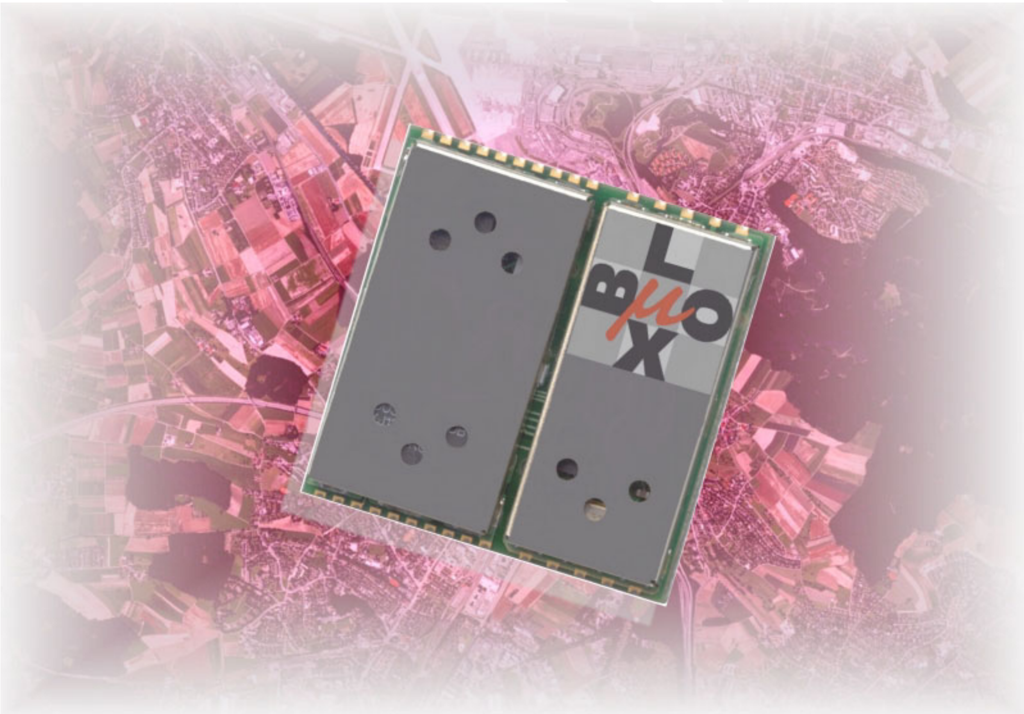
RF Design

FOR GPS RECEIVERS

APPLICATION NOTE



u-blox ag • Zuercherstrasse 68 • CH-8800 Thalwil • Switzerland • Phone +41 1 722 74 44 • Fax +41 1 722 74 47 • www.u-blox.com

Title	**RF Design for GPS receivers**
Doc Type	APPLICATION NOTE PRELIMINARY
Doc Id	GPS-X-02015-A
Author:	Andreas Thiel
Date:	26. June 02
For most recent documents, please visit www.u-blox.com	

CONTENTS

1 OVERVIEW

Given the weakness of the GPS signals sent out by the GPS satellites it's apparently clear that design of the antenna and RF front-end of a GPS receiver requires careful attention. This document gives some insight into the do's and don'ts of RF design for GPS receivers. Antenna selection criteria are as well discussed as layout considerations for proper PCB design.

In Chapter 1 a very brief overview over the characteristics of the GPS RF signal is given and some fundamental definitions and methods of RF design are summarized. Readers who are not attracted by the breathtaking length of the equations listed in this section might prefer to read the following chapters first and only come back to this chapter for specific information.

Chapter 2 presents a discussion of GPS antennas for commercial applications. Selection criteria as well as performance characteristics are summarized.

Chapter 3 deals with all aspects of RF interconnects, like cables, connectors, and RF traces on printed circuit boards.

Chapter 4 is about the most important problem why GPS receivers would not perform as expected in a practical application, i.e. electromagnetic interference. It covers aspects like shielding and techniques for reduction of digital noise emission.

Finally, Chapter 5 provides some useful links to probe further.

1.1 Basic GPS RF-Signal Characteristics

1.1.1 The GPS Signal Structure on L_1

The signal carrier at 1.57542 GHz is modulated using a direct sequence spread spectrum (DS-SS) scheme. Two different coding signals are modulated onto the L_1 carrier: The civilian C/A code (coarse acquisition) and the military P(Y) (precision) code. The chip rates are 1.023 Mcps for C/A and 10.23 Mcps for P(Y) code. The code periods of the pseudo random noise generators are 1023 for C/A code and $2.35469592765 \cdot 10^{14}$ for P code (more than 38 weeks). These code rates were selected so that 1 period of the C/A code corresponds exactly to 1 ms. The Y code is a secure version of the P code where the code is only known to authorized users. Y code replaces P code at U.S. governmental decision.

The combination of the two coding signals onto the single L_1 carrier is realized by quadrature modulation, where the P code modulates the in-phase component and the quadrature component is modulated by the C/A code. When using only C/A code this can be regarded as a binary phase shift keying (BPSK) modulation, disregarding the weaker and higher frequency P-component. A 50 baud binary data stream is additionally modulated onto the code sequence by using modulo-2 addition.

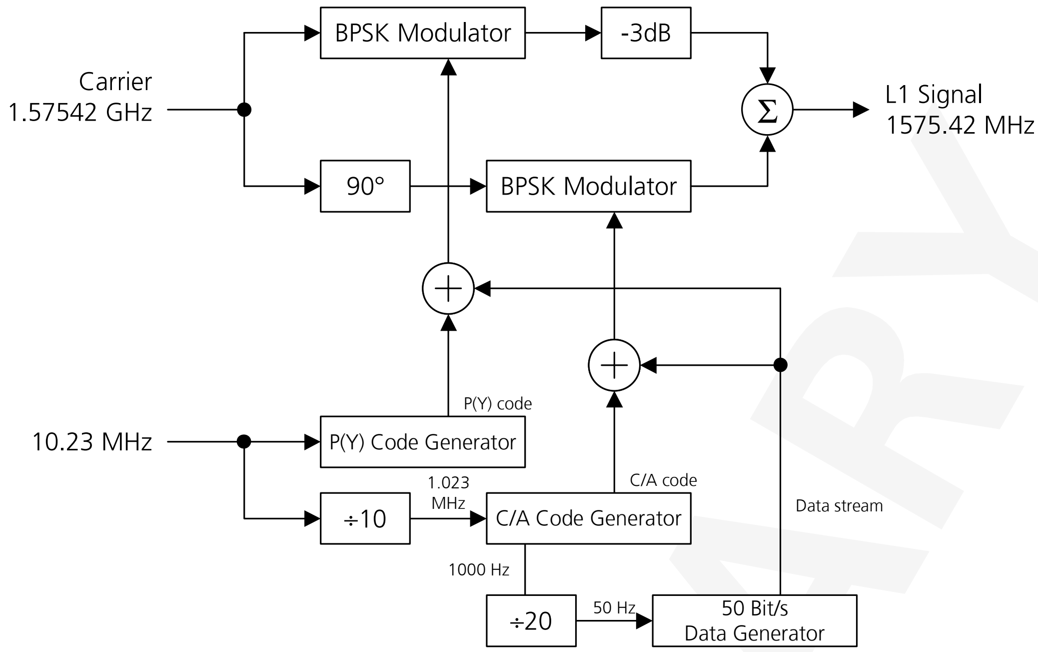


Figure 1-1: GPS L1 Signal Structure

The resulting signal for the L_1 carrier is given in Equation 1-1. Here, P_c and P_P are the C/A and P(Y) signal powers, respectively. P_c is 3 dB stronger than P_p. ϕ represents an arbitrary phase offset. The P code $XP_i(t)$ is a ±1 pseudo noise sequence with a clock rate of 10.23 Mbps. The C/A-code $XG_i(t)$ is a ±1 Gold code sequence with a clock rate of 1.023 Mbps and a period of 1023 bits. $\omega_1 = 2\pi f_1 = 2\pi \cdot 154 \cdot 10.23$ MHz. $D_i(t)$ is the ±1 binary data stream at 50 bps.

$$S_{L_1}(t) = \sqrt{2P_c}\, XG_i(t) D_i(t) \cos(\omega_1 t + \Phi) + \sqrt{2P_P}\, XP_i(t) D_i(t) \sin(\omega_1 t + \Phi)$$

Equation 1-1

1.1.2 Thermal Noise Power

Thermal noise power of a 50 Ohms source (antenna) into a 50 Ohms load is calculated according to Equation 1-2. Substituting the null-to-null bandwidth of 2.046 MHz of the C/A-code modulation for Δf and 290 K (17 degree Celsius, 63 degree Fahrenheit) for the source temperature T one obtains 8.5 pW or -111 dBm noise power or a power spectral density of N_0 = -174 dBm/Hz. We make use of the relation P[dBm] = 10·log(P[W])+30. Depending on the actual implementation of the receiver, a wider bandwidth and therefore higher thermal noise level might result. However, noise power density is not affected by actual bandwidth.

$$P_{Noise} = k \cdot T_S \cdot \Delta f \quad with \quad k = 1.380 \cdot 10^{-23} \frac{Ws}{K}$$

Equation 1-2

Sometimes in literature, a lower noise temperature (e.g. 130 K) of the antenna is assumed. But the influence of antenna temperature on noise power density is only marginal (thermal noise density of 175 K antenna: -176 dBm/Hz). The optimum antenna would only “look” at the “cold” sky with a noise temperature of 4 K compared to the “warm” earth with a temperature of 290 K. In that optimum case we would find the cosmic radiation background noise density of –192.6 dBm/Hz. For simplicity reasons we will continue our discussions based on a thermal noise density of N_0 = -175 dBm/Hz. We will later see that the omni directional characteristic of the GPS antenna will always result in a strong influence of the “warm” earth on antenna noise temperature. Only narrow focused parabolic antennas can get close to cosmic noise temperatures on earth.

Figure 1-2 shows the physical lower limits of the noise temperature in free space antenna environment. One can see that the operating frequency of GPS was selected close to the minimum of cosmic background noise.

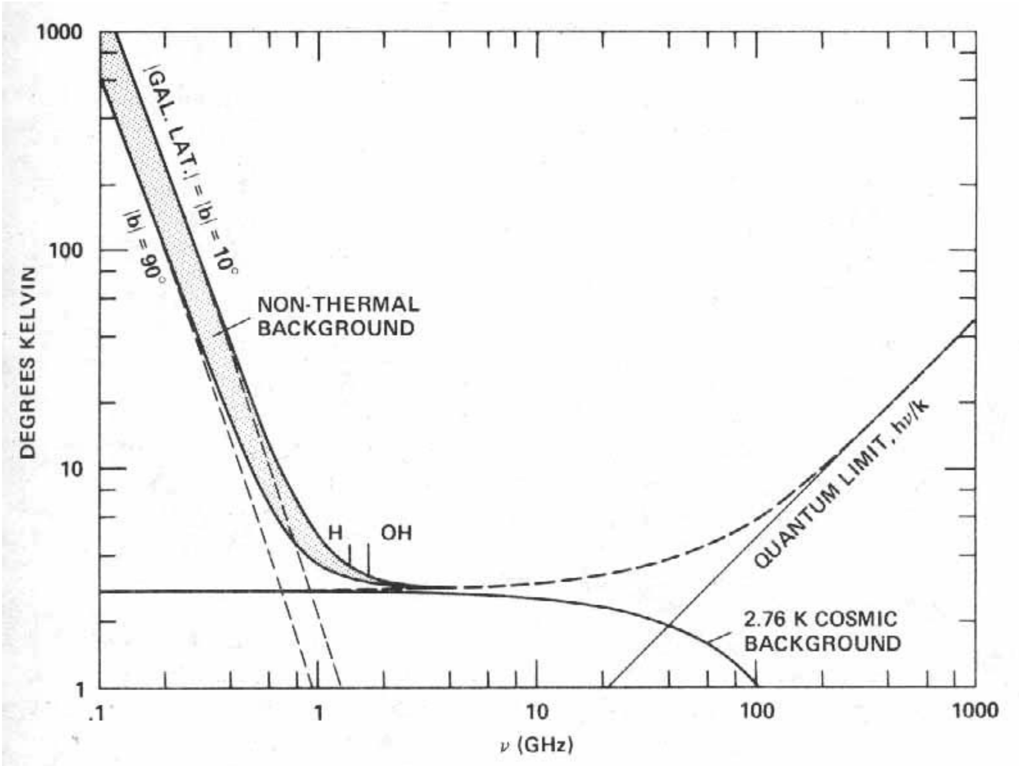


Figure 1-2: Free space noise temperature limits (NASA Document SP-419, http://history.nasa.gov/**)**

1.1.3 GPS Signal Levels on L_1

By GPS system specifications for an earth-bound receiver, the minimum and maximum signal levels for a 0 dBic antenna with right hand circular polarization are defined as -133 dBm to -125 dBm for P_p and –130 dBm to -123 dBm for P_c. The 0 dBic ideal antenna is defined by an antenna aperture of $A = \lambda^2/4\pi \approx 2.9 \cdot 10^{-3}\,m^2$. The term dBic refers to the ideal **i**sotropic **c**ircular polarized radiator, meaning an ideal antenna with omnidirectional directivity characteristic and circular polarization. For a pseudo noise signal with a spectrum of $[si(\pi\, f/f_c)]^2$ the maximum power spectral density is P_s/f_c where f_c is the code rate. For P_C = -130 dBm this translates into a maximum power spectral density of -190.1 dBm/Hz, roughly 15 dB below the thermal noise floor N_0. These low signal levels were defined according to ITU specifications in order to avoid interference with terrestric telecom links.

