

MAX-M5Q

GPS/GNSS modules

Hardware Integration Manual

Abstract

This document describes the features and specifications of the MAX-M5Q in MAX form factor with Mediatek hybrid/parallel GPS/GGLONASS receiver. This stand-alone module is compact and easy-to-integrate, and provides an easy upgrade transition to the future MAX-8 modules.



Document Information	
Title	MAX-M5Q
Subtitle	GPS/GNSS modules
Document type	Hardware Integration Manual
Document number	FTX-HW-13008
Document status	Objective Specification

Document status information	
Objective Specification	This document contains target values. Revised and supplementary data will be published later.
Advance Information	This document contains data based on early testing. Revised and supplementary data will be published later.
Preliminary	This document contains data from product verification. Revised and supplementary data may be published later.
Released	This document contains the final product specification.

This document applies to the following products:

Name	Type number	ROM/FLASH version	PCN reference
MAX-M5Q-0	All	Flash FW320G-UBX.3135	

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.

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 © 2013, u-blox AG.

u-blox® is a registered trademark of u-blox Holding AG in the EU and other countries. ARM® is the registered trademark of ARM Limited in the EU and other countries.



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, Doc No GPS-X-02007 [1]:** This document, also known as the GPS book, provides a wealth of information regarding generic questions about GPS system functionalities and technology.
- **MAX-M5Q Receiver Description including Protocol Specification, Docu. No FTX-SW-13001 [2]:** Messages, configuration and functionalities of the MAX-M5Q software releases and positioning modules are explained in this document.
- **Hardware Integration Manuals:** These manuals provide hardware design instructions and information on how to set up production and final product tests.
- **Application Notes:** These documents provide general design instructions and information that applies to all u-blox GPS/GNSS positioning modules.
- **Related documents** for a list of Application Notes related to your GPS/GNSS receiver.

How to use this Manual

The MAX-M5Q Hardware Integration Manual provides the necessary information to successfully design-in and configure MT3333-based positioning modules. This manual has a modular structure. It is not necessary to read it from the beginning to the end.

The following symbols 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 influence or damage the module.

Questions

If you have any questions about MAX-M5Q 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) 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 ensures that we process your request as soon as possible. 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-7N-0-000), Datacode (e.g. 172100.0100.000) and firmware version (e.g. ROM1.0)
- 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
Contents.....	5
1 Quick reference	8
2 Hardware description	8
2.1 Overview	8
2.1.1 MAX-M5Q	8
2.2 Architecture.....	8
2.2.1 MAX-M5Q Architecture	8
2.3 Power management operating modes	9
2.3.1 Continuous mode	9
2.3.2 Power Save modes	9
2.3.3 Backup State	10
2.4 Configuration	11
2.4.1 Configuration MAX-M5Q	11
2.5 Connecting power.....	11
2.5.1 VCC: Main Supply Voltage	11
2.5.2 V_BCKP: Backup Supply Voltage	11
2.5.3 VCC_IO	11
2.5.4 VCC_RF: Output Voltage RF section	11
2.6 Interfaces.....	12
2.6.1 UART.....	12
2.7 I/O pins.....	12
2.7.1 Antenna input.....	12
2.7.2 RESET_N: Reset input	12
2.7.3 ANT_ON: Active Antenna control (LNA enable)	12
2.7.4 TIMEPULSE	12
3 Design	13
3.1 Design checklist	13
3.1.1 Schematic checklist	13
3.1.2 Layout checklist	14
3.1.3 Antenna checklist.....	14
3.2 Design considerations for minimal designs.....	14
3.2.1 Minimal design.....	15
3.3 Layout	16
3.3.1 Footprint and paste mask	16
4.1.1 Placement	16
4.1.2 Antenna connection and ground plane design	17
4.1.3 General design recommendations:	18

4.1.4	Antenna micro strip.....	19
4.2	Antenna and Antenna supervision	20
4.2.1	Antenna design with passive antenna	20
4.2.2	Active antenna design not using antenna supervisor	21
4.2.3	Design with GLONASS / GPS active antenna	23
4.2.4	Design with GLONASS / GPS passive antenna	24
4.3	Recommended parts.....	25
4.3.1	Recommended GPS & GLONASS active patch antenna (A1)	26
4.3.2	Recommended GPS & GLONASS passive patch antenna	26
4.3.3	Recommended GPS & GLONASS passive chip antenna	26
5	Migration	27
5.1	Migrating from u-blox 7 designs to a MAX-M5Q module	27
5.1.1	Software compatibility.....	27
5.2	Forward compatibility from MAX-M5Q designs to a MAX-8x module	27
5.2.1	Software compatibility.....	27
6	Product handling	28
6.1	Packaging, shipping, storage and moisture preconditioning	28
6.1.1	Population of Modules	28
6.2	Soldering	28
6.2.1	Soldering paste.....	28
6.2.2	Reflow soldering	28
6.2.3	Optical inspection.....	29
6.2.4	Cleaning.....	29
6.2.5	Repeated reflow soldering.....	30
6.2.6	Wave soldering.....	30
6.2.7	Hand soldering.....	30
6.2.8	Rework.....	30
6.2.9	Conformal coating	30
6.2.10	Casting.....	30
6.2.11	Grounding metal covers	31
6.2.12	Use of ultrasonic processes.....	31
6.3	EOS/ESD/EMI precautions	31
6.3.1	Electrostatic discharge (ESD)	31
6.3.2	ESD handling precautions.....	31
6.3.3	ESD protection measures.....	32
6.3.4	Electrical Overstress (EOS).....	33
6.3.5	EOS protection measures.....	33
6.3.6	Electromagnetic interference (EMI)	33
6.3.7	Applications with wireless modules LEON / LISA	34
7	Product testing	36
7.1	u-blox in-series production test.....	36
7.2	Test parameters for OEM manufacturer	36

7.3	System sensitivity test	37
7.3.1	Guidelines for sensitivity tests	37
7.3.2	'Go/No go' tests for integrated devices.....	37
Appendix	38
A Abbreviations	38
Related documents	39
Revision history	39
Contact	40

1 Quick reference

When using this manual for a design, make sure you also have the Data Sheet for the specific positioning module (see the related documents section).

2 Hardware description

2.1 Overview

2.1.1 MAX-M5Q

The u-blox MAX-M5Q is an OEM GNSS receiver module based on the Mediatek MT3333 chip that supports All-in-One GNSS hybrid navigation. The u-blox MAX-M5Q receiver provides extremely low power and very fast TTFF together with weak signal acquisition and tracking capability to meet even the most stringent performance expectations in hybrid navigation, using signals from both GPS + GLONASS GNSS systems.



For product features see MAX-M5Q Data Sheet, Docu. No FTX-HW-13003 [1].

2.2 Architecture

2.2.1 MAX-M5Q Architecture

The MAX-M5Q is the newest standalone GPS/GNSS module from u-blox. With the exceptional utilization of GPS and GLONASS in parallel, MAX-M5Q can enhance the position availability in harsh GNSS satellite visibility conditions or at high latitudes, e.g. in the polar regions of Russia. It is characterized by a small footprint, low power consumption, and sophisticated interference suppression, ensuring maximum performance even in GPS-hostile environments. This makes incorporating MAX-M5Q into customer designs simple and straightforward.

Receiver type: MT3333 engine 99/33-channel (search/track)

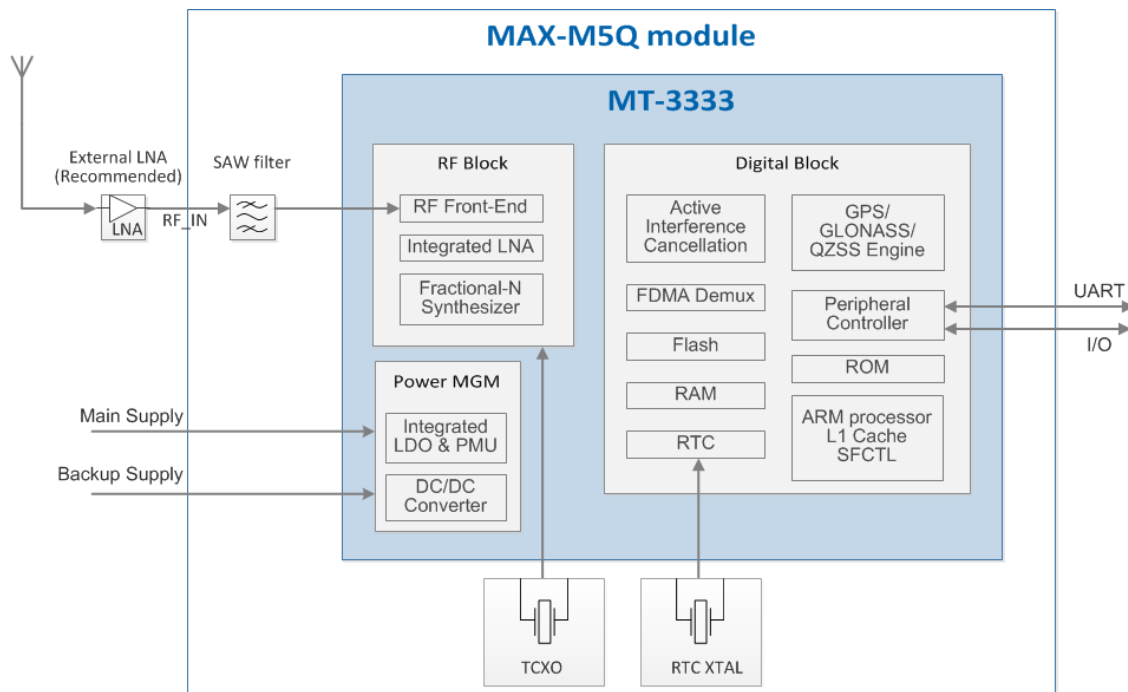


Figure 1: MAX-M5Q Block diagram

2.3 Power management operating modes

u-blox MAX-M5Q technology offers a power-optimized architecture with built-in autonomous power saving functions to minimize power consumption at any given time. The receiver operates in two modes: Continuous mode for best performance or Power Save mode for optimized power consumption. In addition, an integrated high efficiency DC/DC converter allows low power consumption even for higher main supply voltages.

MAX-M5Q modules have the following operating modes:

- Continuous mode for best GPS/GNSS performance
- Power Save modes to optimize power consumption
 - Standby mode
 - Periodic mode
 - AlwaysLocate™ mode
- Backup state

2.3.1 Continuous mode

Continuous mode uses the acquisition engine at full performance resulting in the shortest possible TTFF and the highest sensitivity. It searches for all possible satellites until the almanac is completely downloaded. The receiver switches to the tracking engine to lower the power consumption when:

- A valid GPS/GNSS position is obtained
- The entire almanac has been downloaded
- The ephemeris for each satellite in view is valid

2.3.2 Power Save modes

For power sensitive applications, the MAX-M5Q module also supports low-power operating modes for reduced power consumption by using the embedded power switch. For more information about the power management command, see [2]. Use caution when implementing Backup, Periodic or to AlwaysLocate™ mode since forward compatibility is not guaranteed.

