The GPS antenna connection
Design guide
V1.2

This design guide demonstrates several typical application circuits and should help the application engineer to achieve the best GPS performance.
www.forum.gns-gmbh.com .

GNS Global Navigation Systems GmbH
Adenauerstrasse 18
D – 52146 Würselen
Germany
www.gns-gmbh.com
info@gns-gmbh.com

V1.2, May. 13th 2013

GPS antenna connection Design guide

application note

Index

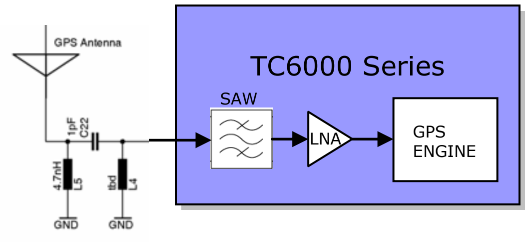
1 Introduction

The "GPS antenna connection design guide" shows the application engineer a practical and simple way to implement RF transmission lines and RF impedance matching for his GPS application.
RF basics are required to follow up this documentation.
It illustrates an overview and comparison about the functional parameters at the market available various GPS antenna types.
Discussed are several noise impacts, which can be generated at a mixed-signal application PCB, and the processes, how to verify and reduce or eliminate them.

2 Antenna design

2.1 Type of antenna

Antenna design is critical because it determines the performance of the entire product.
A poor antenna design will ruin most of the efforts that have been made in receiver development.
The main reasons for a bad performing antenna are: Mismatched antenna frequency and bandwidth, mismatched antenna impedance and noise impacts.
The TC6000 series supports both active and passive antennas. It includes an on-chip LNA (Low-Noise-Amplifier) and an on-module SAW filter. Therefore, a passive antenna can be directly connected to the RF_IN input without any further LNA or filters.



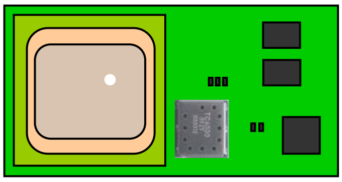
However, a good compromise must be found between small size and antenna performance.
Assuming, that bandwidth and matching is optimized, a bigger (larger area) antenna can generally deliver a better signal than a small one.

2.1.1 Chip antenna

Chip antennas are used for applications which offer very small space, for example mobile phones. Generally, chip antennas are lower in performance due to their small size and linear polarization. Although chip antennas are very small, they are highly dependant on a ground plane and on an "isolation distance" (typically 6mm) where no other components should be placed. Most manufacturers provide designs guides for their GPS chip antennas.
In high density applications like phones and other handhelds, the reception of a chip antenna must be carefully tested in prototype stage, because chip antennas are quite sensitive regarding the components or mechanical parts in their direct proximity.
Chip antennas are not recommended for applications that must reliably work under critical conditions.
For example, the small chip antenna, that is used on the TC6000GN-P1_EM1 board, performs sufficiently under open sky conditions, but performance will degrade, when the receiver is used in deep urban canyons or indoor.

2.1.2 Patch antenna

A common patch antenna of 25mm by 25mm mounted on a metal ground plane is the best choice regarding costs and performance. It should be used, whenever the space conditions allow. The size of the ground plane will essentially influence the performance of the antenna. Best performance (a gain of +4..5 dBi) can be achieved with larger ground plane areas, e.g. 70 by 70mm, while a 30mm by 30mm ground plane will result in a gain of ~1 dBi. The patch antenna should always be placed in the geometrical center of the ground plane. No components should be placed and no tracks should be routed in the area of the ground plane or the patch antenna.



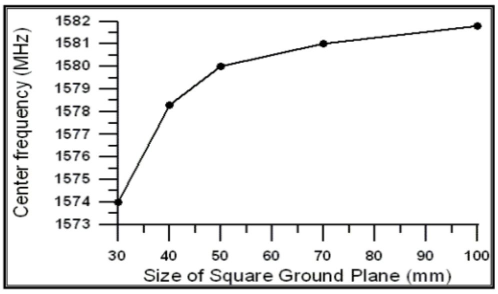
2.1.2.1 Patch antenna size

GPS patch antennas are available in different sizes from 6x6mm^2 to 25x25mm^2 and different thickness. While the 25x25x4mm^2 size is quite robust and high perfoming, the smaller sizes are lower in performance and also more critical regarding their operating parameters.
For a small patch antenna, the resulting center frequency and gain must be carefully observed. For this reason, patch antenna manufacturers offer their products with different center frequencies which allows to choose the antenna that works on the correct frequency together with the customer application.