An important figure of merit for spread spectrum signal quality is the Carrier to Noise Density Ratio C/N_0. Basically, it gives a measure of the signal-to-noise ratio if the signal was completely de-spread. For C/A code and a 0 dBic antenna the maximum number is $P_{c,max}/N_0$ = 50 dBHz. Worst case for a 0 dBic antenna results in 45 dBHz. Typical GPS base band processors require at least a C/N_0 of 30 dBHz for signal tracking and C/N_0 in the order of 40 dBHz for fast acquisition. It's important to notice that carrier to noise density ratio can be completely recovered with an ideal matched receiver. So, the spread-/de-spread procedure is essentially loss-less.

If one wants to compare the numbers given above with the actual C/N_0 values reported by the GPS receiver on the desk, a few things need to be reminded. First, for real GPS antennas - especially miniature versions for mobile phone integration - a gain of –5 dBic is realistic. Very well engineered patch antennas with large ground plane can achieve +5 dBi for satellites at high elevations. Second, it seems that the fresh GPS satellites that were recently put into orbit emit slightly higher signal levels compared to the spec. And, finally, it is completely implementation dependent how an actual receiver calculates C/N_0. In some implementations, even the receivers internal loss is compensated. So, the receiver reports the C/N_0 at the antenna rather than the C/N_0 that the navigation software can make use of. Furthermore, significant signal attenuation in the order of 10 dB can be added by foliage or other obstructions in the signal path. Dense snowfall has also been reported to have some impact, at least a layer of snow on top of the antenna has.

1.2 Noise Figure Theory

The noise figure NF of a radio receiver is a measure that indicates to what extend the signal to noise ratio of an incoming signal is decreased by the additional noise of the receiver. E.g. a noise figure of 5 dB says that a signal to noise ratio of 45 dB at the input is degraded to 40 dB inside the receiver. The noise figure of a receiver is dominantly defined by the first amplification stage as can be seen from Equation 1-3 where the total noise figure of a chain of amplifiers is calculated. Here, NF_1 and G_1 are noise figure and gain of the first amplifier stage, NF_2 and G_2 are noise figure and gain of the second amplifier stage and so forth. Both measures are to be provided on a linear scale. Signal path attenuation adds directly to the noise figure. A 1 dB passive loss is represented as –1 dB gain and a 1 dB noise figure. Concluding, the noise figure of the receiver must be as low as even possible, for GPS receivers typically in the 2-4 dB range. Furthermore, the first stage, typically a low noise amplifier, should have high gain and very low noise figure.

$$NF_{Total} = NF_1 + \frac{NF_2 - 1}{G_1} + \frac{NF_3 - 1}{G_1 \cdot G_2} + \frac{NF_4 - 1}{G_1 \cdot G_2 \cdot G_3} + \ldots$$

Equation 1-3

A different approach to the receiver chain calculation is based on the concept of noise temperature. Here, a noisy building block is virtually replaced by a noiseless gain block fed by a resistor at a temperature that generates the noise power seen at the output of gain block. Noise temperature is always calculated with respect to a "system temperature" usually T_0 = 290 K. In this model, the noiseless amplifier produces an output noise power equal to the output noise of the original amplifier, if the source is now heated to a temperature $T_0 + T_e$. T_e is often referred to as "excessive noise temperature". The increase in output noise power $P_{Noise,additional}$ due to the hotter source is precisely equal to the noise added by the original (noisy) amplifier:

$$P_{Noise,additional} = k \cdot T_e \cdot \Delta f \cdot G$$

Equation 1-4

Noise temperature is used most often in satellite communications systems for several reasons. One is that objects in the sky generally don't have an effective temperature anywhere near 290 K, so choosing such a reference temperature has a weaker physical justification. The other is that space communication systems generally have exceptionally low noise figures, and noise temperature is a higher resolution measure of very low noise figure values.

The relation between noise figure and excessive noise temperature T_e is:

$$T_e = T_0 \cdot (NF - 1) \quad or \quad NF = 1 + \frac{T_e}{T_0}$$

Equation 1-5

For a multi-stage chain of signal processing stages, Equation 1-3 now translates into Equation 1-6.

$$T_{e,Total} = T_{e1} + \frac{T_{e2}}{G_1} + \frac{T_{e3}}{G_1 \cdot G_2} + \ldots$$

Equation 1-6

If one wants to calculate the noise power density at a given point in the processing chain the cascaded noise temperature is of interest:

$$P_{Noise,Total} = k \cdot \Delta f \cdot \left(T_S + T_{e,Total}\right) \cdot G_{Total} \quad with \quad G_{Total} = G_1 \cdot G_2 \cdot \ldots$$

Equation 1-7

Replacing $T_{e,Total}$ in Equation 1-7 with the expression from Equation 1-6 yields a recursive formula:

$$P_{Noise,Total} = k \cdot \Delta f \cdot \left(\left(\left(\left(T_S + T_{e1}\right) \cdot G_1 + T_{e2}\right) \cdot G_2 + T_{e3}\right) \cdot G_3 + \ldots\right)\ldots$$

Equation 1-8

The cascaded noise temperature T_{Cn} after stage n and at the same time the noise power $P_{Noise,n}$ can now recursively determined from the cascaded noise temperature after stage n-1:

$$P_{Noise,n} = k \cdot \Delta f \cdot T_{Cn} \quad with \quad T_{Cn} = \left(T_{Cn-1} + T_{en}\right) \cdot G_n \quad and \quad T_{C0} = T_S$$

Equation 1-9

This formulation is sometime preferred over Equation 1-3 because it allows easier calculation of cascades with only the knowledge of the cascaded noise temperature of the preceding stages.

1.2.1 Example of Receiver Signal to Noise Performance Calculation

Figure 1-3 shows the typical setup of a GPS receiver system. When looking at Equation 1-3, it is apparently clear that the noise figure of the LNA, i.e. F_1 will dominate the noise performance of the receiver. If it is not in place, the losses between antenna and receiver input should be as low as possible. Now, these losses are represented by G_1 and F_1 in Equation 1-3 (G_1 < 1!). If the cable losses are reasonably low, the first amplifier stage of the GPS receiver dominates the noise performance of the system. But even the best receiver cannot compensate the cable losses, in best case it will only add no further losses.

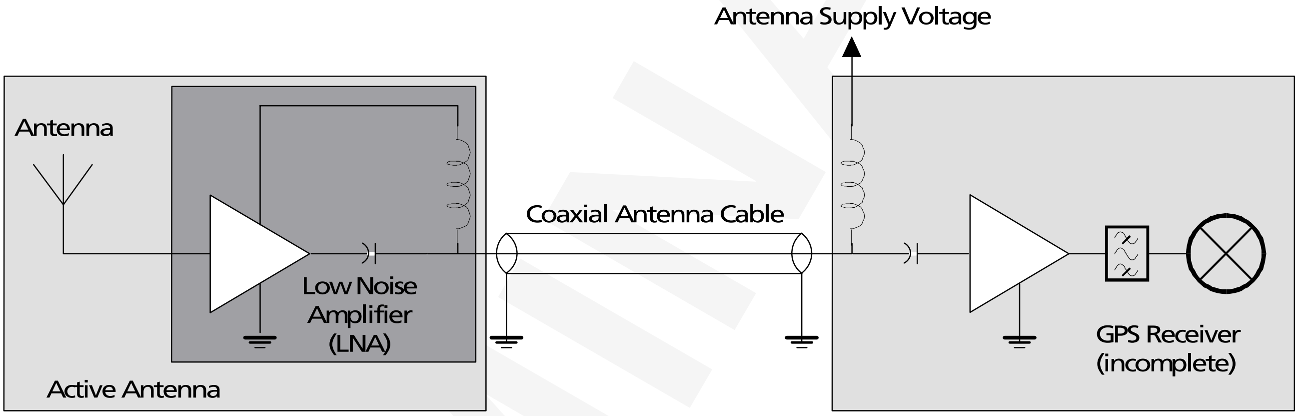


Figure 1-3: GPS receiver setup with antenna

Table 1-1 shows an example calculation for a typical GPS receiver processing chain as depicted in Figure 1-3. The final stage of the receiver is just summarized in the row entitled "Receiver". Gain and noise figure are arbitrarily chosen since this figure will not include the transition to the digital signal processing domain. To the user of a GPS RF chip the internal analog noise figure of the receiver is usually not visible, because he can only access the digitized output signals of this stage. If quantizer architecture and gain control mechanisms are known, one can however reverse calculate the noise figure of the analog part. Finally, in digital signal processing there are noise contributions which do not fit into the concept of noise figure theory. To just name one, quantization errors will always contribute a fixed amount of noise, regardless of the signal pre-amplification.

Stage	Power Gain of Stage	Cascaded Power Gain	Noise Figure of Stage	Cascaded Noise Figure	Carrier Power	Noise Power Density	C/No
	[dB]	[dB]	[dB]	[dB]	[dBm]	[dBm/Hz]	[dBHz]
Antenna				0.00	-130	-173.98	**43.98**
Cable	-1	-1	1	1.00	-131	-173.98	**42.98**
1st LNA	10	9	2	3.00	-121	-161.98	**40.98**
Cable	-3	6	3	3.26	-124	-164.71	**40.71**
2nd LNA	10	16	2	3.56	-114	-154.42	**40.42**
Receiver	60	76	10	3.97	-54	-94.01	**40.01**

Table 1-1: Example of Receiver Gain and Noise calculation

The highlighted fields in Table 1-1 are the input values, the other fields show the results calculated from these numbers. For a higher gain antenna, one could increase the carrier power in the first line by the antenna gain given in dBic. One could also make a different assumption for the antenna noise temperature in the first line and change the initial noise power density accordingly. Of course, gain and noise figure of the analog signal processing stage differ for every receiver implementation.

Figure 1-4 illustrates the direct relation between receiver noise figure and the C/N_0 measure that is available to the receiver. Again, one can easily see the dominant influence of the first processing stages. If the first LNA would have a higher gain, e.g. 25 dB instead of only 10 dB, the contribution of the subsequent stages to the sensitivity degradation would be negligible.