2.3.2.1 Standby mode

In this mode, the receiver stops navigation, the internal processor enters standby state, and the current drain at main supply **VCC** is typically reduced to 0.4 mA. Enter standby mode by sending the NMEA command: \$PMTK161,0*28. The host can wake up the module from Standby mode to Full Power mode by sending any byte via the host port.

2.3.2.2 Periodic mode

This mode allows autonomous power on/off with reduced fix rate to reduce average power consumption, as shown in Figure 2 below. The main power supply **VCC** is still active, but turning the supply on and off is controlled internally by NMEA commands. Enter periodic mode by sending the following NMEA command:

`$PMTK225,<Type>,<Run_time>,<Sleep_time>,<2nd_run_time>,<2nd_sleep_time>*<checksum>`

where:

- Type=1 for Periodic Backup mode
- Run_time = Full Power period (ms)
- Sleep_time = Standby/Backup period (ms)
- 2nd_run_time = Full Power period (ms) for extended acquisition if GNSS acquisition fails during Run_time
- 2nd_sleep_time = Standby/Backup period (ms) for extended sleep if GNSS acquisition fails during Run_time

Example: \$PMTK225,1,3000,12000,18000,72000*16 for periodic mode with 3 s navigation and 12 s sleep in backup state. The acknowledgement response for the command is \$PMTK001,225,3*35. The module can exit Periodic mode by sending the command \$PMTK225,0*2B just after the module wakes up from a previous sleep cycle.

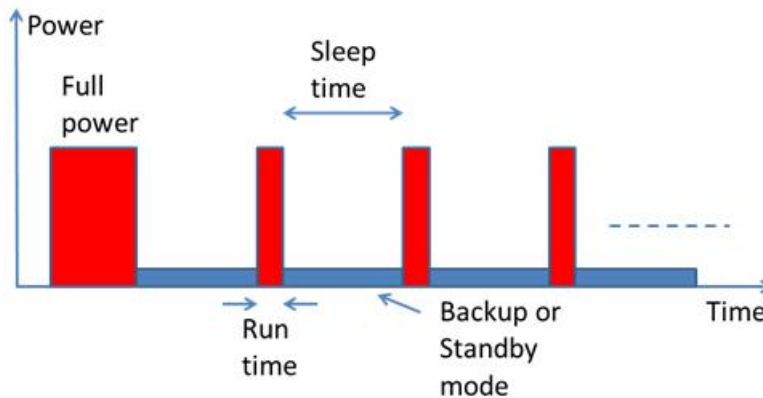


Figure 2: Periodic Mode

2.3.2.3 AlwaysLocate mode

AlwaysLocate™ is an intelligent controller of the Periodic mode; the main power supply **VCC** is still active, but supply is controlled internally by NMEA commands. Depending on the environment and motion conditions, the module can autonomously and adaptively adjust the parameters of the Periodic mode, e.g. on/off ratio and fix rate to achieve a balance in positioning accuracy and power consumption, see figure below. The average power drain can vary based on conditions; typical average power is 7 mW. Associated profiles are: High and Low Speed, Walking, Outdoor Static and Indoor.

Enter AlwaysLocate™ mode by sending the following NMEA command:

```
$PMTK225,<mode>*<checksum><CR><LF>
```

where mode=9 for AlwaysLocate™ in Backup mode. Example: \$PMTK225,9*22. The acknowledgement response for the command is \$PMTK001,225,3*35. The module can exit AlwaysLocate™ mode by sending the command \$PMTK225,0*2B just after the module wakes up from a previous sleep cycle.

The module can control the embedded **VCC** power switch autonomously only after the MAX-M5Q is set to Periodic or to AlwaysLocate™ mode by an NMEA command.

Note that first fix position accuracy can be somewhat degraded in Power Management modes when compared to Full Power operation. The user can improve the position accuracy by taking the second or third fix after waking up.

The user can exit low power modes to Full Power by sending NMEA command \$PMTK225,0*2B just after the module wakes up from a previous sleep cycle. For more information about power management command, see [2].

2.3.3 Backup State

Backup State means a low quiescent (typically 10 μ A at **V_BCKP**) power state where receiver operation is stopped; only the backup supply **V_BCKP** is powered on while the main supply **VCC** is switched off by the host (or autonomously by MAX-M5Q in Periodic mode and AlwaysLocate™ mode). Waking up from Backup State to Full Power is controlled by the host by switching on the **VCC** supply.

After waking up, the receiver uses all internal aiding, including GNSS time, Ephemeris, and Last Position, resulting in the fastest possible TTFF in either hot or warm start modes.

During Backup State, the I/O block is powered off. The suggestion is that the host forces its outputs to a low state or to a high-Z state during the Backup State to minimize small leakage currents (<10 μ A typically) at receiver's input signals.

2.4 Configuration

2.4.1 Configuration MAX-M5Q

2.4.1.1 UART MAX-M5Q

UART Port 0 is normally used for GNSS data reports and receiver control. To configure serial data rates from 4,800 baud to 921,600 baud, use the command:

```
$PMTK251,<baud>*<checksum><CR><LF>
```

Ensure that the message payload fits in selected baud rate. Default baud rate is 115,200 baud; protocol is NMEA. RX signal is pulled up internally and can be left floating (not connected) when not used.



Use caution when implementing MAX-M5Q configuration since forward compatibility is not guaranteed.

2.5 Connecting power

The MAX-M5Q has three power supply pins: **VCC**, **V_BCKP** and **VCC_IO**.

2.5.1 VCC: Main Supply Voltage

The **VCC** pin provides the main supply voltage. During operation, the current drawn by the module can vary by some orders of magnitude, especially if enabling low-power operation modes. For this reason, it is important that the supply circuitry be able to support the peak power for a short time (see datasheet for specification).



When switching from backup mode to normal operation or at startup, MAX-M5Q modules must charge the internal capacitors in the core domain. In certain situations, this can result in a significant current draw. For low power applications using Power Save and backup modes it is important that the power supply or low ESR capacitors at the module input can deliver this current/charge.



Use a proper GND concept. Do not use any resistors or coils in the power line. For ground plane design see section 4.1.2

2.5.2 V_BCKP: Backup Supply Voltage

In case of a power failure on the module supply, **V_BCKP** supplies power to the real-time clock (RTC) and battery backed RAM (BBR). Use of valid time and the GPS/GNSS orbit data at startup will improve the GPS/GNSS performance, i.e. enables hot and warm starts. If no backup battery is connected, the module performs a cold start at power up.



If no backup supply voltage is available, connect the **V_BCKP** pin to **VCC**.



V_BCKP provides supply all the time to backup memory & RTC. Since there is no internal switch to provide backup supply via **VCC_IO**, the **V_BCKP** current will increase while navigating. That will drain the backup battery faster.

2.5.3 VCC_IO



For compatibility (migration path for MAX-7 and MAX-8), Pin 7 (Reserved) must be connected to **VDD**.

2.5.4 VCC_RF: Output Voltage RF section

The **VCC_RF** pin can supply an active antenna or external LNA. For more information, see section 4.2.2.

2.6 Interfaces

2.6.1 UART

The MAX-M5Q positioning module includes a Universal Asynchronous Receiver Transmitter (UART) serial interface **RxD/TxD** that supports configurable baud rates. For information about the supported baud rates, see the MAX-M5Q Receiver Description including Protocol Specification, Docu. No FTX-SW-13001 [2].

The signal output and input levels are 0 V to **VCC**. An interface based on RS232 standard levels (+/- 12 V) can be implemented using level shifters such as Maxim MAX3232. Hardware handshake signals and synchronous operation are not supported.

2.7 I/O pins

2.7.1 Antenna input

The antenna provides simultaneous reception of all frequencies from 1560 to 1606 MHz. The module supports both passive and active antennas. For higher sensitivity, use active antennas or an external LNA with passive antennas. The antenna input RF_IN impedance is 50 Ω . The RF input signal path contains a SAW band-pass filter before the internal LNA, which provides good out-of-band protection against GNSS blocking caused by possible nearby wireless transmitters.



For recommended parts, see section 4.3.

2.7.2 RESET_N: Reset input

Driving **RESET_N** low activates a hardware reset of the system. Use this pin only to reset the module. Do not use **RESET_N** to turn the module on and off, since the reset state increases power consumption.

2.7.3 ANT_ON: Active Antenna control (LNA enable)

In Power Save Mode, the system can turn on/off an optional external LNA using the ANT_ON signal in order to optimize power consumption. Polarity is active high.

2.7.4 TIMEPULSE

The TIMEPULSE (PPS) output signal provides a pulse-per-second output signal for timing purposes. The MAX-M5Q time pulse signal is one pulse per second.

The PPS output is valid when navigation is valid and will also continue to 'freewheel' after a valid fix is lost by a certain navigation DR timeout of typically 10 seconds.

Pulse length (high) is 100ms and UTC sync is at rising edge.

3 Design

3.1 Design checklist

Designing-in a MAX-M5Q module is easy, especially when based on a u-blox reference design. 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 design checklist can also help to avoid an unnecessary PCB respin and achieve the best possible performance. Follow the design checklists when developing any MAX-M5Q GPS/GNSS applications. This can significantly reduce development time and costs.

3.1.1 Schematic checklist

Power supply requirements

- ☐ GPS/GNSS positioning modules require a stable power supply. In selecting a strategy to achieve a clean and stable power supply, any resistance in the **VCC** supply line can negatively influence performance. Consider the following points:
 - Wide power lines or even power planes are preferred.
 - Avoid resistive components in the power line (e.g. narrow power lines, coils, resistors, etc.).
 - Placing a filter or other source of resistance at **VCC** can create significantly longer acquisition times.
 - For ground plane design, see section 3.4.3.
- ☐ Ensure that all power supplies are within the specified range. (See MAX-M5Q Data Sheet, Docu. No FTX-HW-13003 [1])
- ☐ Compare the peak supply current consumption of your MAX-M5Q module with the specification of the power supply. (See MAX-M5Q Data Sheet, Docu. No FTX-HW-13003 [1])
- ☐ At the module input, use low ESR capacitors that can deliver the required current/charge for switching from backup mode to normal operation.



For compatibility (migration path for MAX-7 and MAX-8), Pin 7 (Reserved) must be connected to **VDD**.

Backup battery

- ☐ Use of valid time and the GPS/GNSS orbit data at startup will improve the GPS/GNSS performance, i.e. enables hot and warm starts. To make use of these features connect a battery to **V_BCKP** to continue supplying the backup domain in case of power failure at **VCC**.



If no backup supply voltage is available, connect the **V_BCKP** pin to **VCC**.

3.1.2 Layout checklist



See section 3.3 for more details on the layout.

- ☐ Position the GPS/GNSS module according to the recommendation.
- ☐ Follow the grounding concept.
- ☐ Keep the micro strip as short as possible.
- ☐ Add a ground plane underneath the GPS/GNSS 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/GNSS module, etc.
- ☐ Include appropriate EOS/ESD/EMI protection measures. This is especially important for designs including wireless modules.

