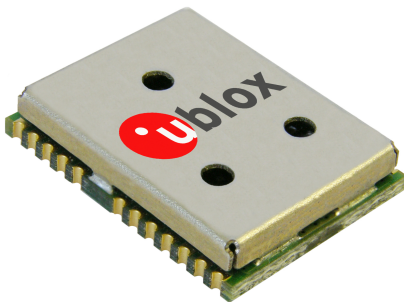


# NEO-5

## u-blox 5 GPS Modules

### Hardware Integration Manual



#### Abstract

This document describes the hardware features and specifications of the u-blox 5 powered NEO-5 series of cost effective, high-performance ROM-Based GPS modules.



Features include AssistNow Online and AssistNow Offline A-GPS services, KickStart accelerated acquisition, SuperSense® Indoor GPS providing best-in-class acquisition and tracking sensitivity, low power consumption and an innovative jamming-resistant RF architecture. The compact 16.0 x 12.2 mm form factor of the highly successful NEO-4S is maintained, enabling easy migration. The NEO-5 series supports passive and active antennas.

<b>Title</b>	NEO-5		
<b>Subtitle</b>	Hardware Integration Manual		
<b>Doc Type</b>	Manual	Preliminary	
<b>Doc Id</b>	GPS.G5-MS5-08003-A2		
<b>Revision Index</b>	<b>Date</b>	<b>Name</b>	<b>Status / Comments</b>
Initial Version	23/04/2008	TG	
A1	28/04/2008	TG	Addition of Hardware Description, Product Handling, Product Testing, Appendix, Power Modes, USB
A2	7/05/2008	TG	Update of pin tables

This document and the use of any information contained therein, is subject to the acceptance of the u-blox terms and conditions. They can be downloaded from [www.u-blox.com](http://www.u-blox.com). u-blox makes no warranties based on the accuracy or completeness of the contents of this document and reserves the right to make changes to specifications and product descriptions at any time without notice.

u-blox reserves all rights to this document and the information contained herein. Reproduction, use or disclosure to third parties without express permission is strictly prohibited. Copyright © 2008, u-blox AG.

For most recent documents, please visit [www.u-blox.com](http://www.u-blox.com)

	Products marked with this lead-free symbol on the product label comply with the "Directive 2002/95/EC of the European Parliament and the Council on the Restriction of Use of certain Hazardous Substances in Electrical and Electronic Equipment" (RoHS).
	This is an Electrostatic Sensitive Device (ESD). Observe precautions for handling.

## Preface

### u-blox Technical Documentation

As part of our commitment to customer support, u-blox maintains an extensive volume of technical documentation for our products. In addition to our product-specific technical data sheets, the following manuals are available to assist u-blox customers in product design and development.

- **GPS Compendium:** This document, also known as the GPS book, provides a wealth of information regarding generic GPS questions about system functionalities and technology.
- **Protocol Specification:** Messages, configuration and functionalities of the u-blox 5 software releases are explained in this document.
- **Hardware Integration Manual:** This Manual provides hardware design instructions and information on how to set up production and final product tests.

### How to use this Manual

The NEO-5 Hardware Integration Manual provides the necessary information to successfully design in and configure these u-blox 5 powered GPS receiver modules. For navigating this document please note the following:

This manual has a modular structure. It is not necessary to read it from the beginning to the end. To help in finding needed information, a brief section overview is provided below:

1. **Hardware Basics:** This chapter introduces the basics of function and architecture of the NEO 5 modules.
2. **Design In:** This chapter provides the Design-In information necessary for a successful design.
3. **Product Handling:** This chapter defines packaging, handling, shipment, storage and soldering.
4. **Product Testing:** This chapter provides information about testing of OEM receivers in production.
5. **Appendix:** The Appendix includes guidelines on how to successfully migrate to u-blox 5 designs, and useful information about the different antenna types available on the market and how to reduce interference in your GPS design.

The following symbols are used to highlight important information within the manual:



An index finger points out key information pertaining to module integration and performance.



**A warning symbol indicates actions that could negatively impact or damage the module.**

## Questions

If you have any questions about u-blox 5 Hardware Integration, please:

- Read this manual carefully.
- Contact our information service on the homepage <http://www.u-blox.com>
- Read the questions and answers on our FAQ database on the homepage <http://www.u-blox.com>

## Technical Support

### Worldwide Web

Our website ([www.u-blox.com](http://www.u-blox.com)) is a rich pool of information. Product information, technical documents and helpful FAQ can be accessed 24h a day.

### **By E-mail**

If you have technical problems or cannot find the required information in the provided documents, contact the nearest of the Technical Support offices by email. Use our service pool email addresses rather than any personal email address of our staff. This makes sure that your request is processed as soon as possible. You will find the contact details at the end of the document.

### **By Phone**

If an email contact is not the right choice to solve your problem or does not clearly answer your questions, call the nearest Technical Support office for assistance. You will find the contact details at the end of the document.

### **Helpful Information when Contacting Technical Support**

When contacting Technical Support please have the following information ready:

- Receiver type (e.g. NEO-5M) and firmware version (e.g. V4.00)
- Receiver configuration
- Clear description of your question or the problem together with a u-center logfile
- A short description of the application
- Your complete contact details

# Contents

Preface .....	3
<b>Contents.....</b>	<b>5</b>
<b>1 Hardware Description .....</b>	<b>8</b>
1.1 Functional Overview .....	8
1.2 Module Selector .....	8
1.3 Architecture.....	9
<b>2 Design-In.....</b>	<b>10</b>
2.1 Power Management.....	10
2.1.1 Connecting Power.....	10
2.1.2 Power Modes .....	11
2.1.3 Active Antenna Supply .....	11
2.2 System Functions.....	11
2.2.1 EXTINT - External Interrupt Pin.....	11
2.2.2 System Monitoring .....	11
2.3 Interfaces.....	11
2.3.1 Serial.....	11
2.3.2 USB .....	11
2.3.3 Display Data Channel (DDC).....	13
2.3.4 Synchronous Peripheral Interface (SPI) .....	13
2.4 I/O Pins .....	13
2.4.1 EXTINT0 .....	13
2.4.2 Configuration Pins (CFG_COM0, CFG_GPS0) .....	13
2.5 Design-In .....	14
2.5.1 Schematic Design-In Checklist for u-blox 5 .....	14
2.5.2 NEO-5 Design.....	14
2.5.3 Design for NEO-5M/ NEO-5Q .....	16
2.6 Layout Design-In Checklist.....	18
2.7 Layout .....	19
2.7.1 Footprint .....	19
2.7.2 Paste Mask .....	19
2.7.3 Placement .....	20
2.7.4 Antenna Connection and Grounding Plane Design.....	21
2.7.5 Antenna Micro Strip .....	23

2.8	Antenna and Antenna Supervisor .....	24
2.8.1	Passive Antenna .....	24
2.8.2	Active Antenna .....	25
2.8.3	Active Antenna Supervisor.....	25
<b>3</b>	<b>Product Handling.....</b>	<b>26</b>
3.1	Packaging.....	26
3.1.1	Reels .....	26
3.1.2	Tapes .....	27
3.2	Shipment, Storage and Handling .....	27
3.2.1	Handling .....	27
3.2.2	Shipment.....	27
3.2.3	Storage .....	28
3.2.4	Handling .....	28
3.2.5	Floor Life .....	29
3.3	Processing .....	30
3.3.1	Moisture Preconditioning .....	30
3.3.2	Soldering Paste.....	30
3.3.3	Reflow Soldering .....	31
3.3.4	Optical Inspection.....	32
3.3.5	Cleaning.....	33
3.3.6	Repeated Reflow Soldering.....	33
3.3.7	Wave Soldering .....	33
3.3.8	Hand Soldering .....	33
3.3.9	Rework.....	33
3.3.10	Conformal Coating .....	34
3.3.11	Casting.....	34
3.3.12	Grounding Metal Covers .....	34
3.3.13	Use of Ultrasonic Processes.....	34
<b>4</b>	<b>Product Testing.....</b>	<b>35</b>
4.1	u-blox In-Series Production Test.....	35
4.2	Test Parameters for OEM Manufacturer .....	35
4.3	System Sensitivity Test .....	36
4.3.1	Guidelines for Sensitivity Tests .....	36
4.3.2	'Go/No go' tests for integrated devices.....	36
<b>A</b>	<b>Migration to u-blox 5 receivers.....</b>	<b>37</b>
A.1	Migration from NEO-4S to NEO-5.....	37

<b>B</b>	<b>Antennas.....</b>	<b>38</b>
B.1	Selecting the right Antenna.....	38
B.2	Active and Passive Antennas.....	39
B.3	Patch Antennas .....	39
B.4	Helix Antennas .....	41
B.4.1	Multilayer Patch .....	41
B.4.2	Dipol .....	41
B.5	Helix or Patch, which selection is best?.....	41
B.6	Antenna Matching .....	42
B.7	Antenna Placement .....	43
<b>C</b>	<b>Interference Issues .....</b>	<b>44</b>
C.1	Sources of Noise .....	44
C.2	Eliminating Digital Noise Sources .....	45
C.2.1	Power and Ground Planes .....	45
C.2.2	High Speed Signal Lines.....	46
C.2.3	Decoupling Capacitors .....	46
C.3	Shielding .....	48
C.3.1	Feed through Capacitors .....	48
C.3.2	Shielding Sets of Sub-System Assembly .....	50
<b>D</b>	<b>Lists .....</b>	<b>51</b>
D.1	List of Figures .....	51
D.2	List of Tables.....	52
<b>E</b>	<b>Glossary .....</b>	<b>52</b>
	<b>Related Documents .....</b>	<b>53</b>
	<b>Contact.....</b>	<b>54</b>

# 1 Hardware Description

## 1.1 Functional Overview

The NEO-5 module series is a family of self-contained GPS receivers featuring the high performance 50-channel u-blox 5 positioning engine. These modules provide exceptional GPS performance in a compact form factor and at an economical price. u-blox 5 sets a new standard in GPS receiver technology. A 32-channel acquisition engine with over 1 million effective correlators is capable of massive parallel searches across the time/frequency space. This enables a Time To First Fix (TTFF) of less than 1 second, while long correlation/dwell times make possible the best-in-class acquisition and tracking sensitivity. Once acquired, satellites are passed on to a dedicated tracking engine. This arrangement allows the GPS engine to simultaneously track up to 16 satellites while searching for new ones. u-blox 5's advanced jamming suppression mechanism and innovative RF architecture provide a high level of immunity to jamming, ensuring maximum GPS performance. u-blox 5 has been designed to be able to support the GALILEO system currently being developed by European authorities. The capability of receiving GALILEO L1 signals will provide increased coverage and even better positioning accuracy when this system comes into operation.

With the NEO-5 series the complete signal processing chain from antenna input to serial output is contained within a single component. NEO-5 modules maintain the compact 17.0 x 22.4 mm form factor of the highly successful NEO-4S predecessor. The NEO-5 modules have been designed with backwards compatibility in mind, enabling ease of upgrade and reducing engineering and design costs.

Their small size makes NEO-5 modules the ideal GPS solution for applications with stringent space requirements. The packaging makes expensive RF cabling obsolete, with the RF input being available directly on a pin. The NEO-5 series are SMT solderable and can be handled by standard pick and place equipment.

NEO-5 modules come equipped with a serial port, which can handle NMEA and UBX proprietary data formats, as well as a high speed USB port. The optional FLASH EPROM provides the capacity to store user-specific configuration settings as well as future software updates.

All NEO-5 modules are RoHS compliant (lead-free) and green (no halogens).

The NEO-5 series of GPS/GALILEO receiver modules are not designed for life saving or supporting devices or for aviation and should not be used in products that could in any way negatively impact the security or health of the user or third parties or that could cause damage to goods.

## 1.2 Module Selector

u-blox provides several modules using the popular NEO Form factor. To select the right product for your design consider Table 1.

	Voltage Range (V)	Thickness (mm)	50-channel engine	KickStart	SuperSense	FW Update / FLASH	Low Power Modes	UART	USB	SPI	DDC	AssistNow Online	AssistNow Offline	Dead Reckoning	Raw Data	Precision Timing	1PPS	CFG Pin	Reset Input	Antenna Supply	Antenna Supervisor
NEO-5M	2.7-3.6	2.4	✓		✓		P	1	1		1	✓	✓				✓	1			
NEO-5Q	2.7-3.6	2.4	✓	✓	✓		P	1	1	1	1	✓	✓				✓	2			

P= Planned availability Q1/09

**Table 1: Features of the NEO-5 Series**

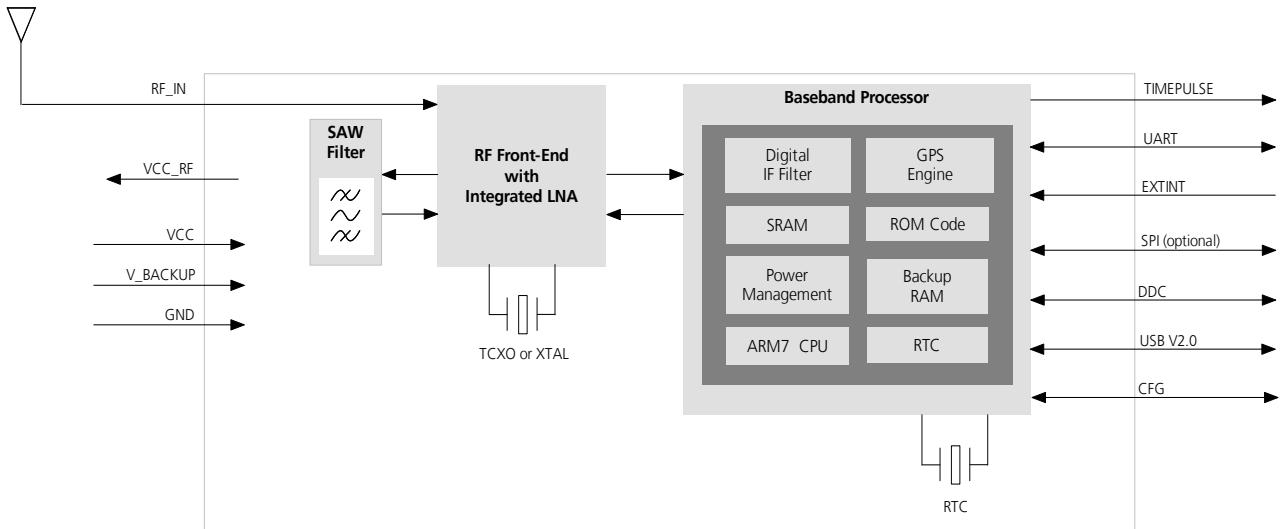


### 1.3 Architecture

NEO-5 modules are divided into two distinct, separately shielded sections. The smaller section is the RF- Section, the larger section contains the Baseband. See Figure 1 for a block diagram of the NEO-5 series.

The RF Front-End contains the integrated Low Noise Amplifier (LNA), the SAW bandpass filter, the u-blox 5 RF-IC and the TCXO or XTO crystal.

The Baseband section contains the digital circuitry comprised of the u-blox 5 Baseband processor, the RTC crystal and additional elements such as the optional FLASH EPROM for enhanced programmability and flexibility.



**Figure 1: NEO-5 Block Diagram**

## 2 Design-In

 For migrating existing ANTARIS®4 product designs to u-blox 5 please refer to *Appendix A*.

In order to obtain good performance with a GPS receiver module, there are a number of points that require careful attention during the design-in. These include:

- Power Supply  
Good performance requires a clean and stable power supply.
- Interfaces  
Ensure correct wiring, rate and message setup on the module and your host system.
- Antenna interface  
For optimal performance seek short routing, matched impedance and no stubs.