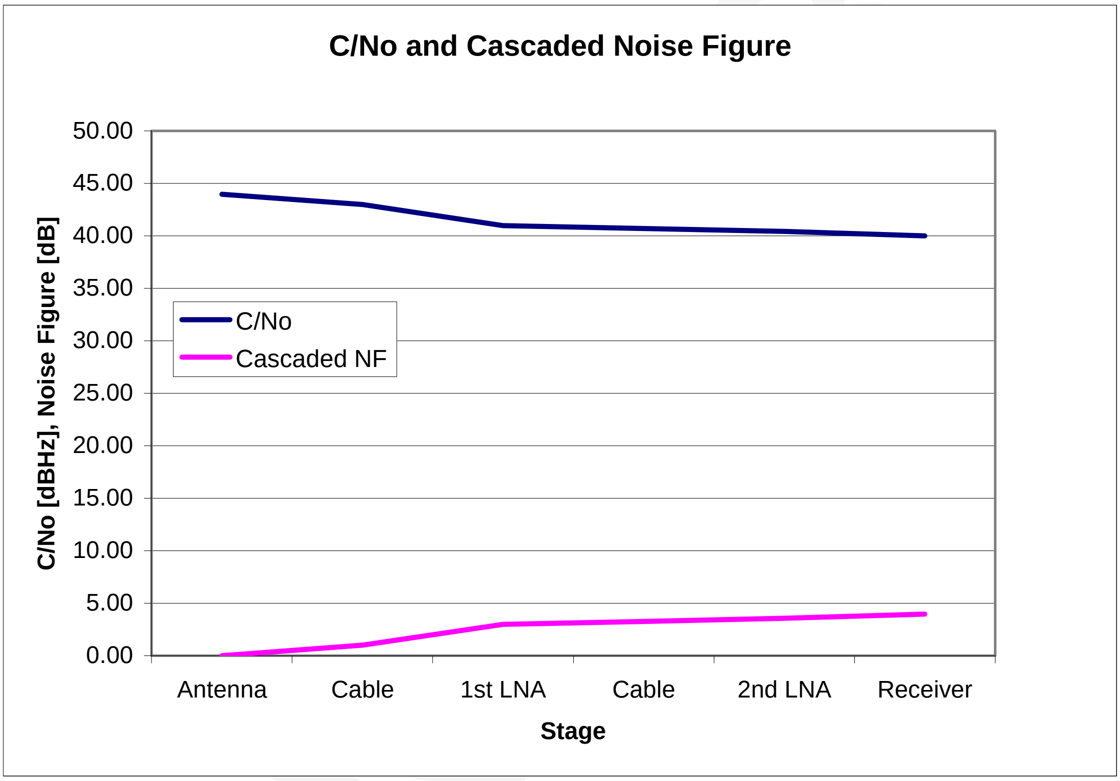


Figure 1-4: C/No and noise figure in receiver processing chain

1.3 Impedance Matching

In RF design it is common practice to refer all impedance specifications to a standard impedance of 50 Ohms. It is out of the scope of this application note to get into the details of S-parameter discussion. However, the concept of impedance matching is critical for understanding of possible implementation losses.

If we have a source, i.e. a generator with signal voltage U_S and a source impedance R_S, we can calculate the power P_L that is delivered to a load impedance R_L as follows:

$$P_L = U_L \cdot I_L = \frac{U_L^2}{R_L} = \frac{U_S^2 \cdot \left(\frac{R_L}{R_L + R_S}\right)^2}{R_L} = U_S^2 \cdot \frac{R_L}{(R_L + R_S)^2}$$

Equation 1-10

It is now straight forward to show that

$$\frac{\partial P_L}{\partial R_L} = U_S^2 \cdot \frac{(R_L + R_S)^2 - R_L \cdot (2R_L + 2R_S)}{(R_L + R_S)^4} = U_S^2 \cdot \frac{R_S^2 - R_L^2}{(R_L + R_S)^4}$$

Equation 1-11

We follow that for $R_S = R_L$ we will get the maximum power transferred form the source power into the load impedance.

There are now many measure to express the quality of the matching between two impedances, Z being the load impedance, e.g. the antenna and Z_0 being the reference impedance, e.g. the coaxial 50 Ohms cable. Let's start with the reflection factor ρ which is defined as

$$\rho = \frac{Z - Z_0}{Z + Z_0} \quad or \quad Z = Z_0 \cdot \frac{1+\rho}{1-\rho}$$

Equation 1-12

For passive impedances (|Z|>0) the magnitude of ρ can never exceed 1. The magnitude of ρ is often expressed in dB and then called return loss R = 20·log(|ρ|). ρ is also equal to what is known as S-parameter S_{11}. Yet another common representation is the voltage standing wave ratio VSWR which is given by

$$VSWR = \frac{1+|\rho|}{1-|\rho|} \quad or \quad |\rho| = \frac{VSWR-1}{VSWR+1}$$

Equation 1-13

The relative signal power loss due to mismatch calculated by $L = 1-|\rho|^2$ or in decibel: $L[dB]=10\cdot\log(1-|\rho|^2)$. Table 1-2 contains some values for reference for Z_0 = 50 Ohms. One should keep in mind that a loss in front of the first amplifier stage directly adds to the receiver noise figure. E.g. if the return loss of the RF input is as poor as –6 dB, about 1.3 dB will be added to the receiver noise figure.

Z [Ohms]	Rho	R [dB]	VSWR	L [dB]
1	-0.96	-0.35	50	-11.14
5	-0.82	-1.74	10	-4.81
6.25	-0.78	-2.18	8	-4.03
12.5	-0.60	-4.44	4	-1.94
16.614	-0.50	-6.00	3.0095	-1.26
25	-0.33	-9.54	2	-0.51
29.924	-0.25	-12.00	1.6709	-0.28
38.818	-0.13	-18.00	1.2881	-0.07
50	0.00	- infinity	1	0.00
64.402	0.13	-18.00	1.288	-0.07
83.545	0.25	-12.00	1.6709	-0.28
100	0.33	-9.54	2	-0.51
150.476	0.50	-6.00	3.0095	-1.26
200	0.60	-4.44	4	-1.94
400	0.78	-2.18	8	-4.03
500	0.82	-1.74	10	-4.81
2500	0.96	-0.35	50	-11.14

Table 1-2: Impedance Matching to 50 Ohms, some examples

1.4 The dB

RF engineers like to express relations on a logarithmic scale, the so-called decibel scale. Maybe this is because the number of transistors on a RF chip is so small that you can only compare it to the number on a digital chip on a logarithmic scale. Confusingly, there are two different conversion equations between dB and linear scale. This depends on whether we are talking about voltage or power ratios. For voltage ratios we find:

$$\frac{V_1}{V_2}[dB] = 20 \cdot \log\left(\frac{V_1}{V_2}[lin]\right) \quad or \quad \frac{V_1}{V_2}[lin] = 10^{\frac{V_1}{V_2}[dB] \cdot \frac{1}{20}}$$

Equation 1-14

For power ratios the following equation is used:

$$\frac{P_1}{P_2}[dB] = 10 \cdot \log\left(\frac{P_1}{P_2}[lin]\right) \quad or \quad \frac{P_1}{P_2}[lin] = 10^{\frac{P_1}{P_2}[dB] \cdot \frac{1}{10}}$$

Equation 1-15

If power and voltage ratios are measured at the same resistance R, we find that power ratio increases with the square of the voltage ratio, i.e. $\Delta P = \Delta V^2 / R$. This means that, when expressed in dB, we will find the same absolute number for the ratios on this particular resistor, regardless whether we are talking about voltage or power ratio.

There are a couple of standard "dB's" used which refer to absolute measures, e.g. to 1 mW or to 1 µV. The following table lists some of the most commonly used.

Abbreviation	Equation	Notes
dBm	10·log(P/1 mW)	dB with respect to 1 mW, equals dBW – 30.
dBW	10·log(P/1 W)	dB with respect to 1 W, equals dBm + 30.
dBuV	20·log(V/1 µV)	dB with respect to 1 µV, equals dBV + 120.
dBV	20·log(V/1 V)	dB with respect to 1 V, equals dBV - 120.
dBi		Antenna gain with respect to an ideal isotropic radiator[1]
dBic		Antenna gain with respect to an ideal isotropic circular polarized radiator
dBd		Antenna gain with respect to an ideal linear polarized half-wavelength dipole

[1] Isotropic radiator. A hypothetical antenna having equal radiation intensity in all directions. Note: An isotropic radiator represents a convenient reference for expressing the directive properties of actual antennas.

1.5 Multi-path

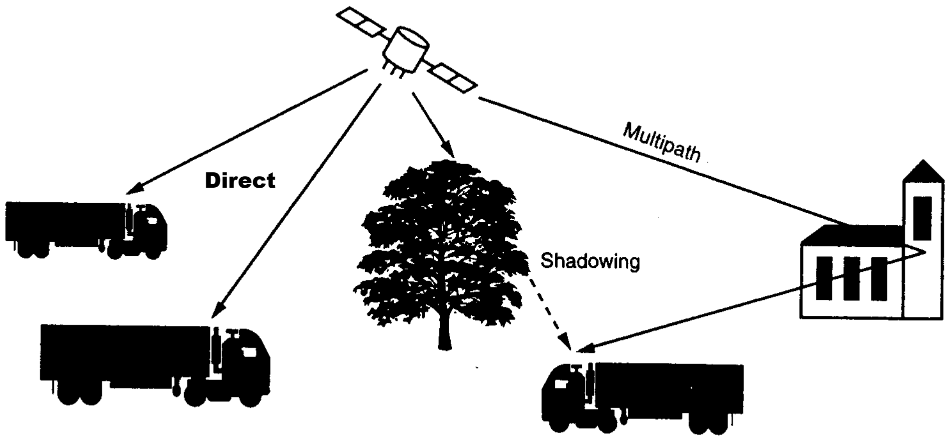


Figure 1-5: A multi-path environment

A multi-path environment exists if GPS signals are not only arriving at the antenna on the direct way (line of sight, LOS) but are also reflected off conductive surfaces, e.g. water or building walls. If there is still a direct path in addition to the reflected path available, the receiver can usually detect the situation and compensate to some extend. If there is a no line of sight (NLOS) situation, the receiver is not able to detect the situation and the range measurement to that particular satellite will present wrong information to the navigation solution, resulting in less accurate position. If there are only few satellites in sight, the navigation solution might be wrong by several 100 m.

If there is a LOS available, the effect of multi-path is actually twofold. First, the correlation peak will be distorted which results in a less precise position. This effect can be compensated by advanced receiver technology. But second, depending on the carrier phase relation of the direct and reflected signal, the received signal strength is subject to an interference effect. The two signals might cancel out each other (out of phase) or add onto each other (in phase). Even if the receiver stands still, the motion of the satellite will change the phase relation between direct and reflected signal constantly, resulting in a periodic modulation of the C/N_0 measured by the receiver. The receiver cannot compensate this second effect, because the signal already cancels out at the antenna. But, since the reflected signal is usually much weaker compared to the direct signal, the two signals will not cancel out completely. The reflected signal will also have an inverted polarity (left hand circular instead of right hand circular), which will reduce its signal level even further if the antenna has good polarization selectivity.

A particularly good reflector is water. So all sea borne applications require special attention to reflected signals arriving at the antenna from its backside, i.e. the water surface. Also, location of the antenna close to vertical metal clad walls can be very harmful since metal is an almost perfect reflector. When mounting an antenna on top of a reflective surface, the antenna should be mounted as close to the surface as possible. Then, the reflective surface will act as an extension of the antennas ground plane and not as a source of multi-path.

Because the periodicity of the modulation of C/N_0 is easily visible in severe multi-path environments, also the multi-path situation itself can be easily detected by the user.

1.6 RF Considerations on GPS Receiver Performance

If we look at it from an RF input side and summarize the findings of the previous sections, the following parameters can significantly degrade performance of a GPS receiver:

- Poor gain of the GPS antenna
- Poor directivity (radiation pattern) of the GPS antenna
- Improper orientation of the antenna to the sky
- Signal path obstruction by buildings, foliage, covers, or snow, etc.
- Poor noise performance of the receivers input stage or the antenna amplifier
- Multi-path effects

- Poor matching between antenna and cable impedance
- Jamming from external signals
- Jamming from signals generated by the receiver itself

The antenna related issues from above list will be further discussed in Chapter 2. The effect of signal path obstructions is self-explanatory. Jamming and interference issues will be extensively discussed in Chapter 4.

2 ANTENNAS

Since even the best receiver can never bring back what has been lost at the antenna, the attention paid to this part of a GPS system cannot be high enough.

2.1 Selecting the right Antenna

Several different antenna designs are available on the market for GPS applications. The GPS signal is right-hand circular polarized (RHCP). This results in a style of antenna that is different from the well-known whip antennas used for linear polarized signals. The most prominent antenna designs for GPS are patch antennas as shown in Figure 2-1.

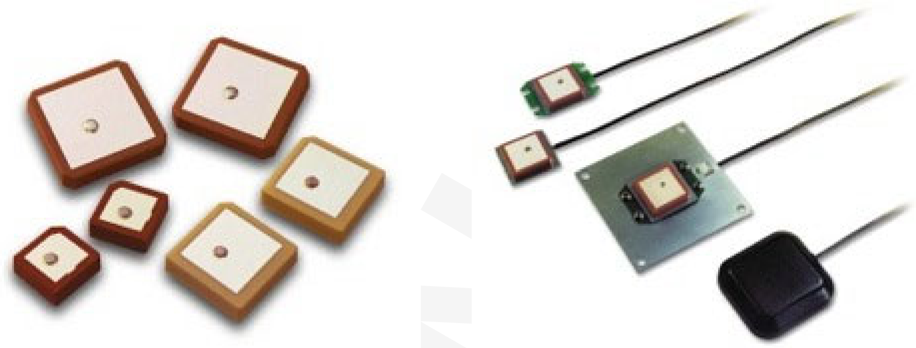


Figure 2-1: Patch Antennas, EMTAC Technology Corp.