3.1.3 Antenna checklist

- ☐ The total noise figure should be well below 3 dB.
- ☐ If a patch antenna is the preferred antenna, choose a patch of at least 15x15x4 mm for standalone GPS/QZSS, or choose a patch of at least 25x25x4 mm for GPS + GLONASS. For smaller antennas, an LNA with a noise figure <2 dB is recommended.
- ☐ Make sure the antenna is not located close to noisy parts of the circuitry (e.g. micro-controller or display).
- ☐ To optimize performance in environments with out-of-band jamming sources, use an additional SAW filter.
- ☐ The micro strip must be 50 Ω 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 first 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. See section 4.1.4.
- ☐ Use an external LNA if your design does not include an active antenna when optimal performance is important.



For information on ESD protection for patch antennas and removable antennas, see section 6.3.3 and if you use GPS for design in combination with GSM or other radio, then check sections 6.3.5 to 6.3.7.



For more information dealing with interference issues, see the GPS Antenna Application Note [3].

3.2 Design considerations for minimal designs

For a minimal design with the MAX-M5Q GPS/GNSS module, the following functions and pins need consideration:

- Connect the Power supply to **VCC**.
- Assure an optimal ground connection to all ground pins of the module.
- Connect the antenna to **RF_IN** over a 50 Ω
- If you need improved startup or use AssistNow Autonomous in your application, connect a backup supply voltage to **V_BCKP**.

3.2.1 Minimal design

This is a minimal setup for a GPS/GNSS receiver:

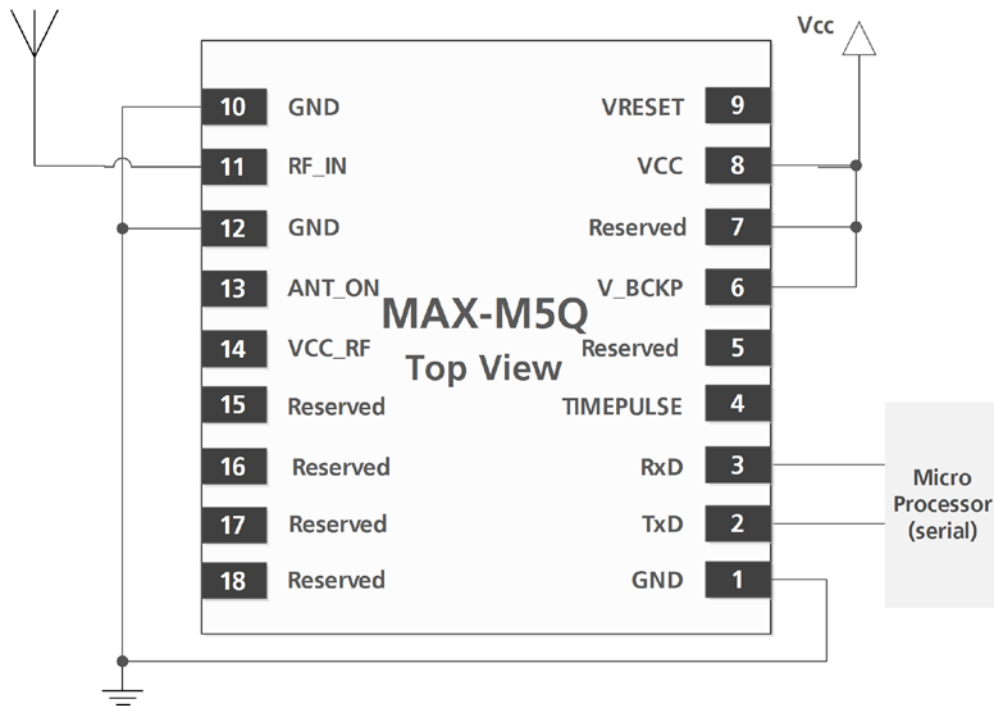


Figure 3: MAX-M5Q passive antenna design



For information on increasing immunity to jammers such as GSM, see section 6.3.7.

Function	PIN	No	I/O	Description	Remarks
Power	VCC	8	I	Supply Voltage	Provide clean and stable supply.
	GND	1,10,12	I	Ground	Assure a good GND connection to all GND pins of the module, preferably with a large ground plane.
	V_BCKP	6	I	Backup Supply Voltage	Backup supply voltage input pin. Connect to VCC if not used.
Antenna	RF_IN	11	I	GPS signal input from antenna	The connection to the antenna has to be routed on the PCB. Use a controlled impedance of 50 Ω to connect RF_IN to the antenna or the antenna connector. DC block inside.
	VCC_RF	14	O	Output Voltage RF section	Can be used for active antenna or external LNA supply.
	ANT_ON	13	O	ANT_ON	Active antenna or ext. LNA control pin in power save mode. ANT_ON pin voltage level is VCC
UART	TXD	2	O	Serial Port	UART, leave open if not used, Voltage level referred VCC .
	RXD	3	I	Serial Port	UART, leave open if not used, Voltage level referred VCC
System	TIMEPULSE	4	O	Timepulse Signal	Leave open if not used, Voltage level referred VCC
	VRESET	9	I	RESET_N	Reset
	Reserved	7	-	Reserved	Connected to VDD (VCC_IO)
	Reserved	5, 15, 16, 17, 18	-	Reserved	Leave open

Table 1: Pinout MAX-M5Q

3.3 Layout

This section provides important information for designing a robust GPS/GNSS system.

GPS/GNSS signals at the surface of the Earth are about 15 dB below the thermal noise floor. Signal loss from the antenna to RF_IN pin of the module must be minimized as much as possible. When defining a GPS/GNSS 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 requiring careful consideration.

3.3.1 Footprint and paste mask

Figure 4 describes the footprint and provides recommendations for the paste mask for MAX-M5Q LCC modules. 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. For the stencil thickness, see section 6.2.1.

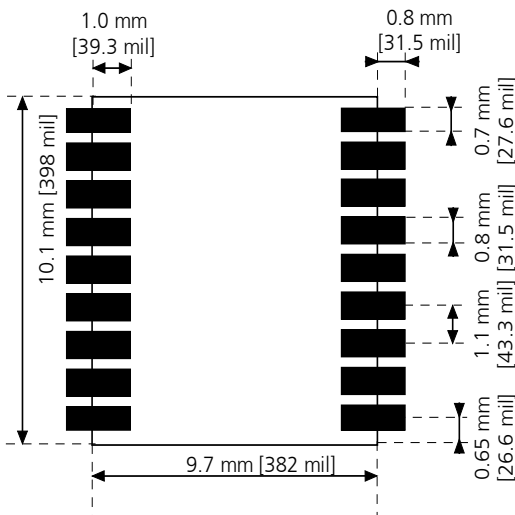


Figure 4: MAX-M5Q footprint

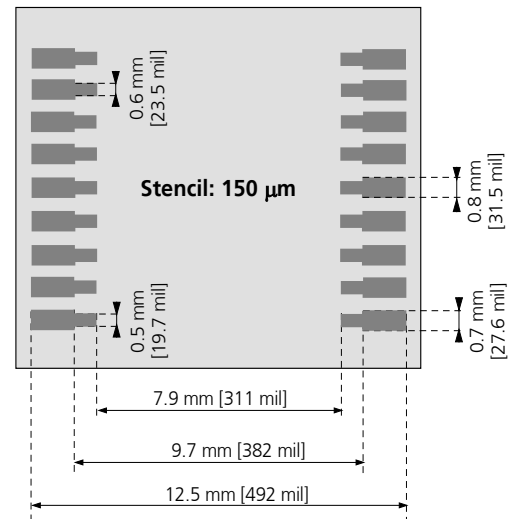


Figure 5: MAX-M5Q paste mask



Consider the paste mask outline when defining the minimal distance to the next component. The exact geometry, distances, stencil thicknesses and solder paste volumes must be adapted to the specific production processes (e.g. soldering) of the customer.

4.1.1 Placement

A very important factor in achieving maximum 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 the RF critical circuits are separate from any other digital circuits on the system board. To achieve this, position the module's digital part towards the digital section on the system PCB. Exercise care if placing the receiver in proximity to heat emitting circuitry. 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.

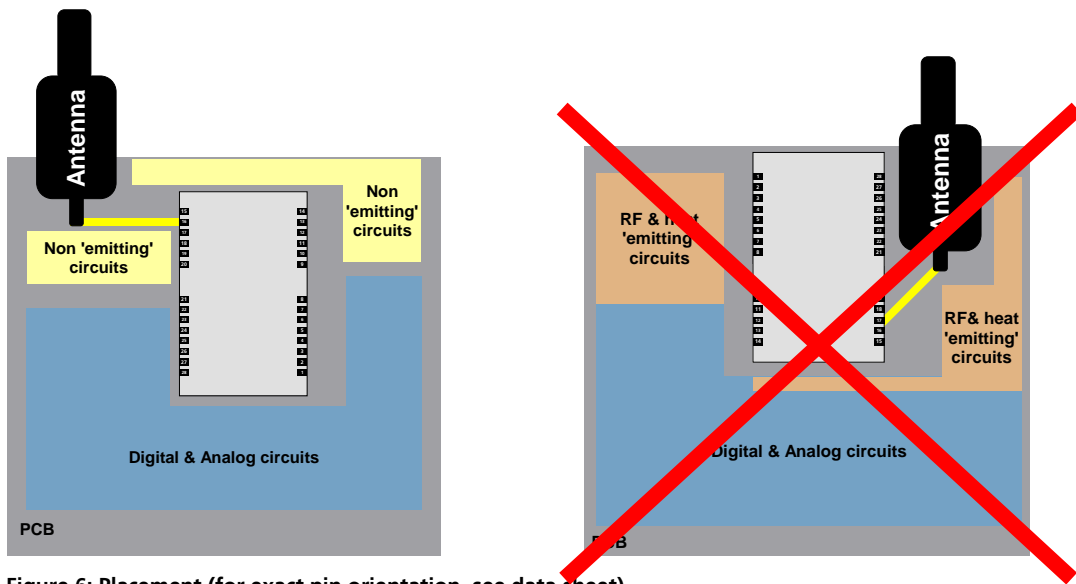


Figure 6: Placement (for exact pin orientation, see data sheet)

4.1.2 Antenna connection and ground plane design

u-blox 7 modules can be connected to passive 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. Figure 7 illustrates connection to a typical five-pin RF connector. One can see the improved shielding for digital lines as discussed in the *GPS Antenna Application Note [3]*. Depending on the actual size of the ground area, if possible place additional vias in the outer region. In particular, terminate the edges of the ground area with a dense line of vias.