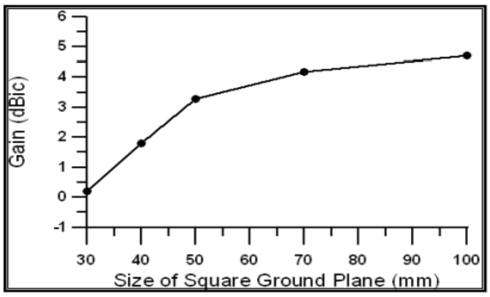
The center frequency alignment should be made using a vector network analyzer (VNA) measuring the return loss (S11).

The following two graphs, as an example, show the relationship between the GND size area and the center frequency as well as the GND size vs. the antenna gain of a typical GPS patch antenna.

frequency vs. ground plane size



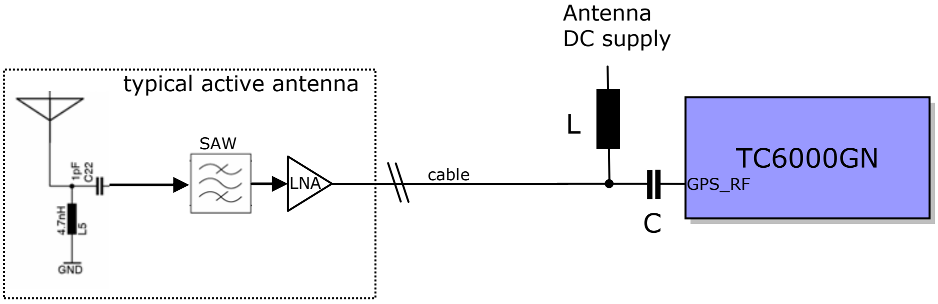
gain vs. ground plane size



2.1.3 Active patch antenna

In use cases where the antenna will be installed separately (more than 10cm away) from the receiver, the antenna should be equipped with an internal LNA, that is placed near to the antenna. The LNA will effectively compensate for cable losses and will minimize the impact of noise.
Such active GPS antennas which contain a LNA and an additional SAW filter are available from many sources on the market.

They require a power supply to power the internal LNA. This power supply is fed over the same wire as the RF (a "phantom" supply) and must be decoupled from the RF_IN line. Please use the following simple circuit to feed-in the antenna supply voltage. The typical power consumption of active antennas is 3 to 30mA, the voltage 1.5 to 5 Volts.



Inductor : L=270nH Capacitor : C=100pF, use low parasitic components. DC voltage : see active antenna spec
The inductor will block the RF against the DC source, the capacitor avoids DC loading of the GPS_RF pin

V1.2, May. 13th 2013

2.1.4 Other antenna types

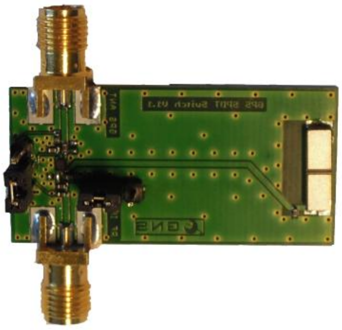
There are some more antenna types available which may be taken into consideration, but are not discussed here because they are rarely used and in special applications, only.
The following table lists some properties of the antenna types.

Type	Performance	Polarisation	Directivity	Costs	Space requirements	implementation
patch	Good	Circular	Hemispheric	Medium	Medium	Easy
Chip	Medium	Linear	Hemispheric, sensitive for disturbances	Low	Very low, but larger area required for good performance	Easy , medium critical
Helix	Good	Circular	Omni	High	Large, best outside of the device	Medium
PCB	Low	Linear	Critical	Very low	PCB space, virtually no hight	Very critical
Fractal	Medium to low	Linear	NN	Low	medium	Medium
F-antenna	Medium	Linear	Omni	Low	Medium	Critical
Loop antenna	Good	NN	Double lobe	Medium	For external use w/cable, large	Easy
Active patch	Very good	Circular	Hemispheric	High	For external use w/cable, large	Very easy
Dipole	Medium to good	Linear	Double lobe	Low	For external use w/cable, large	Critical

