



AirPrime XA1110/XM1110

AirPrime XA1110/XM1110



SIERRA
WIRELESS®

41111116
Rev 1

Important Notice

Due to the nature of wireless communications, transmission and reception of data can never be guaranteed. Data may be delayed, corrupted (i.e., have errors) or be totally lost. Although significant delays or losses of data are rare when wireless devices such as the Sierra Wireless modem are used in a normal manner with a well-constructed network, the Sierra Wireless modem should not be used in situations where failure to transmit or receive data could result in damage of any kind to the user or any other party, including but not limited to personal injury, death, or loss of property. Sierra Wireless accepts no responsibility for damages of any kind resulting from delays or errors in data transmitted or received using the Sierra Wireless modem, or for failure of the Sierra Wireless modem to transmit or receive such data.

Safety and Hazards

Do not operate the Sierra Wireless modem in areas where blasting is in progress, where explosive atmospheres may be present, near medical equipment, near life support equipment, or any equipment which may be susceptible to any form of radio interference. In such areas, the Sierra Wireless modem **MUST BE POWERED OFF**. The Sierra Wireless modem can transmit signals that could interfere with this equipment.

Do not operate the Sierra Wireless modem in any aircraft, whether the aircraft is on the ground or in flight. In aircraft, the Sierra Wireless modem **MUST BE POWERED OFF**. When operating, the Sierra Wireless modem can transmit signals that could interfere with various onboard systems.

Note: Some airlines may permit the use of cellular phones while the aircraft is on the ground and the door is open. Sierra Wireless modems may be used at this time.

The driver or operator of any vehicle should not operate the Sierra Wireless modem while in control of a vehicle. Doing so will detract from the driver or operator's control and operation of that vehicle. In some states and provinces, operating such communications devices while in control of a vehicle is an offence.

Limitation of Liability

The information in this manual is subject to change without notice and does not represent a commitment on the part of Sierra Wireless. SIERRA WIRELESS AND ITS AFFILIATES SPECIFICALLY DISCLAIM LIABILITY FOR ANY AND ALL DIRECT, INDIRECT, SPECIAL, GENERAL, INCIDENTAL, CONSEQUENTIAL, PUNITIVE OR EXEMPLARY DAMAGES INCLUDING, BUT NOT LIMITED TO, LOSS OF PROFITS OR REVENUE OR ANTICIPATED PROFITS OR REVENUE ARISING OUT OF THE USE OR INABILITY TO USE ANY SIERRA WIRELESS PRODUCT, EVEN IF SIERRA WIRELESS AND/OR ITS AFFILIATES HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES OR THEY ARE FORESEEABLE OR FOR CLAIMS BY ANY THIRD PARTY.

Notwithstanding the foregoing, in no event shall Sierra Wireless and/or its affiliates aggregate liability arising under or in connection with the Sierra Wireless product, regardless of the number of events, occurrences, or claims giving rise to liability, be in excess of the price paid by the purchaser for the Sierra Wireless product.

Patents

This product may contain technology developed by or for Sierra Wireless Inc. This product includes technology licensed from QUALCOMM®. This product is manufactured or sold by Sierra Wireless Inc. or its affiliates under one or more patents licensed from InterDigital Group and MMP Portfolio Licensing.

Copyright

© 2017 Sierra Wireless. All rights reserved.

Trademarks

Sierra Wireless®, AirPrime®, AirLink®, AirVantage® and the Sierra Wireless logo are registered trademarks of Sierra Wireless.

Windows® and Windows Vista® are registered trademarks of Microsoft Corporation.

Macintosh® and Mac OS X® are registered trademarks of Apple Inc., registered in the U.S. and other countries.

QUALCOMM® is a registered trademark of QUALCOMM Incorporated. Used under license.

Other trademarks are the property of their respective owners.

Contact Information

Sales information and technical support, including warranty and returns	Web: sierrawireless.com/company/contact-us/ Global toll-free number: 1-877-687-7795 6:00 am to 6:00 pm PST
Corporate and product information	Web: sierrawireless.com

Revision History

Revision number	Release date	Changes
1	June 23, 2017	Initial draft in SWI template.

>> Contents

Introduction	6
Precautions	7
Technical Support	8
General Rules for Design	9
Circuit Design	9
Power Supply - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110	9
VBACKUP backup battery - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110 ..	10
UART / I2C/ SPI Serial Interface	10
GND-Ground - Compatible Models: Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110	12
Antenna Compliance Design - Compatible Models: Gmm-g3, Gmm-g3(B), XM1110, XA1110	12
1PPS - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110	17
3D_Fix - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6	19
1.8V Boost to 3.3V Application	21
Layout Guidelines	22
Layout Underneath the GNSS Module - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110	22
Keep Far Away from High Profile or Metal-Canned Components - Compatible Models: Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XA1110	25
Placement - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110	26
Trace - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110	26
Ground Segmentation - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110	27
Ground Plane - Compatible Models: Gms-g9, PA6E-CAM, Gms-g6, Gms-b6, XA1110	27
Metal Effect on the GNSS Module Compatible Model: Gms-g6a	29

Thermal Profile for SMD Modules	30
SMT Reflow Soldering Temperature Profile	30
SMT Solder Mask	30
Manual Soldering	30
Troubleshooting	31
How to Check the Working Status of the GPS Module	31
Super Capacitor Design	33
About Super Capacitors	33
How to Calculate the Backup Time	33
.....	33
50 Ω Antenna Matching	34
UART to RS232 Interface	35
UART to USB Interface	36
How to Efficiently Transfer 1PPS Through Extended Distances	37
1pps Delay Time	38
The Delay Time Caused by the Cable	39
Calculating the Delay Time in Respect to the Communication Cable Length	39
Waveform Rising Time Caused by OP-amp	40
Recommended OP-amp	40
Voltage degradation of Communication cable	40
Reflow Soldering Precautions	41
Other Cautionary Notes on the Reflow-Soldering Process	43
Manual Soldering	44
Soldering Iron	44

1: Introduction

The purpose of this application note is to assist the system developer and integrator with the MT3333 chip-based Sierra Wireless receiver modules during the process of implementation. This document covers the following receiver modules:

Standard Modules

- XM1110
- Firefly 1 (Gmm-g3)
- Firefly 1b (Gmm-g3b)

Antenna Modules

- XA1110
- Titan 1 (PA6E-CAM)
- Titan 2 (Gms-g6)
- Titan 2B (Gms-b6)
- Titan 3 (Gms-g6)
- Stand Alone GPS module: FireFly 1 & FireFly 1b (Gmm-g3, Gmm-g3(B)) and XM1110

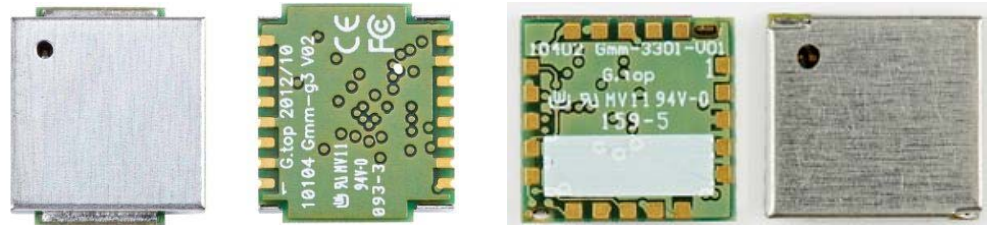


Figure 1-1: Stand Alone GPS modules. Left: FireFly 1 & FireFly 1b (Gmm-g3, Gmm-g3(B)). Right: XM1110

- Built-in Chip Antenna Module: Carrion 2 (Gms-g6a)



Figure 1-2: Built-in Chip Antenna Module: Carrion 2 (Gms-g6a)

- Built-in Patch Antenna Modules: Titan 2 (Gms-g6) and Titan 2B (Gms-b6), Titan 3 (Gms-g9) and Titan1 (PA6E-CAM), and XA1110.