### 2.1 Power Management

#### 2.1.1 Connecting Power

u-blox 5 receivers have three power supply pins: **VCC**, **V\_BCKP** and **VDDUSB**.

##### 2.1.1.1 VCC - Main Power

The main power supply is fed through the **VCC** pin. During operation, the current drawn by the u-blox 5 GPS module can vary by some orders of magnitude, especially, if low-power operation modes are enabled. It is important that the system power supply circuitry is able to support the peak power (see datasheet for specification) for a short time. In order to define a battery capacity for specific applications the sustained power figure shall be used.

##### 2.1.1.2 V\_BCKP - Backup Battery

In case of a power failure on pin **VCC**, the real-time clock and backup RAM are supplied through pin **V\_BCKP**. This enables the u-blox 5 receiver to recover from a power failure with either a Hotstart or a Warmstart (depending on the duration of **VCC** outage) and to maintain the configuration settings. If no backup battery is connected, the receiver performs a Coldstart at power up.

 If no backup battery available connect the **V\_BCKP** pin to **GND (or VCC)**.

As long as **VCC** is supplied to the u-blox 5 receiver, the backup battery is disconnected from the RTC and the backup RAM in order to avoid unnecessary battery drain (see Figure 2). Power to RTC and BBR is supplied from **VCC** in this case.

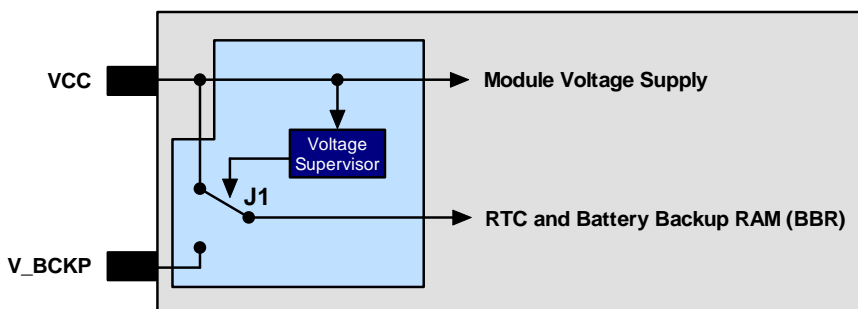


Figure 2: Backup Battery and Voltage

### 2.1.1.3 VDD\_USB - USB Interface Power Supply

**VDD\_USB** supplies the I/Os of the USB interface. If the USB interface is not used, the **VDD\_USB** pin must be connected to GND. For more information regarding the correct handling of VDD\_USB see section 2.3.2.1

## 2.1.2 Power Modes

u-blox 5 technology offers power-optimized architecture with built-in autonomous power saving functions. The receiver uses Autonomous Power Management to minimize the power consumption at any given time. Furthermore, the software shuts down the clock supply to unused peripheral on-chip blocks.

### 2.1.3 Active Antenna Supply

With NEO-5 modules active antennas are supplied via an external coil or circuit. See Section 2.5.2 for more information.

## 2.2 System Functions

### 2.2.1 EXTINT - External Interrupt Pin

**EXTINT0** is an external interrupt pin. It will be used in future NEO-5 releases for wake-up functions in low-power modes.

### 2.2.2 System Monitoring

The u-blox 5 GPS and GALILEO Receiver provides System Monitoring functions that allow the operation of the embedded processor and associated peripherals to be supervised. These System Monitoring functions are being output as part of the UBX protocol, class 'MON'.

Please refer to the *u-blox 5 Protocol Specification* [3]. For more information on UBX messages, serial interfaces for design analysis and individual system monitoring functions.

## 2.3 Interfaces

### 2.3.1 Serial

UART 1 (**RxD1/TxD1**) is the default serial interface. It supports data rates from 4.8 kBit/s to 115 kBit/s. The signal levels are CMOS 0 V to VCC. An interface based on RS232 standard levels (+/- 12 V) can be realized using level shifters such as Maxim MAX3232.



The **RxD1** has fixed input voltage thresholds, which do not depend on **VCC** (see *NEO-5 Data Sheet* [2]). Leave open if unused.

For the default settings see the *NEO-5 Data Sheet* [2].

Hardware handshake signals and synchronous operation are not supported.

### 2.3.2 USB

The u-blox 5 USB interface supports the full-speed data rate of 12 Mbit/s.

#### 2.3.2.1 USB external components

The USB interface requires some external components in order to implement the physical characteristics required by the USB 2.0 specification. These external components are shown in Figure 3 and listed in Table 3.

In order to comply with USB specifications, VBUS must be connected through an LDO (U1) to pin **VDD\_USB** of the module.

If the USB device is **self-powered** it is possible that the power supply (VCC) is shut down and the Baseband-IC core is not powered. Since VBUS is still available, it still would be signaled to the USB host that the device is present and ready to communicate. This is not desired and thus the LDO (U1) should be disabled using the enable signal (EN) of the VCC-LDO or the output of a voltage supervisor. Depending on the characteristics of the LDO (U1) it is recommended to add a pull-down resistor (R11) at its output to ensure **VDD\_USB** is not floating if LDO (U1) is disabled or the USB cable is not connected i.e. VBUS is not supplied. If the device is **bus-powered**, LDO (U1) does not need an enable control.

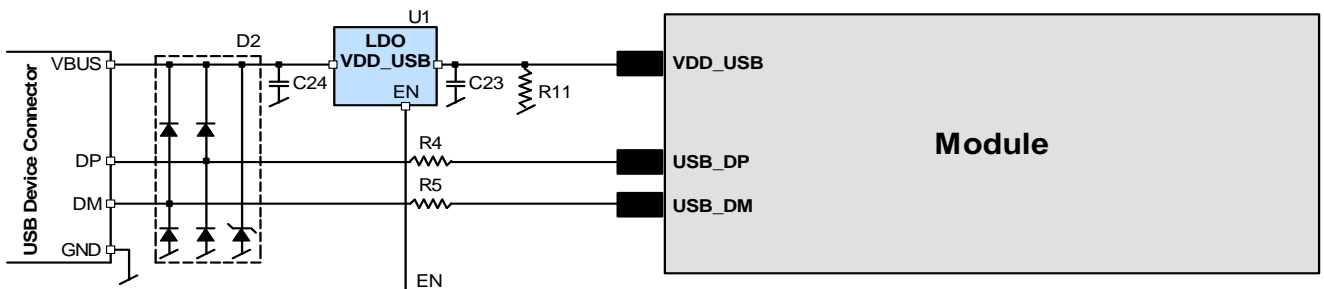
Table 2 lists the present availability of USB on NEO-5 modules. Check the **CFG\_COM0** configuration pin to select between Self and Bus Powered modes. Note that BUS Powered mode is not at this time supported. For more information on configuration see the *NEO-5 Data Sheet* [2].

	Bus Powered	Self Powered
NEO-5Q	P	•
NEO-5M	P	•

•= Supported

P= Planned availability Q1/09

**Table 2: USB availability on NEO-5 modules**



**Figure 3: USB Interface**

Name	Component	Function	Comments
U1	LDO	Regulates VBUS (4.4 ... 5.25 V) down to a voltage of 3.3 V).	Almost no current requirement (~1 mA) if the GPS receiver is operated as a USB self-powered device, but if bus-powered LDO (U1) must be able to deliver the maximum current of ~150 mA. A low-cost DC/DC converter such as LTC3410 from Linear Technology may be used as an alternative.
C23, C24	Capacitors		Required according to the specification of LDO U1
D2	Protection diodes	Protect circuit from overvoltage / ESD when connecting.	
R4, R5	Serial termination resistors	Establish a full-speed driver impedance of 28...44 Ohms	A value of 27 Ohms is recommended.
R11	Resistor	Ensures stable signal at <b>VDD_USB</b> .	

**Table 3: Summary of USB external components**

### 2.3.3 Display Data Channel (DDC)

A DDC interface (**SDA2** / **SCL2**) is available for communication with the host. Pins **SDA2** and **SCL2** have internal pull-ups.

No pull-up resistors are required if the module is used as a DDC/I<sup>2</sup>C slave (see Figure 4).

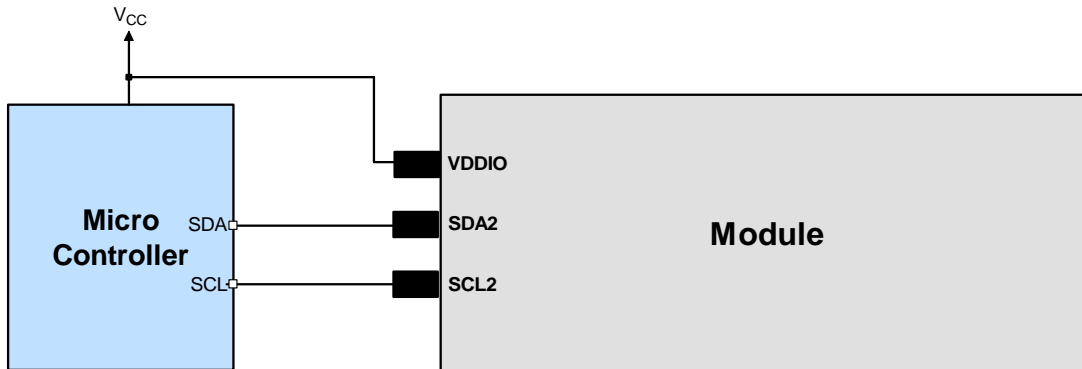


Figure 4: Connecting Module as DDC Slave



No Master Mode: External memory is not supported at this time.

### 2.3.4 Synchronous Peripheral Interface (SPI)

The SPI interface allows to interface to a host CPU. In slave mode a single chip select signal enables communication with the host.



No Master Mode: External memory is not supported at this time.

## 2.4 I/O Pins

### 2.4.1 EXTINT0

**EXTINT0** is an external interrupt pin with fixed input voltage thresholds independent of VCC (see the *NEO-5 Data Sheet* [2]). Leave open if unused.

### 2.4.2 Configuration Pins (CFG\_COM0, CFG\_GPS0)

NEO-5Q and NEO-5M provide one or two pins for boot-time configuration. These pins become effective immediately after start-up. Once the module has started, the configuration settings may be modified with UBX configuration messages. The modified settings remain effective until power-down or reset. If these settings have been stored in battery-backup RAM, then the modified configuration will be retained, as long as the backup battery supply is not interrupted.

Some configuration pins are shared with other functions, e.g. SPI. During start-up, the module reads the state of the configuration pins. Afterwards the other functions can be used.

For more information about settings and messages see the *NEO-5 Data Sheet* [2].

## 2.5 Design-In

This section provides a Design-In Checklist as well as Reference Schematics for new designs with u-blox 5. For migration of existing ANTARIS®4 product designs to u-blox 5 please refer to *Appendix A*.

### 2.5.1 Schematic Design-In Checklist for u-blox 5

Designing-in a NEO-5 GPS receiver is easy especially when a design is based on the reference design in the Hardware Integration Manual. Nonetheless, it pays to do a quick sanity check of the design. This section lists the most important items for a simple design check. The Layout Design-In Checklist also helps to avoid an unnecessary respin of the PCB and helps to achieve the best possible performance.



It is highly recommended to follow the Design-In Checklist when developing any u-blox 5 GPS applications. This can significantly reduce development time and costs.

#### Have you chosen the optimal module?

NEO-5 modules have been intentionally designed to allow GPS receivers to be optimally tailored to specific applications. Changing between the different variants is easy.

- Do you need Kick-start performance – Then choose NEO-5Q.

#### Check Power Supply Requirements and Schematic:

- Is the power supply within the specified range?
- Is the voltage **VDDUSB** within the specified range?
- Compare the peak current consumption of NEO-5 with the specification of your power supply.
- GPS receivers require a stable power supply, avoid ripple on **VCC** (<50mVpp)

#### Backup Battery

- For achieving a minimal Time To First Fix (TTFF) after a power down, make sure to connect a backup battery to **V\_BCKP**.

#### Antenna

- The total noise figure should be well below 3dB.
- If a patch antenna is the preferred antenna, choose a patch of at least 15x15mm. Designs using smaller antennas require aiding.
- Make sure the antenna is not placed close to noisy parts of the circuitry. (e.g. micro-controller, display, etc.)

#### Schematic

- Plan use of 2<sup>nd</sup> interface (Testpoints on serial port, DDC or USB) for firmware updates or as a service connector.

### 2.5.2 NEO-5 Design

For a minimal Design with NEO-5 the following functions and pins need to be considered:

- Connect the Power supply to **VCC**.
- **VDDUSB**: Connect the USB power supply to a LDO before feeding it to **VDDUSB** and **VCC** or connect to GND if USB is not used.
- Ensure an optimal ground connection to all ground pins of the NEO module
- Choose the required serial communication interface (USART USB, SPI or DDC) and connect the appropriate pins to your application
- If you need Hot- or Warmstart in your application, connect a Backup Battery to **V\_BCKP**

- Decide whether **TIMEPULSE** options are required in your application and connect the appropriate pins on your module
- Connect pins 8 and 9 together.
- NEO-5 modules do not provide the antenna bias voltage for active antennas on the **RF\_IN** pin. It is therefore necessary to provide this voltage outside the module via an inductor as indicated in Figure 5. u-blox recommends using an inductor from Coilcraft (0402CS-36NX). Alternative parts can be used if the inductor's resonant frequency matches the GPS frequency of 1575.4MHz.

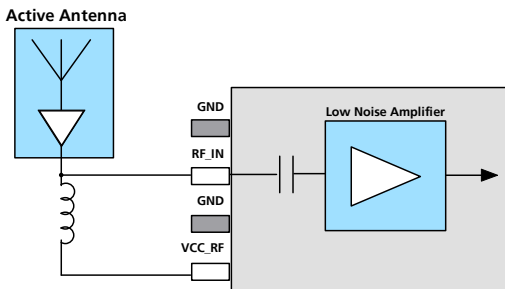


Figure 5: Recommended wiring for active antennas

- For optimal performance, it is important to place the inductor as close to the microstrip as possible. Figure 6 illustrates the recommended layout and how it should not be done.

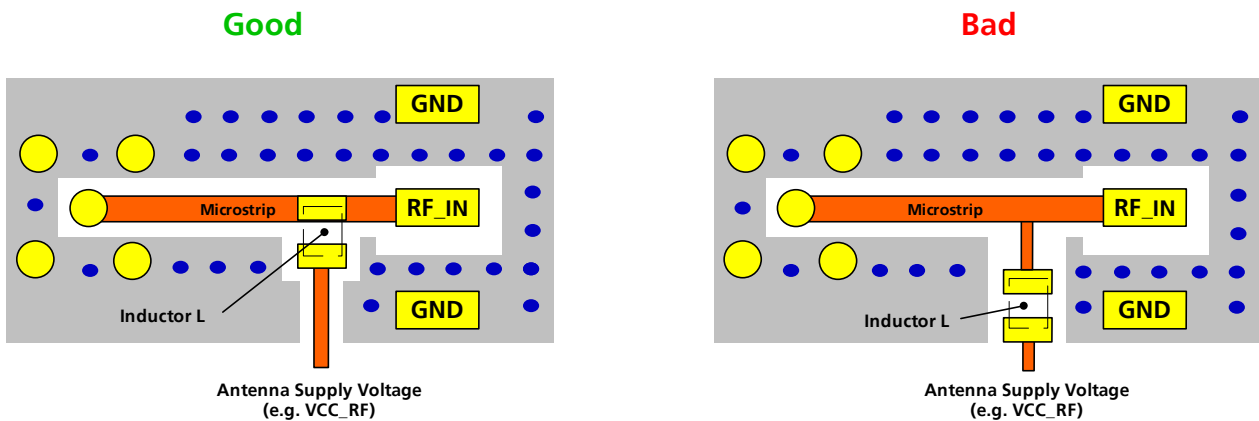


Figure 6: Recommended layout for connecting the antenna bias voltage for NEO-5

### 2.5.3 Design for NEO-5M/ NEO-5Q

This is a minimal setup for a PVT GPS receiver with a NEO-5 module.