2.2 Antenna matching

2.2.1 General

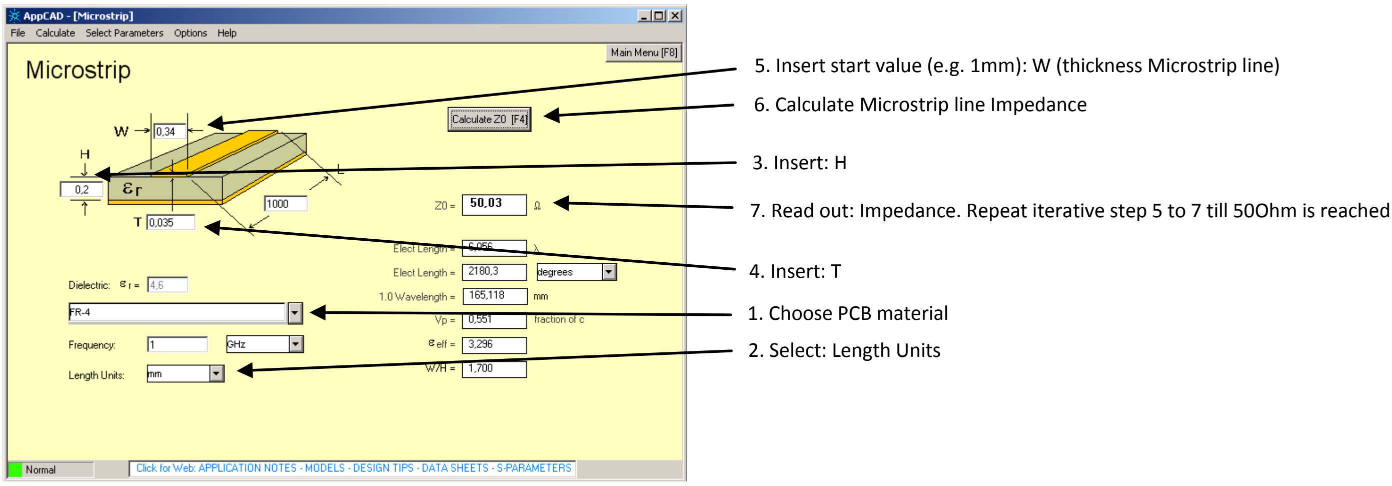
All common GPS antennas are electrically prepared to drive a 50 Ohms load. Therefore the connection (cable or PCB track) and the input of the receiver should be 50 Ohm as well. A deviation will result in power loss and in a possible shift of center frequency.
The antenna must be impedance matched to the input of the TC6000 series, to the PCB, the wiring and the enclosure. The antenna must be impedance matched to the RF input of the TC6000 series for every application. Proper matching will optimize the transmission of antenna input power to the receiver input and therefore improve the overall performance.
Furthermore, please verify your design for correct frequency matching. GPS antennas are relatively narrow band, so there's a risk to tune the antenna out of the band center.
GNS offers support for the antenna matching and provides an evaluation board for a passive GPS chip antenna with an SPDT switch IC on board, which switches between the passive GPS chip or an active/passive GPS patch antenna, GNS part #4037735104549 “GPS Antenna switch evaluation module”.



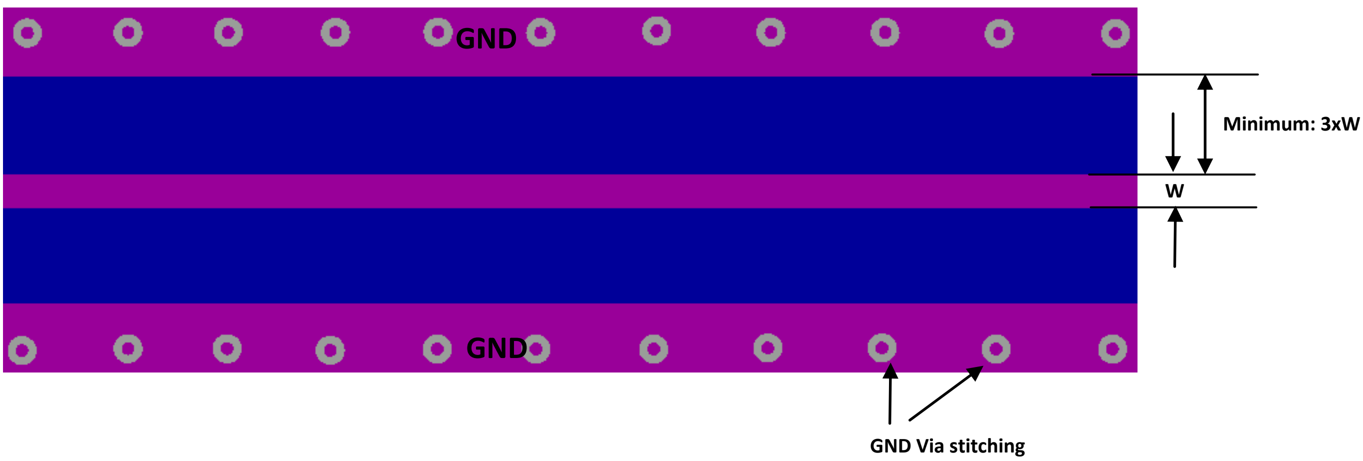
2.2.2 Transmission line type: Microstrip