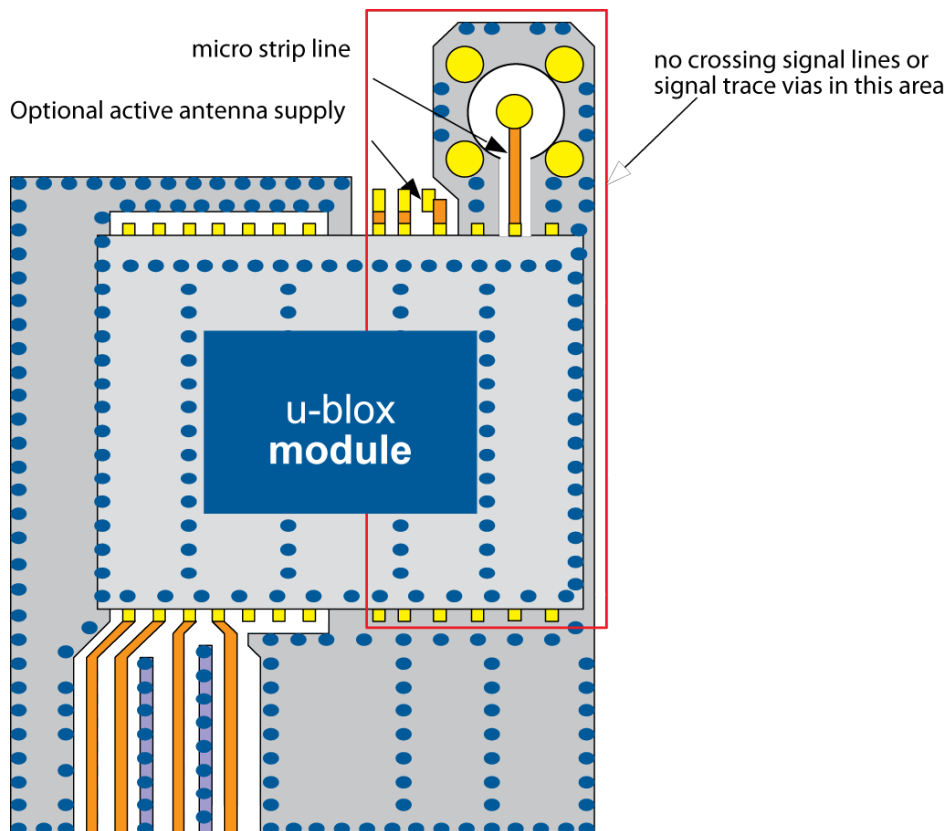


Figure 7: Recommended layout (for exact pin orientation, see data sheet)

As seen in Figure 7, an isolated ground area exists 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 must match the $50\ \Omega$ impedance of the receiver. To achieve an impedance of $50\ \Omega$, the width W of the micro strip line has to be chosen depending on the dielectric thickness H , the dielectric constant ϵ_r of the dielectric material of the PCB and on the build-up of the PCB (see section 4.1.4). Figure 8 shows two different builds: A 2 Layer PCB and a four Layer PCB. The reference ground plane is in both designs on layer 2 (red). Therefore, the effective thickness of the dielectric is different.

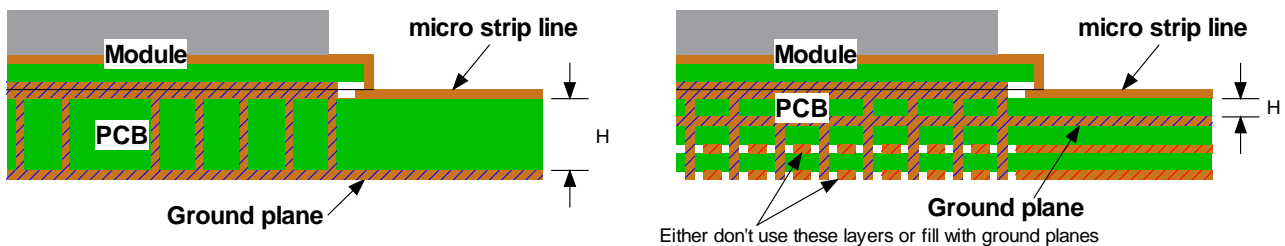


Figure 8: PCB build-up for micro strip line. Left: 2-layer PCB, right: 4-layer PCB

4.1.3 General design recommendations:

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

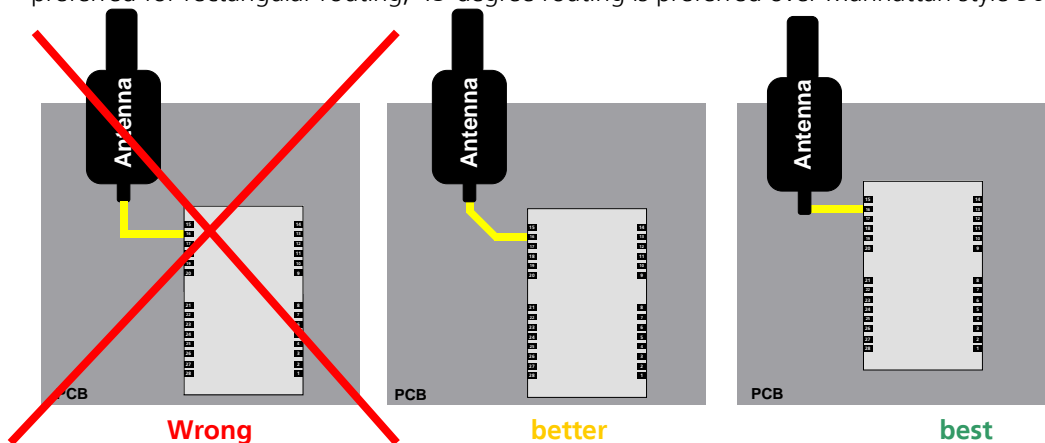


Figure 9: Recommended micro strip routing to RF pin (for exact pin orientation see data sheet)

- Do not route the RF-connection underneath the receiver. 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, cover the area on the PCB under the receiver entirely with solder mask. Vias should not be open. Do not route under the receiver.

4.1.4 Antenna micro strip

There are many ways to design wave-guides on printed circuit boards. A common factor 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 in this regard. Download these tools from www.agilent.com or www.hp.woodshot.com and www.mwoffice.com.

Micro strip is the most commonly used configuration on printed circuit boards; see Figure 10 and Figure 11. As a rule of thumb, to achieve a 50 Ω line impedance with FR-4 material, the width of the conductor is roughly double the thickness of the dielectric.



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 Grounded Coplanar Waveguide model for the calculation of the line dimensions.

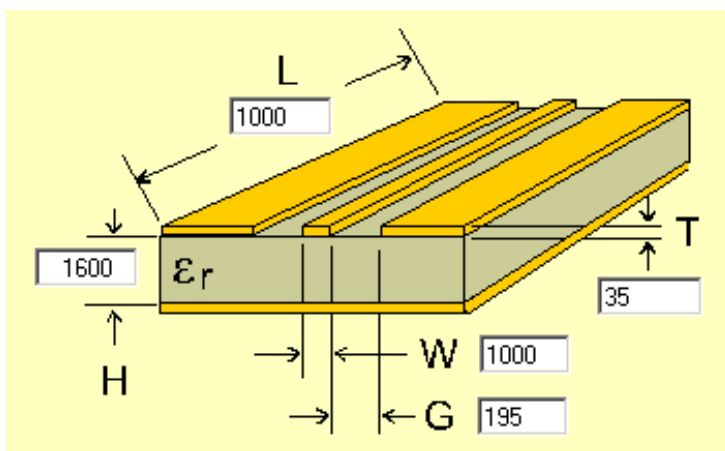


Figure 10: Micro strip on a 2-layer board (Agilent AppCAD Coplanar Waveguide)

Figure 10 shows an example of a 2-layer FR4 board with 1.6 mm thickness (**H**) and a 35 μm (1 ounce) copper cladding (**T**). The thickness of the micro strip is comprised of the cladding (35 μm) plus the plated copper (typically 25 μm). Figure 11 is an example of a multi layer FR4 board with 18 μm ($\frac{1}{2}$ ounce) cladding (**T**) and 180 μm dielectric between layer 1 and 2.

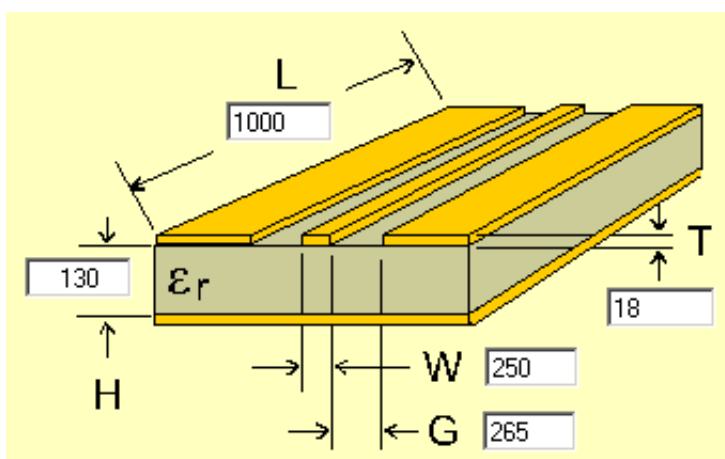


Figure 11: Micro strip on a multi layer board (Agilent AppCAD Coplanar Waveguide)

4.2 Antenna and Antenna supervision

4.2.1 Antenna design with passive antenna

A design using a passive antenna requires more attention to 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 Ω .

4.2.1.1 Minimal setup with a good patch antenna

Figure 3 shows a minimal setup for a design with a good GPS patch antenna.

4.2.1.2 Setup for best performance with passive antenna

Figure 12 shows a design using an external LNA to increase the sensitivity for best performance with passive antenna.

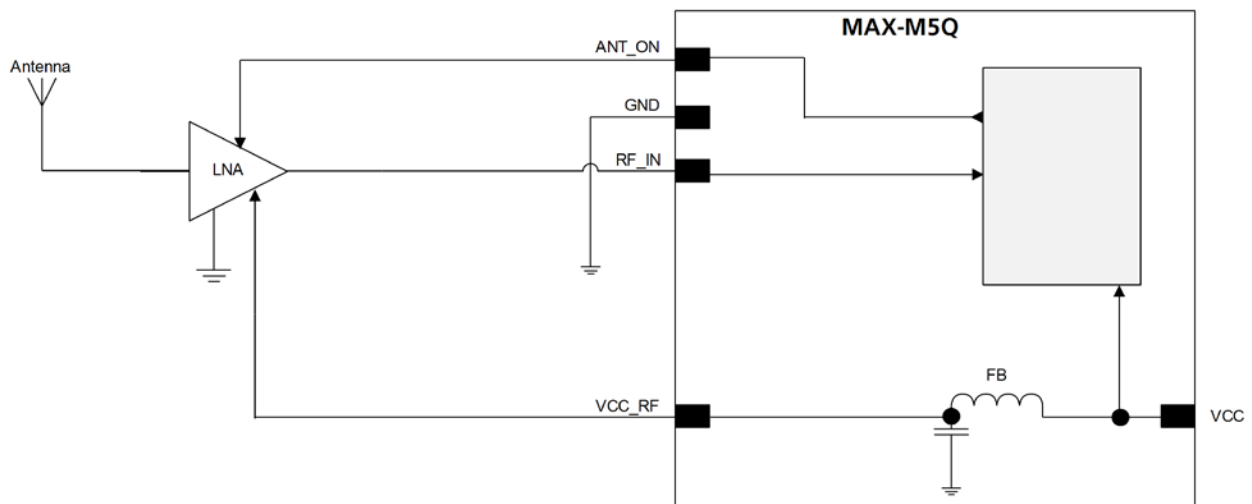


Figure 12: Module design with passive antenna and an external LNA (for exact pin orientation, see data sheet)

ANT_ON (antenna on) can be used to turn on and off an optional external LNA.