Figure 1-3: Titan 2 (Gms-g6) and Titan 2B (Gms-b6)

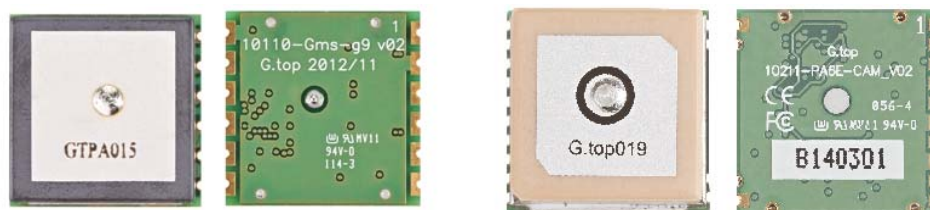


Figure 1-4: Titan 3 (Gms-g9) and Titan1 (PA6E-CAM)



Figure 1-5: XA1110

Precautions

Please read carefully before you start

If you use the GNSS receiver inside buildings, tunnels, or beside any huge objects, the GNSS signals might be cut-off or weakened. Please do not assume the receiver has malfunctioned.

This application note provides the necessary guidelines for a successful system design using GNSS modules. For detailed module specifications, please refer to the corresponding datasheet of the GNSS module.

The GNSS module is an electrostatic sensitive device, please DO NOT touch the GPS module directly. Follow ESD safety rules when handling.

When using the device for the first time, it is strongly recommended to test the device outdoors with open sky for at least 10 to 15 minutes to ensure that full ephemeris data received.

Technical Support

If you have any technical problems or cannot find the required information in our documents, please feel free to contact us for technical support. Below is a list of information which you can provide to us to help us in determining the source of the problem and the necessary solution:

1. Your company name and website.
2. Description about application and system.
3. GNSS module type.
4. GNSS firmware version.
5. Description of the question or problems encountered, together with figures, pictures or videos files.
 - a. Test setup.
 - b. The problem or issue shown in figures.

2: General Rules for Design

This section provides some rules when using the GNSS module to obtain good GNSS performance.

Circuit Design

Power Supply - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

It is necessary to provide a clean and stable power supply for our GNSS module in order to obtain good performance. An unstable power source will have a significant negative impact on the performance. To achieve high-quality performance, the VCC ripple must be controlled under 50mVpp. In addition, there are also some important suggestions for main power circuit design:

1. Add a ferrite bead, power choke, or low pass filter for power noise reduction.
2. A linear regulator (LDO) is better than a switch DC/DC power supplier in the ripple.
3. Use enough decoupling capacitors beside the VCC for stable voltage.

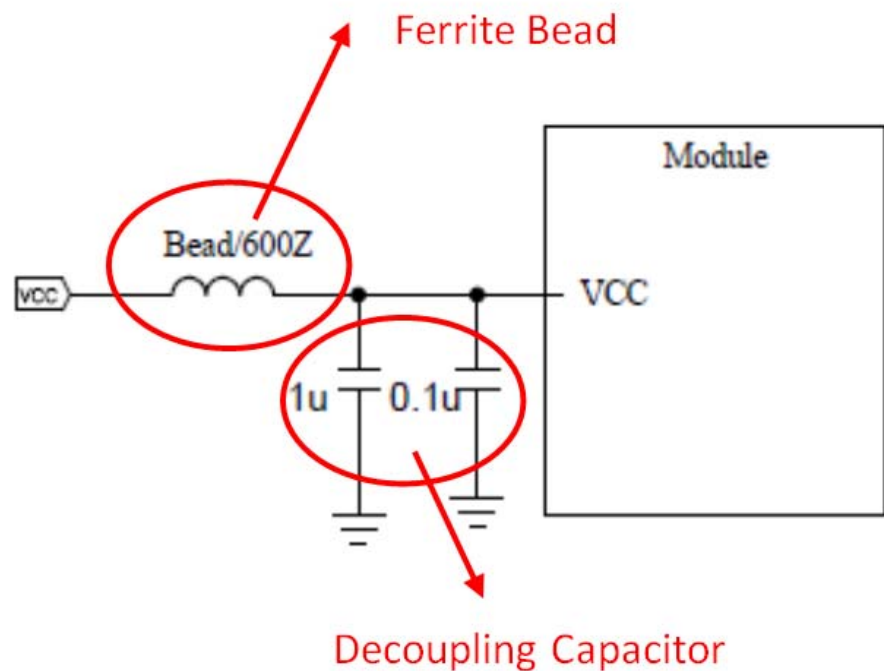


Figure 2-1: Power Design for the GNSS Module

VBACKUP backup battery - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

The GNSS module has a built-in charging circuit which charges the rechargeable coin battery.

For most systems, it is recommended to provide the module with backup power (e.g. Li-Ion rechargeable coin battery, super capacitor). See [Figure 2-2](#) for a reference design.

For information on the super capacitor reference design, please refer to [Super Capacitor Design](#).

Backup power is useful in order to maintain RTC operation and retain Ephemeris data in flash memory which can get a faster TTFF and acquire PVT (Position, Velocity, Time) information.

If *VBACKUP* isn't connected to any coin battery, the GNSS module will execute a cold start whenever the system is restarted.

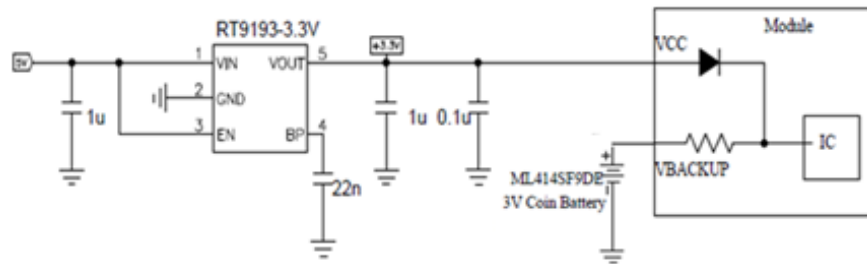


Figure 2-2: Built-in a charging circuit for GNSS module

UART / I²C/ SPI Serial Interface

UART 0 (RX/TX) Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

1. UART is the TTL level interface that carries the baud rate ranging from 4800 bps to 115200 bps.
2. Placing a damping resistor on the RX and TX of the GNSS module could limit the interference from the host MCU or high speed digital logics. Fine tuning the damping resistor is required to efficiently suppress interference. The damping resistor is a wire wound component and may function as a choke coil.

3. Please don't connect diode(s) to RX/TX as it will decrease the signal driving capability which might adversely affect the RX/TX signal level. In some cases no data output will occur.
4. If RS232 logic-level is needed for any particular application, the level shifter will be necessary. For more information please refer to [UART to RS232 Interface](#).
5. If USB logic-level is needed for any particular application please refer to [UART to USB Interface](#).