To connect a GPS antenna it is required to use a 50Ohm impedance transmission line between the RF input of the TC6000GN and the antenna. These impedance lines are GND related transmission lines, which have to be calculated. A free RF software tool from Agilent Technologies "AppCAD" can be used. To setup up calculating transmission line, the "ε_r-relative Permitivity" of the PCB material, for example: ε_r =4.6 for FR-4 material, and the distance between signal/top layer and GND layer (layer2 or bottom) "H" must be known. These values are provided by PCB manufacturer.

Example: H=0,2mm; ε_r = 4,6; T=35um copper cladding



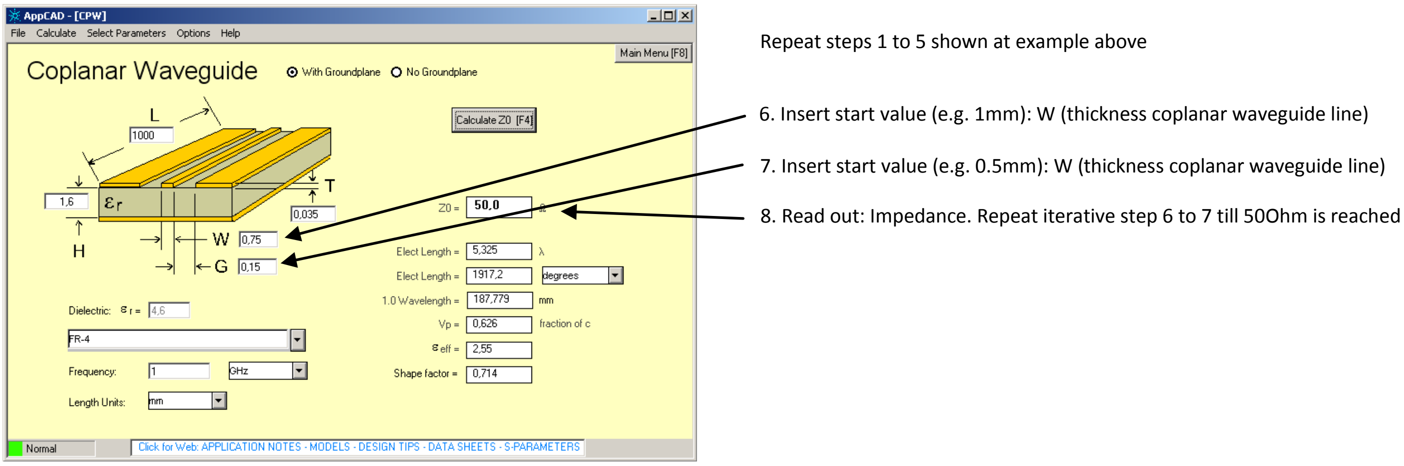
Using a microstrip line it is recommended to have a distance between the microstrip line and enveloping GND plane of minimum 3xW (Rule of thumb)!