The **VCC_RF** output can be used to supply the LNA with a filtered supply voltage.



For recommended parts, see section 4.3



A standard GPS LNA has enough bandwidth to amplify GPS and GLONASS signals.

4.2.2 Active antenna design not using antenna supervisor

Active antennas have an integrated low-noise amplifier. Active antennas require a power supply that will contribute to the total GPS system power consumption budget with additional 5 to 20 mA typically.

If the supply voltage of the MAX-M5Q receiver matches the supply voltage of the antenna (e.g. 3.0 V), use the filtered supply voltage **VCC_RF** output to supply the antenna. See section 4.2.2.1.

In case of different supply voltage, use a filtered external supply as shown in section 4.2.2.2

4.2.2.1 Active antenna design, VCC_RF used to supply active antenna

Figure 13 shows an active antenna design supplied by VCC_RF.

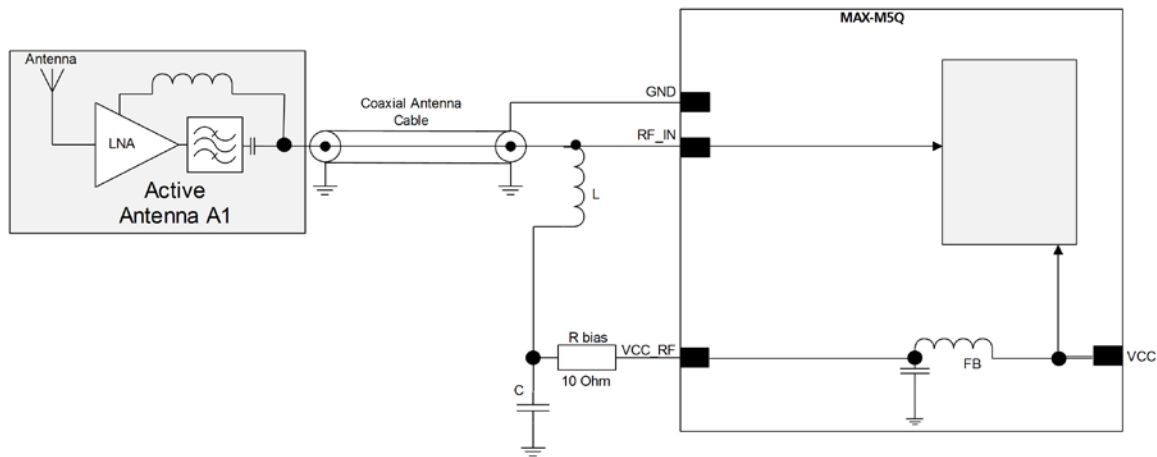


Figure 13: Active antenna design, external supply from VCC_RF (for exact pin orientation, see data sheet [1])



For recommended parts, see section 4.3.

4.2.2.2 Active antenna design powered from external supply

Figure 14 shows a design with direct externally powered active antenna.

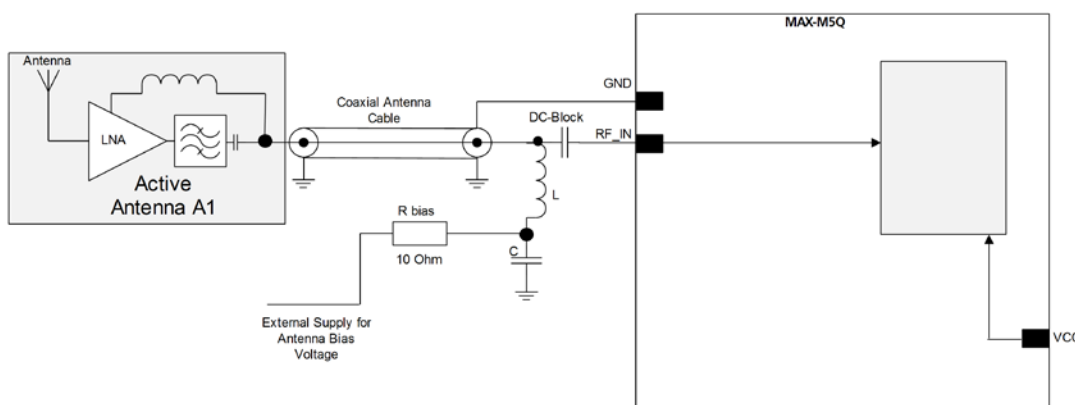


Figure 14: Active antenna design, direct external supply (for exact pin orientation, see data sheet [1])



For recommended parts, see section 4.3.



In case VCC_RF voltage does not match with the antenna supply voltage, use a filtered external supply as shown in Figure 14.

4.2.2.3 External active antenna supervisor using customer uP

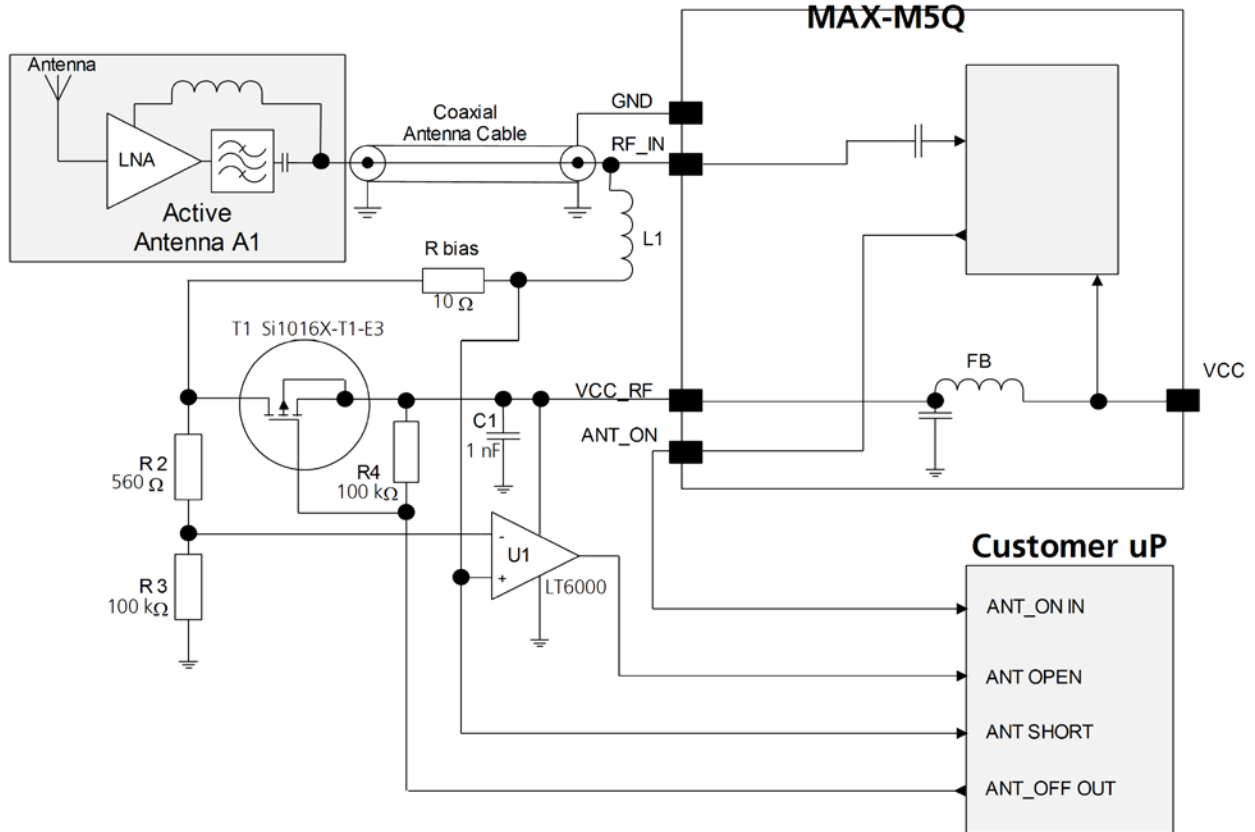


Figure 15: External active antenna supervisor using ANT_ON

$$I = \frac{\left(\frac{R2}{R2 + R3} \right)}{R_{bias}} \cdot V_{cc_RF}$$

Equation 1: Calculation of threshold current for open circuit detection



For recommended parts, see section 4.3.

4.2.2.4 External active antenna control

The ANT_ON signal can be used to turn on and off an external LNA. This reduces power consumption in Power Save Mode (Backup mode).

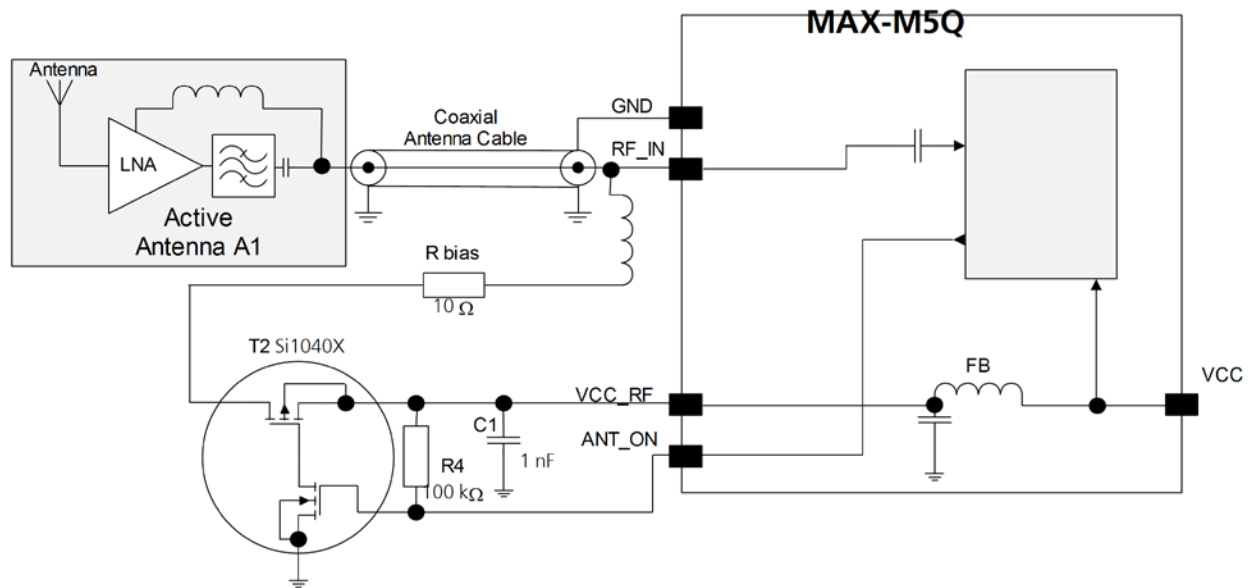


Figure 16: External active antenna control



For recommended parts, see section 4.3.