Another style is the quadrifilar helix antenna shown in Figure 2-2. The actual geometric size of both antenna designs depends on the dielectric that fills the space between the active parts of the antenna. If the antenna is only loaded with air it will be comparatively large, high dielectric constant ceramics result in a very small form factor. The smaller the dimensions of the antenna are, the more performance critical are tight manufacturing tolerances. Furthermore, a smaller antenna will show a smaller aperture that could collect the signal energy from sky resulting in a lower overall gain of the antenna. This is result of pure physics and there is no "magic" to get around this problem. Amplifying the signal after the antenna will not improve the signal to noise ratio, see section 1.1.

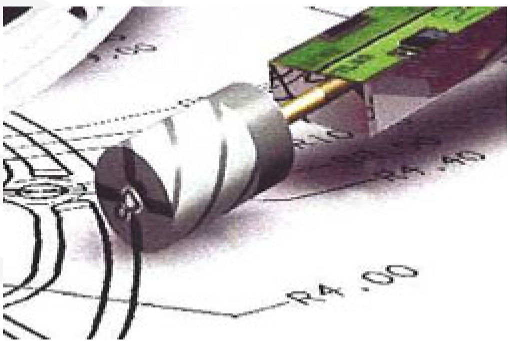


Figure 2-2: Quadrifilar Helix Antenna, Sarantel, Ltd.

In contrast to helix antennas, patch antennas require a ground plane for operation. Helix antennas can be designed for use with or without a ground plane.

For precision application like surveying or timing, some very high-end systems do exist. Common to these designs is large size, huge power consumption and high price; see Figure 2-3 for an example. These designs are mainly optimized to suppress multi-path signals reflected off the ground (choke ring antennas, multi-path limiting antennas, MLA). Another area of optimization is accurate determination of the phase center of the antenna. For precision GPS applications with position resolution in the millimeter range it is important that signals from satellites at all elevations virtually meet at exactly the same point inside the antenna. For this type of application often receivers with multiple antenna inputs are required.

Figure 2-3: Reference GPS Antenna optimized for WAAS augmentation and certified by the FAA, by Mircropulse, Inc.

At the low end of the spectrum of possible antenna solutions - if the user is willing to accept significant signal losses - a simple linear polarized whip or strip antenna will work. Compared to a circular polarized antenna, 3 dB of signal to noise ratio will get lost at least.

2.2 Active and passive antennas

Passive antennas contain only the radiating element, e.g. the ceramic patch or the helix structure. Sometimes they also contain a passive matching network to match the electrical connection to 50 Ohms.

Active antennas do have an integrated low-noise amplifier. This is beneficial in two respects. First, the losses of the cable do no longer affect the overall noise figure of the GPS receiver system. And second, even the receiver noise figure can be much higher without sacrificing performance. Therefore, some receivers will only work with active antennas. Active antennas need a power supply that will contribute to GPS system power consumption with some 10 to 20 mA, typically. Usually, the supply voltage is fed to the antenna through the coaxial RF cable. Inside the antenna, the DC component on the inner conductor will be separated from the RF signal and routed to the supply pin of the LNA (see Figure 1-3).

The use of an active antenna is always advisable if the RF-cable length between receiver and antenna exceeds about 10 cm. Care should be taken that the gain of the LNA inside the antenna does not lead to an overload condition at the receiver. For receivers that also work with passive antennas – like many u-blox products – an antenna LNA gain of 15 dB is usually sufficient, even for cable lengths up to 5 m. There's no need for the antenna LNA gain to exceed 25 dB for use with u-blox receivers. With short cables and an gain above 25 dB, an overload condition might occur on some receivers.

When comparing gain measures of active and passive antennas one has to keep in mind that the gain of an active antenna is composed of two components, the antenna gain of the passive radiator, given in dBic, and the LNA power gain given in dB. As we learned in section 1.2.1, low antenna gain cannot be compensated by high

LNA gain. If a manufacturers do only provide one total gain figure, this is not sufficient to judge the quality of the antenna. At least, one would need information on antenna gain (in dBic), amplifier gain, and amplifier noise figure.

2.3 Patch antennas

Patch antennas are ideal for application where the antenna sits on a flat surface, e.g. the roof of a car. Patch antennas can show a very high gain, especially if they are mounted on top of a large ground plane. Ceramic patch antennas are very popular because of the small size, typically measuring 25 x 25 mm^2 down to 12 x 12 mm^2. Very cheap construction might also use ordinary circuit board material like FR-4 or even air as dielectric, but this will result in a much larger size, typically in the order of some 10 x 10 cm^2. Figure 2-4 shows a typical example of the radiation pattern of a 16 x 16 mm^2 ceramic patch antenna. This measurement does only show the upper sphere of the radiation pattern. Depending on ground plane size there will also be a prominent back lobe present.

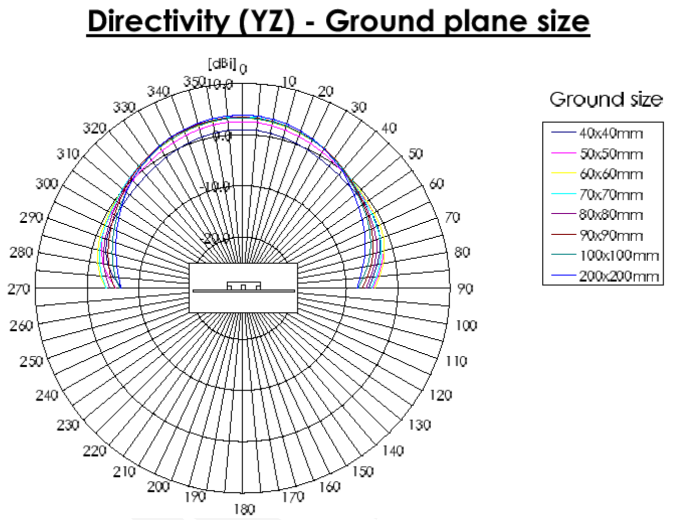


Figure 2-4: Typical Radiation Pattern of a Patch Antenna, MuRata, Inc.

One can easily see that the so-called axial ratio, i.e. the relation between maximum antenna gain at the zenith and gain at 90 degree can reach the order of 10 dB for large ground planes. Therefore, the correct dimensioning of the size of the ground plane is always a compromise between maximum gain at high elevations and reasonable gain even at low elevations.

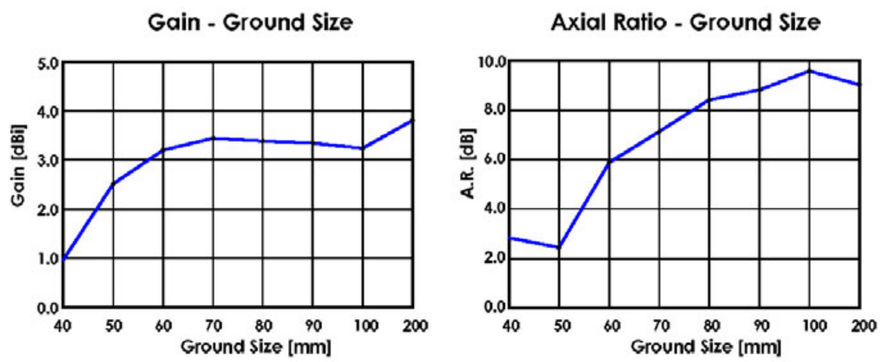


Figure 2-5: Typical Gain and Axial Ratio of a Patch antenna with respect to ground plane size, MuRata, Inc.

A good compromise for the ground plane size is typically in the area of 50 to 70 mm square. This number is largely independent of the size of the patch itself (as long as we are talking about ceramic patches). Patch antennas with small ground planes will also have a certain back-lobe in their radiation pattern, making them susceptible to radiation coming from the backside of the antenna, e.g. multi-path signals reflected off the ground. The larger the size of the ground plane, the less severe is this effect.

Smaller size patches will usually reach their maximum gain with a slightly smaller ground plane compared to a larger size patch. However, the maximum gain of a small sized patch with optimum ground plane might still be much lower than the gain of a large size patch on a ground plane which is smaller than the respective optimum size.

But, not only gain and axial ratio of the patch antenna are affected by the size of the ground plane but also the matching of the antenna to the 50 Ohms impedance of the receiver. See section 2.6 for more information on matching.

2.4 Helix antennas

Helix antenna can be designed for use with or without ground plane. For example, the radiating elements on board the GPS satellites do have a ground plane. Using an array of helix antennas, the GPS satellites can control the direction of the emitted beam. If a helix antenna is designed without ground plane it can be tuned such to show a more omni directional radiation pattern as shown in Figure 2-6.

Figure 2-6: Radiation pattern of helix antenna without ground plane, Sarantel, Ltd.

Although we also can determine an axial ratio close to 9 dB between zero degree and 90 degree elevation, which compares to the patch antenna, the back lobe of the helix generally degrades much smoother and does not show any sensitivity at the –180 degree direction. In contrast, back lobe of the patch antenna depends very much on size and shape of the ground plane. As with patch antennas, the size of helix antennas can be reduced by filling the antenna with a high dielectric constant material. Sizes in the order of 18 mm length and 10 mm diameter are being offered on the market. Again, antenna gain will decrease with decreasing size of the antenna.

2.5 Helix or Patch, which selection is best?

For practical applications the possibilities of integrating a certain style of antenna into the actual device is of primary concern. Some designs naturally prefer the patch type of antenna, e.g. for roof-top applications. Others prefer the pole like style of the helix antenna which is quite similar to the style of mobile phone antennas. Furthermore, it is important that the antennas main lobe points to the sky in order to receive most of the satellites with the maximum antenna gain. If the application is a hand held device, the antenna should be designed in a way that natural user operation results in optimum antenna orientation. The helix antenna seems to be more appropriate in this respect.

However, one has to keep in mind that comparable antenna gain requires comparable size of the antenna aperture which will lead to a larger volume filled by a helix antenna in comparison to a patch antenna. Helix antennas with a "reasonable" size will therefore quite often show a lower sensitivity compared to a "reasonably" sized patch antenna.

A helix antenna might result in a "more satellites on the screen" situation in difficult signal environments when directly compared with a patch antenna. This is due to the fact that the helix will more easily pick up reflected signals through its omni directional radiation pattern. However, the practical use of these signals is very limited because of the uncertain path of the reflected signals. So, the receivers sees more satellites but the navigation solution will be degraded because of distorted range measurements in a multi-path environment. Depending on the actual receiver performance under multi-path conditions it might be either preferable to present even distorted signals to the navigation engine, or to better hide them from the position solution.

It's always a good idea to test the actual performance of different antenna types in a real use environment before even starting the mechanical design of the GPS enabled product.

2.6 Antenna Matching

All common GPS antennas are designed for a 50 Ohms electrical load. Therefore, one should select a 50 Ohms cable to connect the antenna to the receiver. However, there are several circumstances under which the matching impedance of the antenna might shift considerably. Expressed in other words, this means that the antenna does no longer represent a 50 Ohms source impedance. What typically happens is that the center frequency of the antenna is shifted away from GPS frequency - usually towards lower frequencies – by some external influence. The reason for this effect are primarily disturbing objects in the near field of the antenna. This can either be a ground plane which does not have the same size as the antenna was designed for, or it can be an enclosure with a different dielectric constant than air.

In order to analyze effects like this one would normally employ electrical field simulations which will result in exact representation of the electric fields in the near field of the antenna. Furthermore, these distortions of the near field will also show their effect in the far field, changing the radiation pattern of the antenna. Unfortunately, there is no simple formula to calculate the frequency shift of a given antenna in a specified environment. So one has to go either for extensive simulation or do some experimental work. Luckily, antenna manufacturers typically offer a selection of pre-tuned antennas, so the user can test and select the version that fits his environment best. However, at least a scalar network analyzer is needed to verify the matching.