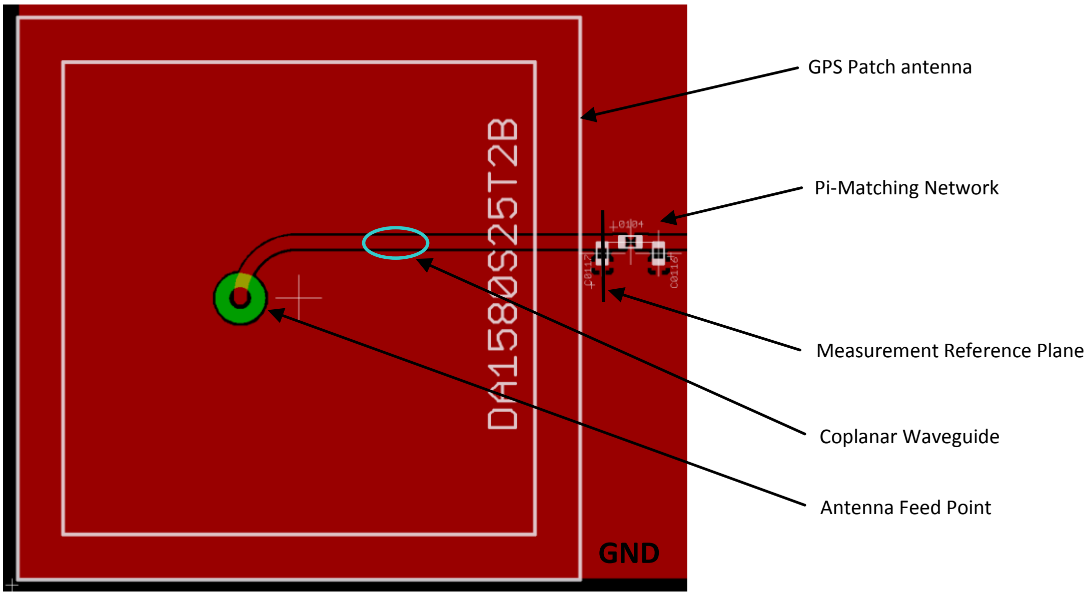
If the GPS patch antenna will be on the same layer as the GPS module or thickness of the Microstrip line will be too thick, because of the PCB build-up or relative Permitivity, a "Coplanar waveguide line (CPW)" should be used, refer to GNS part #4037735104549 "GPS Antenna switch evaluation module".

2.2.3 Transmission line type: Coplanar Waveguide

Example: H=1,6mm; ε_r = 4,6; T=35um copper cladding



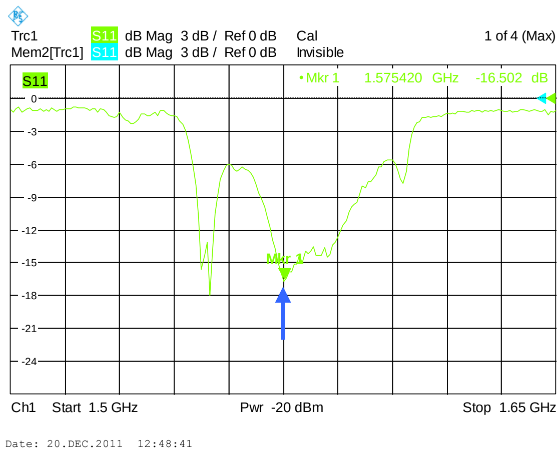
Designing the transmission line between the RF input of the GPS module and the GPS antenna is finished. To match the GPS antenna to the application board, a matching network is required and should be placed as near as possible to the feed point of the GPS antenna. Three lumped elements should be placed as a PI-network closely to the GPS antenna feed point. These three elements are placeholders for the antenna matching network, which has to be simulated or calculated and verified by measurement.



2.2.4 Evaluating matching network values

The following steps should be done for evaluating the matching network components

1. Measure S-Parameter-set using a network analyzer (VNA). Measurement reference plane should be at the first shunt element (marked in above figure)
2. Use the network analyzer measurement results file (*.s2p) as input for a RF simulation software (e.g. *Agilent ADS*, *Ansoft Designer* or similar), which supports an impedance matching tool.
3. Selecting the same frequency grid in the simulation as in the s2p file will provide the best simulation result.
4. After identification of the two matching elements and their values, they should be soldered on the application board carefully at the appropriate positions
5. Next, verify the return loss (S11) again using a VNA. The return loss should be -10dB or better at 1575.42 MHz.
6. In most cases, a fine tuning of the matching components will be necessary to tune the point of lowest return loss to the exact frequency. This tuning should be made by trying the next value from the E12 series and repeating the return loss measurement.
   The graph shows a measurement of return loss with a good result of more than -16dB.



If no RF software tool is available, the impedance measurement procedure can be done alternatively by measuring impedance at the reference line at 1575.42MHz. Then insert normalized impedance to a Smith Chart and calculate the two elements there. The details of this procedure cannot be explained in detail here, therefore please refer to specialized literature.

The GPS antenna matching is finished.

