Product gigaNOVA® GPS SMD Antennas
Document Application Note
Release Date 13 February 2013
Version 3.0

Application Note

Scope

This document outlines matters to be considered when designing a product embodying gigaNOVA GPS SMD antennas A10137 or A10204. These include selecting a suitable location for the antenna and designing the RF feed between the antenna and the radio chip. Advice is provided on the measurement techniques and instrumentation to carry out all the design steps; this will enable users to establish whether they can carry out the integration in-house or should contact Antenova M2M to make use of Antenova M2M's integration service.

1 Introduction

gigaNOVA GPS SMD antennas are electrically small antennas whose electrical characteristics depend to some extent on the size of the ground plane on which they are mounted and the layout of circuit components in the area around the antenna. For many applications the antenna is mounted on a small printed circuit board, in close proximity to other components or circuit boards and with a variety of different connection methods.

The A10137 and A10204 offer a choice of both electrical and mechanical design.

- The A10137 has an electrically balanced design that reduces its dependence on the size and shape of the ground plane. It is mounted in a cut-out notch at the edge of the supporting PCB
- The A10204 uses an unbalanced design and is mounted over the supporting ground plane.

The choice between the two designs may be made on the basis of their form factors. But for applications on ground planes smaller than about 50mm x 50mm, the balanced A10137 may be preferred because of its balanced design and whose performance is less dependent on ground plane dimensions.

The very low levels of the satellite signals arriving at the antenna mean that particular attention must be paid to avoiding or suppressing local noise sources. These are functions of the design of the entire host device and not just of the arrangements for feeding and grounding of the antenna.

To provide the best results in any particular application, the designer needs to have some understanding of the possible effects of the configuration on RF performance. Once the configuration is established the performance of the antenna can be optimised by measuring its input impedance in situ, and designing an appropriate matching network so the LNA is operating into a 50-ohm source impedance.

The A10137 includes an internal transmission line balun. Both types of antenna are connected to the GPS receiver by a standard unbalanced 50-ohm transmission line of co-planar waveguide (CPW) format over a ground plane.

2 Placing the antenna

gigaNOVA GPS SMD antennas must be mounted at the edge of the motherboard of the host device. The mounting position should have a view of the sky, clear of the user's hand, when the host device is in the position in which it will be used. It should preferably be clear of other circuit boards, metallic objects or interconnecting cables. The signal level from any GPS antenna is very low and to avoid loss of signal and increased noise figure it is important that the LNA is placed close to the antenna.

3 Antennas on small RF modules

As a rule of thumb, the smaller the PCB on which an antenna is mounted, the more sensitive the performance of the antenna will be to the environment in which the PCB is mounted and connected. Things become more critical when the longest dimension of the PCB is less than about half a wavelength (~95mm at 1.575GHz). If the module carrying the antenna is mounted on a motherboard (or on a conductive surface of a host device), the ground plane of the module should be grounded to it at a fixed point. A flexible printed circuit (FPC) connection is not a good RF ground, because of its high inductance and poor screening. If the module fitted with the antenna is connected to the host motherboard using a multi-pole connector it is good practice to provide ground pins at both ends of the connector. Fig. 1 shows some typical arrangements.