Again, it must be pointed out that the smaller the size of the antenna, the more sensitive it will be to distortions in the near field. Even worse, the antenna bandwidth will decrease with decreasing size, making it even harder to hit the optimum tuning.

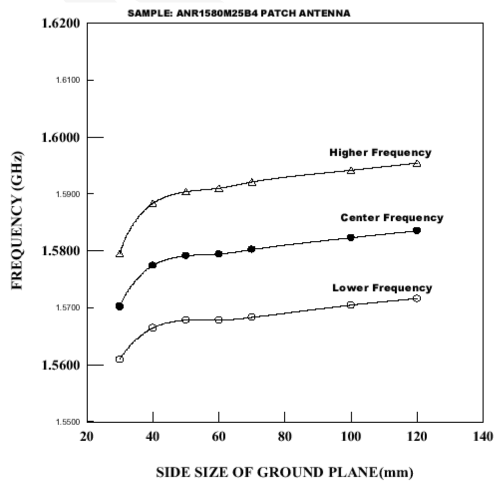


Figure 2-7: Dependency of center frequency on ground plane dimension for a 25 x 25 mm^2 patch, EMTAC

Once again, a LNA placed very close to the antenna can help in relaxing the matching requirements. If the interconnect length between antenna and LNA is much shorter than the wavelength (9.5 cm on FR-4), the matching losses become less important. It is now more important how the noise figure of the LNA depends on its input match. Within a reasonable mismatch range, integrated LNAs might show a decrease in gain in the order of dBs but only an increase of noise figure in the order of some tenths of a dB. So, if your application requires a very small antenna, a LNA can help to match the hard to control impedance of the antenna to a 50 Ohms cable. This effect is even beneficial if the cable length between antenna and receiver is only short. In this case, there's no need for the gain of the LNA to exceed 10-15 dB. Its sole purpose is impedance matching and not signal amplification.

3 CABLING

3.1 Coaxial RF-Cables

If antenna and receiver are not to be placed on the same PC board, a cable is needed for connection. Only coaxial cables with a 50 Ohms impedance should be used.

Figure 3-1: RF coaxial cables, Huber + Suhner AG

There is a nearly unlimited number of cables for RF connections. Normally, they are listed according to the Radio Guide scheme (RG). Additionally, there's another numbering scheme according to MIL-C-17 standard. Table 3-1 lists a selection of the most popular cables.

Type	**Cable Diameter [mm]**	**Loss @ 1.6 GHz [dB/10m]**	**Remarks**
RG 58 C/U M17/155-00001 M17/28-RG58	4.9	7	Polyethylen
RG 141 /U	4.4	6	Teflon®
RG 142 B/U M17/158-00001 M17/60-RG142	4.95	6	Teflon®
RG 165 /U M17/159-00001 M17/65-RG165	10.4	3.5	Teflon®
RG 174 A/U M17/173-00001 M17/119-RG174	2.8	13	Polyethylen
RG 178 B/U M17/169-00001 M17/93-RG178	1.8	20	Teflon®
RG 188 A/U	2.6	12	Teflon®
RG 196 A/U	1.95	20	Teflon®
RG 213 /U	10.3	3.5	Polyethylen

Type	Cable Diameter [mm]	Loss @ 1.6 GHz [dB/10m]	Remarks
M17/163-00001 M17/74-RG213			
RG214 /U M17/164-00001 M17/75-RG214	10.8	3.5	Polyethylen
RG 217 /U M17/165-00001 M17/78-RG217	11.2	2.5	Polyethylen
RG 223 /U M17/167-00001 M17/84-RG223	5.4	6.5	Polyethylen
RG 225 /U M17/86-00001	10.9	3.5	Teflon®
RG 303 /U M17/170-00001 M17/111-RG303	4.3	6	Teflon®
RG 316 /U M17/172-00001 M17/113-RG316	2.5	12	Teflon®
RG 393 /U M17/174-00001 M17/127-RG393	9.9	3.5	Teflon®
RG 400 /U M17/175-00001 M17/128-RG400	4.95	6.5	Teflon®
RG 401 /U M17/129-RG401 H+S EZ250/M17 or SM250	6.3	3.0	Semi-Rigid, Teflon®
RG 402 /U M17/130-RG402 H+S EZ141/M17 or SM141	3.6	4.8	Semi-Rigid, Teflon®
RG 405 /U M17/133-RG405 H+S EZ86/M17 or SM86	2.2	8.6	Semi-Rigid, Teflon®

Table 3-1: Comparison of standard 50 Ohms Coaxial Cable, Data according to Huber + Suhner

The simple conclusion from this table is that the larger the diameter of the cable, the lower the respective losses. Whenever possible one should use one of the low-loss cables. If cable losses of an actual connection reach the order of dB's, an active antenna is definitely required.

As long as there is sufficient gain in the LNA at the antenna side of the cable, there's virtually no limit in cable length. The LNA gain can be calculated from the anticipated losses in the cable. It's important to remember that the LNA will only help if it is placed on the antenna side of the cable, before the losses occur.

The receiver will always calculate the position and time of the antenna. If you need precise time at the receiver, the cable length has to be compensated. Some receivers offer user-definable settings to feed in the correct compensation values.

3.2 RF Interconnects on Printed Circuit Boards

There are many ways to design wave-guides on printed circuit boards. Common to all is that calculation of the electrical parameters is not straightforward. A free-ware tool like AppCAD from Agilent is of great help. It can be downloaded from www.agilent.com.

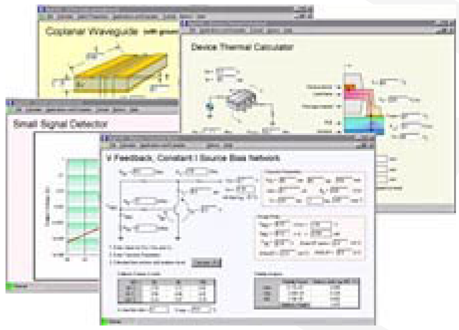


Figure 3-2: Screenshots from AppCAD, Agilent

The micro strip is the most common configuration for printed circuit boards. The basic configuration is shown in Figure 3-3. As a rule of thumb, for a FR-4 material the width of the conductor is roughly double the thickness of the dielectric for a 50 Ohms line impedance.

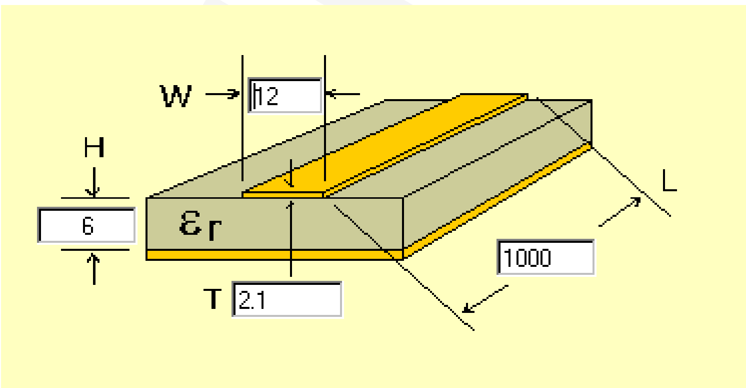


Figure 3-3: Microstrip, Agilent AppCAD

The micro strip is very easy to design, it is however susceptible to interference since the field also travels through the air above the conductor. Furthermore, neighboring conductors should be kept far away from the signal line to avoid cross talk. As a rule of thumb, the next conductor should be several times the dielectric thickness apart of the signal line. Ground traces between signal lines will reduce cross talk. Ground lines should be connected to the ground plane with as many vias as possible. They should also not be routed too close to the signal line to avoid reduction of the impedance.

Table 3-2 and Figure 3-4 show the almost linear relationship between conductor trace width W and dielectric thickness H for a 50 Ohms line. It can also be noted that the influence of different copper plating thickness T is negligible for common values. Most manufacturers specify the dielectric constant of the FR-4 PCB material with a dielectric constant somewhere in between 4.1 and 4.6. Make sure that the numbers provided by the PCB manufacturer are also valid in the 1.5 GHz range.

H [mm]	W $\varepsilon_r = 4.1$ T = 35 µm [mm]	W $\varepsilon_r = 4.1$ T = 18 µm [mm]	W $\varepsilon_r = 4.6$ T = 35 µm [mm]	W $\varepsilon_r = 4.6$ T = 18 µm [mm]
0.25	0.47	0.49	0.43	0.44
0.50	0.97	0.99	0.89	0.90
0.75	1.47	1.49	1.35	1.36
1.00	1.97	1.99	1.81	1.83
1.25	2.47	2.49	2.27	2.29
1.50	2.98	3.00	2.73	2.75
1.75	3.48	3.50	3.19	3.21
2.00	3.98	4.00	3.65	3.67

Table 3-2: Micro strip line widths for FR-4 material, copper plating: 35 µm and 18 µm, Agilent AppCAD

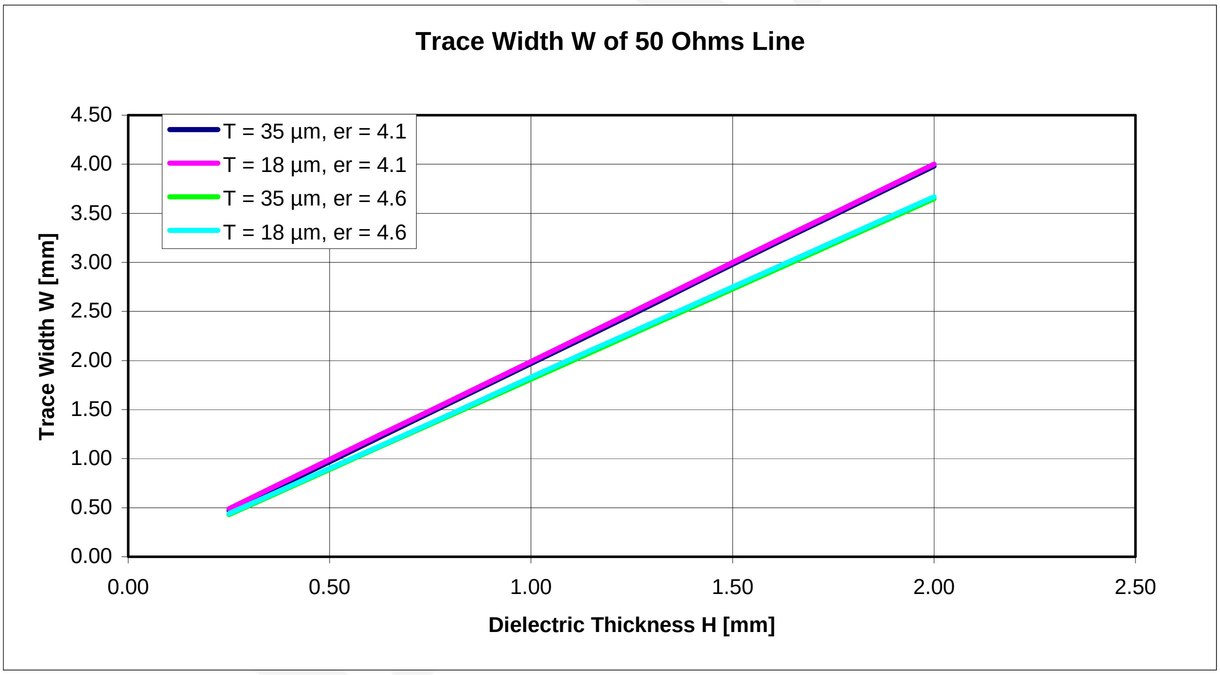


Figure 3-4: Dependency of Trace width of a 50 Ohms line on dielectric constant ε_r, and copper plating thickness T

Another quite common configuration results if the micro strip is buried between two shielding layers. The basic configuration is shown in Figure 3-5. It's important that the distance to the two ground planes from the inner conductor is equal, i.e. the conductor sits exactly in the middle of the two planes.