2.3 Antenna placement and directivity

The GPS system is based on a system of satellites, which are moving across the sky. For a good performance of the receiver, the viewing angle of the antenna should be as wide as possible to be able to track the biggest number of satellites. Ideally it should cover the whole hemisphere.
For this reason, the directivity of the antenna, the placement of the antenna and the placement of the whole receiver should allow a free view to the whole hemisphere. When moving through urban canyons or mountainous landscapes, a wide viewing angle will also optimize the view to the available satellites. Furthermore, any material between the antenna and the satellites has more or less influence on the signal. Metal, metalized glass and thick stone will block the signals almost totally, while plastics or glass cause some attenuation.

2.4 Indoor reception

Modern high-sensitivity GPS receivers like the TC6000 series allow indoor operation in many cases. However, it's important to know about the shortcomings of indoor GPS:

1. The indoor signal has high directivity. In many situations, the signals will come in through a window and therefore the available viewing angles will be quite narrow. Since the satellites move over the sky within ~6hrs, the signals will be lost when the satellites leave the viewing angle. Maybe, new satellites will come into view, but in many scenarios the fix may get lost.
2. After turning on the GPS receiver, gaining a position solution depends on receiving the data telegrams from the satellites. This process needs much more signal strength than the tracking mode does. Therefore, time to first fix (TTFF) may be very long or the fix may be impossible.
3. When using GPS indoor in an urban environment, there might be a lot of signals that are reflected from other buildings. These signals will have longer signal paths and therefore, the calculated position will be moved away from the real position. Reflections can easily generate position errors of 50m or more.

→ For indoor applications, we recommend to use an outdoor external antenna whenever possible. Indoor antennas are always likely to cause uncertain (or no) position fixes.

2.5 External LNA

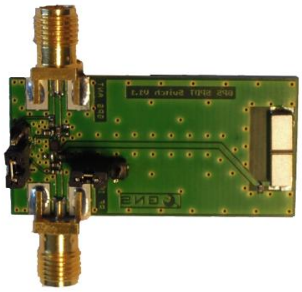
TC6000 series includes a complete RF frontend with integrated LNA and can be directly connected to a passive antenna.
However, in some applications an external LNA can improve the overall performance. An external LNA can be beneficial if the antenna tracks or cables have some loss or if the antenna has low gain.
Furthermore, a very low noise LNA might improve the GPS reception by lowering the noise figure of the entire input stage.
When adding an additional LNA special care should be taken to avoid any saturation of the input.

2.6 Noise impact at antenna input

GPS receivers work at very low signal levels down to -162dBm which allows to track a position in very demanding environments like urban canyons and indoor locations.
Therefore it is very important to avoid any noise signal impact on the antenna or the RF_INPUT tracks.
Thanks to advanced filtering and signal processing, this problem is primarily the result of noise that is in band. For all out-band impacts there's a good suppression because of using a internal SAW filter on all TC6000 series modules.
On the other hand, if there is in-band noise, the receiver may be extremely sensible. Even a very low noise level (that might be really difficult to locate by Spectrum Analyzer measurements without external high gain LNA) can have a strong impact on GPS performance.
If an in-band noise source is identified in your application, better try first to remove the source of problem or move the frequency value away instead of trying to filter or to shield the noise.
If you observe a bad GPS sensitivity during development, please check the following:

- *Are there any frequencies at 1575.42MHz ±10 MHz in the system?*
  → Try to locate the source. Move the source frequency away whenever possible.
- *Are there any (digital) oscillators (e.g. from the microcontroller) that have harmonics on GPS frequency? Even high order harmonics of clocks can disturb GPS.*
  → use a different clock frequency. Decouple RF GND and analog/digital GND.
- *Are these oscillators of R/C type? Such oscillators may move their working frequency with temperature. A high order harmonic of a R/C clock may "move" into the GPS band by thermal effects. Furthermore, R/Cs have big tolerances in initial frequency.*
  → A safe way is to use an Xtal instead of R/C oscillator for your microcontroller.
- *Are there any broadband noise sources in the system? e.g. Switch-mode-voltage regulators ?*
  → Try to optimize noise sources. Shielding or improved PCB tracks should help
- *Last but not least:*
  → In case of unknown noise sources : check your laboratory for noise sources. Computers and even measurement equipment may be sources of noise.

3 Available evaluation kits

type	description	picture	available from
GPS Antenna switch evaluation module	A GPS antenna evaluation board to switch between a passive GPS chip antenna and an active/passive patch antenna.		GNS GNS part#: 4037735104549

4 Document revision history

date	author	comment
12/20/2011	MR	Initial document V 1.0
12/23/2011	PS	Reviewed V1.1
05/13/2013	MR	Document structure reworked V1.2