www.neoway.com

Neoway 有方

Antenna Port Application Note

Version 1.0

有 物 联　方 智 能

GET CONNECTED GET SMART

Copyright

Notice

This document provides guide for users to use Neoway modules.

This document is intended for system engineers (SEs), development engineers, and test engineers.

The information in this document is subject to change without notice due to product version update or other reasons

Every effort has been made in preparation of this document to ensure accuracy of the contents, but all statements, information, and recommendations in this document do not constitute a warranty of any kind, express or implied.

Neoway provides customers complete technical support. If you have any question, please contact your account manager or email to the following email addresses:

Sales@neoway.com

Support@neoway.com

Website: http://www.neoway.com

Revision Record		
Issue	**Changes**	**Date**
V1.0	Initial version	2017-03

Contents

About this document

This document describes the antenna port design points of customer in detail.

This document is applicable to all the modules of Neoway Technology Co., Ltd.

1 Circuit design

In this chapter, we will introduce the circuit design of the antenna port, including the circuit design of the power supply and the circuit design of the antenna impedance matching .

1.1 Circuit design of power supply

In the worst case, the communication system of the module may need the 2W power, which may lead to the decrease of the power supply voltage of the module and unstable working status. In order to ensure that the voltage of power supply of module is stable, recommended to increase a large tantalum capacitor (100uF or higher), near power supply pin position of the module.

High frequency noise of the power supply will affect the RF performance of module, recommend to increase multiple ceramic capacitor with small value (33pF, 100pF). If you use the DC-DC to supply the module, in order to avoid the switching frequency of the DC-DC interfering radio frequency, recommend to add a magnetic beads in the power(it is able to withstand 2A current at least), it is recommended 0805 package. In actual, the capacitor can be increased or decreased according to the actual situation and the value of capacitor can be adjusted. The magnetic beads of the power can use 0Ω resistor as replacement.

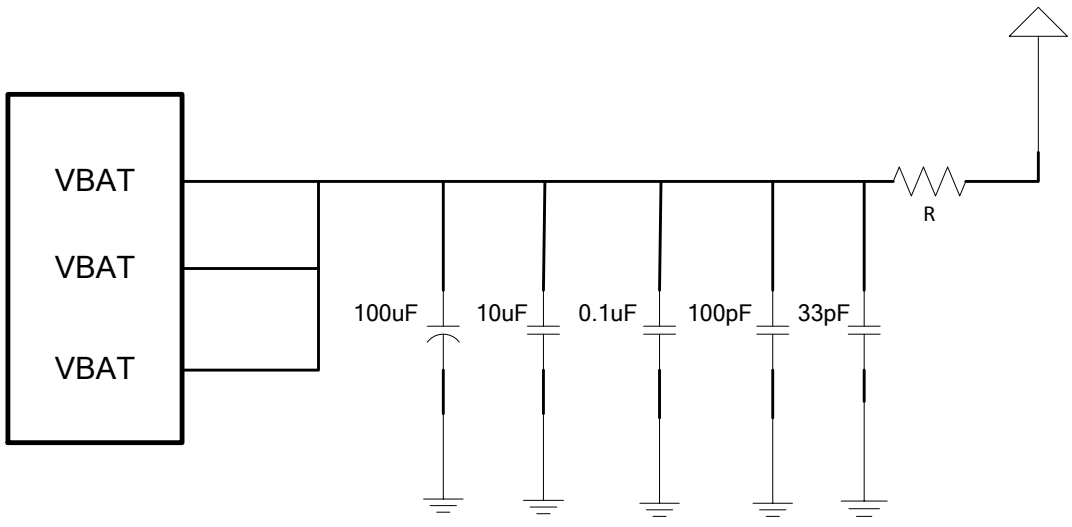

Figure 1-1 Reference circuit for power supply

1.2 Antenna matching circuit design

In order to ensure the best performance of the antenna, it is necessary to add the matching circuit between the module and the antenna interface, please refer to Figure 1-2 and Figure 1-3.