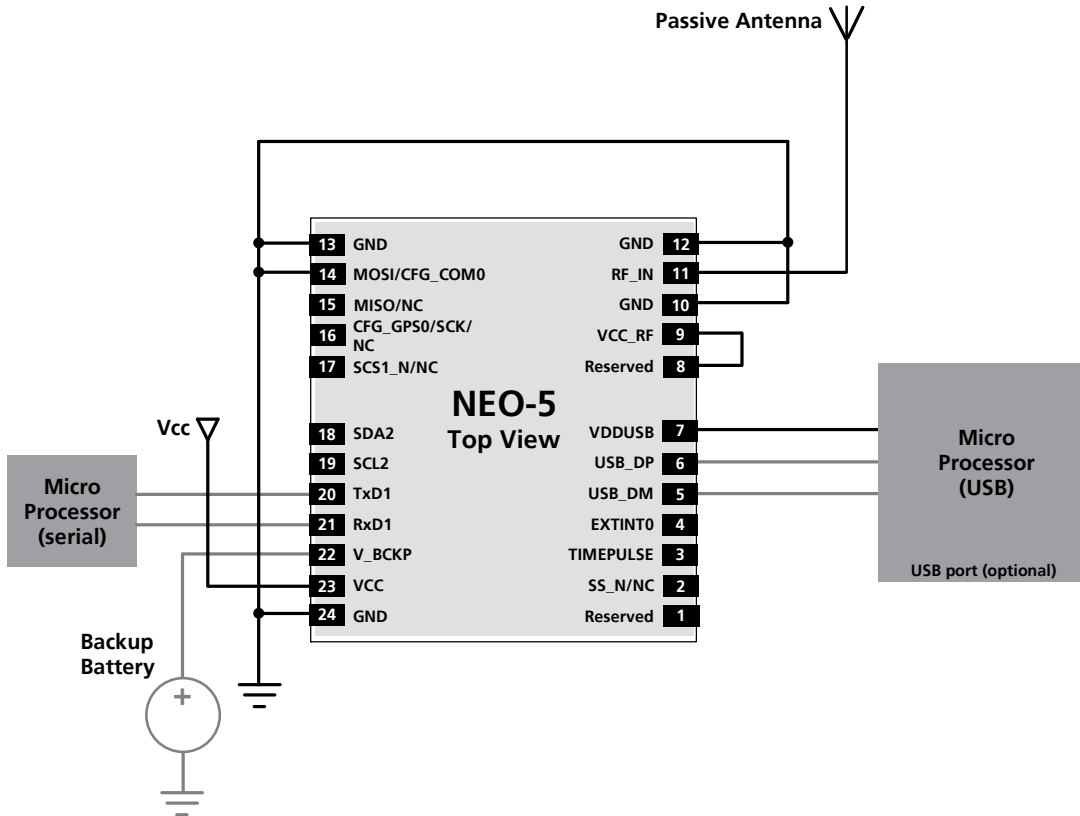


Figure 7: Passive Antenna Design for NEO-5 Receivers



Standard Function				Remarks
No	Name	I/O	Description	
1	Reserved	I	Reserved	Leave open
2	NC/SS_N	I	Not Connected/ SPI Slave Select	NEO-5Q: Slave select input for SPI. Leave open if not used.
3	TIMEPULSE	O	Time pulse (1PPS)	Configurable Timepulse signal (one pulse per second by default). Leave open if not used.
4	EXTINT0	I	External Interrupt Pin	Leave open if not used.
5	USB_DM <sup>1</sup>	I/O	USB Data	USB bidirectional communication pin. Implementation see Section 2.3.2.
6	USB_DP <sup>1</sup>	I/O	USB Data	
7	VDDUSB <sup>1</sup>	I	USB Supply	To use the USB interface connect this pin to 3.0 – 3.6V. If no USB serial port used connect to GND.
8	Reserved	I	Reserved	Pins 8 and 9 <b>must</b> be connected. VCC_RF can also be used to power an external active antenna.
9	VCC_RF	O	Output Voltage RF section	
10	GND	I	Ground	Assure a good GND connection to all GND pins of the module, preferably with a large ground plane.
11	RF_IN	I	GPS signal input	The connection to the antenna has to be routed on the PCB. Use a controlled impedance of 50 Ohm to connect RF_IN to the antenna or the antenna connector.
12	GND	I	Ground	See pin 10.
13	GND	I	Ground	See pin 10.
14	MOSI/CFG_COM0	O	SPI MOSI/ CFG_COM0	NEO-5Q: Leave open if not used. See Section 2.3.2. NEO-5M: Leave open. <span style="float: right;">Connect to GND to use USB in Self Powered mode.</span>
15	MISO/NC	I	SPI MISO/ Not Connected	NEO-5Q: SPI MISO. Leave open if not used.
16	SCK/CFG_GPS0/ NC	I/O	SPI Clock/Power Mode Configuration Pin/ Not Connected	NEO-5Q: CFG_GPS0 pin shared with the SPI Clock pin. When using Eco Mode and SPI, pull CFG_GPS0 low during startup and then release it.
17	SCS1_N/ NC	O	SPI chip select <sup>1</sup> / Not Connected	NEO-5Q: Leave open if not used.
18	SDA2	I/O	DDC Data	
19	SCL2	I/O	DDC Clock	
20	TxD1	O	Serial Port 1	
21	RxD1	I	Serial Port 1	3.6V tolerant serial input. Internal pull-up resistor to VCC. Leave open if not used. Don't use an external pull up resistor.
22	V_BCKP	I	Backup voltage supply	It's recommended to connect a backup battery to V_BCKP in order to enable Warm and Hot Start features on the receiver. Otherwise connect to GND.
23	VCC	I	Supply voltage	Max allowed ripple on VCC=50mVpp
24	GND	I	Ground	See pin 10.

**Table 4: Pinout NEO-5**
<sup>1</sup> Planned availability of Bus Powered Mode: Q1/09

## 2.6 Layout Design-In Checklist

Follow this checklist for the Layout design to get an optimal GPS performance.

### Layout optimizations (Section 2.7)

- Is the GPS module placed according to the recommendation in Section 2.7.2?
- Has the Grounding concept been followed (see Section 2.7.3)?
- Has the micro strip been kept as short as possible?
- Add a ground plane underneath the GPS module to reduce interference.
- For improved shielding, add as many vias as possible around the micro strip, around the serial communication lines, underneath the GPS module etc.

### Calculation of the micro strip (Section 2.7.5)

- The micro strip must be 50 Ohms and be routed in a section of the PCB where minimal interference from noise sources can be expected.
- In case of a multi-layer PCB, use the thickness of the dielectric between the signal and the 1st **GND** layer (typically the 2nd layer) for the micro strip calculation.
- If the distance between the micro strip and the adjacent **GND** area (on the same layer) does not exceed 5 times the track width of the micro strip, use the "Coplanar Waveguide" model in AppCad to calculate the micro strip and not the "micro strip" model.

## 2.7 Layout

This section provides important information for designing a reliable and sensitive GPS/GALILEO system.

GPS signals at the surface of the Earth are about 15dB below the thermal noise floor. Signal loss at the antenna and the RF connection must be minimized as much as possible. When defining a GPS receiver layout, the placement of the antenna with respect to the receiver, as well as grounding, shielding and jamming from other digital devices are crucial issues and need to be considered very carefully.

### 2.7.1 Footprint

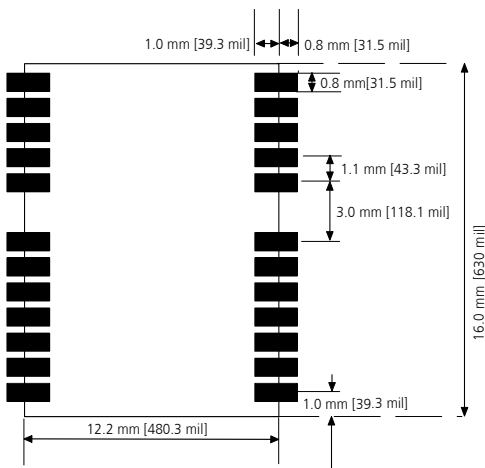


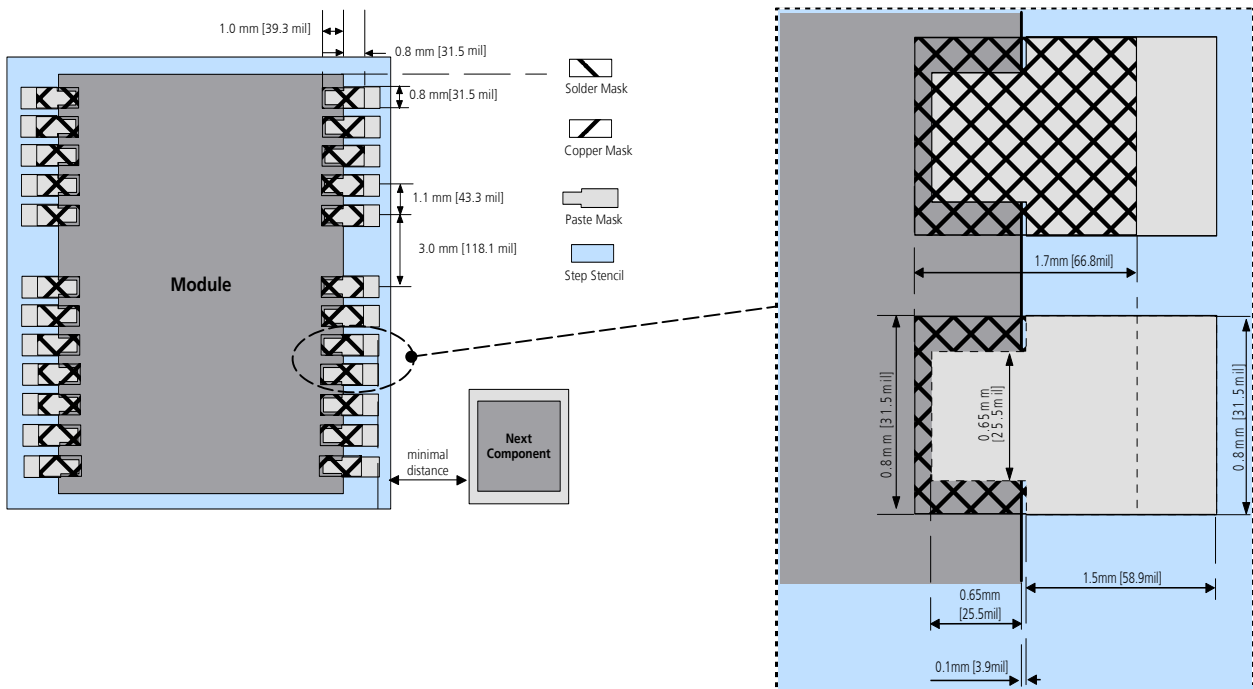
Figure 8: Recommended footprint

### 2.7.2 Paste Mask

Figure 9 shows the recommended positioning of the Paste Mask, the Copper and Solder masks, as well as the step stencil. These are recommendations only and not specifications. Note that the Copper and Solder masks have the same size and position.

To improve the wetting of the half vias, reduce the amount of solder paste under the module and increase the volume outside of the module by defining the dimensions of the paste mask to form a T-shape (or equivalent) extending beyond the Copper mask as shown in Figure 9. In addition, use a step stencil covering the entire area of the module and beyond the paste mask to increase the volume of solder paste here. The solder paste at the step stencil should have a total thickness of 175 to 200  $\mu\text{m}$ .

If a step stencil is not used it is still advisable to increase the volume of solder paste **outside** the module to attain the desired level of wetting. This must be done by modifying the shape of the paste mask outside the module, to allow for the increased volume of solder paste.



**Figure 9: Recommendations for copper, solder and paste masks with enlargement**

- The paste mask outline needs to be considered when defining the minimal distance to the next component.
- These are recommendations only and not specifications. The exact geometry, distances, stencil thicknesses, step heights and solder paste volumes must be adapted to the specific production processes (e.g. soldering etc.) of the customer.

### 2.7.3 Placement

A very important factor in achieving maximum GPS and GALILEO performance is the placement of the receiver on the PCB. The connection to the antenna must be as short as possible to avoid jamming into the very sensitive RF section.

Make sure that RF critical circuits are clearly separated from any other digital circuits on the system board. To achieve this, position the receiver digital part towards your digital section of the system PCB. Care must also be exercised with placing the receiver in proximity to circuitry that can emit heat. The RF part of the receiver is very sensitive to temperature and sudden changes can have an adverse impact on performance.

- The RF part of the receiver is a temperature sensitive component. Avoid high temperature drift and air vents near the receiver.**