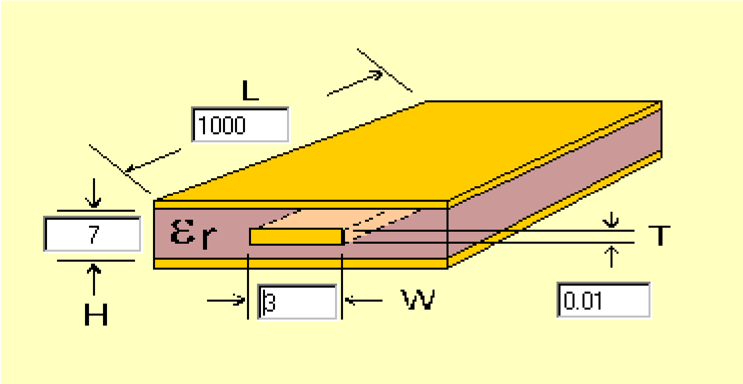


Figure 3-5: Strip line, Agilent AppCAD

This configuration provides better shielding compared to the micro strip. Its three layer configuration might be difficult to manufacture, however. Again, a ground trace with lots of vias can help in isolation of neighboring signal lines. Both ground planes should be connected with as many vias as possible, but not too close to the signal line.

3.3 RF-Connectors

There's a virtually unlimited number of RF connectors available on the market. In the field of GPS, one may come across one of the RF connector systems listed in Table 3-3. Very general spoken the design options are as with the cables: the larger the size of the connector, the better its RF performance and its reliability. Pre-configured antennas are often offered with a choice of connectors. Any of the connectors from Table 3-3 will work with GPS. The primary selection criteria will then be size, availability, reliability, and price.

When using a passive antenna without additional LNA, the losses of the connector system should be kept as low as possible, leading to the selection of one of the high-grade connectors (e.g. SMA). Of course, it would be even better to not use a connector at all and only use solder or crimp joints.

Furthermore, not all connectors will fit to all types of cables. Small size connectors will only fit onto thin cables with higher losses. Low loss cables with large diameter will require large size connectors (e.g. TNC or N). Connector manufacturers usually offer a number of versions of one connector style fitting precisely to a selection of RF-cables.

Tooling for the cable assembly is very important. An improper cable to connector assembly can result in large RF losses. All high quality RF connector systems require special tooling for assembly. Frequently, connector manufacturers also offer pre-configured cable assemblies according to customer specifications. This is the solution of choice if tooling is not available in house.

Connector Series	**Picture (not to scale)**	**Remarks**
N		N connectors are available with 50 and 75 impedance. The frequency range extends to 18 GHz, depending on the connector and cable type. The screw-type coupling mechanism provides a sturdy and reliable connection. N connectors are available for flexible cables, for semi-rigid cables and for corrugated copper tube cables. Size is typically too large for GPS applications. RF-performance is superior.

Connector Series	Picture (not to scale)	Remarks
BNC		BNC is the most popular RF connector series, featuring a two stud bayonet coupling mechanism, which is particularly useful for frequently coupled and uncoupled RF connections with frequencies up to 4 GHz. Size is quite large. RF-performance is fair.
TNC		TNC connectors are threaded RF connectors applicable from DC up to 11 GHz. Precision designs using a dielectric bead are suitable for use up to 18 GHz. The threaded coupling mechanism improves control over the interface dimensions and allows them to be used under a higher environmental load than BNC, especially under a high vibration load. Size is quite large. RF-performance is very good.
SMA		SMA connectors are precision connectors for microwave applications up to 18 GHz/ 26.5 GHz. They distinguish themselves through their high mechanical strength, high durability, high reliability and low VSWR. SMA launchers are the preferred connection element for varied microwave circuits including hermetically sealed designs. There is a huge variety of applications for SMA connectors, such as test + measurement, mobile communication, avionics, etc. Size is o.k. for most applications. RF-performance is superior.
PC 3.5		PC 3.5 connectors are precision connectors for use in microwave applications up to 33 GHz. They are especially suitable for use with semi-rigid cables and microwave components. They are intermateable with SMA, K and SK connectors. Due to an air dielectric interface and the more durable construction a superior repeatability further enhances the performance. Very expensive. Ultimate RF performance. Used for precision measurement applications.
SMB		SMB subminiature connectors are suitable for applications from DC up to 4 GHz. The SMB snap-on mechanism provides a fast and reliable connection for applications with high packing density. They are used in fixed and mobile communication equipment for internal wiring. In accordance with the international specifications, plugs are fitted with female center contacts and jacks with male center contacts. *The subminiature connectors Series SMB, SMC, SMS have the same basic design. They only differ with regard to their coupling mechanism.* Easy coupling mechanism. Fair RF-performance. Small size.

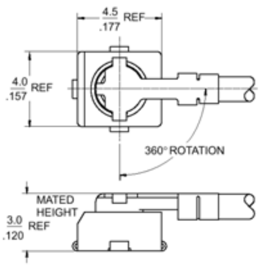
Connector Series	Picture (not to scale)	Remarks
SMC		The SMC subminiature connector series is based on the same design as the SMB and SMS series. But due to its screw-on coupling mechanism, the SUHNER SMC subminiature connectors are suitable for applications up to 10 GHz. This threaded coupling mechanism permits a vibration-proof connection suitable for semi-permanent connections and for use in mobile equipment with low VSWR requirements. Reliable coupling mechanism. Small size. Good RF-performance.
SMS		SMS connectors are used in applications requiring rapid connection and disconnection. The low engaging and disengaging forces due to the slide-on mechanism make this connectors specially suited for rack or panel systems. These connectors are suitable for applications up to 4 GHz. Use only for laboratory applications, no locking mechanism. Fair RF-performance.
MCX (OSX)		MCX connectors show an essential space reduction of 30% when compared with the series SMB. These MCX connectors with snap-on coupling mechanism offer you an excellent blend of size, weight, durability and performance for applications such as GPS, wireless or fixed communications systems and instruments. MCX connectors can be used from DC to 6 GHz and are tested in accordance with CECC 22220. Very popular for GPS. Good RF-performance. Small size.
MMCX		MMCX connectors are intended for use in applications where the smallest dimensions have to be achieved. MMCX connectors can be used in applications from DC to 6 GHz. The reliable "snap-on" coupling mechanism ensures that the electrical parameters are consistently reproduced. Due to its non-slotted outer contact, the MMCX series provides a low RF-leakage. Even smaller than MCX. Same good RF-performance.
SSMT		The SSMT Interconnect System consistently achieves broadband electrical performance through 6 GHz with a maximum VSWR of 1.20:1 at 2 GHz. The SSMT utilizes a common OSMT plug receptacle, which is designed for high volume assembly using surface mount technology and is available in tape and reel packaging for automatic pick and place board assembly. The mating cable jack is available terminated to a highly flexible micro-coax cable as either a pigtail, jumper or standard interseries connector assembly to meet various application needs. The SSMT Interconnect System can be manually mated, facilitating high volume assembly and eliminating the need for special engagement tooling. Interface durability is rated at 100 mating cycles. Used on GPS-MS1E and GPS-PS1E. Very small. Only available from Tyco, AMP-Division, formerly M/A-Com.

Table 3-3: RF Connectors, Huber + Suhner, Tyco Electronics

4 INTERFERENCE ISSUES

The low signal levels of GPS have already been discussed extensively. A typical GPS receiver has a very low dynamic range. This is because the antenna should only see thermal noise in GPS frequency band, given that the peak power of the GPS signal is 15 dB below thermal noise floor. And, this thermal noise floor is usually very constant over time. Most receiver architectures use an automatic gain control (AGC) circuitry to automatically adjust to the input levels presented by different antenna and pre-amplifier combinations. The control range of these AGCs can be as large as 50 dB. However, the dynamic range for a jamming signal exceeding the thermal noise floor is typically only 6 to 12dB, due to the one or two bit quantization schemes commonly used in GPS receivers. If there are jamming signals present at the antenna and the levels of these signals exceed the thermal noise power, the AGC will regulate on the jamming signal, suppressing the GPS signal buried in thermal noise even further. Depending on the filter characteristics of the antenna and the front end of the GPS receiver, the sensitivity to such in-band jamming signals decreases more or less rapidly if the frequency of the jamming signal moves away from GPS signal frequency. However, we can conclude that a jamming signal exceeding thermal noise floor within a reasonable bandwidth, e.g. 100 MHz, around GPS signal frequency will degrade the performance remarkably.

But, even out-of band signals might affect GPS receiver performance. If these jamming signals are strong enough that even antenna and front end filter attenuation are not sufficient, the AGC will still regulate on the jamming signal. Moreover, very high jamming signal levels can result in non-linear effects in the pre-amplifier stages of the receiver, resulting in desensitizing of the whole receiver. One such particular difficult scenario is the transmitting antenna of a DCS handset (max. 30 dBm at 1710 MHz) in close proximity to the GPS antenna. When integrating GPS with other RF transmitters special care is necessary.

If the particular application requires integration of the antenna with other digital systems, one should make sure that jamming signal levels are kept to an absolute minimum. Even harmonics of a CPU clock can reach as high as 1.5 GHz and still exceed thermal noise floor.

On the receiver side there's not much that can be done to relax the situation with reasonable effort. Of course, high price military receivers have integrated counter-measures against intentional jamming. But the methods employed are out of the scope of this note and might even conflict with export restrictions for dual-use goods.

This whole section contains a number of general recommendations and ideas. It is however totally dependent on the actual application if any of these concepts will apply.

In applications where an active antenna is used in a remote position, e.g. >1 m away from other electronics, interference should not be an issue.

If antenna and electronics are to be integrated tightly, the following sections should be read very carefully.

4.1 Sources of noise

Basically two sources of noise are responsible for most of the interference issues with GPS receivers:

1. Strong RF transmitters close to GPS frequency, e.g. PCS at 1710 MHz or radars at 1300 MHz.
2. Harmonics of the clock frequency emitted from digital circuitry.

The first problem can be very hard to solve, but if GPS and RF transmitter are to be integrated close to each other, there's also the engineer at hand who knows the specifications of the RF transmitter. In most cases, counter measures like filters will be required on the transmitter side to limit its spurious emissions below noise floor in the vicinity of GPS frequency.

Even if the transmitter is quiet in the GPS band, a very strong emission close to GPS band can cause saturation in the front-end of the receiver. Typically, the receiver's front-end stage will reach its compression point which will in turn increase the overall noise figure of the receiver. In that case, only special filtering between GPS antenna and receiver input will help to reduce signal levels to the region of linear operation of the front-end.

The second problem is more common but also proves to be hard to solve regularly. Here, the emitting source is not well specified and the emission can be of broadband nature, making specific countermeasures very difficult. Moreover, the GPS band is far beyond the 1 GHz limit that applies to almost all EMC regulations. So, even if a device is compliant with respect to EMC regulations it might disturb a GPS receiver severely.

If the GPS antenna is to be placed very close to some other electronics, e.g. the GPS receiver itself or a PDA-like appliance, the EMC issue has to be taken very seriously right from the concept phase of the design. It is one of the most demanding tasks in electrical engineering to design a system that is essentially free of measurable emissions in a certain frequency band.

4.2 Eliminating digital noise sources

Digital noise is caused by short rise-times of digital signals. Data and address buses with rise-times in the nanosecond range will emit harmonics up to several GHz. These sections contain some general hints on how to decrease the level of noise emitted from a digital circuit board that eventually sits close to the GPS receiver or the antenna.

4.2.1 Power and ground planes

Use solid planes for power and ground interconnect. This will typically result in a PCB with at least four layers but will also result in a much lower radiation. Solid ground planes ensure that there is a defined return path for the signals routed on the signal layer. This will reduce the "antenna" area of the radiating loop. Planes should be solid in a sense that there are no slots or large holes inside the plane.

The outer extent of the power plane should be within the extent of the ground plane. This avoids that the edges of the two planes form a slot antenna at the board edges. It's a good idea to have a ground frame on the circumference of every layer that is connected to the ground plane with as many vias as possible. If necessary, a shield can then be easily mounted on top of this frame (see Figure 4-2). Furthermore, free space on the outermost Layers can be filled with ground shapes connected to the ground plane to shield radiation from internal layers.