J1 is the antenna port, have a variety of forms, such as pad, RF connector, antenna shrapnel and so on, customers choose according to their own design

The antenna matching design in Figure 1-2 is suitable for the main antenna, LTE diversity antenna, Bluetooth /WIFI antenna, GPS passive antenna. In this design, the antenna matching can be made into the π type (C1, L1, L2), can also be made of T type (C1, C2, L2). L3 is inductance, close to the antenna port, in addition to being used as matching, as well as to improve the performance of ESD. More than 27nH winding inductance is recommended.

The actual device type selection is not shown in the figure. For example, according to debugging, C1 may be inductance, L1 may be capacitance.

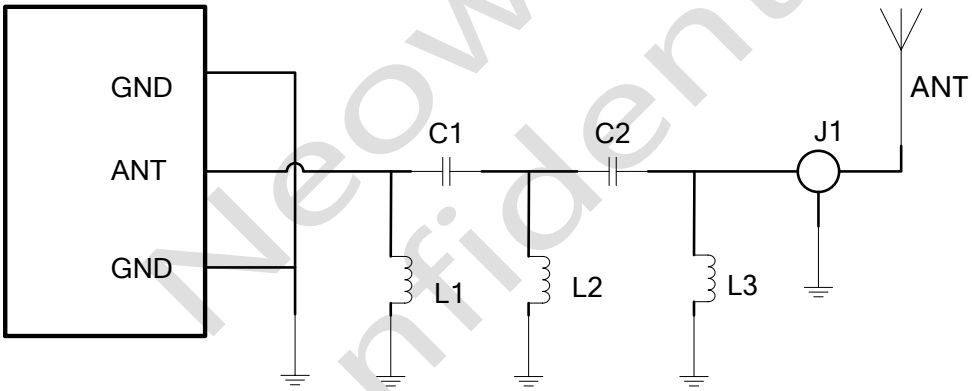

Figure 1-2 Reference circuit for passive antenna matching

Figure 1-3 is an active antenna reference design, commonly used in GPS external antenna. VREF need to choose low noise power supply, such as low-noise LDO which used in the RF circuit. C3 is a separate capacitor, can choose 33pF or 100pF. L3 recommends to use high Q winding inductance.

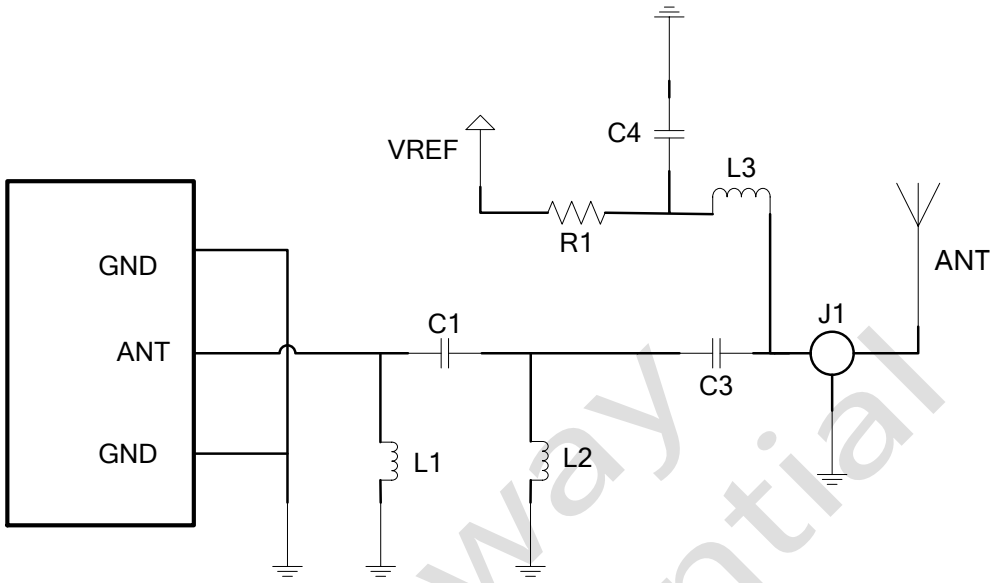

Figure 1-3 Reference circuit for active antenna matching

For antenna matching circuit design, please note the following:

- Please ensure that one completed π type matching network between ANT and J1 at least, which is shown in the figure as L1, C1, L2.
- The component of antenna matching circuit is only capacitance, inductance, 0Ω resistance. Recommend to use high Q capacitance inductance.
- The distance between ANT and J1 should be not too far. If distance is too far, it is recommended to add the matching circuit close to the ANT and the J1.
- If there are equipment lightning protection requirements, you will need to do professional lightning protection design.

2 Layout and line design

Layout and line design between ANT and J1, please follow the below principle to design：

- The matching circuit should be far away from the noise source, such as high-speed digital circuit, if not, please shield the noise source.
- The antenna matching circuit cannot cross with power supply, decoupling capacitor in the power supply of module should be close to the power supply pin of module.
- Ensure that the impedance of the transmission line is 50Ω, and keeps the PCB line as short as possible.

- PCB LAYOUT is as straight as possible and try to avoid through the hole line to another layer; at the same time, avoid the right angle line or acute angle line.
- The surrounding of PCB line should have a good reference place to avoid other signal lines near the antenna line isolating without place.
- A completed layer as place is recommended, can use this completed layer as reference place.
- The area around the antenna must be connected to the main ground through the hole, placed near the edge of the ground and track the alignment.

The following is the complete structure of the 50 micro strip and strip line and recommended stack reference design.