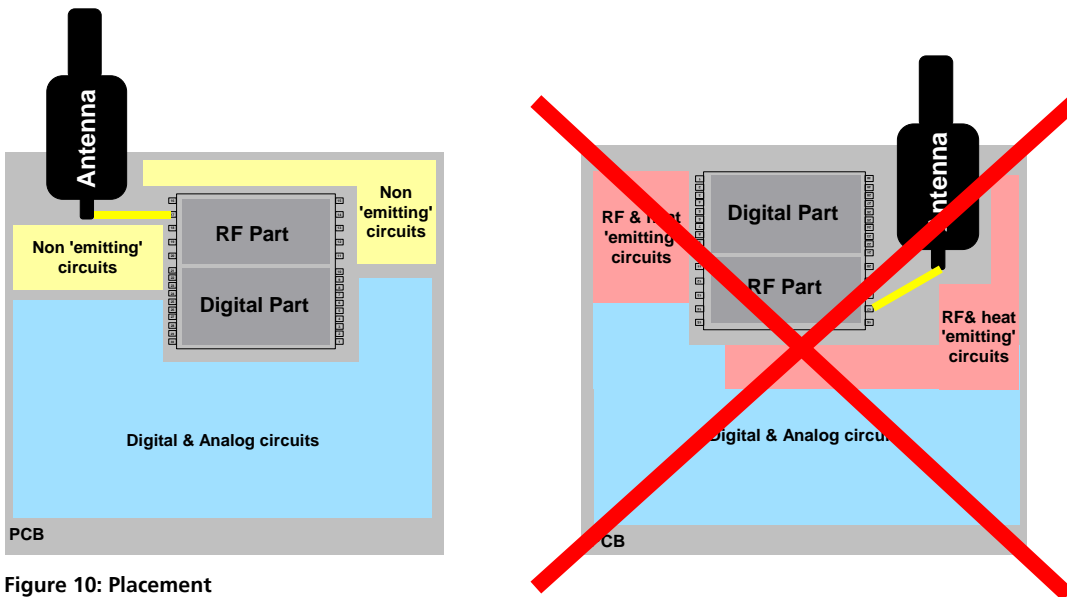
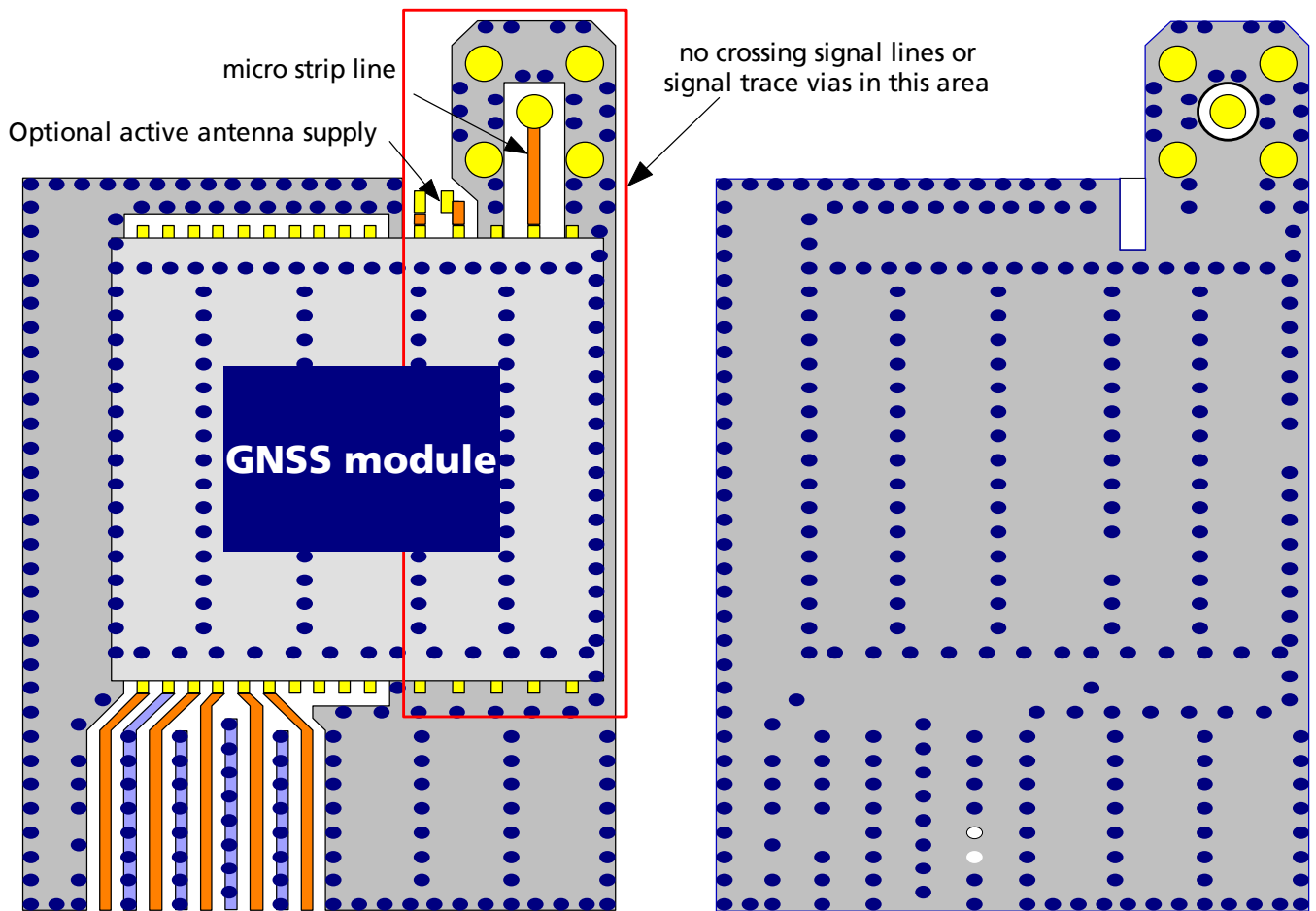


Figure 10: Placement

### 2.7.4 Antenna Connection and Grounding Plane Design

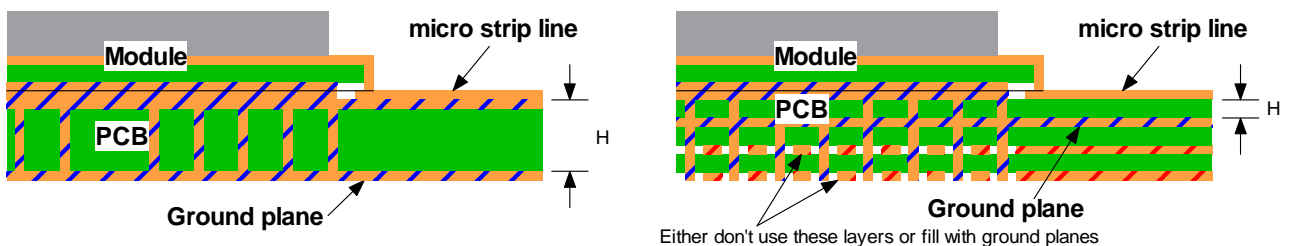
u-blox 5 modules can be connected to passive patch or active antennas. The RF connection is on the PCB and connects the **RF\_IN** pin with the antenna feed point or the signal pin of the connector, respectively. *Figure 11* illustrates connection to a typical five-pin RF connector. One can see the improved shielding for digital lines as discussed in *Appendix C.3*. Depending on the actual size of the ground area, additional vias should be placed in the outer region. In particular, the edges of the ground area should be terminated with a dense line of vias.



**Figure 11: Recommended layout**

As seen in *Figure 11*, an isolated ground area is created around and below the RF connection. This part of the circuit **MUST** be kept as far from potential noise sources as possible. Make certain that no signal lines cross, and that no signal trace vias appear at the PCB surface within the area of the red rectangle. The ground plane should also be free of digital supply return currents in this area. On a multi layer board, the whole layer stack below the RF connection should be kept free of digital lines. This is because even solid ground planes provide only limited isolation.

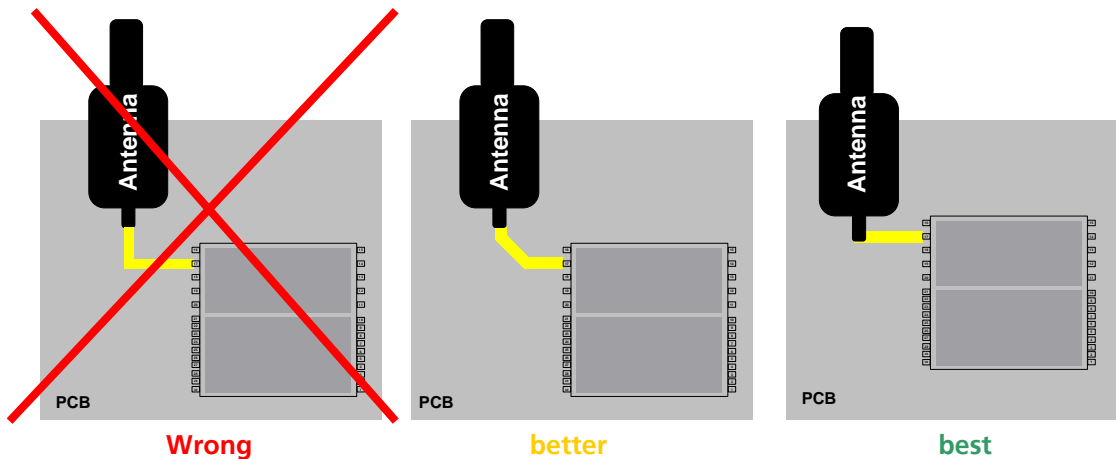
The impedance of the antenna connection has to match the 50 Ohm impedance of the receiver. To achieve an impedance of 50 Ohms, the width  $W$  of the micro strip has to be chosen depending on the dielectric thickness  $H$ , the dielectric constant  $\epsilon_r$  of the dielectric material of the PCB and on the build-up of the PCB (see *Section 2.7.5*). *Figure 12* shows two different builds: A 2 Layer PCB and a 4 Layer PCB. The reference ground plane is in both designs on layer 2 (red). Therefore the effective thickness of the dielectric is different.



**Figure 12: PCB build-up for Micro strip line. Left: 2-layer PCB, right: 4-layer PCB**

General design recommendations:

- The length of the micro strip line should be kept as short as possible. Lengths over 2.5 cm (1 inch) should be avoided on standard PCB material and without additional shielding.
- Distance between micro strip line and ground area on the top layer should at least be as large as the dielectric thickness.
- Routing the RF connection close to digital sections of the design should be avoided.
- To reduce signal reflections, sharp angles in the routing of the micro strip line should be avoided. Chamfers or fillets are preferred for rectangular routing; 45-degree routing is preferred over Manhattan style 90-degree routing.



- Routing of the RF-connection underneath the receiver should be avoided. The distance of the micro strip line to the ground plane on the bottom side of the receiver is very small (some 100  $\mu\text{m}$ ) and has huge tolerances (up to 100%). Therefore, the impedance of this part of the trace cannot be controlled.
- Use as many vias as possible to connect the ground planes.
- In order to avoid reliability hazards, the area on the PCB under the receiver should be entirely covered with solder mask. Vias should not be open.

### 2.7.5 Antenna Micro Strip

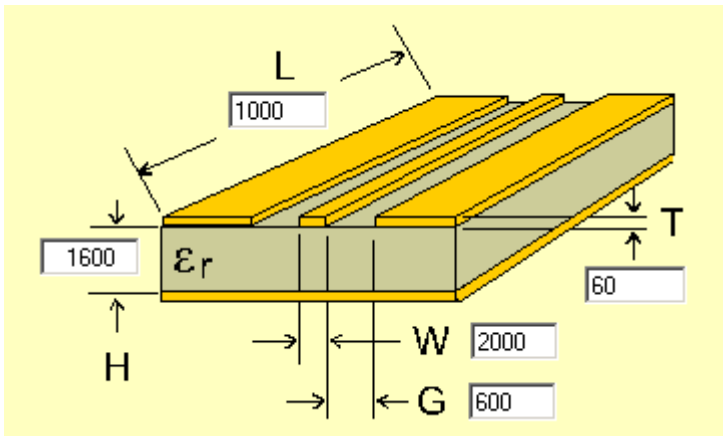
There are many ways to design wave-guides on printed circuit boards. Common to all is that calculation of the electrical parameters is not straightforward. Freeware tools like AppCAD from Agilent or TXLine from Applied Wave Research, Inc. are of great help. They can be downloaded from [www.agilent.com](http://www.agilent.com) and [www.mwoffice.com](http://www.mwoffice.com).

The micro strip is the most common configuration for printed circuit boards. The basic configuration is shown in *Figure 13* and *Figure 14*. As a rule of thumb, for a FR-4 material the width of the conductor is roughly double the thickness of the dielectric to achieve 50 Ohms line impedance.

For the correct calculation of the micro strip impedance, one does not only need to consider the distance between the top and the first inner layer but also the distance between the micro strip and the adjacent GND plane on the same layer

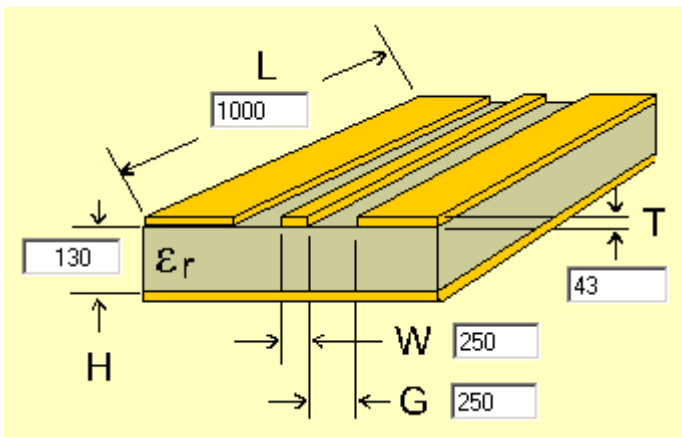


Use the Coplanar Waveguide model for the calculation of the micro strip.



**Figure 13: Micro strip on a 2-layer board (Agilent AppCAD Coplanar Waveguide)**

Figure 13 shows an example of a 2-layer FR4 board of 1.6 mm thickness and a 35µm (1 ounce) copper cladding. The thickness of the micro strip is comprised of the cladding (35µm) plus the plated copper (typically 25µm). Figure 14 depicts an example of a multi layer FR4 board with 18µm (½ ounce) cladding and 180µ dielectric between layer 1 and 2.



**Figure 14: Micro strip on a multi layer board (Agilent AppCAD Coplanar Waveguide)**

## 2.8 Antenna and Antenna Supervisor

u-blox 5 modules receive L1 band signals from GPS and GALILEO satellites at a nominal frequency of 1575.42 MHz. The RF signal is connected to the **RF\_IN** pin.

u-blox 5 modules can be connected to passive or active antennas.



For u-blox 5 receivers, the total preamplifier gain (minus cable and interconnect losses) must not exceed 50 dB. Total noise figure should be below 3 dB.

### 2.8.1 Passive Antenna

A design using a passive antenna requires more attention regarding the layout of the RF section. Typically a passive antenna is located near electronic components; therefore care should be taken to reduce electrical 'noise' that may interfere with the antenna performance. Passive antennas do not require a DC bias voltage and can be directly connected to the RF input pin **RF\_IN**. Sometimes, they may also need a passive matching network to match the impedance to 50 Ohms.





Some passive antenna designs present a DC short to the RF input, when connected. If a system is designed with antenna bias supply AND there is a chance of a passive antenna being connected to the design, consider a short circuit protection.



All u-blox 5 receivers have a built-in LNA required for passive antennas.

## **2.8.2 Active Antenna**

NEO-5 modules do not provide the antenna bias voltage for active antennas at the RF\_IN pin. See Section 2.5.2 for more information.

## **2.8.3 Active Antenna Supervisor**

NEO-5 modules do not support the Antenna Supervisor.

### 3 Product Handling



All NEO-5 modules are RoHS compliant (lead-free).

#### 3.1 Packaging

NEO-5 modules are delivered as hermetically sealed, reeled tapes in order to enable efficient production, production lot set-up and tear-down.



Figure 15: Reeled u-blox 5 Modules

##### 3.1.1 Reels

NEO-5 modules for GPS and GALILEO are deliverable in quantities of 250pcs on a reel. The dimensions of the reel are shown in Figure 16.

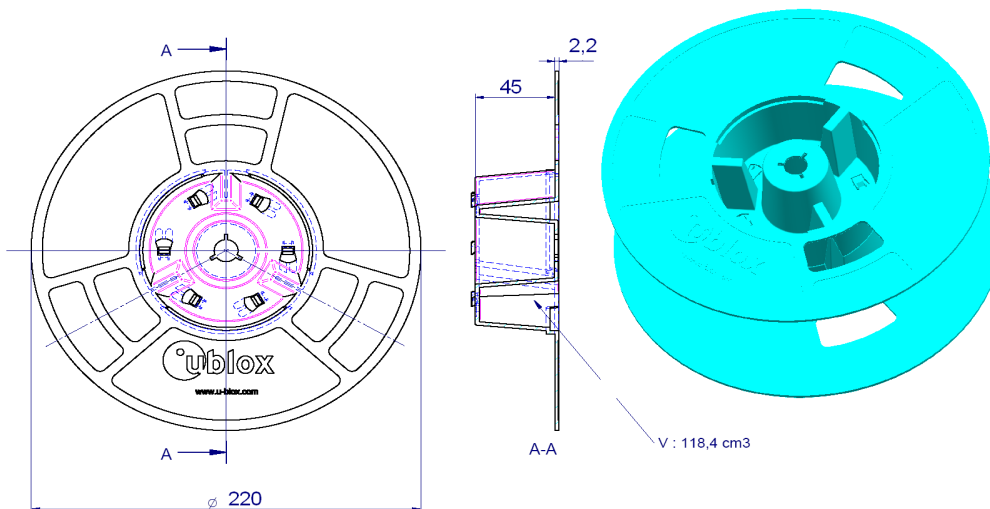


Figure 16: Dimension of reel for 250 pieces (dimensions unless otherwise specified in mm)

### 3.1.2 Tapes

The dimensions and orientations of the tapes for NEO 5 modules are specified in Figure 17.