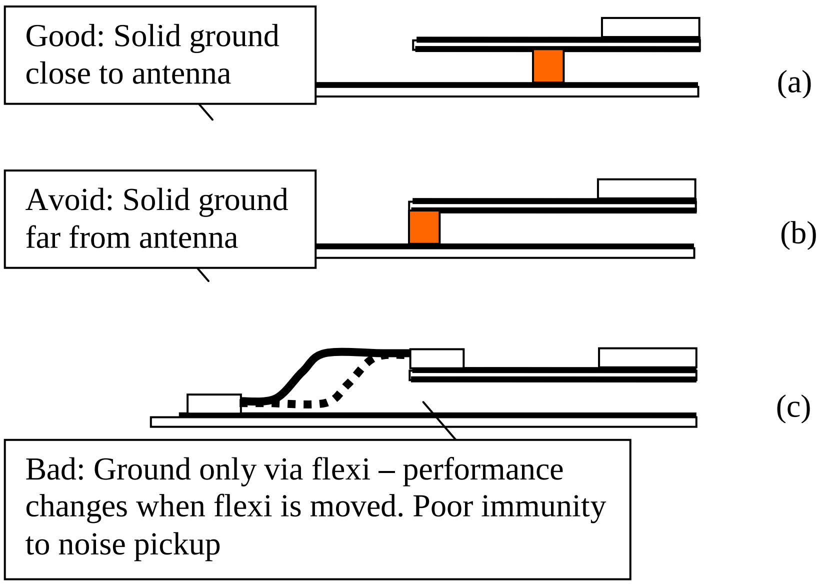


Fig 1: Grounding arrangements for modules on motherboards

The space between an RF module and its motherboard will always tend to be ‘lively’; the arrangement in Fig 1(a) shows solid ground point(s) around the middle of a small module. Assuming only two ground connections will be provided (for economic reasons), this is probably the best method. The ground connections should be made using screws with bushes in contact with the ground plane of the module.

The arrangement of Fig 1(b) in which the ground point is remote from the antenna is likely to be more problematical because the module will itself tend to operate as an antenna – especially if the overhang from the ground point is around a quarter wavelength. In this event the performance of the antenna will be influenced by the connection geometry; the radiation pattern of the ‘real’ antenna and the module (operating together) may not be what is expected. It may be found that adding RF chokes into the interconnecting lines will suppress unwanted currents and reduce potential noise pickup.

The arrangement of Fig 1(c) in which the only ground connection is made via an interconnecting cable or flexi-PCB is always to be avoided. A long flexible connection is an unsuitable RF ground: the antenna impedance will vary according to the exact disposition of the connection, and it is probable that noise from the data circuits and other lines will be coupled into the receiver, reducing the sensitivity of the system and increasing the time to first fix (TTFF).

4 Placing the antenna on the PCB

The antenna must be placed on the edge of a PCB, where it will have a ‘view’ out of the equipment in which it is installed.

A small clearance is required between the antenna and other components on the same side of the board, but if the components are only of low height (small SMD R, L or C), the clearance can be only 2mm. Taller components or components should be kept further away, with a clearance up to 5mm. A reasonable rule of thumb for components not sensitive to RF fields and not radiating RF noise is that they should lie below the line shown in Fig 2.

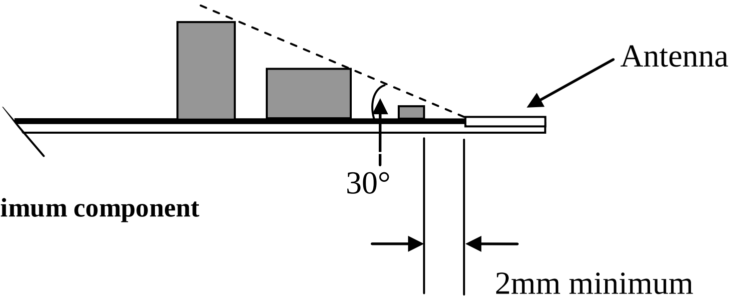


Fig 2: Typical minimum component clearances

The effect of infringing the suggested limit will be a progressive reduction of the efficiency of the antenna, caused by an increased risk of loss of RF energy by coupling into other circuits. It is very difficult to provide hard quantitative guidelines because of interactions between the antenna, the PCB dimensions and the particular layout, component types and circuit functions of the relevant components. Detailed electromagnetic modelling is possible, but it is very expensive to model a complex circuit in the necessary detail, especially when some of the interactions involve complex digital ICs.