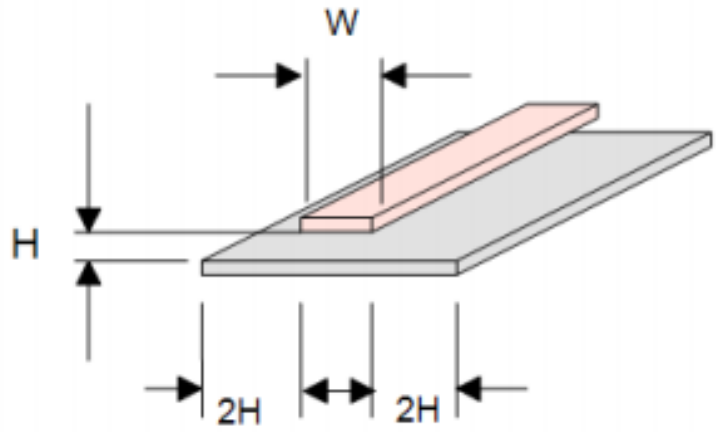

Figure 2-1 Completed structure with micro strip

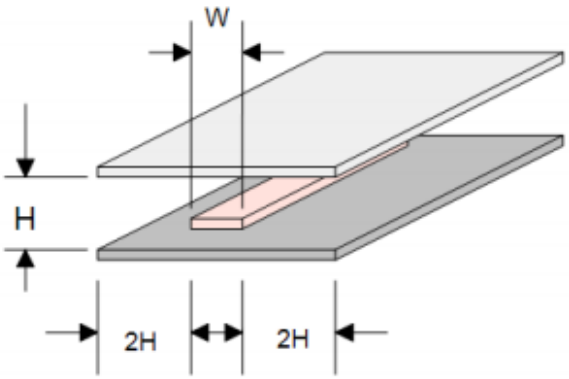

Figure 2-2 Completed structure with stripline

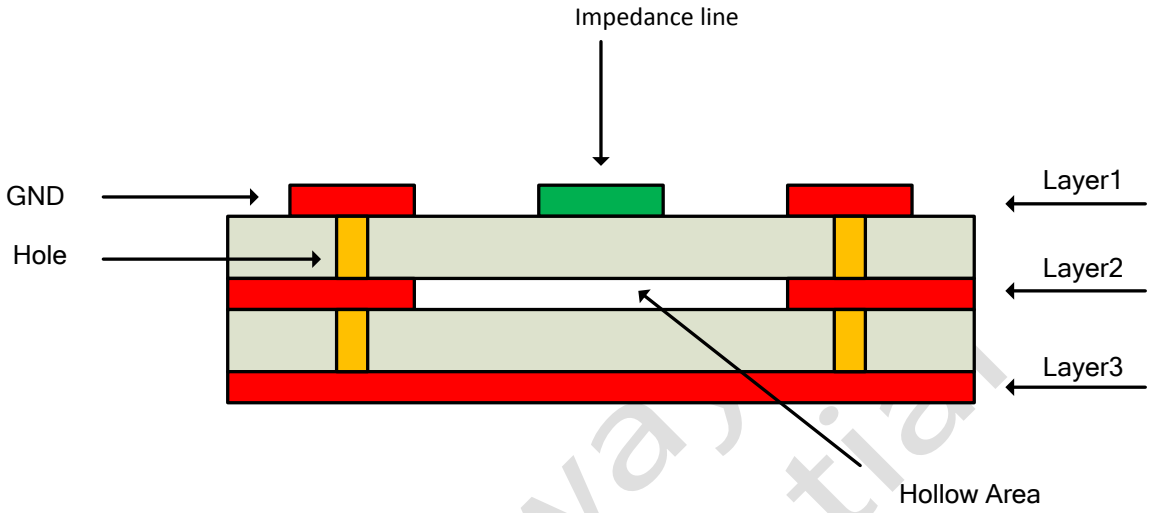

Figure 2-3 Recommended stack reference design

Trace impedance is related with the PCB board used, PCB thickness, alignment width and other parameters, there are much software can calculate the impedance of the alignment, such as Polar SI9000. But also in the outgoing PCB production documents, told the PCB manufacturer that the general PCB manufacturers in accordance with the customer's impedance requirements to adjust the width of the impedance line.

3 Antenna port type

- The pad welding, it is low cost, but the performance is not good. This way cannot do completely electromagnetic shielding, so we must pay attention to that there is no strong radiation in the vicinity of the pad. At the same time, there are some reliability problems, such as the welding is not strong and the shielding wire is shaking, it will cause the core line to break.
- Antenna shrapnel. Built in passive FPC antenna. In this way, the structure correlation between the antenna and the whole device is very high, Should deal with the environment around the antenna.
- SMA radio frequency cable. Easy to connect, It should have good shielding between external antenna and SMA connector. The external RF cable should stay away from all interference sources, especially the high speed digital signal and switch power supply, etc.
- Radio frequency connector. It can be completely shielded by RF coaxial line, but it is still far away from the interference source. RF coaxial line should be taken into account when adjusting antenna. RF connector is recommended. Suggest to increase the antenna test socket for the module calibration and various certification testing.

4 Antenna design requirements

4.1 Antenna efficiency

Antenna efficiency is the ratio of radiated power and useful power input to the antenna. The radiation power of the antenna is usually smaller than the input power due to the loss of the following antenna parts: return loss, dielectric loss and coupling loss. The antenna efficiency is related to the electrical size, and the antenna efficiency increases with the increase of the electrical size. The transmission line is also part of the antenna, and the line loss increases with the increase of line length and frequency. It is recommended the cable loss as low as possible in the selection.