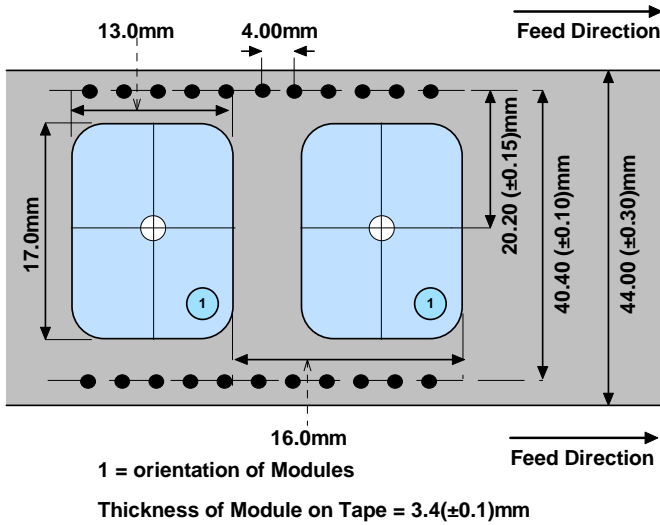


Figure 17: Dimensions and orientation for NEO-5 modules on tape

## 3.2 Shipment, Storage and Handling

### 3.2.1 Handling

u-blox 5 modules are designed and packaged to be processed in an automatic assembly line, and are shipped in Tape-and-Reel.



**These components contain highly sensitive electronic circuitry. Handling the NEO-5 modules without proper ESD protection may destroy or damage them permanently.**



**According to JEDEC ISP, NEO-5 modules are moisture sensitive devices. Appropriate handling instructions and precautions are summarized in Sections 3.2.2 to 3.2.5. Read them carefully to prevent permanent damages due to moisture intake.**

### 3.2.2 Shipment

The NEO-5 modules are delivered on Tape-and-Reels in a hermetically sealed package ("dry bag") to prevent moisture intake and protect against electrostatic discharge. For protection from physical damage, the reels are individually packed in cartons.

The dry bag provides a JEDEC compliant MSD label (Moisture Sensitive Devices) describing the handling requirements to prevent humidity intake.

	<b>CAUTION</b> This bag contains <b>MOISTURE-SENSITIVE DEVICES</b>	LEVEL <div style="border: 1px solid black; padding: 2px; display: inline-block;"> <b>4</b> </div>
	<ol style="list-style-type: none"> <li>1. Calculated shelf life in sealed bag: 12 months at &lt;40°C and &lt;90% relative humidity (RH)</li> <li>2. Peak package body temperature: <b>260</b> °C</li> <li>3. After this bag is opened, devices that will be subjected to reflow solder or other high temperature process must be             <ol style="list-style-type: none"> <li>a) Mounted within <b>72</b> hours of factory conditions &lt;30°C / 60% RH, or</li> <li>b) Stored at &lt;10% RH</li> </ol> </li> <li>4. Devices require baking, before mounting, if:             <ol style="list-style-type: none"> <li>a) Humidity Indicator Card is &gt;10% when read at 23° ± 5°C, or</li> <li>b) 3a or 3b not met</li> </ol> </li> <li>5. If baking is required, devices may be baked for 48 hours at 125° ± 5°C</li> </ol> <p>Note: If device containers cannot be subjected to high temperature or shorter bake times are desired, reference IPC/JEDEC J-STD-033 for bake procedure.</p> <p>Bag Seal Date: <b>01.01.2008</b></p> <p>Note: Level and body temperature defined by IPC/JEDEC J-STD-020</p>	

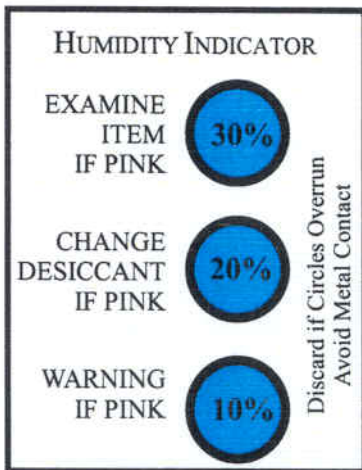
**Figure 18: Applicable MSD Label (See Section 3.1 for baking instructions)**

### 3.2.3 Storage

Shelf life in sealed bag is 12 months at <40°C and <90% relative humidity.

### 3.2.4 Handling

A humidity indicator card and a desiccant bag to absorb humidity are enclosed in the sealed package. The parts are shipped on tape-and-reel in a hermetically sealed package. If no moisture has been absorbed, the three fields in the humidity indicator card indicate blue color.



**Figure 19: Humidity Indicator Card, good condition**

### 3.2.5 Floor Life

For products with moisture sensitivity level 4, the floor life is 72 hours, or precisely three days. Under factory floor temperature and humidity conditions (<30°C, <60% relative humidity), the parts must be processed and soldered within this specified period of time.

Once the sealed package of the reel is opened and the parts exposed to humidity, they need to be processed within 72 hours (precisely three days) in a reflow soldering process. If this time is exceeded, or the sticker in the sealed package indicates that the goods have been exposed to moisture, the devices need to be pre-baked before the flow solder process. Please refer to *Section 3.3* for instructions on how to pre-bake the components.

## 3.3 Processing

### 3.3.1 Moisture Preconditioning

Both encapsulant and substrate materials absorb moisture. JEDEC specification J-STD-020 must be observed to prevent cracking and delamination associated with the "popcorn" effect during solder reflow. The popcorn effect can be described as miniature explosions of evaporating moisture. Baking before processing is required in following cases:

- Humidity indicator card: At least one circular indicator is no longer blue
- Floor life or environmental requirements after opening the seal is opened has been exceeded, e.g. exposure to excessive seasonal humidity.

#### **Recommended baking procedure:**

Duration: 48 hours

Temperature: 125°C

Humidity: Below 5%. Desiccant must be placed into the oven to keep humidity low.

Oven: Convection flow oven. Also put desiccant pack into the oven for dehydration.

After work: Put the baked components with desiccant and moisture indicator into a humidity proof bag and use a vacuum hot barrier sealing machine for sealing if not processed within specified floor time. Storage in a nitrogen cabinet or dry box is also a possible approach to prevent moisture intake.



**Do not attempt to bake the NEO-5 modules contained in tape and rolled up in reels. If necessary, bake the NEO-5 modules quickly at 125°C for 48 hours, remove them from the belt and place them individually onto the oven tray.**



A repeated baking process will reduce the wetting effectiveness of the pad contacts. This applies to all SMT devices.

### 3.3.2 Soldering Paste

Use of "No Clean" soldering paste is strongly recommended, as it does not require cleaning after the soldering process has taken place. The paste listed in the example below meets these criteria.

Soldering Paste: LFSOLDER TLF-206-93F (Tamura Kaken (UK) Ltd.)

Alloy specification: Sn 95.5/ Ag 3.9/ Cu 0.6 (95.5% Zinc/ 0.6 % Silver/ 0.6% Copper)

Melting Temperature: 216 - 221°C

Stencil Thickness: 150 µm for base boards

The final choice of the soldering paste depends on the approved manufacturing procedures.

The paste-mask geometry for applying soldering paste should meet the recommendations in section 2.7.2



The quality of the solder joints on the connectors ('half vias') should meet the appropriate IPC specification.

### 3.3.3 Reflow Soldering

**A convection type-soldering oven is strongly recommended** over the infrared type radiation oven. Convection heated ovens allow precise control of the temperature and all parts will be heated up evenly, regardless of material properties, thickness of components and surface color.

Consider the "IPC-7530 Guidelines for temperature profiling for mass soldering (reflow and wave) processes, published 2001".

#### Preheat Phase

Initial heating of component leads and balls. Residual humidity will be dried out. Please note that this preheat phase will not replace prior baking procedures.

- Temperature rise rate: 1 - 4°C/s      If the temperature rise is too rapid in the preheat phase it may cause excessive slumping.
- Time: 60 – 120 seconds      If the preheat is insufficient, rather large solder balls tend to be generated. Conversely, if performed excessively, fine balls and large balls will be generated in clusters.
- End Temperature: 150 - 200°C      If the temperature is too low, non-melting tends to be caused in areas containing large heat capacity.

#### Heating/ Reflow Phase

The temperature rises above the liquidus temperature of 216 - 221°C. Avoid a sudden rise in temperature as the slump of the paste could become worse.

- Limit time above 220°C liquidus temperature: 20 - 40s
- Peak reflow temperature: 230 - 250°C

#### Cooling Phase

A controlled cooling avoids negative metallurgical effects (solder becomes more brittle) of the solder and possible mechanical tensions in the products. Controlled cooling helps to achieve bright solder fillets with a good shape and low contact angle.

- Temperature fall rate: max 3°C / s



To avoid falling off, the u-blox 5 GPS/GALILEO module should be placed on the topside of the motherboard during soldering.

The final soldering temperature chosen at the factory depends on additional external factors like choice of soldering paste, size, thickness and properties of the base board, etc. Exceeding the maximum soldering temperature in the recommended soldering profile may permanently damage the module.

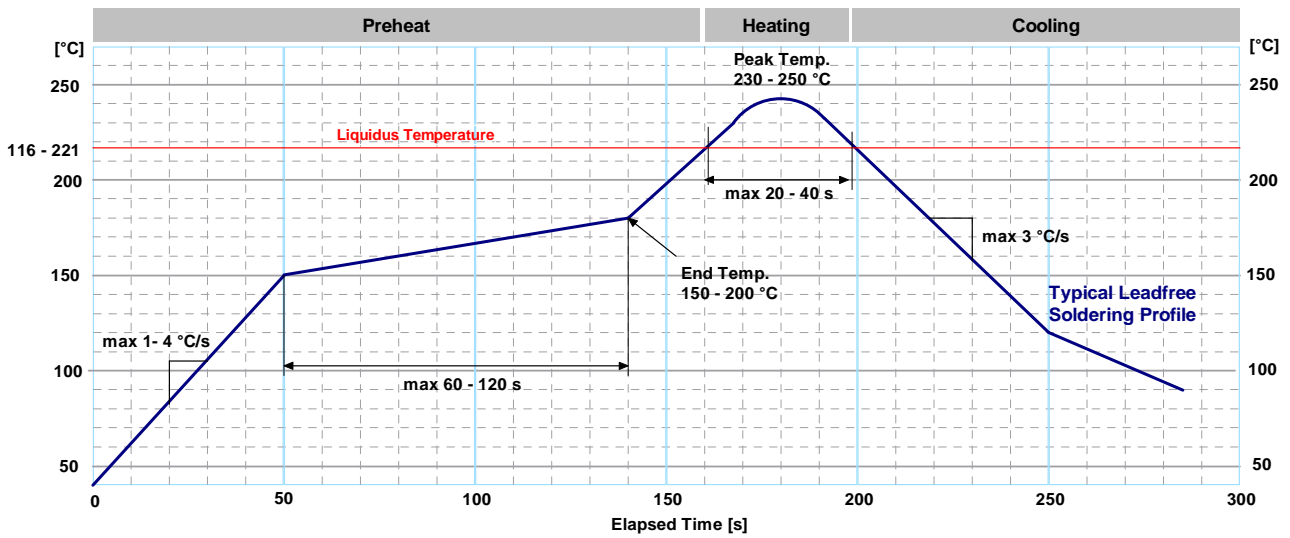


Figure 20: Recommended soldering profile



When soldering leadfree (u-blox 5) modules in a leaded process, check the following temperatures:

- PB- Technology Soaktime: 40-80sec
- Time above Liquidus: 40-90 sec
- Peak temperature: 225-235 °C



NEO-5 modules must not be soldered with a damp heat process.

### 3.3.4 Optical Inspection

After soldering the NEO-5 module, consider an optical inspection step to check whether:

- The module is properly aligned and centered over the pads
- All pads are properly soldered
- No excess solder has created contacts to neighboring pads, or possibly to pad stacks and vias nearby.



### 3.3.5 Cleaning

In general, cleaning the populated modules is strongly discouraged. Residues underneath the modules cannot be easily removed with a washing process.

- Cleaning with water will lead to capillary effects where water is absorbed in the gap between the baseboard and the module. The combination of residues of soldering flux and encapsulated water leads to short circuits or resistor-like interconnections between neighboring pads. Water will also damage the sticker and the ink-jet printed text.
- Cleaning with alcohol or other organic solvents can result in soldering flux residues flooding into the two housings, areas that are not accessible for post-wash inspections. The solvent will also damage the sticker and the ink-jet printed text.
- Ultrasonic cleaning will permanently damage the module, in particular the quartz oscillators.

The best approach is to use a "no clean" soldering paste and eliminate the cleaning step after the soldering.

### 3.3.6 Repeated Reflow Soldering

Only a single reflow soldering process is encouraged for boards with a NEO-5 module populated on it. The reason for this is the risk of the module falling off due to high weight in relation to the adhesive properties of the solder.

### 3.3.7 Wave Soldering

Base boards with combined through-hole technology (THT) components and surface-mount technology (SMT) devices require wave soldering to solder the THT components. Only a single wave soldering process is encouraged for boards populated with NEO-5 modules.

### 3.3.8 Hand Soldering

Hand soldering is allowed. Use a soldering iron temperature setting of "7" which is equivalent to 350°C and carry out the hand soldering according to the IPC recommendations / reference documents IPC7711.

Place the module precisely on the pads. Start with a cross-diagonal fixture soldering (e.g. pins 1 and 16), and then continue from left to right.

### 3.3.9 Rework

The NEO-5 module can be unsoldered from the baseboard using a hot air gun.



**Avoid overheating the module.**

After the module is removed, clean the pads before placing and hand-soldering a new module.



**Never attempt a rework on the module itself, e.g. replacing individual components. Such actions immediately terminate the warranty.**

### 3.3.10 Conformal Coating

Certain applications employ a conformal coating of the PCB using HumiSeal® or other related coating products. These materials affect the HF properties of the GPS module and it is important to prevent them from flowing into the module.

The RF shields do not provide 100% protection for the module from coating liquids with low viscosity, therefore care is required in applying the coating.



Conformal Coating of the module will void the warranty.

### 3.3.11 Casting

If casting is required, use viscose or another type of silicon pottant. The OEM is strongly advised to qualify such processes in combination with the NEO-5 module before implementing this in the production.



Casting will void the warranty.

### 3.3.12 Grounding Metal Covers

Attempts to improve grounding by soldering ground cables, wick or other forms of metal strips directly onto the EMI covers is done at the customer's own risk. The numerous ground pins should be sufficient to provide optimum immunity to interferences and noise.



u-blox makes no warranty for damages to the NEO-5 module caused by soldering metal cables or any other forms of metal strips directly onto the EMI covers.

### 3.3.13 Use of Ultrasonic Processes

Some components on the NEO-5 module are sensitive to Ultrasonic Waves. Use of any Ultrasonic Processes (cleaning, welding etc.) may cause damage to the GPS Receiver.



u-blox offers no warranty against damages to the NEO-5 module caused by any Ultrasonic Processes.

## 4 Product Testing

### 4.1 u-blox In-Series Production Test

u-blox focuses on high quality for its products. To achieve a high standard it's our philosophy to supply fully tested units. Therefore at the end of the production process, every unit is tested. Defective units are analyzed in detail to improve the production quality.