5 Designing for low noise

A GPS receiver typically operates with signals as low as -160dBm, so it is essential that the whole platform on which the antenna is placed is designed with noise limitation as a primary design objective. The following rules should always be observed:

a. Place the antenna as far away as possible from noise sources such as microprocessors, memory chips and cameras.
b. As far as possible, use the outer layers of a multi-layer PCB as ground layers; tie these layers together at their edges using vias spaced not more than 10mm apart.
c. Fence-in noise sources by placing vias between the outer ground planes around the noise source; decouple traces which pass through this fence.
d. Screen the GPS LNA/receiver and decouple DC, data and control lines passing to/from them.
e. If the same platform carries other radio transmitters, make sure that there is sufficient isolation between antennas to avoid receiver blocking and also verify that the wideband spurious outputs from the transmitters will not impair the sensitivity of the GPS receiver. The susceptibility to blocking can be improved by the addition of suitable SAW filters at the LNA and/or receiver inputs, but broadband noise must be reduced at the transmitter.

6 Designing the antenna feed circuit

Before doing anything, it is essential to download the receiver and LNA manufacturer's application note and to take special notice of the requirements for the chip-to-antenna circuit. Enough PCB space must be provided between the chip and the antenna to accommodate these circuits. A matching circuit and/or a filter may be needed between the antenna and the LNA/receiver.

To achieve maximum gain the antenna is etched on low-loss PTFE/glass laminate. To prevent the noise figure of the system being significantly increased by the attenuation of the transmission line between the receiver and the antenna, these components must always be mounted as close together as possible.

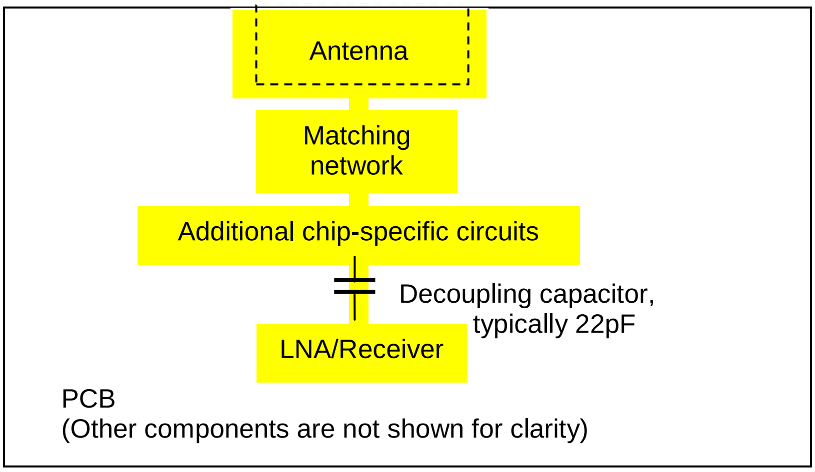
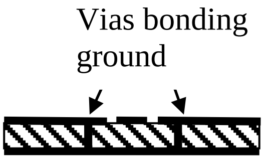


Fig 3: Matching network position. For reference to the additional specified circuits, see above.

7 Designing the feed line

The RF transmission line connecting the antenna to the receiver chip should be configured as a hybrid co-planar waveguide with ground plane as shown in Fig 4.



Co-planar waveguide with ground

Fig 4: Transmission line configuration

The dimensions of this line will vary according to the spacing between the ground plane and the track and also with the permittivity of the substrate. They can be calculated using a simple Windows computer program TXLINE, available free of charge from: http://www.taconic-add.com/en--downloads.php.

To use this program:

- Download and install it;
- Launch the program;
- Select the tab corresponding to the chosen line configuration (CPW Ground)
- Note that the default units may not be the ones you want to use – you may wish to adjust them before you go further
- Enter the permittivity of the PCB material (typically 4.6 for FR4, but less than this for some modern ‘high speed’ board materials (ignore the material names listed by the program unless you happen to be using one of them – the permittivity you enter will be used)
- Enter the effective board thickness (H) (i.e. the distance from the RF ground plane to the line conductor).
- Enter the thickness of the outer layer copper foil (0.017mm for ½-oz copper)
- Enter the required characteristic impedance (usually 50 ohms)
- Enter the chosen gap spacing (G) between the line and the upper ground plane (typically between 0.5mm and 1mm); press the lower calculation arrow and the line width is calculated.