I²C (SCL/SDA) - Compatible Models: XM1110, XA1110

1. The I²C interface is mainly connected to external devices. MT3333 supports only slave mode (default slave address is 0x10). MT3333 has 256 bytes URAM mode and 8-byte FIFO mode for transmitting and receiving data. The bit rate is up to 400 kb/s for fast mode. In addition, MT3333 supports manual or automatic indicator for data transfer in the slave mode. Device addresses in the slave mode are programmable.
2. Pull-up resistor must be added for I²C bus:

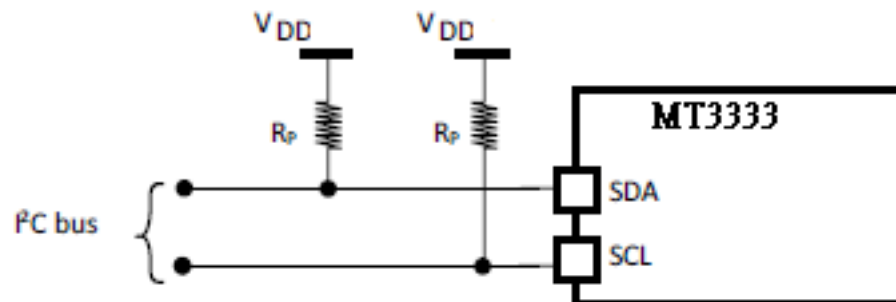


Figure 2-3: Addition of Pull-up Resistor

SPI (CS/CLK/MISO/MOSI) - Compatible Model: Gmm-3301, XA1110

1. The serial peripheral interface port manages the communication between the digital BB and external devices. The MT3333 supports only slave mode. The slave has a 4 byte-register mode or URAM mode. In URAM mode, the transmitted and received data size is 256 bytes. The clock phase and clock polarity are selectable. The MT3333 supports a manual or automatic indicator for data transfer in slave mode. The bit rate is up to 700kb/s.

GND-Ground - Compatible Models: Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

Make sure all GND pins of the module are connected to a completed ground plane.

The separation of ground between the GNSS module and the rest of the system are recommended to avoid interference. If this setup is not possible, it is best to follow these rules: divide the segmentation of the ground between the digital and analogue system, high current and low current system, and different radiation systems in general (such as GNSS and GPRS).

One method to segment the ground is to place digital and noise components at one corner of the board, while placing analog and quiet components at the opposite corner of the board. Make sure there is no crossing of microstrip or current between the two component sets with the ground of each set to be connected to one point only.

Another method is to place the two different sets at different layers of the board while the ground of each layer is connected to one point only which is preferably located at the border of the board practically.

Antenna Compliance Design - Compatible Models: Gmm-g3, Gmm-g3(B), XM1110, XA1110

The GNSS antenna is a receiving part of the device to acquire weak GNSS signals from the sky. A common solution would be to use a ceramic patch antenna because of its small form factor and low cost. There are two types of antennas: passive and active.

A passive antenna, like a patch antenna and chip antenna, is solely by itself without signal amplifier such as LNA. It must be fine tuned for the specific module model to obtain the best signal receiving performance.

An external antenna is a standalone device, which integrates an LNA and patch antenna with an RF cable for higher gain. This performs better than a passive antenna.

The antenna can be chosen according to the radiation efficiency, radiation pattern, gain, bandwidth, form factor, and cost. Make sure the ground plane is sufficient for the antenna to ensure better performance.

Design Patch Antenna with GNSS module.

1. In general, a 50Ω patch antenna will work well with the GNSS module. The antenna can be connected to the *Antenna IN* pin with a 50Ω impedance trace.
2. Please keep the patch antenna far away from noise sources such as the switching power supply and high speed digital logic and radar wave guide.

3. The 50Ω trace should be kept as short as possible to reduce the chance of picking up noise from the air and PCB. A simple direct-line trace is recommended.
4. If needed, a matching circuit could be placed between the patch antenna and the GNSS module. The matching circuit design should be discussed with the module and patch antenna manufacturer.
5. For 50Ω matching, please refer to [50 Ω Antenna Matching](#).

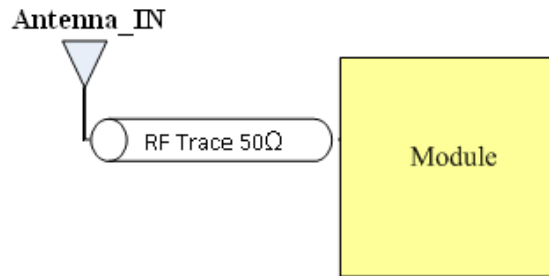


Figure 2-4: PCB trace design for antenna impedance matching

Selecting an External Antennal Architecture for the GNSS Module

An external antenna requires DC power in order to work properly. A typical method is to feed DC into the RF trace. The external antenna then extracts the DC from the RF trace. Thus the RF trace transports both the RF signal and DC power. An RF chock coil couples the DC power to the RF trace to perform this method.

The Sierra Wireless modules provide a circuit for connecting an external antenna and power feeding method as depicted in [Figure 2-5](#), [Figure 2-6](#), and [Figure 2-7](#) below. Please select the module prudently while applying the design with an external antenna.

- **Mode1 (i.e. Gmm-g3, Gmm-g3(B)):** the power supply for the external antenna needs to be externally fed into the module via the *VANT* pin and the antenna connected to the *Antenna IN* pin:

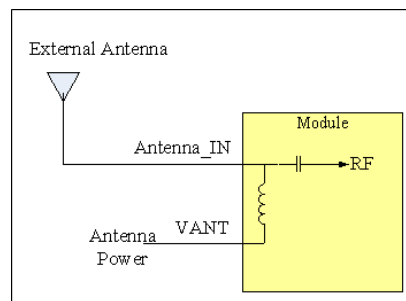


Figure 2-5: Mode 1: (Internal chock coil)

- **Mode2 (i.e. XM1110):** The power supply needs to be externally provided and is connected directly to the external antenna via a chock coil.

Note: Chock Coil for reference: LQG15HS33NJ02D (Murata)

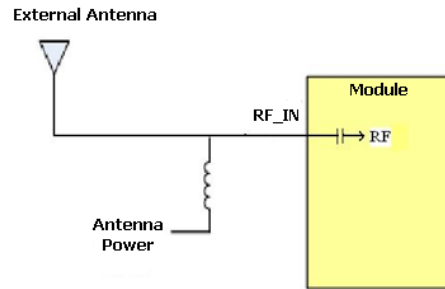


Figure 2-6: Mode 2: (External chock coil)

Note: Chock Coil for reference: LQG15HS33NJ02D (Murata)

- **Mode3 (i.e. PA6E-CAM, XA1110):** The power supply for the external antenna needs to be externally fed into the module from the VCC pin directly and connect antenna to EX_ANT pin.

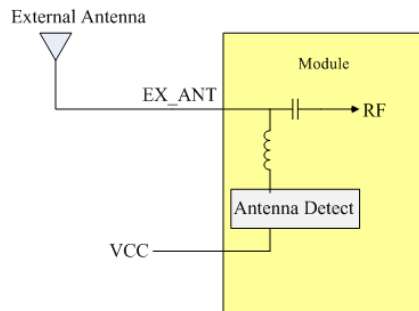


Figure 2-7: Mode 3: (Internal chock coil)

Mode \ Module	Gmm-g3, Gmm-g3(B)	XM1110	PA6E-CAM XA1110
Mode 1(via the VANT) (Internal chock coil)	✓		
Mode 2 (External chock coil)		✓	
Mode 3(from the VCC) (Internal chock coil)			✓

Figure 2-8: Modes

Design chip antenna with GNSS module

Different vendors *have* different reference designs, so we list two chip antenna sources: *Pulse* and *Unictron*.

1. Antenna source: Pulse

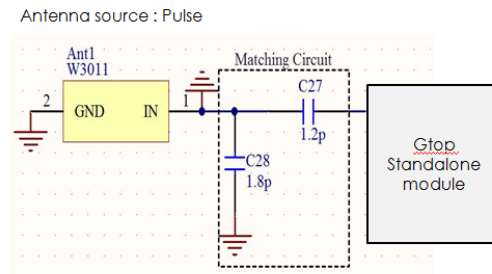


Figure 2-9: Pulse Schematic Design

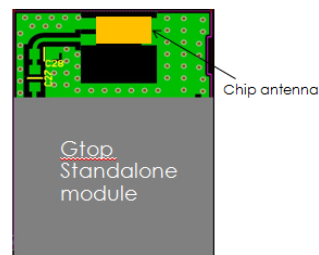


Figure 2-10: Pulse PCB Layout

When selecting the Antenna source, check its datasheet first before tuning an RF match component's footprint (such as: C27, C28). You can base this on its PCB size and housing to tune for an optimal value and meet the GNSS's frequency in order to get a good reception effect.