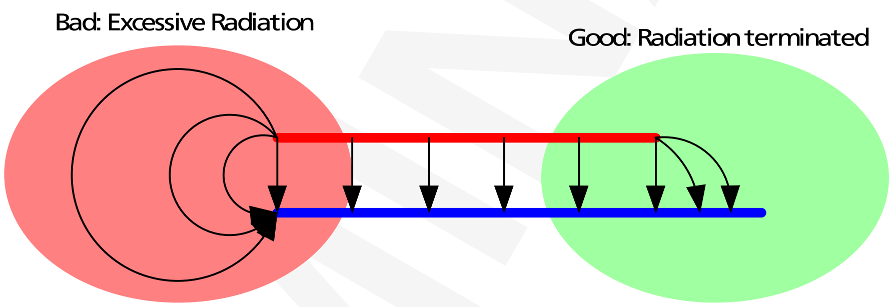


Figure 4-1: Signal and power plane extends should lie within ground plane extends

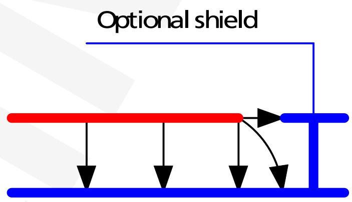


Figure 4-2: Further improvement of reduction of power plane radiation

4.2.2 High speed signal lines

Keep high-speed lines as short as possible. This will reduce the area of the noise-emitting antenna, i.e. the conductor traces. Furthermore, use of line drivers with controlled signal rise-time is suggested whenever it comes to driving large bus systems. Alternatively, high-speed signal lines can be terminated with resistors or even active terminations to reduce high frequency radiations originating from overshoot and ringing on these lines.

If dielectric layers are thick compared to the line width route ground traces between the signal lines to increase shielding. This is especially important if only two layer boards are used (see Figure 4-3).

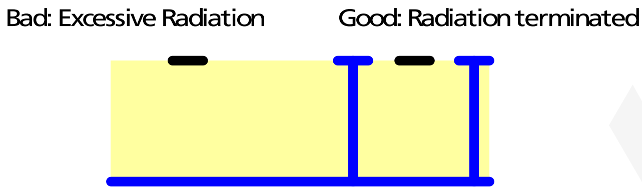


Figure 4-3: Terminating radiation of signal lines

4.2.3 Decoupling Capacitors

Use a sufficient number of decoupling capacitors in parallel between power and ground nets. Small size, small capacitance types reduce high-frequency emissions. Large size high capacitance types stabilize low frequency variations. It's preferred to have a large number of small value capacitors in parallel rather than having a small number of large value capacitors. Every capacitor has an internal inductance in series with the specified capacitance. Above resonance, the capacitor will behave like an inductor. If many capacitors are connected in parallel, total inductance will decrease while total capacitance will increase. Figure 4-4 shows the impedance dependency of SMD capacitors.

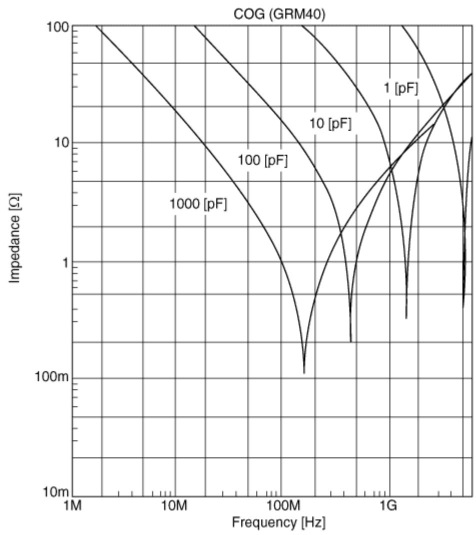


Figure 4-4: Impedance of 0805 size SMD capacitors vs. frequency, MuRata

If power and ground plane are not connected by an efficient capacitor network, the power plane may act as a radiating patch antenna with respect to ground.

Furthermore, ceramic capacitors come with different versions of dielectric material. These materials show different temperature behavior. For industrial temperature range applications, at least a X5R quality should be selected. Y5V or Z5U types may loose almost all of their capacitance at low temperatures, resulting in potential system failure at low temperatures because of excessive noise emissions from the digital part. Tantalum capacitors show good thermal stability, however, their high ESR (equivalent series resistance) limits the usable frequency range to some 100 kHz.

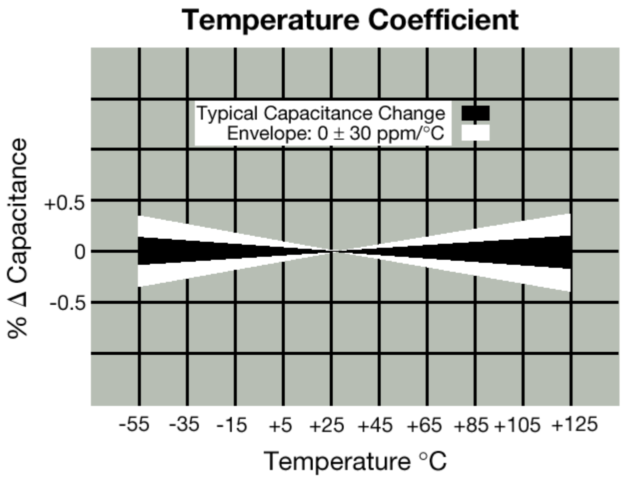


Figure 4-5: Temperature dependency of COG/NPO dielectric, AVX

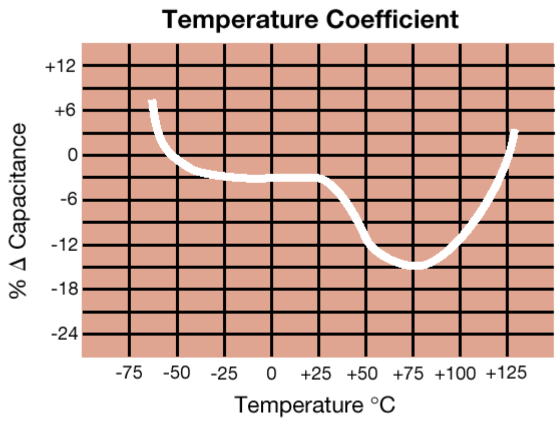


Figure 4-6: Temperature dependency of X7R dielectric, AVX

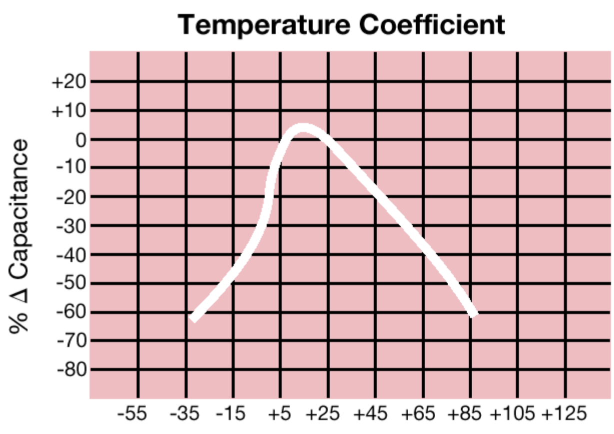


Figure 4-7: Temperature dependency of Y5V dielectric, AVX

4.3 Shielding

If EMI problems cannot be solved by employing the countermeasures listed in section 4.2, the ultimate solution will be shielding of the noise source. But, even a real-world shield is not perfect. The shielding effectiveness you can expect from a solid metal shield is somewhere in the order of 30-40 dB. If a thin PCB copper layer is used as a shield, these numbers might even be lower. Perforation of the shield will also lower its effectiveness.

Be aware of the negative effects that holes in the shield can have on shielding effectiveness. Lengthy slots might even turn a shield into a radiating slot antenna. Therefore, a proper shield has to be tightly closed and very well connected to the circuit board.

4.3.1 Feed through capacitors

The basic concept of shielding is that a metal box will terminate all electrical fields on its surface. In practice we have the problem that we need to route some signals from inside to outside of this box.

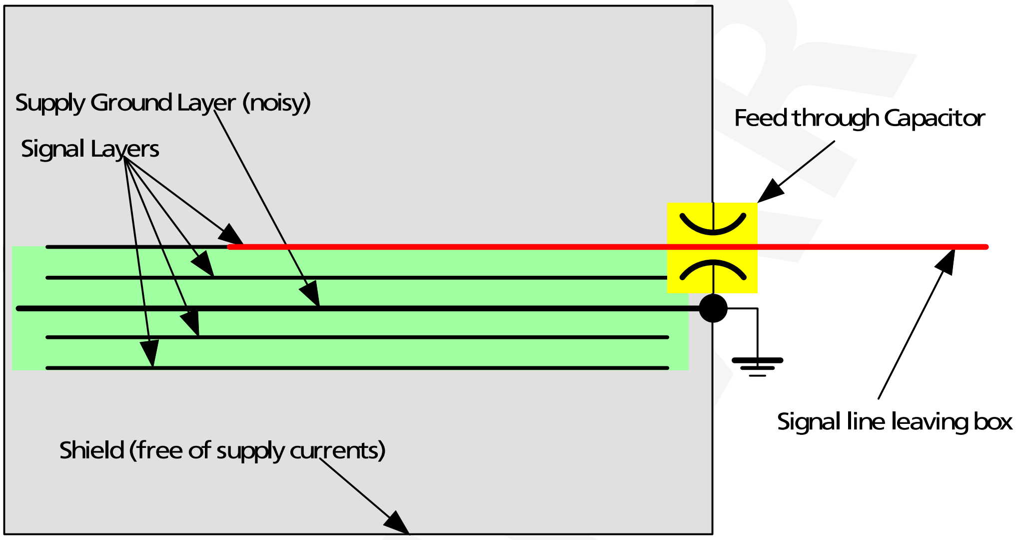


Figure 4-8: Ideal shielding

The proposed setup for such a system is shown in Figure 4-8. A feed through capacitor removes all high frequency content from the outgoing signal line. It's important to notice that any conductor traveling through the shielding box is subject to picking up noise inside and re-radiating it outside, regardless of the actual signal it is intended to carry. Therefore, also DC lines, e.g. the power supply should be filtered with feed through capacitors. When selecting feed through capacitors, it's important to choose components with appropriate frequency behavior. As with the ordinary capacitors, small value types will show better attenuation at high frequencies, see Figure 4-9.

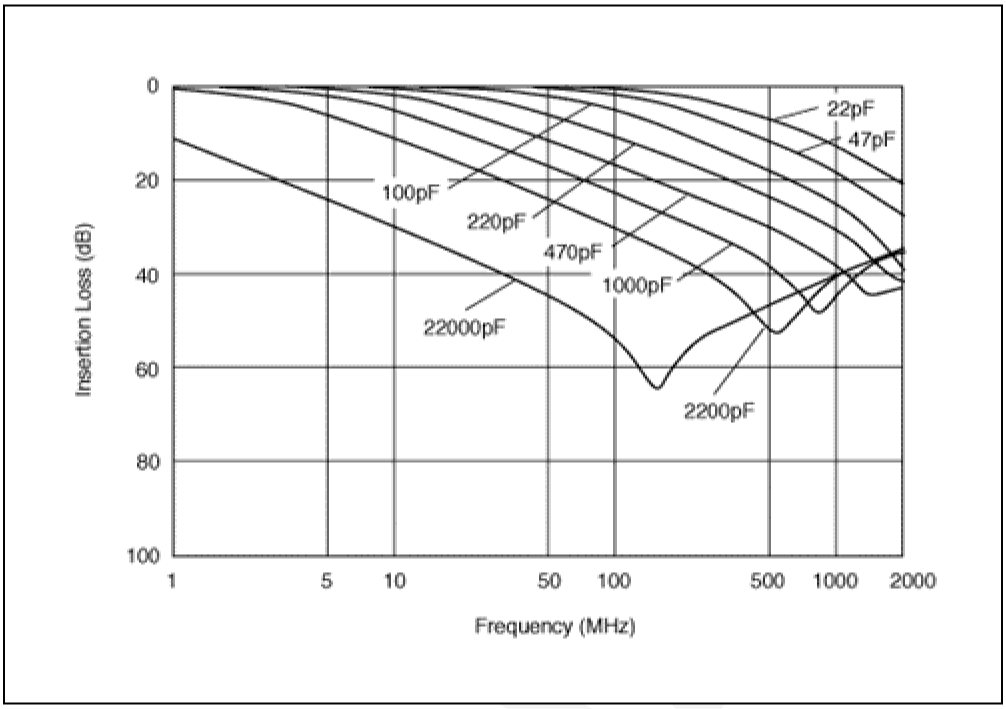


Figure 4-9: MuRata's NFM21C Feed Trough Capacitors

Any feed through capacitor will only achieve its specified performance if it has a proper ground connection.

If use of a special feed through capacitor is not feasible for a particular design, a simple capacitor between signal line and shielding ground placed very close to the feed through of the signal line will also help. It has been found that a 12 pF SMD capacitor works quite well at GPS frequency range. Larger capacitance values will be less efficient.