Other transmission line configurations are possible, but their use is not recommended for this application.

8 Static control

It is strongly advised that the electronic assembly is handled in a static controlled environment, but both gigaNOVA GPS SMD antennas are intrinsically grounded and touching the antenna with a finger will not expose the chip to significant electrostatic charge.

9 Designing the matching circuit

9.1 Configuration

The matching circuit usually takes the form of an L-C L, Pi- or Tee-network connected in series with the antenna. This is often configured as a low-pass network (with series L and parallel C) because this configuration tends to reduce the levels of any emitted harmonics which the chip may produce.

Note that the decoupling capacitor is placed between the matching circuit and the amplifier. It provides a DC block, required by a number of receiver chips, and also provides some degree of isolation of electrostatic charge. It should be a high-Q component to avoid introducing loss. The total RF path between the antenna and the receiver should, as already noted, be kept as short as possible.

When designing the first version of the PCB it is good practice to include the necessary accommodation for two series and two parallel components, allowing freedom to configure the matching circuit as required.

9.2 Measurement

The first step is to measure the complex input impedance of the antenna at the input of the matching circuit (i.e. at the terminal of the matching network that is connected to the antenna).

The equipment necessary to do this is a vector network analyser (VNA), and if you have access to this type of equipment there is no reason why you cannot make the necessary measurements and design the matching network successfully yourself.

> If you do not have access to a VNA or do not have anyone available in your organisation who is familiar with its use, Antenova M2M can design your matching network and will be happy to provide a quotation for this service. If you wish to use this service it will simplify matters greatly if you have followed the suggestions contained in this Application Note in the design of your application.

In addition to the VNA and a calibration kit suitable for use at the appropriate frequency, the other equipment you will need comprises:

- A quarter-wave choke fitted with an SMA plug at the open circuit end;
- A short length of UT-47 coaxial cable with a fitted SMA socket.

A suitable quarter-wave choke is shown in Fig 5. It comprises a short length of coaxial cable fitted with an SMA connector at each end – UT141 is ideal for this. A quarter-wavelength piece of copper tube is connected to the outer conductor of the cable at one end and is left open-circuit at the end close to the test item.

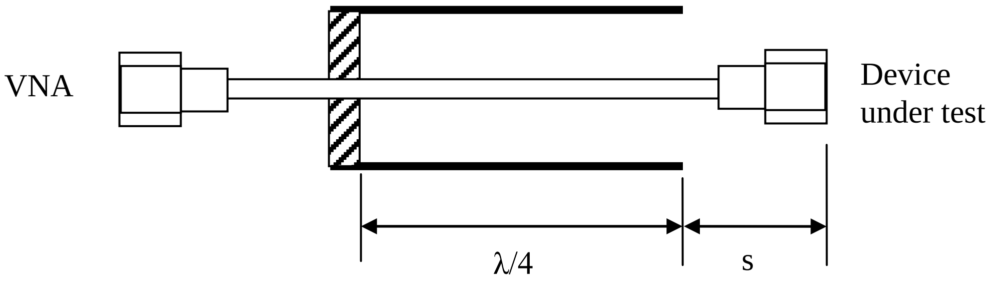


Fig 5: Quarter-wave choke (balun).
Dimension s is kept as short as possible.

The sleeve choke (or balun) is re-usable. Its function is to prevent the cable carrying currents induced by the antenna and inadvertently acting as part of the antenna – in this event the measured impedance is sensitive to the length and position of the cable, When the choke is fitted the measured impedance becomes insensitive to the influence of the cable. If you are interested in a fuller explanation, there is a white paper on the use of baluns for measurement purposes available on Antenova's website.