Pulse web site:

www.pulseeng.com/antennas

Note: C27, C28's value are still based on your actual trace to tune it.

2. Antenna source: Unictron

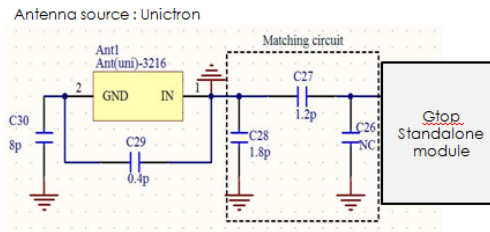


Figure 2-11: Unictron Schematic

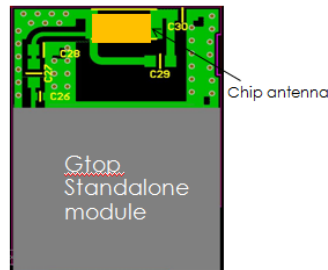


Figure 2-12: Unictron PCB

When you select this source, check its datasheet first before tuning the RF match component's footprint (such as: C26, C27, C28, C29, C30). You can tune based on the PCB size and housing to tune for an optimal value in order to meet the GNSS's frequency and obtain good reception.

Unictron web site:

<http://www.unictron.com/index/>

Note: C26,C27,C28,C29,C30's value are still based on your actual trace to tune it.

External Antenna of Specification

When select your external antenna, refer to the specification shown in [Table 2-1](#) and [Table 2-2](#) below.

The Gmm-g3 module uses the GPS/GLONASS External Antenna:

Table 2-1: GPS/GLONASS External Antenna

Characteristic	Specification
Polarization	Right-hand circular polarized
Frequency Received	1.575GHz~1.615GHz
Power Supply	3.3V
DC Current	5mA<IDC<11mA at 3.3V
Total Gain	27±3dB
Output VSWR	2.0
Impedence	50Ω
Noise Figure	2dB

The Gmm-g3(B) module uses the Beidou/GPS External Antenna:

Table 2-2: Beidou/GPS External Antenna

Characteristic	Specification
Polarization	Right-hand circular polarized
Frequency Received	1.561GHz~1.575GHz
Power Supply	3.3V
DC Current	3mA<IDC<30mA at 3.3V
Total Gain	27±1dB
Output VSWR	2.0
Impedence	50Ω
Noise Figure	1.5dB

1PPS - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

1PPS signal is an output pulse signal used for timing applications. Its electrical characteristics are:

- Low Voltage level: 0~0.4V
- High Voltage level: 2.8~3.1V

- Period: 1s
- Accuracy (jitter): ± 20 ns
- 100ms pulse width duration (firmware customization for pulse width duration is available)

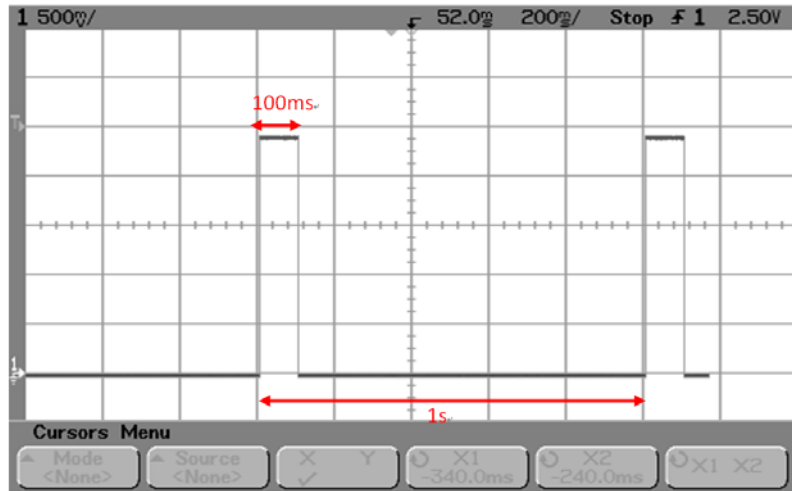


Figure 2-13: 1PPS signal and its pulse width with 100ms duration

Free run 1PPS Output Before 3D_FIX

The Sierra Wireless standard GNSS module outputs 1PPS signal after the module obtains a 3D_FIX. This is a factory default setting.

Cable Delay Compensation

In some cases a long distance connection (~300m) may be needed. For a timing application, the cable length is critical. For more information on 1PPS signal transmission delay please refer to [How to Efficiently Transfer 1PPS Through Extended Distances](#).

LED Indicator for 1PPS Signal

For 1PPS LED indication, you may connect an LED indicator with a 330ohm resistor in series.

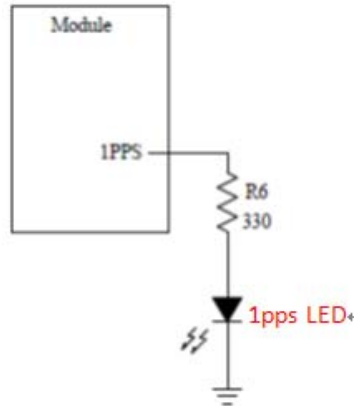


Figure 2-14: 1PPS Signal Design for IO

3D_Fix - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6

3D_Fix signal is an output pulse signal used for GPS fix applications. Its electrical characteristics are:

- Low Voltage level: 0~0.4V
- High Voltage level: 2.8~3.1V
- Source current (Max.): 14mA, sink current 14mA

Signal Output Before 2D Fix:

Duration: 1 second high and 1 second low.

Period: 2s

(Firmware customizable)

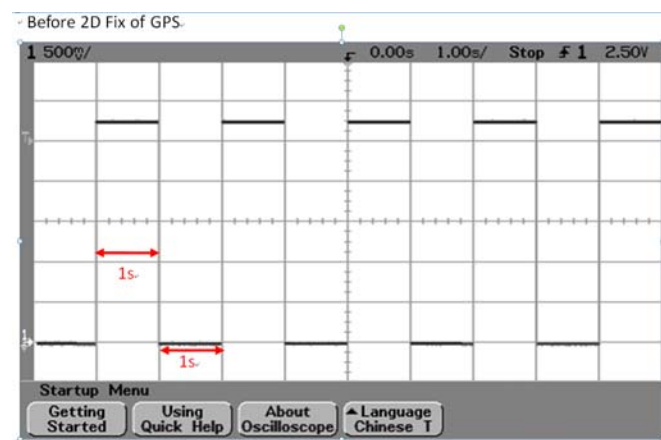


Figure 2-15: Signal Output Before 2D Fix

Signal Output After 2D or 3D Fix

Duration: Constant low level output

(Firmware customizable)

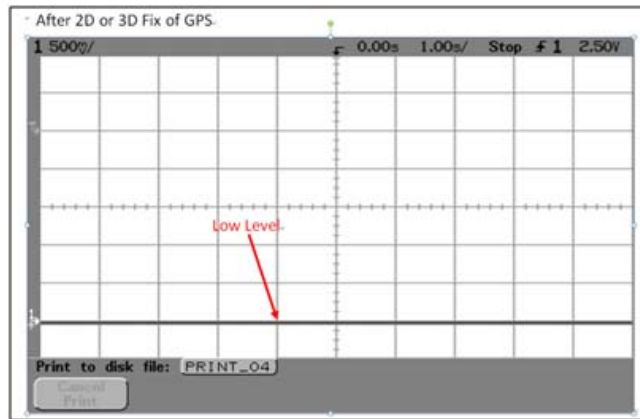


Figure 2-16: Signal output after 2D or 3D Fix

For 3D_Fix indication, you may connect an LED with a 330ohm resistor in series in two ways as shown in [Figure 2-17](#) and [Figure 2-18](#).