4.2.3 Design with GLONASS / GPS active antenna

The Russian GLONASS satellite system is an alternative system to the US-based Global Positioning System (GPS). u-blox 7 modules can receive and process GLONASS signals. GLONASS and GPS satellite signals are not transmitted at the same frequency (as seen in Figure 17). In existing designs that were only intended for GPS reception, the RF path has to be modified (the LNA, filter, and antenna) accordingly to let both signals pass.

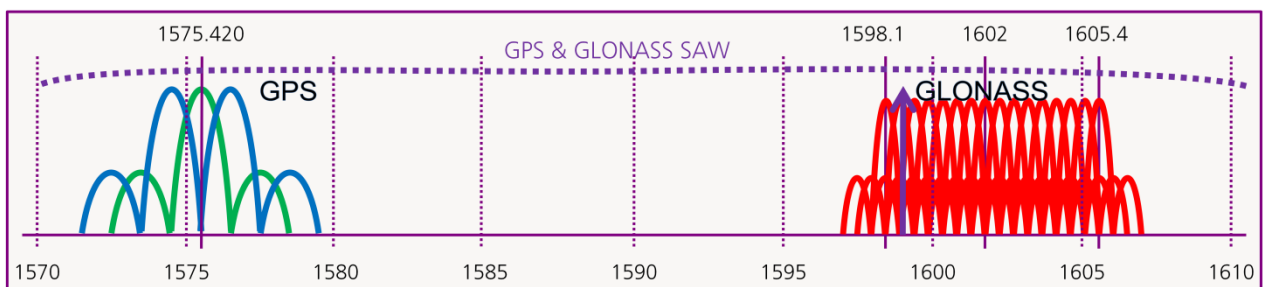


Figure 17: GPS & GLONASS SAW filter

Usually an active GPS antenna includes a GPS band pass filter, which may filter out the GLONASS signal (see Figure 17). For this reason, make sure that the filter in the active antenna is wide enough to let the GPS and GLONASS signals pass. Use a good performance GPS & GLONASS active antenna (for recommended components see section 1.1.1880621787).

In a combined GPS & GLONASS antenna, be sure to tune the antenna for receiving both signals. In addition, any internal filter has a larger bandwidth to provide optimal GPS & GLONASS signal reception (see Figure 17).



Use a good performance GPS & GLONASS active antenna (for recommended components see section 1.1.1880621787).

4.2.4 Design with GLONASS / GPS passive antenna

In general, GPS patch antennas only receive GPS signals well. A typical return plot (S11 measurement) shows that the GLONASS signal is highly attenuated. (See Figure 18)

u-blox 7 modules supporting GLONASS have a GPS & GLONASS SAW filter (see Figure 17) that lets both GPS and GLONASS signals pass. For best performance with passive antenna designs, use an external LNA. (See section 4.2.1.2).

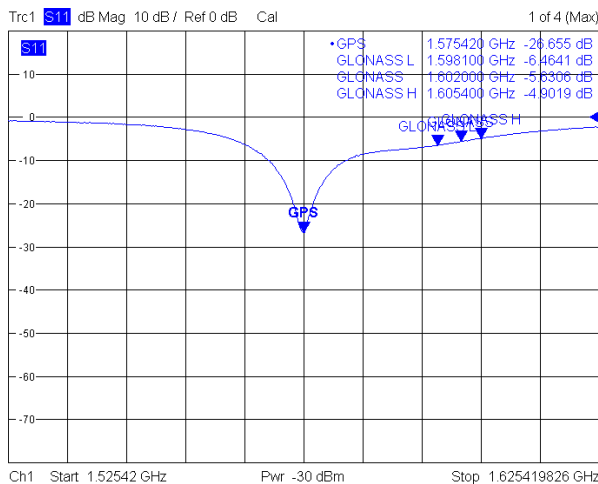


Figure 18: 25*25*4 mm GPS patch antenna on 70*70 mm GND plane

To receive GPS and GLONASS, a special antenna patch (which can receive both GPS and GLONASS) is needed. The return plot (S11 measurement) in Figure 19 below shows the two areas of lower attenuation.

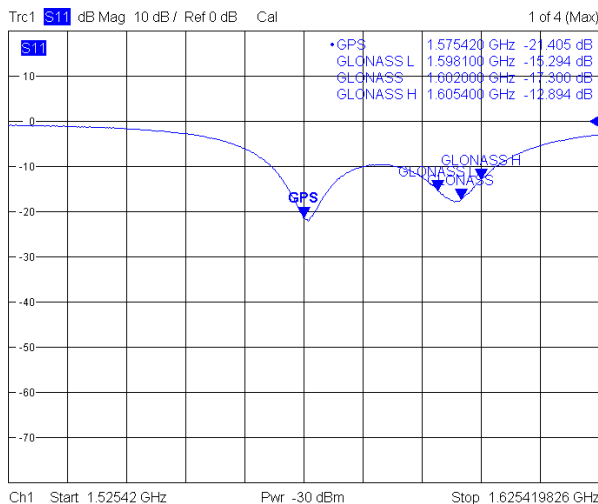


Figure 19: 25*25*4 mm GPS / GLONASS patch antenna on 70*70 mm GND plane

4.3 Recommended parts

u-blox has tested and recommends the parts listed in Table 2. Other untested components may also be used.

	Manufacturer	Part ID	Remarks	Parameters to consider
Diode	ON Semiconductor	ESD9R3.3ST5G	Standoff Voltage>3.3 V	Low Capacitance < 0.5 pF
		ESD9L3.3ST5G	Standoff Voltage>3.3 V	Standoff Voltage > Voltage for active antenna
		ESD9L5.0ST5G	Standoff Voltage>5 V	Low Inductance
SAW	Infineon	ESD0P2RF		low capacitance (<0.5 pF)
	TDK/ EPCOS	B8401: B39162-B8401-P810	GPS+GLONASS	High attenuation
	TDK/ EPCOS	B3913: B39162B3913U410	GPS+GLONASS	For automotive application
	TDK/ EPCOS	B9850: B39162B9850P810	GPS	Low insertion loss
	TDK/ EPCOS	B8400: B39162B8400P810	GPS	ESD protected and high input power
	muRata	SAFEA1G58KB0F00	GPS+GLONASS	Low insertion loss, only for mobile application
	muRata	SAFEA1G58KA0F00	GPS+GLONASS	High attenuation, only for mobile application
	muRata	SAFFB1G58KA0F0A	GPS+GLONASS	High attenuation, only for mobile application
	muRata	SAFFB1G58KB0F0A	GPS+GLONASS	Low insertion loss, Only for mobile application
LNA	Avago	ALM-1106	LNA	pHEMT (GaAS)
		ALM-1412	LNA + FBAR Filter	
		ALM-1712	Filter + LNA + FBAR Filter	
		ALM-1912	LNA	Module, for GPS only, also including FBAR filter in front of LNA
		ALM-2412	LNA + FBAR Filter	
		ALM-2712	LNA	Module, for GPS only, FBAR filter-LNA filter FBAR
	MAXIM	MAX2659ELT+	LNA	Low noise figure, up to 10 dBm RF input power
	JRC	NJG1143UA2	LNA	Low noise figure, up to 15 dBm RF input power
	Infineon	BGM1032N16	Filter + LNA	
		BGM981N11	Filter + LNA + Filter	
		BGM1052N16	LNA + Filter	
	Triquint	TQM640002	Filter + LNA + Filter	
Inductor	Murata	LQG15HS27NJ02	L, 27 nH	Impedance @ freq GPS > 500 Ω
Capacitor	Murata	GRM1555C1E470JZ01	C, 47 pF	DC-block
Ferrite Bead	Murata	BLM15HD102SN1	FB	High IZI @ fGSM
Feed thru Capacitor for Signal	Murata	NFL18SP157X1A3	Monolithic Type	Load Capacitance appropriate to Baud rate
		NFA18SL307V1A45	Array Type	CL < xxx pF
Feed thru Capacitor	Murata	NFM18PC	0603 2A	Rs < 0.5 Ω
		NFM21P....	0805 4A	
Resistor		10 $\Omega \pm 10\%$, min 0.250 W	R _{bias}	
		560 $\Omega \pm 5\%$	R2	
		100 k $\Omega \pm 5\%$	R3, R4	
Op Amp	Linear Technology	LT6000	U1	Rail to Rail
Transistor	Vishay	Si1016X	T1	
Transistor	Vishay	Si1040X	T2	

Table 2: Recommended parts

4.3.1 Recommended GPS & GLONASS active patch antenna (A1)

Manufacturer	Order No.	Comments
Taoglas (www.taoglas.com)	AA.160.301111	36*36*4 mm, 3-5V 30mA
Taoglas (www.taoglas.com)	AA.161.301111	36*36*3 mm, 1.8 to 5.5V / 10mA at 3V

Additional antenna Manufacturer: INPAQ, Hirschmann,
Allis Communications, 2J, Tallysman Wireless

Table 3: Recommend GPS & GLONASS active antenna (A1)

4.3.2 Recommended GPS & GLONASS passive patch antenna

Manufacturer	Order No.	Comments
Amotech (www.amotech.co.kr)	B35-3556920-2J2	35x35x3 mm GPS+GLONASS
Amotech (www.amotech.co.kr)	A25-4102920-2J3	25x25x4 mm GPS+GLONASS
Amotech (www.amotech.co.kr)	A18-4135920-AMT04	18x18x4 mm GPS+GLONASS

Table 4: Recommend GPS & GLONASS passive patch antenna

4.3.3 Recommended GPS & GLONASS passive chip antenna

Manufacturer	Order No.	Comments
INPAQ (www.inpaq.com.tw)	ACM4-5036-A1-CC-S	5.2 x 3.7 x 0.7 mm GPS+GLONASS

Table 5: Recommend GPS & GLONASS passive chip antenna

5 Migration

5.1 Migrating from u-blox 7 designs to a MAX-M5Q module

Due to differences between Mediatek and u-blox chips, 100% compatibility is not possible

No I2C, EXTINT, VCC_IO, SAFE_BOOT, Antenna Sense

MAX-M5Q I/O voltage is 2.8V. MAX-8x I/O voltage is VCC_IO

For forward compatibility Pin 7 (Reserved) must be connected to **VDD (VCC_IO)**

Pin 6, **V_BCKP** current drain:

V_BCKP provides supply all the time to backup memory & RTC. Since there is no internal switch to provide backup supply via **VCC_IO**, the **V_BCKP** current will increase while navigating; that will drain the backup battery faster.



