design Service

Quectel antenna design service team will be able to meet each of your antenna design consulting/evaluation, antenna design/testing, certification support, as well as antenna manufacturing demands.

For any antenna design issue, please contact Quectel antenna design service team at antenna.support@quectel.com.