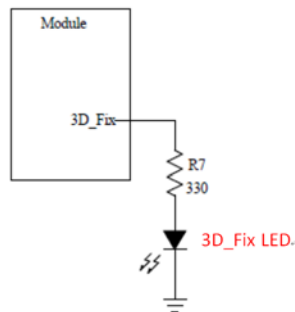


Figure 2-17: 3D_Fix signal design for IO (LED ON when high, LED OFF when low)

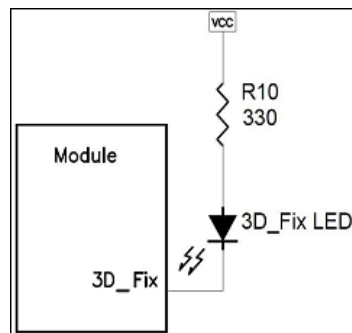


Figure 2-18: 3D_Fix signal design for IO (LED ON when low, LED OFF when high)

1.8V Boost to 3.3V Application

If you want to use the 3.3V's GNSS module in the 1.8V system there are two considerations: power supply translation and signal level shift. For power supply translation, you can use a boost circuit which can boost 1.8V to 3.3V (refer to [Figure 2-19](#) below).

TPS61097A-33 is the Ti's boost IC which can support boost functionality. In its application C_1 and C_2 need to use 10 μ F and L_1 is 10 μ H. It can support approximately 100mA of output. For information about the capacitor and inductor's placement, refer to the application note on the Ti web site:

<http://www.ti.com/lit/ds/symlink/tps61097a-33.pdf>

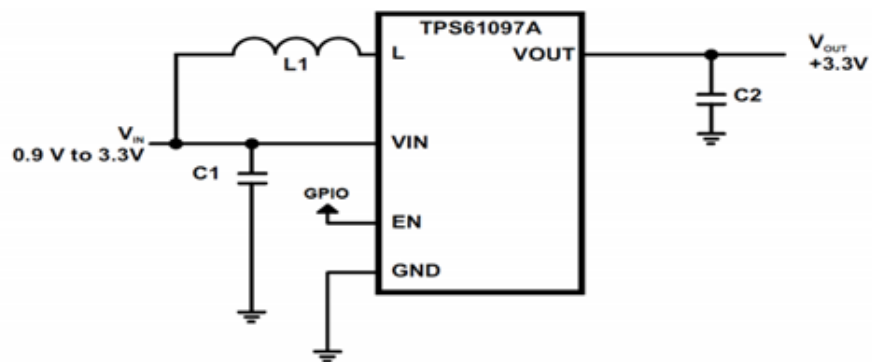


Figure 2-19: Application Schematic

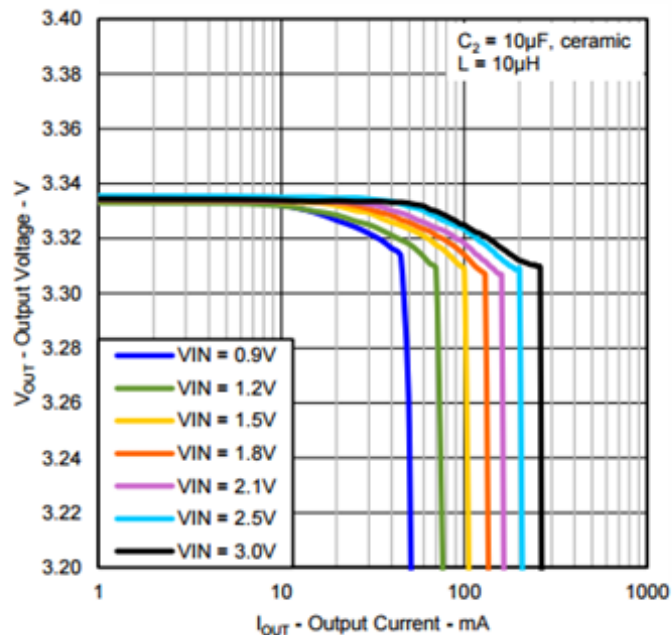


Figure 2-20: Output Voltage vs Output Current

For signal level shift, when your host system is 1.8V and the Sierra Wireless module is 3.3V, the host system can control the Sierra Wireless module by using a signal level shift circuit (refer to the [Figure 2-21](#) below). R1 to R4's values are default values. In actual design, you can adjust their values to achieve control. The *2N7002L* can select low RDS(On) to reduce power consumption through a voltage drop.

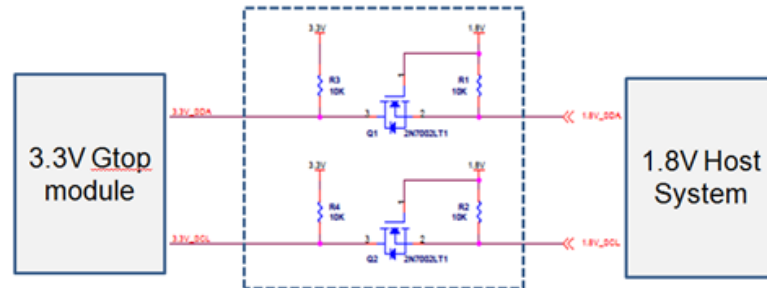


Figure 2-21: Signal Level Shift Circuit

Layout Guidelines

Please follow the layout guideline for the right GPS module during the design process.

Layout Underneath the GNSS Module - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E- CAM, XM1110, XA1110

In general, GNSS modules have high receiving sensitivity at around -165dBm. During hardware integration, try to avoid noise or harmonics in the following bands to prevent unnecessary reception degradation:

- Beidou 1561.098MHz \pm 2.046MHz and GPS 1572.42MHz \pm 2MHz
- GLONASS 1598.0625~1605.375MHz

Modern GNSS positioning products include many components such as an LCD screen, MCU, high speed digital signal (access memory card), and an RF system (i.e. Cellular module, BT, Wi-Fi, DVB-T). Keep these components away from the GNSS module in order to avoid the noise interference, otherwise poor GNSS RF reception may result.

For modules Gmm-g3, Gmm-g3(B) and XM1110

Don't leave any trace or mark underneath the GNSS module as indicated by the circled area in figure [Figure 2-22](#) below. In other words, provide the GNSS module with a clean GND plane.

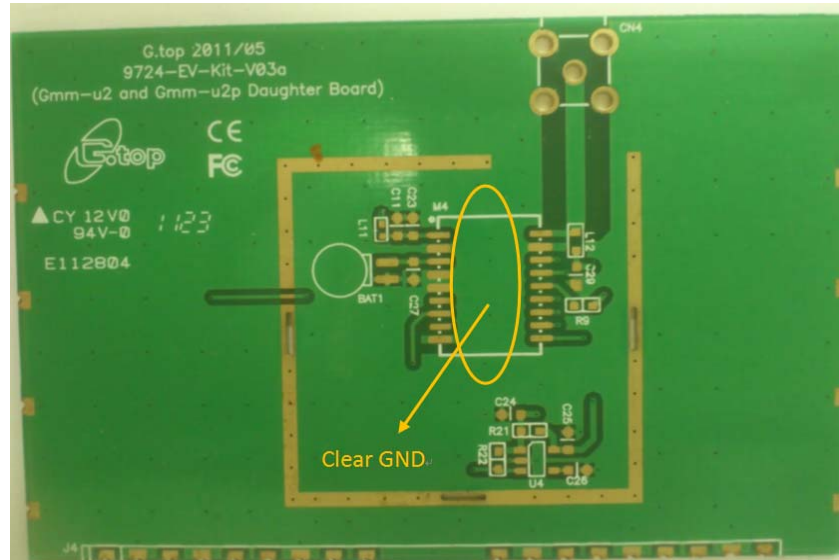


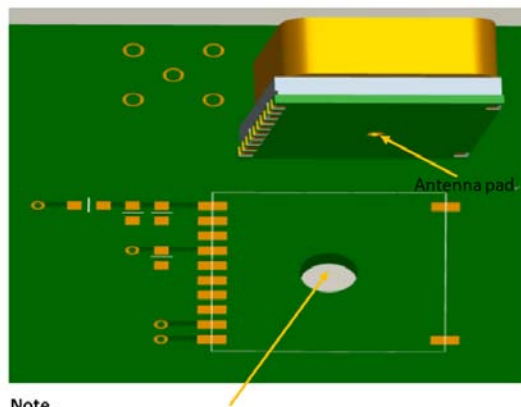
Figure 2-22: GPS Module on a clean GND Plane