9.2.1 Procedure

a. Cut and form the short terminated length of UT-47 to fit the test board. Make sure that the minimum possible length of inner conductor extends beyond the outer – just enough to make contact with the pad at the input end of the feed (the receiver end of the matching network). Connect zero ohm resistors in the series component spaces between this point and the antenna; do not fit any parallel components. Do not connect the inner at this stage. Ensure the connector overhangs the edge of the board by the minimum distance possible (it is often possible to solder the connector to the ground at the edge of the board.)

b. Set the VNA *Start* and *Stop* frequencies either side of the band of interest and markers to the lower, centre and upper frequencies of the band. Set the display format to *Smith Chart*

c. Make sure the *Electrical delay* parameter on the VNA is set to zero.

d. Attach the balun to the VNA cable and calibrate the VNA to the end of the balun.

e. Connect the balun to the input of the cable you have just attached to the device under test. The far end of this cable is still open-circuit (as step a).

f. Increase the *Electrical Delay* on the VNA (usually *Scale*/*Electrical Delay*) until the displayed plot coalesces into a dot at the open circuit position on the Smith Chart. Record the value of delay that achieves this. (This can be used later if you need to recalibrate the VNA after connecting the cable inner conductor.)

g. Solder the inner of the test cable to the pad you placed it above (step a).

h. The VNA is now displaying the complex impedance of the antenna at the point of connection of the matching network.

i. If you are testing an antenna mounted on a small module, at this stage it is worth experimenting with the environment of the module to examine the variability of the antenna impedance. Once you are satisfied that you understand the variability that may be found in practice, place the module in an environment that provides some kind of typical result – roughly in the middle of the area of the Smith Chart where the plot moved during this experiment.

j. From the impedance measured, compute the necessary values required for the matching circuit. These can be computed using a program such as *CSmith*, or *Advanced Automated Smith Chart*, or calculated with the aid of a paper Smith Chart, a ruler, a pair of compasses and a calculator.

k. Solder the computed components into the matching network, removing the series zero ohm components where necessary.

l. Measure the input impedance you have now matched and make any necessary component adjustments. Validate the impedance match for various module environments.

Both the bandwidth and the stability of the impedance you will achieve are dependent on the physical size of the module the antenna is mounted on. With very small modules you may need to choose the mounting and/or grounding positions carefully to make it possible to achieve the required performance. This characteristic is typical of any small unbalanced antenna and is not particular to gigaNOVA.

10 Footnote

We hope this application note is useful in implementing gigaNOVA GPS SMD antennas in your application. If you have not found the information you were looking for, please e-mail sales@antenova-m2m.com with your queries and we will be delighted to assist you. This application note will be subject to future revisions, so please contact Antenova M2M sales for the latest version.

antenova®
m2m

www.antenova-m2m.com

Corporate Headquarters

Antenova Ltd.
Far Field House
Albert Road
Stow-cum-Quy
Cambridge
CB25 9AR

Tel: +44 1223 810600
Fax: +44 1223 810650
Email: sales@antenova-m2m.com

North America Headquarters

Antenova Ltd.
Rogers Business Park
2541 Technology Drive Suite 403
Elgin, IL 60124

Tel: +1 (847) 551 9710
Fax +1 (847) 551 9719
Email: sales@antenova-m2m.com

Asia Headquarters

Antenova Asia Ltd.
4F, No. 324, Sec. 1, Nei-Hu Road
Nei-Hu District
Taipei 11493
Taiwan, ROC

Tel: +886 (0) 2 8797 8630
Fax: +886 (0) 2 8797 6890
Email: sales@antenova-m2m.com

ISOQAR
QUALITY ASSURED
ISO 9001
UKAS
QUALITY MANAGEMENT
026
Certificate No: 4598