This is achieved with automatic test equipment, which delivers a detailed test report for each unit. The following measurements are done:

- Digital self-test (Software Download, verification of FLASH firmware, etc.)
- Measurement of voltages and currents
- Measurement of RF characteristics (e.g. C/No)

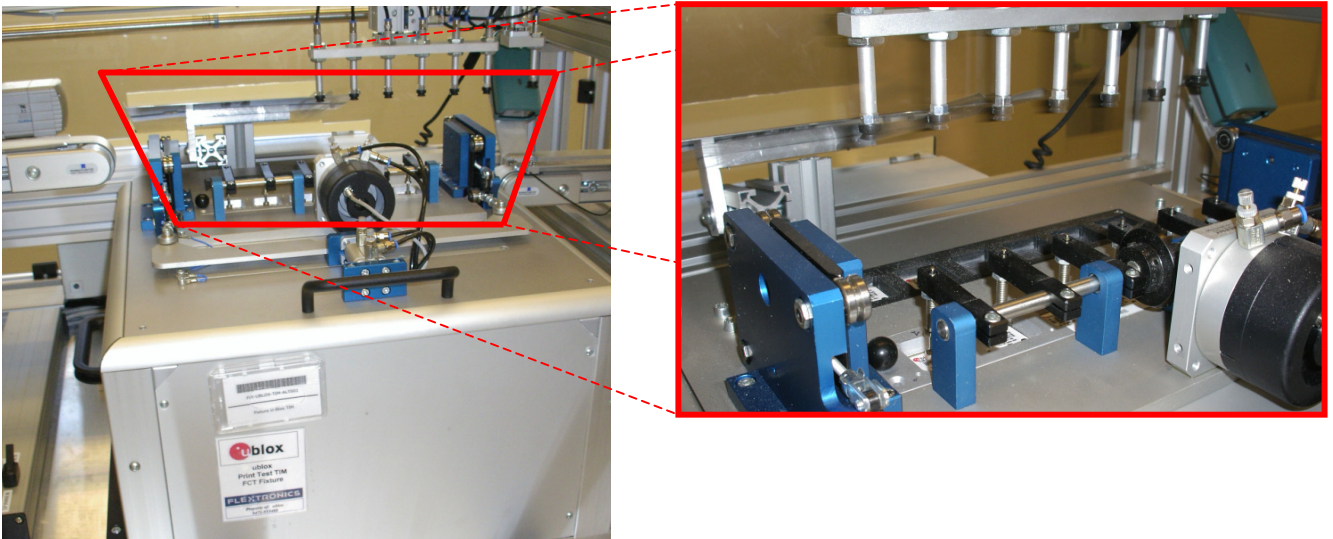


Figure 21: Automatic Test Equipment for Module Tests

### 4.2 Test Parameters for OEM Manufacturer

Because of the testing done by u-blox (with 100% coverage), it is obvious that an OEM manufacturer doesn't need to repeat firmware tests or measurements of the GPS parameters/characteristics (e.g. TTFF) in their production test.

An OEM Manufacturer should focus on

- Overall sensitivity of the device (including antenna, if applicable)
- Communication to a host controller

## 4.3 System Sensitivity Test

The best way to test the sensitivity of a GPS device is with the use of a 1-channel GPS simulator. It assures reliable and constant signals at every measurement.



**Figure 22: 1-channel GPS simulator**

u-blox recommends the following Single-Channel GPS Simulator:

- Spirent GSS6100  
Spirent Communications Positioning Technology  
(previously GSS Global Simulation Systems)

[www.positioningtechnology.co.uk](http://www.positioningtechnology.co.uk)

### 4.3.1 Guidelines for Sensitivity Tests

1. Connect a 1-channel GPS simulator to the OEM product
2. Choose the power level in a way that the “Golden Device” would report a C/No ratio of 38-40 dBHz
3. Power up the DUT (Device Under Test) and allow enough time for the acquisition
4. Read the C/No value from the NMEA GSV or the UBX-NAV-SVININFO message (e.g. with u-center AE)
5. Compare the results to a “Golden Device” or a u-blox 5 Evaluation Kit.

### 4.3.2 ‘Go/No go’ tests for integrated devices

The best test is to bring the device to an outdoor position **with excellent sky view** (HDOP < 3.0). Let the receiver acquire satellites and compare the signal strength with a “Golden Device”.



As the electro-magnetic field of a redistribution antenna is not homogenous, indoor tests are in most cases not reliable. These kind of tests may be useful as a ‘go/no go’ test but not for sensitivity measurements.

# A Migration to u-blox 5 receivers

Migrating ANTARIS®4 to a u-blox 5 GPS/GALILEO receiver is a straightforward procedure. Nevertheless there are some points to be considered during the migration.



Not all of the functionalities available with ANTARIS®4 are supported by u-blox 5. These include:

- FixNow Mode
- Low Power Modes
- RTCM
- UTM

## A.1 Migration from NEO-4S to NEO-5

The pin-outs of NEO-4S and NEO-5M/NEO-5Q differ slightly. Table 5 compares the modules and highlights the differences to be considered.

Pin	NEO-4S		NEO-5		Remarks for Migration
	Pin Name	Typ. Assignment	Pin Name	Typ. Assignment	
1	<b>BOOT_INT</b>	NC	<b>Reserved</b>	NC	do not drive low.
2	<b>SELECT</b>	VDDIO level I/O; not connected	<b>NC/ SS_N</b>	NC	NEO-5M: NC NEO-5Q: SS_N
3	<b>TIMEPULSE</b>	VDDIO level I/O	<b>TIMEPULSE</b>	Output	
4	<b>EXTINT0</b>	NC	<b>EXTINT0</b>	NC	
5	<b>USB_DM</b>	NC	<b>USB_DM</b>	NC	
6	<b>USB_DP</b>	NC	<b>USB_DP</b>	NC	
7	<b>VDDUSB</b>	Connected to GND or VDD_USB	<b>VDDUSB</b>	Connected to GND or VDD_USB	Do not leave open. (VDD_USB is 3.3V regulated power supply from VBUS.)
8	<b>Reserved</b>	NC	<b>Reserved</b>	NC	Pins 8 and 9 must be connected.
9	<b>VCC_RF</b>	VCC-0.1V	<b>VCC_RF</b>	VCC-0.1V	No difference
10	<b>GND</b>	GND	<b>GND</b>	GND	No difference
11	<b>RF_IN</b>	RF_IN	<b>RF_IN</b>	RF_IN	No difference
12	<b>GND</b>	GND	<b>GND</b>	GND	No difference
13	<b>GND</b>	GND	<b>GND</b>	GND	No difference
14	<b>MOSI</b>	NC	<b>MOSI/CFG_COM0</b>	NC	
15	<b>MISO</b>	NC	<b>MISO/NC</b>	NC	
16	<b>SCK/ CFG_USB</b>	RF_IN	<b>SCK/CFG_GPS0/ NC</b>	NC	Leave open if not used.
17	<b>NCS</b>	NC	<b>SCS1_N/ NC</b>	NC	No difference
18	<b>Reserved</b>	NC	<b>SDA2</b>	NC	
19	<b>Reserved</b>	NC	<b>SCL2</b>	NC	
20	<b>TXD1</b>	VDDIO level I/O	<b>TxD1</b>	Output	
21	<b>RXD1</b>	VDDIO level I/O	<b>RxD1</b>	Input	Leave open if not used.
22	<b>V_BAT</b>	1.5-3.6V	<b>V_BCKP</b>	1.4-4.8V	Wider voltage range but needs more current. Check your backup supply, regarding the higher consumption.
23	<b>VCC</b>	2.7-3.3V	<b>VCC</b>	2.7-3.6V	Higher peak supply current
24	<b>GND</b>	GND	<b>GND</b>	GND	No difference

 : Pins to be checked carefully; NC: Not connected

**Table 5: Pin-out comparison NEO-4S vs. NEO-5**

## B Antennas

Even the best receiver cannot bring back what has been lost at the antenna. The importance of the attention paid to this part of a GPS/GALILEO system cannot be stated highly enough.

### B.1 Selecting the right Antenna

Several different antenna designs are available on the GPS/GALILEO applications market. The GPS/GALILEO L1 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 foremost antennas for GPS/GALILEO application designs are the patch antennas as shown in Figure 23.



Figure 23: Patch Antennas, EMTAC Technology Corp.

Another style is the quadrifilar helix antenna shown in Figure 24. 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 much smaller form factor. The smaller the dimensions of the antenna, the more performance critical tight manufacturing tolerances become. Furthermore, a smaller antenna will present a smaller aperture to collect the signal energy from sky resulting in a lower overall gain of the antenna. This is the 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.

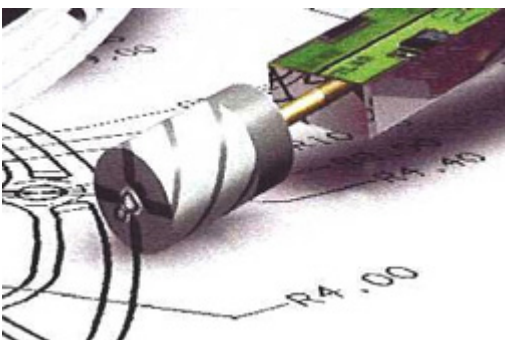


Figure 24: Quadrifilar Helix Antenna, Sarantel, Ltd.

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 applications such as surveying or timing, some very high caliber antenna systems exist. Common to these designs are large size, high power consumption and high cost. These designs are highly optimized to suppress multi-path signals reflected from 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/GALILEO 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 these types of applications receivers with multiple antenna inputs are often required.

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, a minimum of 3 dB of signal to noise ratio will be lost.

## B.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 impedance.

Active antennas have an integrated low-noise amplifier. This is beneficial in two respects. Firstly, the losses of the cable no longer affect the overall noise figure of the GPS/GALILEO receiver system. Secondly, the receiver noise figure can be much higher without sacrificing performance. Therefore, some receivers will only work with active antennas. Active antennas require a power supply that contributes to total GPS/GALILEO system power consumption, typically in the region of 5 to 20mA. 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.

The use of an active antenna is always advisable if the RF-cable length between receiver and antenna exceeds approximately 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, 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 26 dB for use with u-blox receivers. With shorter cables and a 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. A low antenna gain cannot be compensated by high LNA gain. If a manufacturer provides one total gain figure, this is not sufficient to judge the quality of the antenna. One would need information on antenna gain (in dBic), amplifier gain, and amplifier noise figure.

## B.3 Patch Antennas

Patch antennas are ideal for an application where the antenna sits on a flat surface, e.g. the roof of a car. Patch antennas can demonstrate a very high gain, especially if they are mounted on top of a large ground plane. Ceramic patch antennas are very popular because of their small size, typically measuring 25 x 25 mm<sup>2</sup> down to 12x12mm<sup>2</sup>. Very cheap construction techniques might use ordinary circuit board material like FR-4 or even air as a dielectric, but this will result in a much larger size, typically in the order of 10 x 10 cm<sup>2</sup>. Figure 25 shows a typical example of the radiation pattern of a 16 x 16 mm<sup>2</sup> ceramic patch antenna. This measurement only shows the upper sphere of the radiation pattern. Depending on ground plane size there will also be a prominent back lobe present.

### Directivity (YZ) - Ground plane size

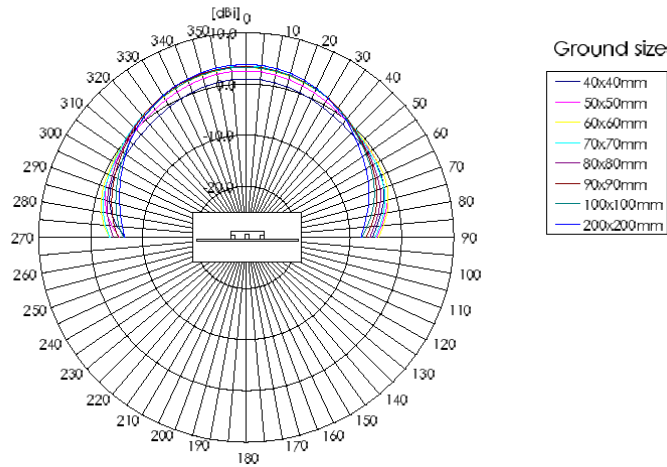


Figure 25: Typical Radiation Pattern of a Patch Antenna, MuRata, Inc.

For the specific example shown in Figure 26 one can easily see that the so-called axial ratio, the relation of major to minor axis of the elliptical polarization has a minimum at the 50 mm<sup>2</sup> square ground plane. At this point, the polarization of the antenna is closest to an ideal circular polarization (axial ratio = 0 dB). At a 100 mm<sup>2</sup> square ground plane size this particular patch shows an axial ratio in the order of 10 dB, which is closer to linear polarization than to circular and will result in respective losses. This effect can also be seen in the left graph of the figure, where gain no longer increases with increasing ground plane size. In conclusion, the correct dimensions for the size of the ground plane can serve as a useful compromise between maximum gain and reasonable polarization loss.

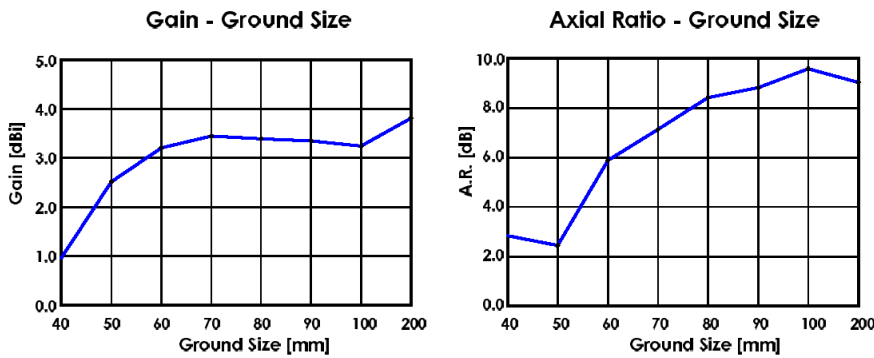


Figure 26: Typical Gain and Axial Ratio of a Patch antenna with respect to ground plane size, MuRata, Inc.

A good allowance for ground plane size is typically in the area of 50 to 70 mm<sup>2</sup>. This number is largely independent of the size of the patch itself (when considering 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 this effect becomes.

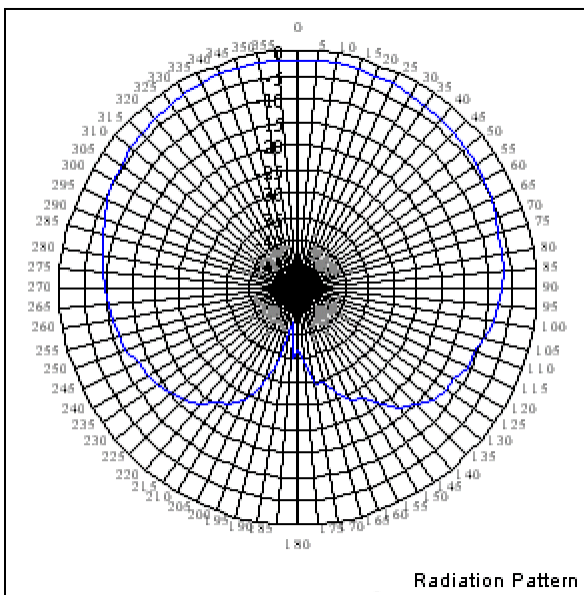
Smaller sized patches will usually reach their maximum gain with a slightly smaller ground plane compared to a larger size patch. However, the maximum gain of a small sized patch with optimum ground plane may still be much lower than the gain of a large size patch on a less than optimal ground plane.



It is not only gain and axial ratio of the patch antenna that is affected by the size of the ground plane but also the matching of the antenna to the 50 Ohms impedance of the receiver. See *Appendix B.6* for more information on matching.

## B.4 Helix Antennas

Helix antennas can be designed for use with or without ground plane. For example, the radiating elements on board the GPS and GALILEO satellites have a ground plane. Using an array of helix antennas, the GPS and GALILEO 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 27.



**Figure 27: Radiation pattern of helix antenna without ground plane, Sarantel, Ltd.**

Although we can determine an axial ratio close to 9 dB between zero degree and 90 degrees 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, the back lobe of a patch antenna depends very much on size and shape of the ground plane. As with patch antennas, filling the antenna with a material with a high dielectric constant can reduce the size of helix antennas. Sizes in the order of 18 mm length and 10 mm diameter are being offered on the market. Again, antenna gain will decrease with decreased size.