Don't place any trace such as I²C (SCL/SDA), SPI (CLK/MISO/MOSI), UART (TX/RX) underneath the GNSS module, otherwise it will cause a sensitivity decrease.

For Modules Gms-g9, Gms-g6, Gms-b6 and PA6E-CAM

Place a hole according to the location of the RF feeding pin of the module. The hole will provide an area to fit the RF feeding pin on your system PCB and provide the appropriate hole size.

Don't place any high speed trace such as I²C (SCL/SDA), SPI (CLK/MISO/MOSI), UART (TX/RX) underneath the GNSS module, otherwise it will cause a sensitivity decrease.



Note.
Place one hole (diameter = 3.0mm) under this module as part of the antenna pad and please don't let any trace and via pass the area.

Figure 2-23: Hole Placement

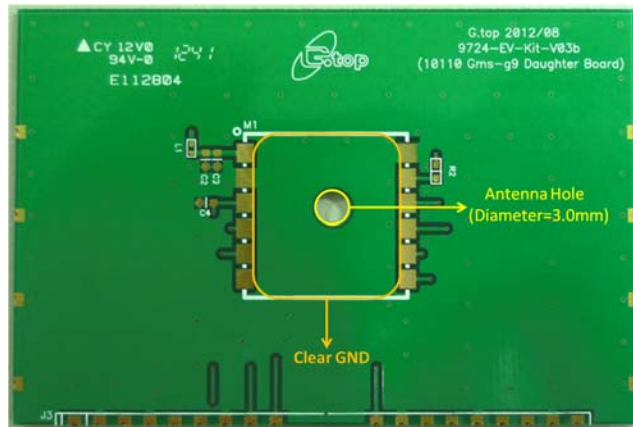


Figure 2-24: GPS on a clean GND plane and antenna hole

For Module XA1110

In order to avoid the interference, place many vias on the two sides of the RF trace which is from the module to the SMA/RF connector on your system PCB as illustrated in xxx below.

Don't place any high speed trace such as I²C (SCL/SDA), SPI (CLK/MISO/MOSI), UART (TX/RX) underneath the GNSS module, otherwise it will cause a sensitivity decrease.

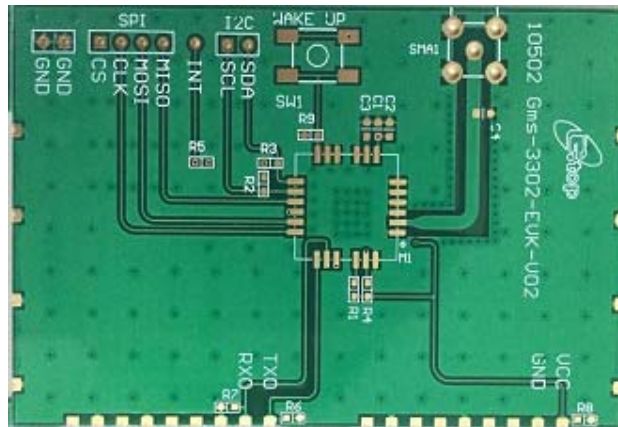


Figure 2-25: Avoiding Interference

Keep Far Away from High Profile or Metal-Canned Components - Compatible Models: Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XA1110

It is good practice to place the GNSS module far away from any high-profile components especially with metal case enclosed components such as the E-CAP, coin battery, and Pin Header. The antenna field pattern can be affected and pattern distortion can occur. At worst, this will decrease the GNSS signal up to ~10dB.



Figure 2-26: First Example of bad and good GNSS module placements

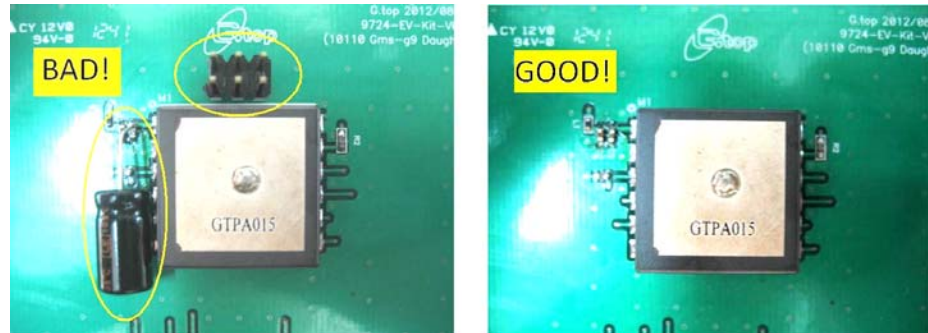


Figure 2-27: Second Example of bad and good GNSS module placements

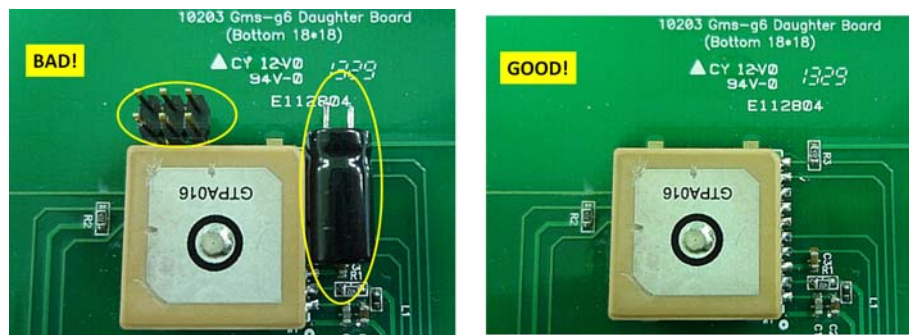


Figure 2-28: Third Example of bad and good GNSS module placements

Placement - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

- Place the decoupling capacitors for the *VCC* close to the GNSS module.
- Place the damping resistors for *TX/RX* close to GNSS module.

Do not place:

- the GNSS module in proximity to high-speed digital processing circuitry
- the GNSS module in proximity to high-current switching power circuitry
- the GNSS module in proximity to clock sources circuitry

Trace - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

1. The USB differential signals should be traced closely and of equal length for better noise immunity and minimum radiation.
2. Apply a 50 ohm impedance RF trace for correct impedance matching.
3. Any right angle turn in trace routing should be done with two 135 degree turns or an arc turn.