One should keep in mind that a feed-through capacitor is basically a high frequency "short" between signal line and ground. If the ground point that the capacitor is connected to is not ideal, meaning the ground connection or plane has a finite resistance, noise will be injected into the ground net. Therefore, one should try to place any feed trough capacitor far away from the most noise sensitive parts of the circuit. And, to stress this once again, one should ensure a very good ground connection for the feed through capacitor.

If there is no good ground connection available at the point of the feed through, or injection of noise into the non-ideal ground net must be avoided totally, inserting a component with a high resistance at high frequencies might be a good alternative. Ferrite beads are the components of choice if a high DC resistance cannot be accepted. Otherwise, for ordinary signal lines one could insert a 1 K series resistor which would then form a low-pass filter together with the parasitic capacitance of the conductor trace.

See also MuRata web page for extensive discussion on EMC countermeasures.

4.3.2 Shielding sets of sub-system assembly

Yet another problem arises if more than one building block are combined in a single system. Figure 4-10 shows one possible scenario. In this case, the supply current traveling through the inductive ground connection between the two sub-systems will cause a voltage difference between the two shields of the sub-system. The shield of the other system will then act as a transmitting antenna, radiating with respect to the ground and shield of the GPS receiver and the attached antenna.

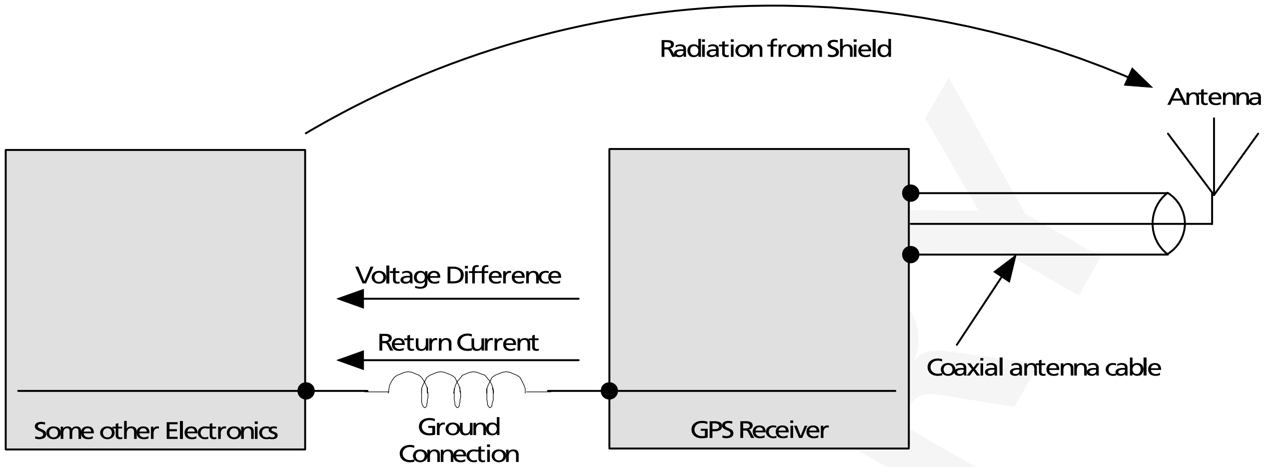


Figure 4-10: Two shielded sub-systems, connected by a "poor" ground

This situation can be avoided by ensuring a low inductivity ground connection between the two shields. But now, it might be difficult to control the path of the ground return currents to the power supply since the shield is probably connected to the supply ground at more than one location. The preferred solution is shown in Figure 4-11. Again, it is important to have a good, i.e. low inductance interconnection between the outer shield and the shielding ground of the GPS receiver.

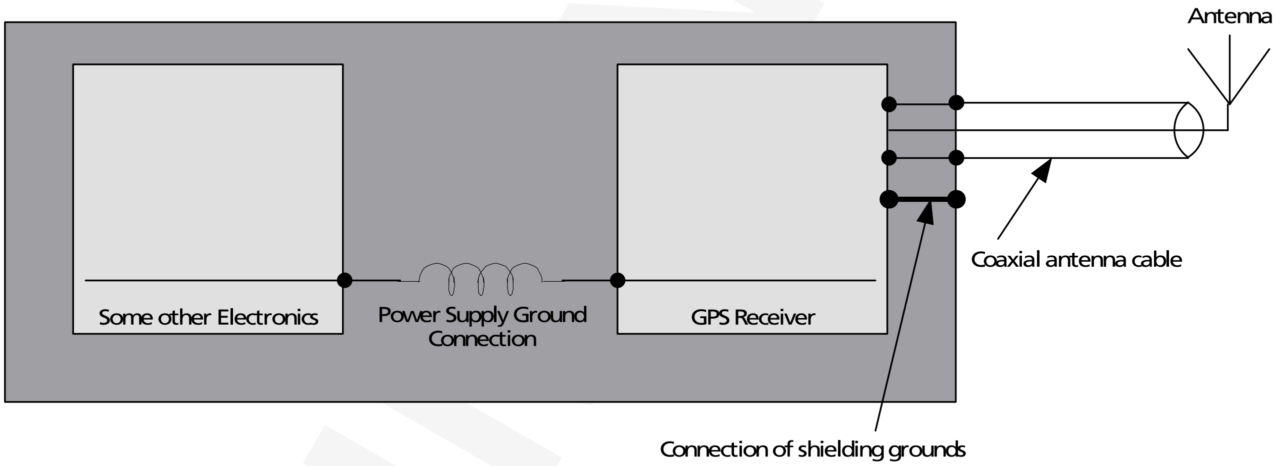


Figure 4-11: Proper shielding of a sub-system assembly

It is clear that the situation depicted in Figure 4-11 might get arbitrarily complex if "Some other electronics" contains another wireless transmitter system, requiring a second antenna which is referenced to the systems shielding ground. As already pointed out, in a setup like this it is important to keep the shield free from supply currents with high frequency spectral content. If there are to be additional connections to the shielding ground, these should be of high inductivity nature.

4.3.3 Shielding concept of TIM

Inside TIM the shielding concepts pointed out before have been applied. Figure 4-12 shows cross-section as well as layout of the inner ground layer of TIM, Rev. B. Since many applications do not require absolutely zero noise emissions, TIM does not contain any feed through capacitors on its signal and VCC pins. For specific noise emission requirements it might be advisable to add 12 pF capacitors or even better feed-through capacitors to shielding ground on those signals that are being routed outside of TIM. There's no need to also add capacitors to pins that have no connecting line attached outside of TIM. In the u-blox SAM smart antenna products, it was found that 1 K series resistors in the digital I/O lines are sufficient to reduce radiation from the flat ribbon cable to an acceptable level.

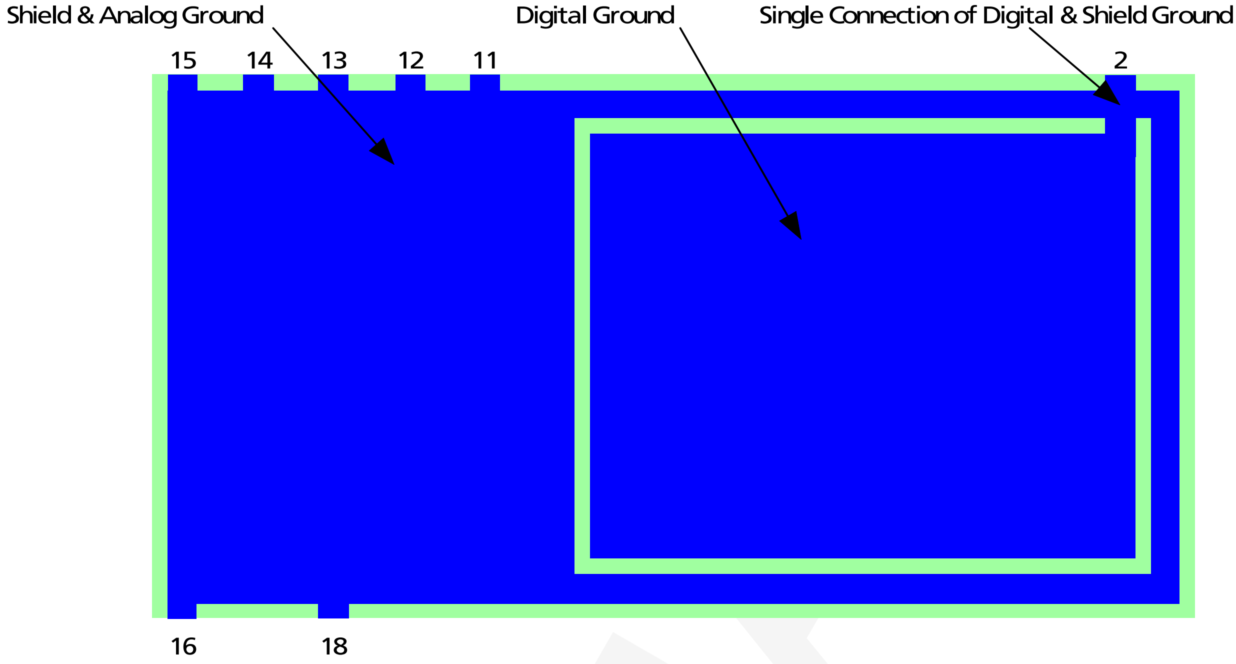


Figure 4-12: Shield & ground concept of TIM, Rev. B, with pin numbers

Pins number 10 and 30 are connected to the internal digital ground plane for backward compatibility with TIM, Rev. A. However, these two pins should not be used for new designs. The performance penalty of theses pins being connected to the ground plane of the application board is very low. So, there's no need to change a board based on the TIM, Rev. A layout recommendations. But for future designs, these two pins might become officially NCs and later being used for some I/O signals.

5 To Probe Further

5.1 EMI Suppression Components

MuRata	http://www.murata.com/
AVX	http://www.avx.com/

5.2 Antenna Manufacturers

Sarantel	http://www.sarantel.com/
EMTAC	http://www.emtac.com.tw/
Micropulse	http://www.micropulse.com/
NAIS	http://dmedia.mew.co.jp/nais-automotive/e-index.html

5.3 Serial Interface Drivers

Maxim	http://www.maxim-ic.com/

5.4 Voltage Regulators

Analog Devices	http://www.analog.com/
Linear Technology	http://www.linear.com/
Maxim	http://www.maxim-ic.com/
National	http://www.national.com/

5.5 Design Tools

Agilent	http://www.agilent.com/
Ansoft	http://www.ansoft.com/
Advanced Wave Research	http://www.mwoffice.com/

A RELATED DOCUMENTS

[1] TIM GPS Receiver Macro Component – Data Sheet, GPS.G2-MS2-01001

[2] SAM GPS Smart Antenna – Data Sheet, GPS.G2-SA-02004

[3] The GPS Dictionary, GPS-X-00001

All these documents are available on our homepage (http://www.u-blox.com).

B GLOSSARY

Please refer to the *GPS Dictionary*, [3].

C CONTACT

For further info, please contact us:

Headquarters	Subsidiaries	
u-blox ag Zuercherstrasse 68 CH-8800 Thalwil Switzerland Phone: +41 1 722 74 44 Fax: +41 1 722 74 47 E-mail: info@u-blox.com www.u-blox.com	**u-blox Deutschland GmbH** Berliner Ring 89 D-64625 Bensheim Germany Phone: +49 (0) 6251 17566-0 Fax: +49 (0) 6251 17566-11 E-mail: info_de@u-blox.de www.u-blox.de **Tech. Support:** Phone: +41 1 722 74 74 support_de@u-blox.de	**u-blox Asia Pacific Ltd.** 22/F., City Landmark I 68 Chung On Street Tsuen Wan, Hong Kong Phone: +852-2940-0085 Fax: +852-2615-2285 E-mail: info_ap@u-blox.com www.u-blox.com **Tech. Support:** Phone: +852-2940-0085 support_ap@u-blox.com
	u-blox Europe Ltd. Barham Court Maidstone, Kent ME18 5BZ United Kingdom Phone: +44 1622 618628 Fax: +44 1622 618629 E-mail: info_uk@u-blox.co.uk www.u-blox.co.uk **Tech. Support:** Phone: +44 1622 618628 support_uk@u-blox.co.uk	**u-blox America, Inc.** 13800 Coppermine Road Herndon, VA 20171 USA Phone: +1 (703) 234 5290 Fax: +1 (703) 234 5770 E-mail: info_us@u-blox.com www.u-blox.com **Tech. Support:** Phone: + 1 (703) 234 5290 support_us@u-blox.com