To reduce the **V_BCKP** current while navigating, use an external schottky diode pair to provide **V_BCKP** supply via **VCC** (when it is active) and when **VCC** is off, then the other diode passes current from battery.

5.1.1 Software compatibility

No u-blox binary protocol, configuration etc.

For a typical NMEA interface usage the MAX-M5Q can be considered as compatible to MAX-7x

Review the MAX-M5Q Receiver Description including Protocol Specification, Docu. No FTX-SW-13001 [2] for details.

5.2 Forward compatibility from MAX-M5Q designs to a MAX-8x module

Due to differences between Mediatek and u-blox chips, 100% compatibility is not possible.

MAX-M5Q I/O voltage is 2.8V. MAX-8x I/O voltage is VCC_IO

For forward compatibility Pin 7: (Reserved) must be connected to **VDD (VCC_IO)**

5.2.1 Software compatibility

For a typical NMEA interface usage the MAX-M5Q can be considered as compatible to MAX-8x



Use caution when implementing Backup, Periodic or AlwaysLocate™ mode, since forward compatibility is not guaranteed.

6 Product handling

6.1 Packaging, shipping, storage and moisture preconditioning

For information pertaining to reels and tapes, Moisture Sensitivity levels (MSD), shipment and storage information, as well as drying for preconditioning see MAX-M5Q Data Sheet, Docu. No FTX-HW-13003 [1].

6.1.1 Population of Modules



When populating our modules make sure that the pick and place machine is aligned to the copper pins of the module and not on the module edge.

6.2 Soldering

6.2.1 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: OM338 SAC405 / Nr.143714 (Cookson Electronics)
 Alloy specification: Sn 95.5/ Ag 4/ Cu 0.5 (95.5% Tin/ 4% Silver/ 0.5% Copper)
 Melting Temperature: 217 °C
 Stencil Thickness: see section 3.3.1

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.



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

6.2.2 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: max. 3 °C/s If the temperature rise is too rapid in the preheat phase it may cause excessive slumping.
- Time: 60 - 120 s 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 217°C. Avoid a sudden rise in temperature as the slump of the paste could become worse.

- Limit time above 217 °C liquidus temperature: 40 - 60 s
- Peak reflow temperature: 245 °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 4 °C/s



To avoid falling off, the u-blox 7 GPS/GNSS 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.

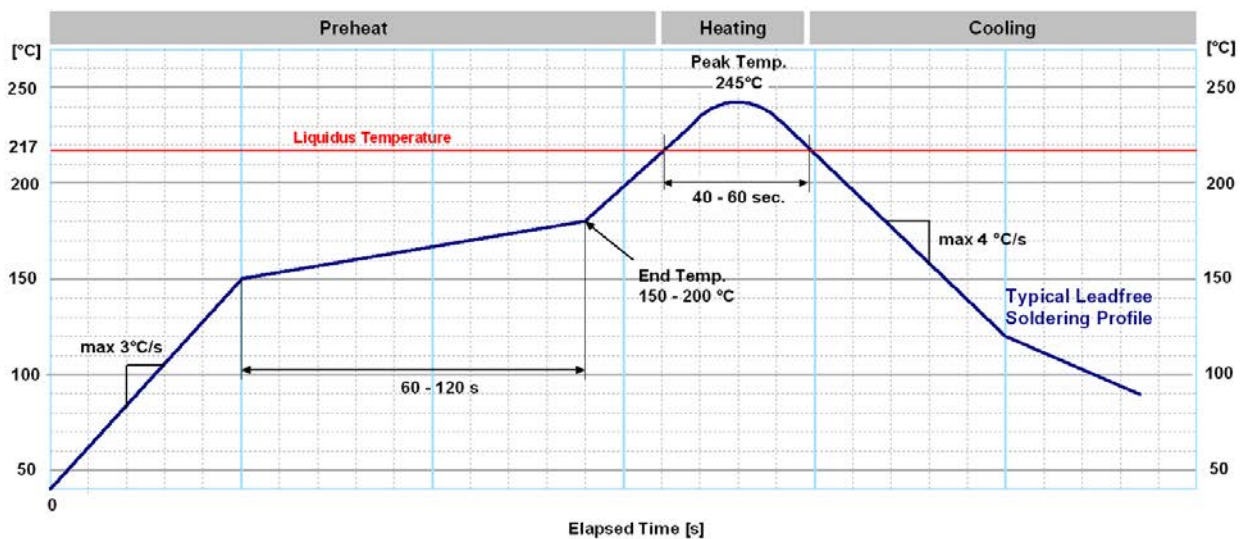


Figure 20: Recommended soldering profile



u-blox 7 modules **must not** be soldered with a damp heat process.

6.2.3 Optical inspection

After soldering the u-blox 7 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

6.2.4 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.
- 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.

6.2.5 Repeated reflow soldering

Only single reflow soldering processes are recommended for boards populated with u-blox 7 modules. u-blox 7 modules should not be submitted to two reflow cycles on a board populated with components on both sides in order to avoid upside down orientation during the second reflow cycle. In this case, the module should always be placed on that side of the board, which is submitted into the last reflow cycle. The reason for this (besides others) is the risk of the module falling off due to the significantly higher weight in relation to other components.

Two reflow cycles can be considered by excluding the above described upside down scenario and taking into account the rework conditions described in Section 6.2.8.



Repeated reflow soldering processes and soldering the module upside down are not recommended.

6.2.6 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 u-blox 7 modules.

6.2.7 Hand soldering

Hand soldering is allowed. Use a soldering iron temperature setting 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 15), and then continue from left to right.

6.2.8 Rework

The u-blox 7 module can be unsoldered from the baseboard using a hot air gun. When using a hot air gun for unsoldering the module, max one reflow cycle is allowed. In general, we do not recommend using a hot air gun because this is an uncontrolled process and might damage the module.



Attention: use of a hot air gun can lead to overheating and severely damage the module. Always 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.

In addition to the two reflow cycles, manual rework on particular pins by using a soldering iron is allowed. For hand soldering the recommendations in IPC 7711 should be followed. Manual rework steps on the module can be done several times.

6.2.9 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/GNSS 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.

6.2.10 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 u-blox 7 module before implementing this in the production.



Casting will void the warranty.

6.2.11 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 u-blox 7 module caused by soldering metal cables or any other forms of metal strips directly onto the EMI covers.

6.2.12 Use of ultrasonic processes

Some components on the u-blox 7 module are sensitive to Ultrasonic Waves. Use of any Ultrasonic Processes (cleaning, welding etc.) may cause damage to the GPS/GNSS Receiver.



u-blox offers no warranty against damages to the u-blox 7 module caused by any Ultrasonic Processes.

6.3 EOS/ESD/EMI precautions

When integrating GPS/GNSS positioning modules into wireless systems, careful consideration must be given to electromagnetic and voltage susceptibility issues. Wireless systems include components, which can produce Electrical Overstress (EOS) and Electro-Magnetic Interference (EMI). CMOS devices are more sensitive to such influences because their failure mechanism is defined by the applied voltage, whereas bipolar semiconductors are more susceptible to thermal overstress. The following design guidelines are provided to help in designing robust yet cost effective solutions.



To avoid overstress damage during production or in the field it is essential to observe strict EOS/ESD/EMI handling and protection measures.



To prevent overstress damage at the RF_IN of your receiver, never exceed the maximum input power (see Data Sheet).

6.3.1 Electrostatic discharge (ESD)

Electrostatic discharge (ESD) is the sudden and momentary electric current that flows between two objects at different electrical potentials caused by direct contact or induced by an electrostatic field. The term is usually used in the electronics and other industries to describe momentary unwanted currents that may cause damage to electronic equipment.

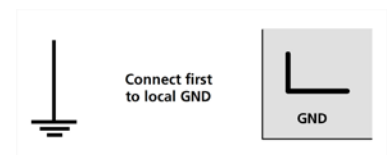


6.3.2 ESD handling precautions

ESD prevention is based on establishing an Electrostatic Protective Area (EPA). The EPA can be a small working station or a large manufacturing area. The main principle of an EPA is that there are no highly charging materials near ESD sensitive electronics, all conductive materials are grounded, workers are grounded, and charge build-up on ESD sensitive electronics is prevented. International standards are used to define typical EPA and can be obtained for example from International Electrotechnical Commission (IEC) or American National Standards Institute (ANSI).

GPS/GNSS positioning modules are sensitive to ESD and require special precautions when handling. Particular care must be exercised when handling patch antennas, due to the risk of electrostatic charges. In addition to standard ESD safety practices, the following measures should be taken into account whenever handling the receiver.

- Unless there is a galvanic coupling between the local GND (i.e. the work table) and the PCB GND, then the first point of contact when handling the PCB must always be between the local GND and PCB GND.
- Before mounting an antenna patch, connect ground of the device



- When handling the RF pin, do not come into contact with any charged capacitors and be careful when contacting materials that can develop charges (e.g. patch antenna ~10 pF, coax cable ~50 - 80 pF/m, soldering iron, ...)
- To prevent electrostatic discharge through the RF input, do not touch any exposed antenna area. If there is any risk that such exposed antenna area is touched in non ESD protected work area, implement proper ESD protection measures in the design.
- When soldering RF connectors and patch antennas to the receiver's RF pin, make sure to use an ESD safe soldering iron (tip).



Failure to observe these precautions can result in severe damage to the GPS/GNSS module!

6.3.3 ESD protection measures



GPS/GNSS positioning modules are sensitive to Electrostatic Discharge (ESD). Special precautions are required when handling.



For more robust designs, employ additional ESD protection measures. Using an LNA with appropriate ESD rating can provide enhanced GPS/GNSS performance with passive antennas and increases ESD protection.

Most defects caused by ESD can be prevented by following strict ESD protection rules for production and handling. When implementing passive antenna patches or external antenna connection points, then additional ESD measures as shown in Figure 21 can also avoid failures in the field.

Small passive antennas (<2 dBic and performance critical)	Passive antennas (>2 dBic or performance sufficient)	Active antennas
<p>A</p> <p>LNA with appropriate ESD rating</p>	<p>B</p>	<p>C</p>

Figure 21: ESD Precautions



Protection measure A is preferred because it offers the best GPS performance and best level of ESD protection.

6.3.4 Electrical Overstress (EOS)

Electrical Overstress (EOS) usually describes situations when the maximum input power exceeds the maximum specified ratings. EOS failure can happen if RF emitters are close to a GPS/GNSS receiver or its antenna. EOS causes damage to the chip structures. If the RF_IN is damaged by EOS, it is hard to determine whether the chip structures have been damaged by ESD or EOS.

6.3.5 EOS protection measures



For designs with GPS/GNSS positioning modules and wireless (e.g. GSM/GPRS) transceivers in close proximity, ensure sufficient isolation between the wireless and GPS antennas. If wireless power output causes the specified maximum power input at the GPS RF_IN to be exceeded, employ EOS protection measures to prevent overstress damage.