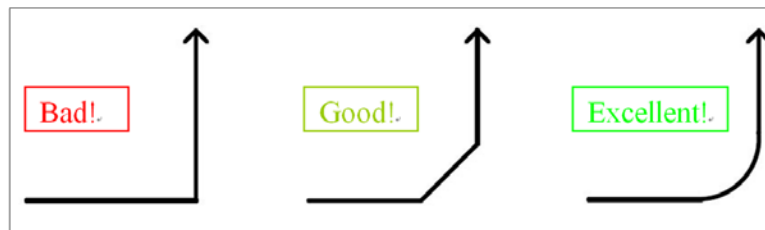


Figure 2-29: Examples of turns in trace routing

It is better to have an independent trace of the power source for any device:

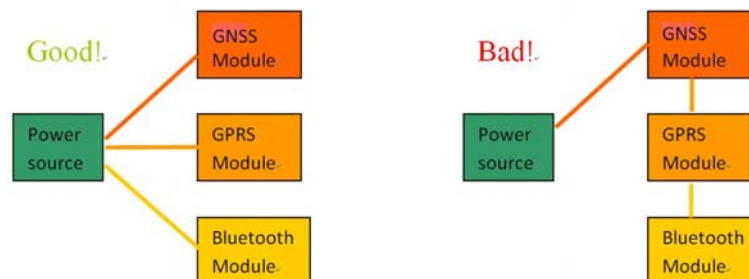


Figure 2-30: Examples of Independent Trace

Ground Segmentation - Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, XM1110, XA1110

In general, the separation of ground between the GNSS module and the rest of the system is recommended to avoid interference. If this is not possible, it is best to follow these typical rules: segmentation of ground between digital and analogue system, high current and low current system, and different radiation systems such as GNSS and GPRS. One way to segment the ground is to place digital and noise components at one corner of the board, while placing analog and quiet components at the opposite corner of the board. Make sure there is no crossing of microstrip or current between the two component sets with ground of each sets to be contacted to one point only.

Another method is to place the two different sets at different layers of the board, while the ground of each layer is contacted at one point only which is practically located at the border of the board.

Ground Plane - Compatible Models: Gms-g9, PA6E-CAM, Gms-g6, Gms-b6, XA1110

For Gms-g9, PA6E-CAM, Gms-g6, Gms-b6, XA1110

A large GND plane directly underneath the module could enhance the magnetic-field line of the antenna for better GNSS signal reception. Typically it will improve ~2dB maximum. It is strongly recommended to have a ground plane designed underneath the GNSS module as big as possible.

The recommended thickness for the ground layer is 0.5 to 1 OZ (0.0175 to 0.035 mm). It is best to place the ground plane on the top layer of the PCB, directly underneath the GNSS module as shown in [Figure 2-31](#):

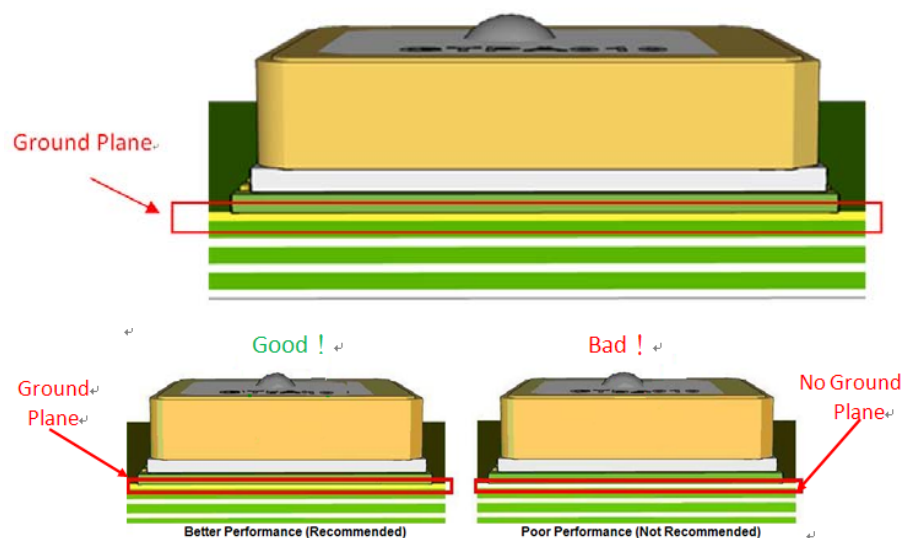


Figure 2-31: Ground Plane for FR4 Design

Compatible Model: Gms-g6a

For the Gms-g6a with a built-in chip antenna module, a special ground plane is required to improve chip antenna reception performance.

As shown in the circled area of figure [Figure 2-32](#) where the chip antenna area is when placed on the PCB, no ground plane should be placed on this chip antenna area or all other layers underneath:

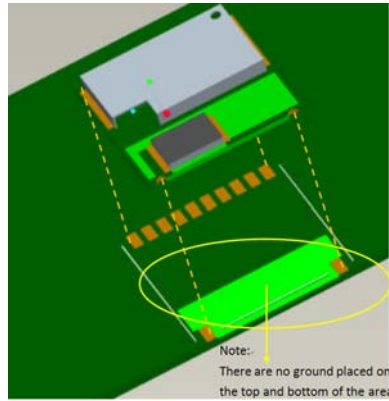


Figure 2-32: Area where no ground plane should be placed

Note: The clearance area size at least 15x6.3mm.

To enhance the antenna efficiency for better GNSS signal reception, it is recommended to cover the rest of the PCB top layer with ground plane except where the chip antenna seats as shown in [Figure 2-33](#):

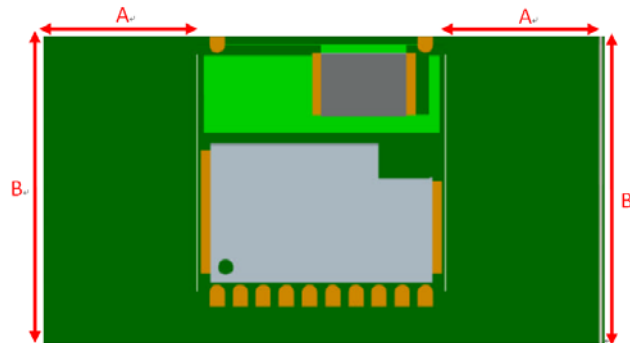


Figure 2-33: Variation in reception sensitivity according to the different size of the GND plane

Metal Effect on the GNSS Module

Compatible Model: Gms-g6a

The Gms-g6a chip antenna is a sensitive component with the environment.

Care consideration should be taken regarding the housing and the environment where it is placed:

1. The housing (case) of the host device should not be made of or include any conductive material for the basic antenna reception.
2. The host device should not be placed underneath conductive material such as a metal case, metal film, or plastic housing coated in conductive material, which blocks the GNSS signal from reaching the antenna. If there is metal material placed underneath the chip antenna, this will also decrease the antenna reception performance.

As shown in the figure, the distance (d) between metal and module should be kept as far as possible.

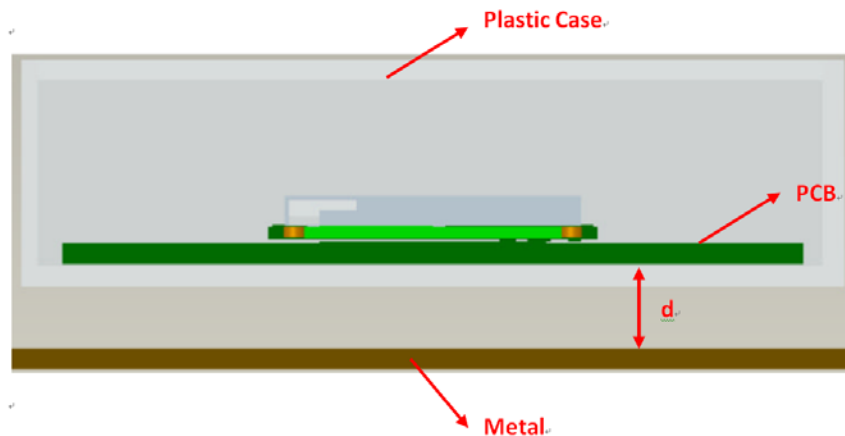


Figure 2-34: Distance between metal and module should be as large as possible

Table 2-3 shows how reception performance can be negatively affected in regard to the distance between metal and the mounted module.