## B.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 rooftop applications. Others prefer the pole like style of the helix antenna, which is quite similar to the style of antennas used in mobile phones. Furthermore, it is important that the antenna's main lobe points to the sky in order to receive as many satellites as possible with maximum gain. If the application is a hand held device, the antenna should be designed in such 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 typically 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 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. Therefore, the receivers can see more satellites but the navigation solution will be degraded because of distorted range measurements in a multi-path environment.

If possible test the actual performance of different antenna types in a real life environment before starting the mechanical design of the GPS/GALILEO enabled product.

## B.6 Antenna Matching

All common GPS/GALILEO 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 no longer presents a 50 Ohms source impedance. Typically what happens is that the center frequency of the antenna is shifted away from GPS/GALILEO frequency - usually towards lower frequencies – by some external influence. The reasons for this effect are primarily disturbances in the near field of the antenna. This can either be a ground plane, that does not have the size for which the antenna was designed , 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 any specified environment. So one must do either extensive simulation or experimental work. Usually, antenna manufacturers offer a selection of pre-tuned antennas, so the user can test and select the version that best fits the given environment. However, testing equipment such as 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. Also the antenna bandwidth will decrease with decreasing antenna size, making it harder to achieve optimum tuning.

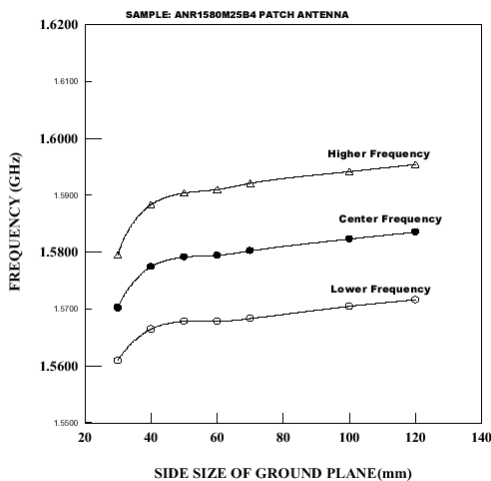


Figure 28: Dependency of center frequency on ground plane dimension for a 25 x 25 mm<sup>2</sup> patch, EMTAC

A LNA placed very close to the antenna can help to relieve 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. Under these conditions the matching of the input to the LNA becomes more important. Within a reasonable mismatch range, integrated LNAs can show a gain decrease in the order of a few dBs versus an increase of noise figure in the order of several tenths of a dB. 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 indeed beneficial if the antenna cable between the antenna and the receiver is only short. In this case, there’s no need

for the gain of the LNA to exceed 10-15 dB. In this environment the sole purpose of the LNA is to provide impedance matching and not signal amplification.

## B.7 Antenna Placement

The location where the antenna is mounted is crucial for optimal performance of the GPS/GALILEO receiver. When using patch antennas, the antenna plane should be parallel to the geographic horizon. The antenna must have full view of the sky ensuring a direct line-of-sight with as many visible satellites as possible.

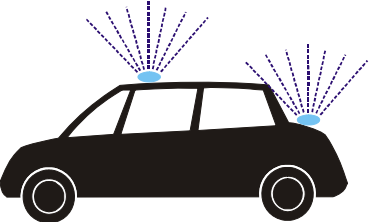

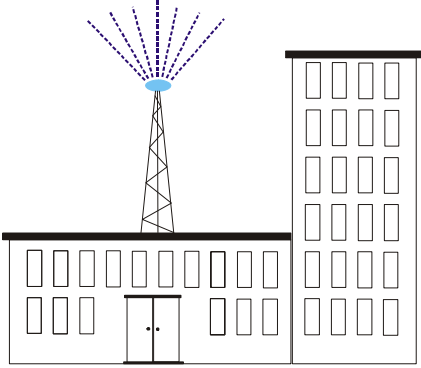
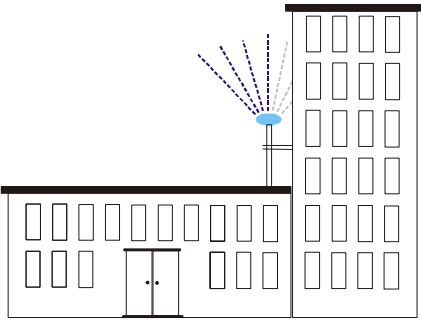
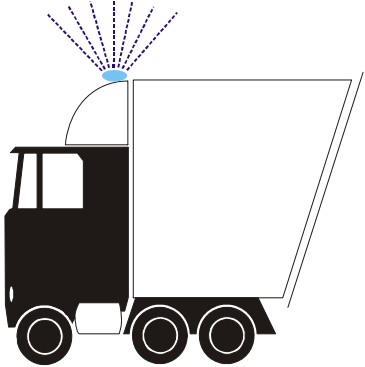
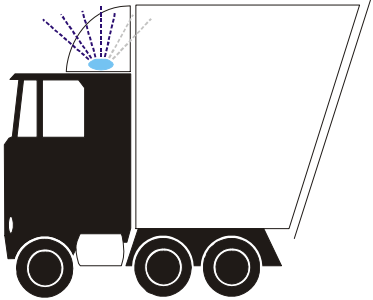
1 <sup>st</sup> Choice Placement	2 <sup>nd</sup> Choice Placement
Recommended Antenna positions	Performance may be degraded! If recommended placements are not available, these may also viable.
	 <p><b>Note:</b> Window and roof reduce GPS signal and obstruct sky view<sup>2</sup></p>
	 <p><b>Note:</b> There may be multipath signals and a obstructed sky view</p>
	 <p><b>Note:</b> Fiberglass airfoil attenuates the GPS/GALILEO signal</p>

Table 6: Optimal antenna placement

<sup>2</sup> Some cars have a metallic coating on the windscreens. GPS/GALILEO reception may not be possible in such a car without the use of SuperSense<sup>®</sup> Technology. There is usually a small section, typically behind the rear view mirror, reserved for mobile phone and GPS/GALILEO antennas.

## C Interference Issues

A typical GPS/GALILEO receiver has a very low dynamic range. This is because the antenna should only detect thermal noise in the GPS/GALILEO frequency band, given that the peak power of the GPS/GALILEO signal is 15 dB below the thermal noise floor. 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 AGC's 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/GALILEO 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 the jamming signal, suppressing the GPS/GALILEO signal buried in thermal noise even further. Depending on the filter characteristics of the antenna and the front end of the GPS/GALILEO 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/GALILEO signal frequency. We can conclude that a jamming signal exceeding thermal noise floor within a reasonable bandwidth (e.g. 100 MHz) around GPS/GALILEO signal frequency will degrade the performance significantly.

Even out-of-band signals can affect GPS/GALILEO 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 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 particularly difficult scenario is the transmitting antenna of a DCS handset (max. 30 dBm at 1710 MHz) in close proximity to the GPS/GALILEO antenna. When integrating GPS/GALILEO 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 improve the situation without significant 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 document and might even conflict with export restrictions for dual-use goods.

The recommendations and concepts in this section are completely dependent on the specific applications. In situations 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 tightly integrated, the following sections should be read very carefully.

### C.1 Sources of Noise

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

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

The first problem can be very difficult to solve, but if GPS/GALILEO and RF transmitter are to be integrated close to each other, there's a good chance that there is an engineer at hand who knows the specifications of the RF transmitter. In most cases, counter measures such as filters will be required for the transmitter to limit disruptive emissions below the noise floor near the GPS/GALILEO frequency.

Even if the transmitter is quiet in the GPS/GALILEO band, a very strong emission close to it 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 the GPS/GALILEO antenna and receiver input will help to reduce signal levels to the level of linear operation at the front-end.

The second problem is more common but also regularly proves to be hard to solve. Here, the emitting source is not well specified and the emission can be of broadband nature, making specific countermeasures very difficult. Moreover, the GPS/GALILEO 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 severely disturb a GPS receiver.

If the GPS/GALILEO antenna is to be placed very close to some other electronics, e.g. the GPS/GALILEO receiver itself or a PDA-like appliance, the EMC issue must 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 given frequency band.

## C.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. The following sections contain some general hints on how to decrease the level of noise emitted from digital circuit board that are potentially in close proximity to the GPS receiver or the antenna.

### C.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 29). Furthermore, free space on the outermost Layers can be filled with ground shapes connected to the ground plane to shield radiation from internal layers.

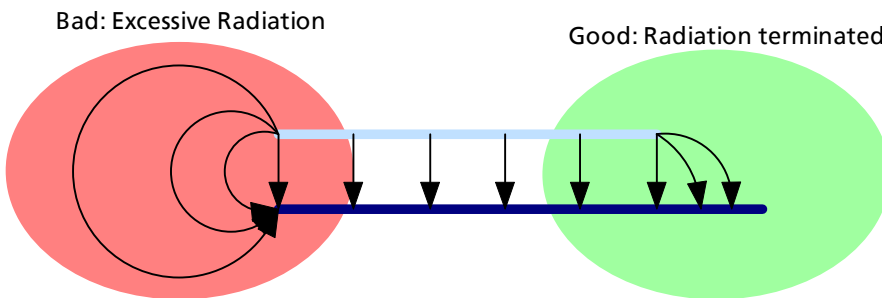


Figure 29: Signal and power plane extends should lie within ground plane extends

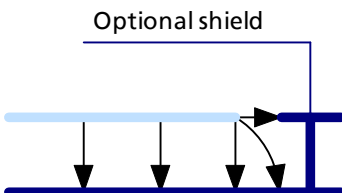


Figure 30: Further improvement of reduction of power plane radiation

## C.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, the use of line drivers with controlled signal rise-time is suggested whenever driving large bus systems. Alternatively, high-speed signal lines can be terminated with resistors or even active terminations to reduce high frequency radiation 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 31).

Bad: Excessive Radiation      Good: Radiation terminated



Figure 31: Terminating radiation of signal lines

## C.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. In addition to resonance, the capacitor will also behave like an inductor. If many capacitors are connected in parallel, total inductance will decrease while total capacitance will increase. Figure 32 shows the impedance dependence of SMD capacitors.

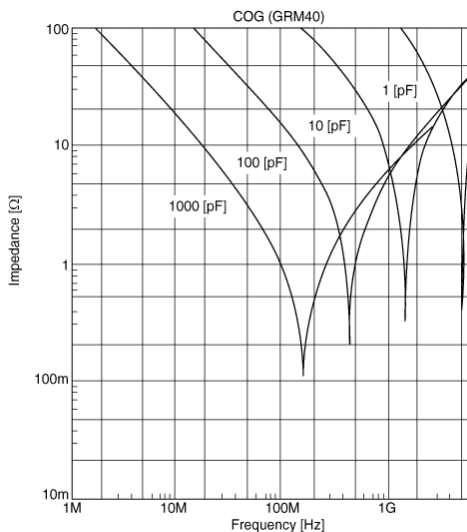


Figure 32: Impedance of 0805 size SMD capacitors vs. frequency, MuRata

If the 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 the ground. Furthermore, ceramic capacitors come with different dielectric materials. These materials show different temperature behavior. For industrial temperature range applications, at least a X5R quality should be selected. Y5V or Z5U types may lose almost all of their capacitance at extreme 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.

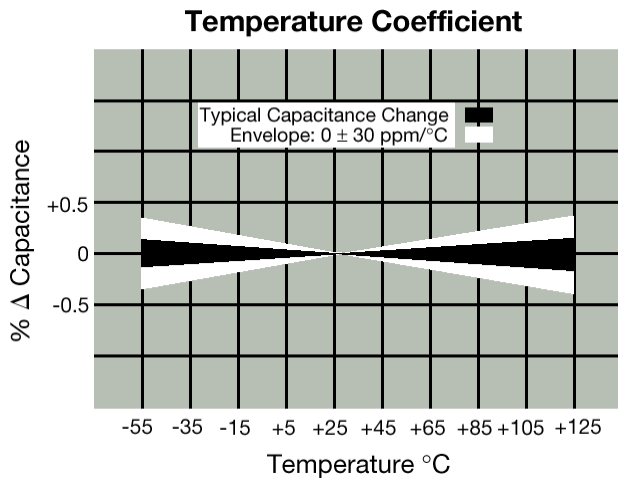


Figure 33: Temperature dependency of COG/NPO dielectric, AVX

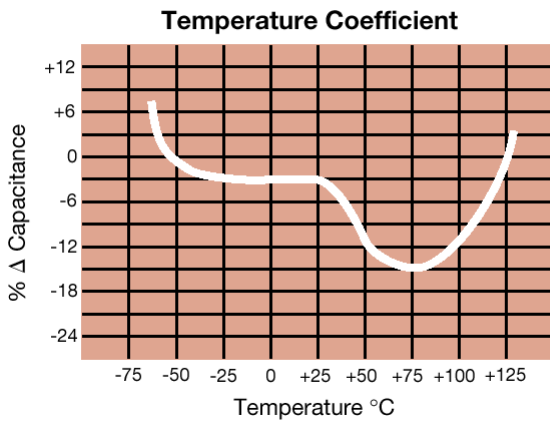


Figure 34: Temperature dependency of X7R dielectric, AVX

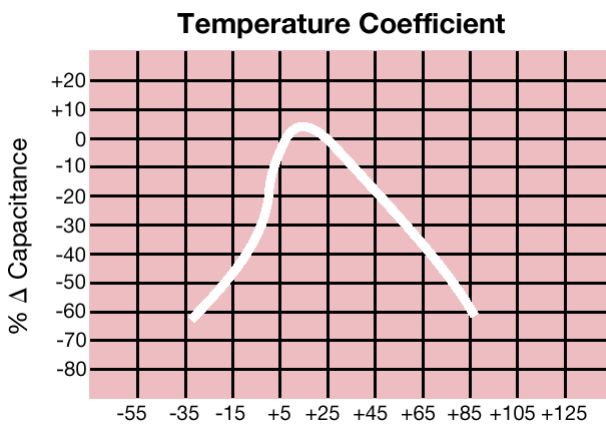


Figure 35: Temperature dependency of Y5V dielectric, AVX

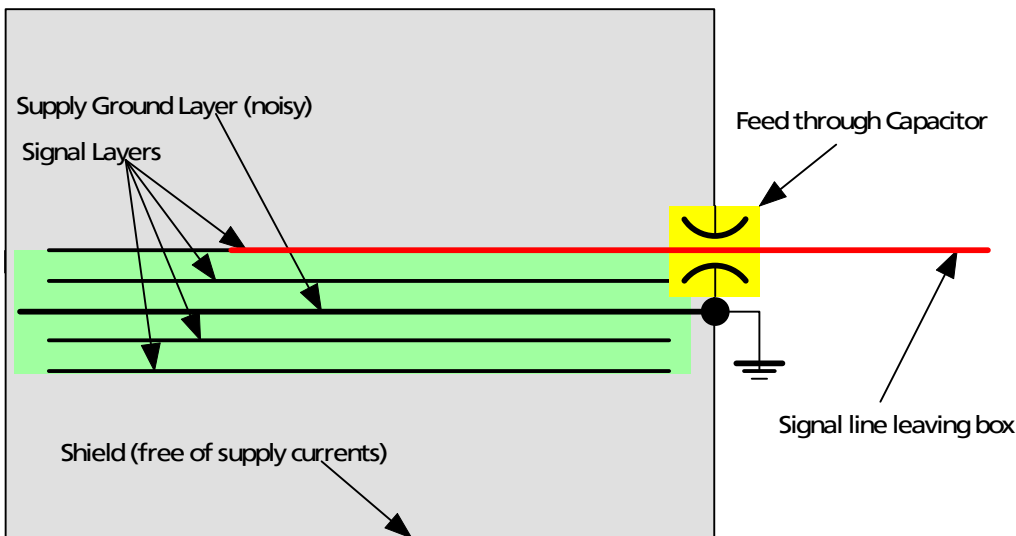
### C.3 Shielding

If employing the countermeasures listed in *Appendix C.2* cannot solve EMI problems, the solution may be shielding of the noise source. In the real world, shields are 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 values can be even 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.