6.3.6 Electromagnetic interference (EMI)

Electromagnetic interference (EMI) is the addition or coupling of energy originating from any RF emitting device. This can cause a spontaneous reset of the GPS/GNSS receiver or result in unstable performance. Any unshielded line or segment (>3mm) connected to the GPS/GNSS receiver can effectively act as antenna and lead to EMI disturbances or damage.

The following elements are critical regarding EMI:

- Unshielded connectors (e.g. pin rows etc.)
- Weakly shielded lines on PCB (e.g. on top or bottom layer and especially at the border of a PCB)
- Weak GND concept (e.g. small and/or long ground line connections)

EMI protection measures are recommended when RF emitting devices are near the GPS/GNSS receiver. To minimize the effect of EMI a robust grounding concept is essential. To achieve electromagnetic robustness follow the standard EMI suppression techniques.

<http://www.murata.com/products/emc/knowhow/index.html>

<http://www.murata.com/products/emc/knowhow/pdf/4to5e.pdf>

Improved EMI protection can be achieved by inserting a resistor (e.g. $R > 20 \Omega$) or better yet a ferrite bead (BLM15HD102SN1) or an inductor (LQG15HS47NJ02) into any unshielded PCB lines connected to the GPS/GNSS receiver. Place the resistor as close as possible to the GPS/GNSS receiver pin.

Example of EMI protection measures on the RX/TX line using a ferrite bead:

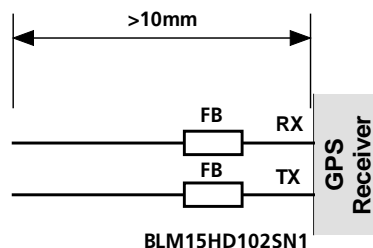


Figure 22: EMI Precautions

VCC can be protected using a feed thru capacitor. For electromagnetic compatibility (EMC) of the RF_IN pin, refer to section 6.3.5

6.3.7 Applications with wireless modules LEON / LISA

GSM uses power levels up to 2 W (+33 dBm). Consult the Data Sheet for the absolute maximum power input at the GPS/GNSS receiver.

6.3.7.1 Isolation between GPS and GSM antenna

In a handheld type design, an isolation of approximately 20 dB can be reached with careful placement of the antennas. If such isolation cannot be achieved, e.g. in the case of an integrated GSM/GPS antenna, an additional input filter is needed on the GPS side to block the high energy emitted by the GSM transmitter. Examples of these kinds of filters would be the SAW Filters from Epcos (B9444 or B7839) or Murata.

6.3.7.2 Increasing jamming immunity

Jamming signals come from in-band and out-band frequency sources.

6.3.7.3 In-band jamming

With in-band jamming the signal frequency is very close to the GPS frequency of 1575 MHz (see Figure 23). Such jamming signals are typically caused by harmonics from displays, micro-controller, bus systems, etc.

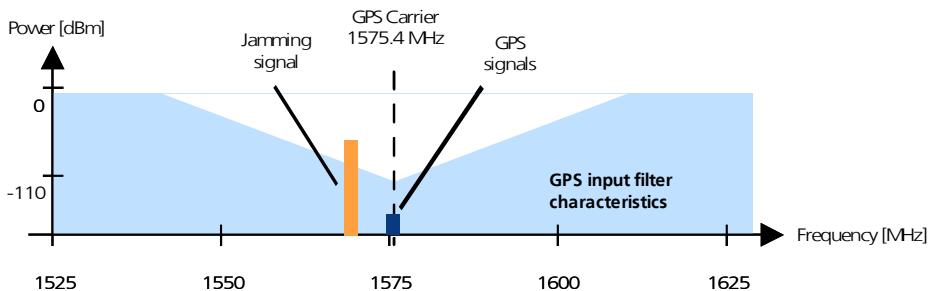


Figure 23: In-band jamming signals

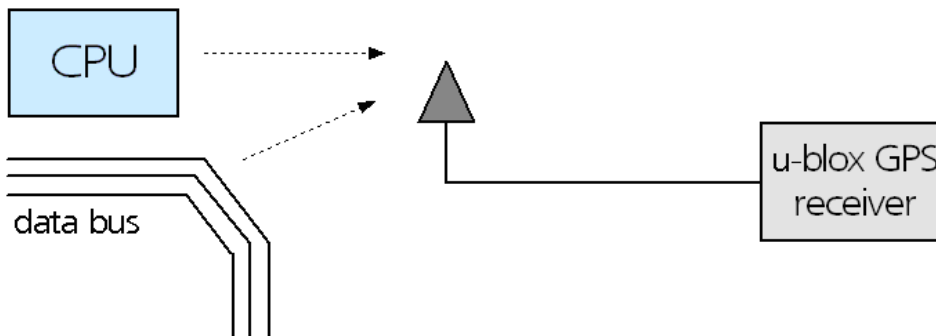


Figure 24: In-band jamming sources

Measures against in-band jamming include:

- Maintaining a good grounding concept in the design
- Shielding
- Layout optimization
- Filtering
- Placement of the GPS antenna
- Adding a CDMA, GSM, WCDMA band pass filter before handset antenna

6.3.7.4 Out-band jamming

Out-band jamming is caused by signal frequencies that are different from the GPS carrier (see Figure 25). The main sources are wireless communication systems such as GSM, CDMA, WCDMA, Wi-Fi, BT, etc.

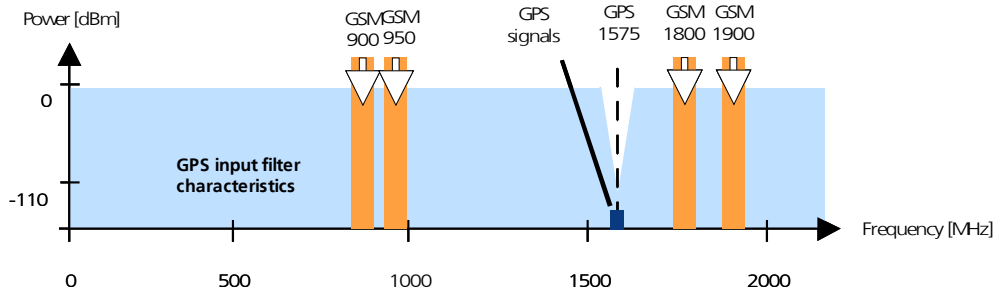


Figure 25: Out-band jamming signals

Measures against out-band jamming include maintaining a good grounding concept in the design and adding a SAW or band pass ceramic filter (as recommend in section 6.3.5) into the antenna input line to the GPS/GNSS receiver (see Figure 26).



Figure 26: Measures against in-band jamming

7 Product testing

7.1 u-blox in-series production test

u-blox focuses on high quality for its products. To achieve a high standard it is 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)
- Traceability down to component level
- X-Ray and Automated Optical Inspection (AOI)
- Ongoing Reliability Tests

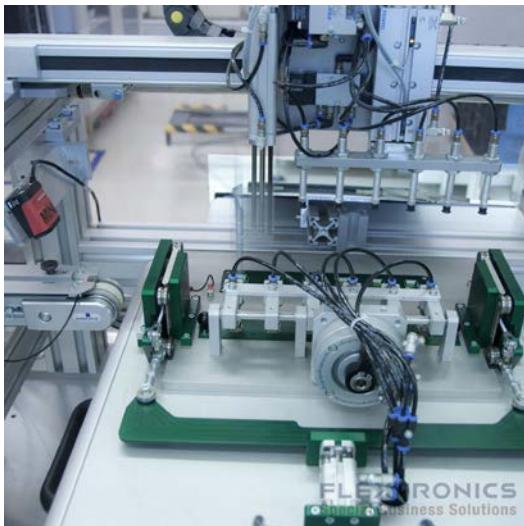


Figure 27: Automatic Test Equipment for Module Tests



Figure 28: X-Ray Inspection

7.2 Test parameters for OEM manufacturer

Because of the testing done by u-blox (with 100% coverage), it is obvious that an OEM manufacturer does not 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

7.3 System sensitivity test

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



Figure 29: 1-channel GPS simulator

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

- Spirent GSS6100 (GPS)
 - Spirent GSS6300 (GPS/GLONASS)
- Spirent Communications Positioning Technology www.spirent.com

7.3.1 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-SVINFO message (e.g. with u-center)
5. Compare the results to a "Golden Device" or a u-blox 7 Evaluation Kit.

7.3.2 '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.

Appendix

A Abbreviations

Abbreviation	Definition
ANSI	American National Standards Institute
CDMA	Code Division Multiple Access
EMC	Electromagnetic compatibility
EMI	Electromagnetic interference
EOS	Electrical Overstress
EPA	Electrostatic Protective Area
ESD	Electrostatic discharge
GLONASS	Russian satellite system
GND	Ground
GNSS	Global Navigation Satellite System
GPS	Global Positioning System
GSM	Global System for Mobile Communications
IEC	International Electrotechnical Commission
PCB	Printed circuit board
QZSS	Quasi-Zenith Satellite System

Table 6: Explanation of abbreviations used

Related documents

- [1] MAX-M5Q Data Sheet, Docu. No FTX-HW-13003
- [2] MAX-M5Q Receiver Description including Protocol Specification, Docu. No FTX-SW-13001
- [3] GPS Antenna Application Note, Docu. No GPS-X-08014
- [4] GPS Compendium, Doc No GPS-X-02007

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



For regular updates to u-blox documentation and to receive product change notifications please register on our homepage (<http://www.u-blox.com>)

Revision history

Revision	Date	Name	Status / Comments
-	May 30, 2013	jfur	Objective Specification

Contact

For complete contact information, visit us at www.u-blox.com

u-blox Offices

North, Central and South America

u-blox America, Inc.

Phone: +1 703 483 3180
E-mail: info_us@u-blox.com

Regional Office West Coast:

Phone: +1 408 573 3640
E-mail: info_us@u-blox.com

Technical Support:

Phone: +1 703 483 3185
E-mail: support_us@u-blox.com

Headquarters Europe, Middle East, Africa

u-blox AG

Phone: +41 44 722 74 44
E-mail: info@u-blox.com
Support: support@u-blox.com

Asia, Australia, Pacific

u-blox Singapore Pte. Ltd.

Phone: +65 6734 3811
E-mail: info_ap@u-blox.com
Support: support_ap@u-blox.com

Regional Office China (Beijing):

Phone: +86 10 68 133 545
E-mail: info_cn@u-blox.com
Support: support_cn@u-blox.com

Regional Office China (Shenzhen):

Phone: +86 755 8627 1083
E-mail: info_cn@u-blox.com
Support: support_cn@u-blox.com

Regional Office India:

Phone: +91 959 1302 450
E-mail: info_in@u-blox.com
Support: support_in@u-blox.com

Regional Office Japan:

Phone: +81 3 5775 3850
E-mail: info_jp@u-blox.com
Support: support_jp@u-blox.com

Regional Office Korea:

Phone: +82 2 542 0861
E-mail: info_kr@u-blox.com
Support: support_kr@u-blox.com

Regional Office Taiwan:

Phone: +886 2 2657 1090
E-mail: info_tw@u-blox.com
Support: support_tw@u-blox.com