In order to ensure the RF radiation performance of the module, the antenna efficiency is recommend:

- The main antenna efficiency：≥ 40%
- The diversity antenna efficiency：≥50% efficiency of receiving frequency band of the main antenna

4.2 Passive antenna parameters

S11 (reflection coefficient) refers to the input impedance of the antenna with the reference impedance (50) matching index. VSWR (Voltage Standing Wave Ratio, in Bobbi) is another statement of S11. S11 is related to antenna efficiency, can use network analyzer to measure

Recommended in Bobbi：VSWR≤ 2

Recommended return loss value S11:

- The main antenna≤-12dB
- The diversity antenna ≤-12dB

In addition, antenna efficiency is more important than S11, and the correlation between S11 and wireless performance is small.

4.3 Quarantine

If a wireless device has multiple antennas, the power of any two antennas will be coupled together. Antenna isolation is used to measure the coupling power. The power radiated from the antenna may be received by the adjacent antenna, which will reduce the radiation efficiency of the antenna and affect the operation of other equipment. In order to prevent this situation, the antenna isolation is large enough in the early stage of design.

The antenna isolation is determined by the following factors:

- The distance between the antenna
- The type of antenna
- The direction of antenna

The main antenna must be close to the module to shorten the length of the cable. The diversity antenna is perpendicular to the main antenna. The distance between the diversity antenna and the module can be a little far, so we can use the two port vector network analyzer to measure the antenna isolation.
Recommended isolation between each antenna is 20dB.

4.4 Polarization

The polarization of the antenna is the rotation direction of the electric field vector of the antenna with the maximum radiation direction. Linear polarization or vertical polarization is recommended in the module.

4.5 Radiation pattern

The radiation pattern of the antenna reflects the radiation characteristics of the antenna in the far field. The antenna radiation means the power or the magnetic field intensity of the radiated electromagnetic wave from the antenna in all direction. The power or magnetic field strength will be different with different angular coordinates (θ and φ), but independent of the angular coordinate.
The radio frequency mode of half wave dipole antenna is omnidirectional in the horizontal. The incident wave at the base station is usually horizontal. Therefore, the receiving performance is the best
Module recommended RF mode main / diversity antenna: omnidirectional.
In addition, the radiation pattern of the diversity antenna should be complementary to the radiation pattern of the main antenna.

4.6 Gain and direction

The radiation pattern of the antenna represents the intensity of the radiated electromagnetic wave in all directions. Antenna direction is to measure power density of antenna radiation.
The gain is another important parameter of the antenna, which is also closely related to the direction.
Antenna gain is related to the direction and antenna efficiency. Proper antenna gain can extend battery life
The recommended antenna gain is greater than 0dBi.

4.7 TRP/TIS

TRP (Total Radiated Power)，recommended common frequency bands are as follows:

- GSM850/900>28dBm
- GSM1800/1900>26dBm
- CDMA BC0/BC1>19dBm
- WCDMA Band 1/2/4/5/8/19>19dBm
- TD-SCDMA Band 34/39>19dBm

- LTE FDD Band 1/3/4/5/7/8/17/20 >18dBm
- LTE TDD Band 38/39/40/41>18dBm

TIS (Total Isotropic Sensitivity)，recommended common frequency bands are as follows:

- GSM850/900<-102dBm
- GSM1800/1900<-104dBm
- WCDMA Band 1<-106dBm
- WCDMA Band 2/4/5/8/19<-104dBm
- CDMA BC0/BC1<-102dBm
- TD-SCDMA Band 34/39<-106dBm
- LTE FDD Band 1/2/3/4/5/7/8/17/20 <-92dBm (10MHz Band width)
- LTE TDD Band 38/39/40/41 <-90dBm (10MHz Band width)

4.8 Interference

In addition to the antenna performance, the other interference of PCB board also affects the receiving performance of the antenna. In order to ensure the receiving performance of module, it is necessary to control the interference. For example, Speaker, LCD, CPU, FPC line, audio circuits, power supply should be away as far as possible from the antenna, and do the appropriate isolation and shielding, or do filter on the path.

In the structure stack, it should do the corresponding clearance design for the surrounding environment of antenna.