#### C.3.1 Feed through Capacitors

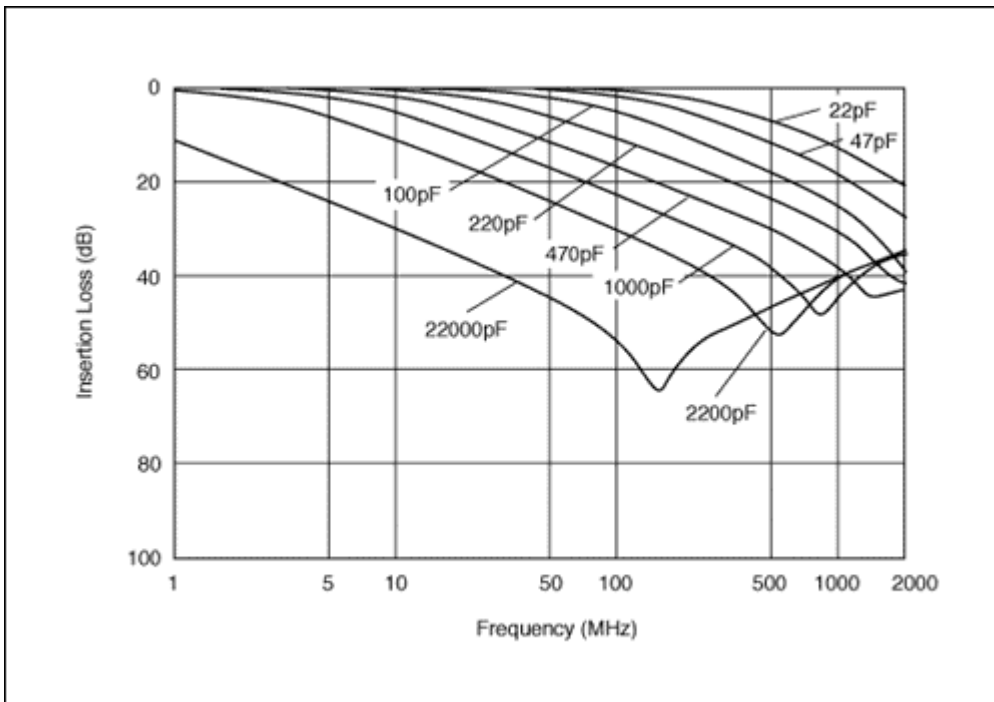
The basic concept of shielding is that a metal box will terminate all electrical fields on its surface. In practice we have the problem that we need to route some signals from inside to outside of this box.



**Figure 36: Ideal shielding**

The proposed setup for such a system is shown in Figure 36. A feed through capacitor removes all high frequency content from the outgoing signal line. It's important to note that any conductor projecting 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 37). For the GPS/GALILEO frequency band the 470pF capacitor is the optimum choice of the Murata NFM21C series.





**Figure 37: MuRata’s NFM21C Feed Through Capacitors**

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

If the use of a special feed through capacitor is not feasible for a particular design, a simple capacitor between the 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 the GPS/GALILEO frequency range. Larger capacitance values will be less efficient.

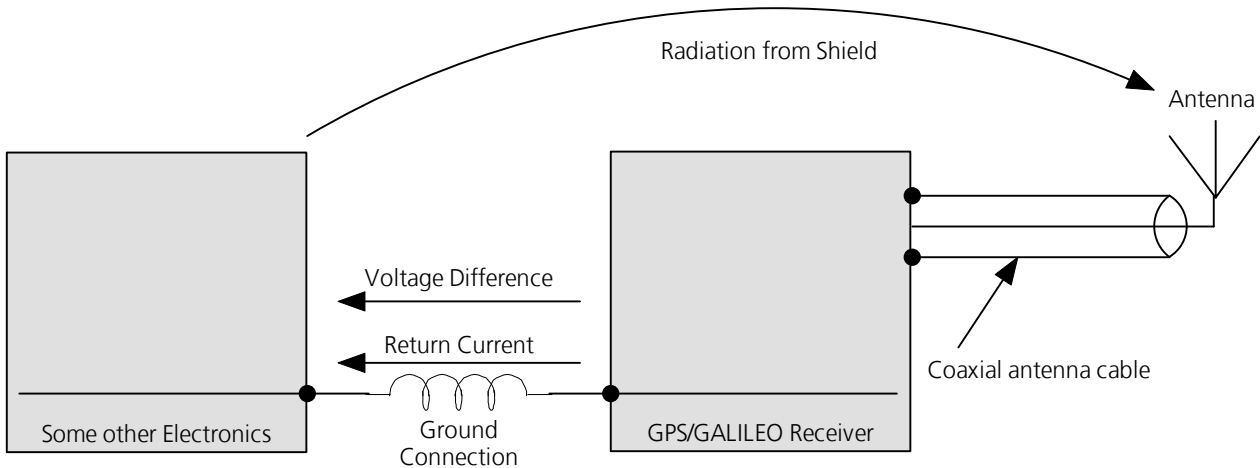
One should keep in mind that a feed through capacitor is basically a high frequency “short” between the signal line and ground. If the ground point that the capacitor is connected to is not ideal, meaning the ground connection or plane has a finite resistance, noise will be injected into the ground net. Therefore, one should try to place any feed through capacitor far away from the most noise sensitive parts of the circuit. To emphasize 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 the MuRata web page for extensive discussion on EMC countermeasures.

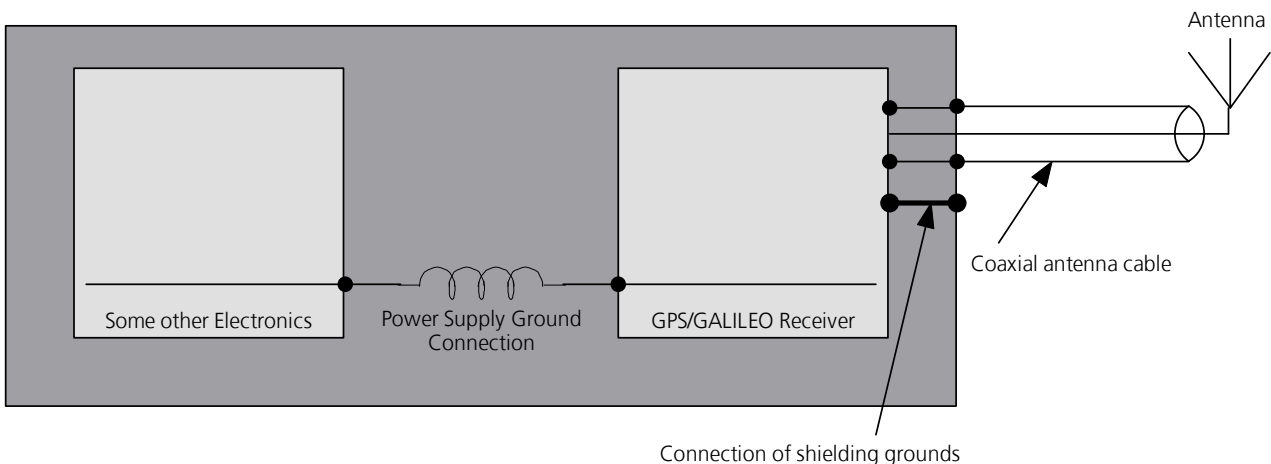
### C.3.2 Shielding Sets of Sub-System Assembly

Yet another problem arises if multiple building blocks are combined in a single system. Figure 38 shows a 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/GALILEO receiver and the attached antenna.



**Figure 38: 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 39. Again, it is important to have a good (i.e. low inductance) interconnection between the outer shield and the shielding ground of the GPS/GALILEO receiver.



**Figure 39: Proper shielding of a sub-system assembly**

It is clear that the situation illustrated in Figure 39 can become complex if the component “Some other electronics” contains another wireless transmitter system with 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 a highly inductive nature.

# D Lists

## D.1 List of Figures

Figure 1: NEO-5 Block Diagram .....	9
Figure 2: Backup Battery and Voltage .....	10
Figure 3: USB Interface .....	12
Figure 4: Connecting Module as DDC Slave .....	13
Figure 5: Recommended wiring for active antennas .....	15
Figure 6: Recommended layout for connecting the antenna bias voltage for NEO-5 .....	15
Figure 7: Passive Antenna Design for NEO-5 Receivers .....	16
Figure 8: Recommended footprint .....	19
Figure 9: Recommendations for copper, solder and paste masks with enlargement .....	20
Figure 10: Placement .....	21
Figure 11: Recommended layout .....	22
Figure 12: PCB build-up for Micro strip line. Left: 2-layer PCB, right: 4-layer PCB .....	22
Figure 13: Micro strip on a 2-layer board (Agilent AppCAD Coplanar Waveguide) .....	24
Figure 14: Micro strip on a multi layer board (Agilent AppCAD Coplanar Waveguide) .....	24
Figure 15: Reeled u-blox 5 Modules .....	26
Figure 16: Dimension of reel for 250 pieces (dimensions unless otherwise specified in mm) .....	26
Figure 17: Dimensions and orientation for NEO-5 modules on tape .....	27
Figure 18: Applicable MSD Label (See Section 3.1 for baking instructions) .....	28
Figure 19: Humidity Indicator Card, good condition .....	29
Figure 20: Recommended soldering profile .....	32
Figure 21: Automatic Test Equipment for Module Tests .....	35
Figure 22: 1-channel GPS simulator .....	36
Figure 23: Patch Antennas, EMTAC Technology Corp. ....	38
Figure 24: Quadrifilar Helix Antenna, Sarantel, Ltd. ....	38
Figure 25: Typical Radiation Pattern of a Patch Antenna, MuRata, Inc. ....	40
Figure 26: Typical Gain and Axial Ratio of a Patch antenna with respect to ground plane size, MuRata, Inc. ....	40
Figure 27: Radiation pattern of helix antenna without ground plane, Sarantel, Ltd. ....	41
Figure 28: Dependency of center frequency on ground plane dimension for a 25 x 25 mm <sup>2</sup> patch, EMTAC .....	42
Figure 29: Signal and power plane extends should lie within ground plane extends .....	45
Figure 30: Further improvement of reduction of power plane radiation .....	45
Figure 31: Terminating radiation of signal lines .....	46
Figure 32: Impedance of 0805 size SMD capacitors vs. frequency, MuRata .....	46
Figure 33: Temperature dependency of COG/NPO dielectric, AVX .....	47
Figure 34: Temperature dependency of X7R dielectric, AVX .....	47
Figure 35: Temperature dependency of Y5V dielectric, AVX .....	47
Figure 36: Ideal shielding .....	48
Figure 37: MuRata's NFM21C Feed Through Capacitors .....	49
Figure 38: Two shielded sub-systems, connected by a "poor" ground .....	50

Figure 39: Proper shielding of a sub-system assembly ..... 50

## D.2 List of Tables

Table 1: Features of the NEO-5 Series ..... 8  
Table 2: USB availability on LEA-5 modules ..... 12  
Table 3: Summary of USB external components ..... 12  
Table 4: Pinout NEO-5 ..... 17  
Table 5: Pin-out comparison NEO-4S vs. NEO-5 ..... 37  
Table 6: Optimal antenna placement ..... 43

## E Glossary

API	Application Programming Interface
BBR	Battery backup RAM
ECEF	Earth Centered Earth Fixed
ESD	Electro Static Discharge
HAE	Height Above WGS84-Ellipsoid
LNA	Low Noise Amplifier
LOS	Line of sight,
NMEA 0183	ASCII based standard data communication protocol used by GPS receivers.
PUBX	u-blox proprietary extension to the NMEA protocol
PVT	Position Velocity Time
SA	Selective Availability
SV	Satellite Vehicle
SBAS	Satellite Based Augmentation Systems
UBX	File extension for u-center log file or short form for the UBX protocol
UBX Protocol	A proprietary binary protocol used by the ANTARIS™ GPS technology

## Related Documents

- [1] GNSS Compendium, Doc No GPS-X-02007
- [2] NEO-5 Data Sheet, Doc No GPS.G4-MS5-07025
- [3] u-blox 5 Protocol Specification, Doc No GPS.G5-X-07063

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



*your position is our focus*

## Contact

For further info, please contact us:

### Headquarters

#### u-blox AG

Zuercherstrasse 68  
CH-8800 Thalwil  
Switzerland

Phone: +41 44 722 74 44  
Fax: +41 44 722 74 47  
E-mail: [info@u-blox.com](mailto:info@u-blox.com)

[www.u-blox.com](http://www.u-blox.com)

### Offices

#### North, Central and South America

##### u-blox America, Inc.

1902 Campus Commons Drive  
Suite 310  
Reston, VA 20191  
USA

Phone: +1 (703) 483 3180  
Fax: +1 (703) 483 3179  
E-mail: [info\\_us@u-blox.com](mailto:info_us@u-blox.com)

##### Regional Office West Coast:

8600 Lemon Ave #1  
La Mesa, CA 91941  
USA

Phone: +1 (619) 741 3011  
Fax: +1 (619) 741 4334  
E-mail: [info\\_us@u-blox.com](mailto:info_us@u-blox.com)

##### Technical Support:

Phone: +1 (703) 483 3185  
E-mail: [support\\_us@u-blox.com](mailto:support_us@u-blox.com)

#### Europe, Middle East, Africa

##### u-blox AG

Zuercherstrasse 68  
CH-8800 Thalwil  
Switzerland

Phone: +41 44 722 74 44  
Fax: +41 44 722 74 47  
E-mail: [info@u-blox.com](mailto:info@u-blox.com)

##### Technical Support:

Phone: +41 44 722 74 74  
E-mail: [support@u-blox.com](mailto:support@u-blox.com)

#### Asia, Australia, Pacific

##### u-blox Singapore Pte. Ltd.

435 Orchard Road  
#17-01, Wisma Atria,  
Singapore 238877

Phone: +65 6734 3811  
Fax: +65 6736 1533  
E-mail: [info\\_ap@u-blox.com](mailto:info_ap@u-blox.com)  
Support: [support\\_ap@u-blox.com](mailto:support_ap@u-blox.com)

##### Regional Office China:

Room 716-718  
No. 65 Fuxing Road  
Beijing, 100036, China

Phone: +86 10 68 133 545  
Fax: +86 10 68 217 890  
E-mail: [info\\_cn@u-blox.com](mailto:info_cn@u-blox.com)  
Support: [support\\_cn@u-blox.com](mailto:support_cn@u-blox.com)

##### Regional Office Japan:

22F Shibuya Mark City West,  
1-12-1 Dogenzaka Shibuya-ku  
Tokyo 150-0043 Japan

Phone: +81 3 4360 5343  
Fax: +81 3 4360 5301  
E-mail: [info\\_jp@u-blox.com](mailto:info_jp@u-blox.com)  
Support: +81 3 4360 5344  
[support\\_jp@u-blox.com](mailto:support_jp@u-blox.com)

##### Regional Office Korea:

Room 501, Gyeong Hui Building 109-18,  
Samseong-Dong,  
GangNam-Gu,  
Seoul, Korea 135-090

Phone: +82 2 542 0861  
Fax: +82 2 542 0862  
E-mail: [info\\_kr@u-blox.com](mailto:info_kr@u-blox.com)  
Support: [support\\_kr@u-blox.com](mailto:support_kr@u-blox.com)

##### Regional Office Taiwan:

Room 305  
3F, #181, ZouTze Street  
Neihu Dis.  
Taipei, Taiwan

Phone: +886 2 2657 1090  
Fax: +886 2 2657 1097  
E-mail: [info\\_tw@u-blox.com](mailto:info_tw@u-blox.com)  
Support: [support\\_tw@u-blox.com](mailto:support_tw@u-blox.com)