Table 2-3: Decrease in Sensitivity due to Distance

Distance (MM)	Sensitivity Decrease in dB
0mm	-14 dB ~ -18dB
1mm	-7dB~ -9dB
3mm	-6dB ~ -7dB
5mm	-3dB~ -5dB
7mm	-2dB~ -4dB
10mm	0dB

3: Thermal Profile for SMD Modules

The following information is Pb-Free compliant. The details are for reference only.

SMT Reflow Soldering Temperature Profile

- Average ramp-up rate (25 ~ 150°C): 3°C/sec. max.
- Average ramp-up rate (270°C to peak): 3°C/sec. max.
- Preheat: 175 ± 25°C, 60 ~ 120 seconds
- Temperature maintained above 217°C: 60~150 seconds
- Peak temperature: 250 +0/-5°C, 20~40 seconds
- Ramp-down rate: 6°C/sec. max.
- Time 25°C to peak temperature: 8 minutes max.

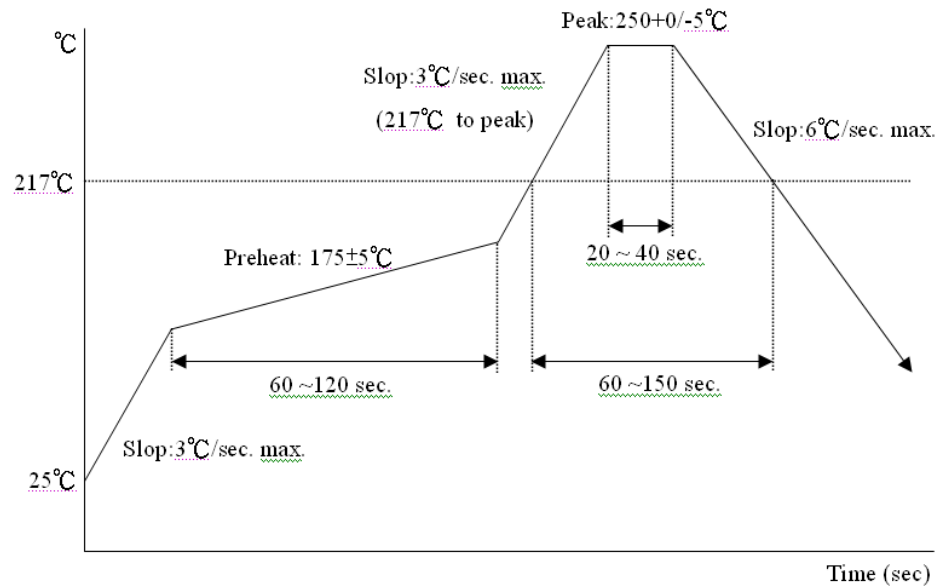


Figure 3-1: SMT Reflow Soldering Temperature Profile

SMT Solder Mask

Please use the dimension of PCB pad as reference and shrink the size by 0.1 to 0.2 mm and use that as layout for paste mask (for PCB pad layout, please see "Recommended PCB pad layout" on the individual GPS module data sheet).

Manual Soldering

- Soldering iron: Bit Temperature: under 380°C | Time: under 3 second.

4: Troubleshooting

How to Check the Working Status of the GPS Module

The first thing to check for is the NMEA sentence output through *TX* using various application tools. For example: you may use the Windows tool: Hyperterminal or you may use another GPS application program.

If there is no NMEA output at the *TX* pin, this indicates that the module is currently not working. Please double check your schematic design. The following is a list of possible items to check:

Item 1: VCC

The voltage should be kept between 3V to 4.3V (typical: 3.3V).

Item 2: VBACKUP

The voltage should be kept between 2.0V~4.3V (typical: 3.0V). If a backup battery is not installed for the *VBACKUP* pin, the pin could still be measured as the voltage comes from the built-in battery recharging circuit. It is recommended to provide power to the *VBACKUP* pin as it is used to keep RTC time running and preserve stored navigation data.

Item 3: 3D-FIX

If all the measurements are within the specifications, please also measure the *3D-FIX* signal. Before 2D Fix, the pin should output one second high followed by a one second low signal.

Note: this is not applicable to those with a customized 1PPS which always has a 1PPS signal configuration as described in [1PPS - Compatible Models: Gmm-g3, Gmm-g3\(B\), Gms-g6a, Gms-g9, Gms-g6, PA6E-CAM, XM1110, XA1110](#).

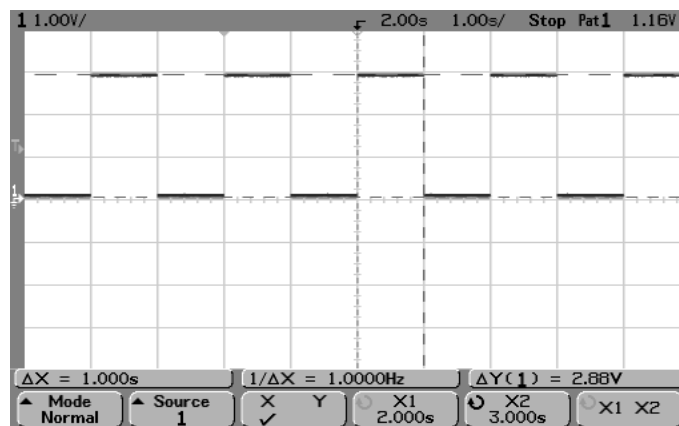


Figure 4-1: 3D-Fix

Item 4: TX0

The UART transmitter of the module outputs the GPS NMEA information for the application.

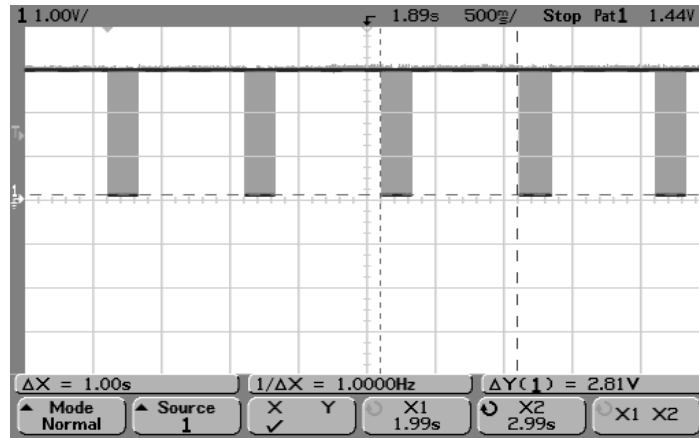


Figure 4-2: TX0

Appendix A: Super Capacitor Design

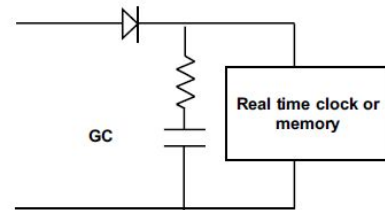
About Super Capacitors

Super capacitors have a lifetime similar to that of aluminum electrolytic capacitors. The service life of a super capacitor is greatly dependent on the operating temperature, humidity, applied voltage, current, and backup time. Therefore, the service life is determined based on the backup time set by the customer.

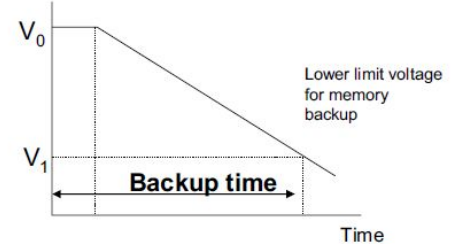
How to Calculate the Backup Time

The example below shows how to calculate the backup time:

$$T = C (V_0 - V_1) / (I + I_L)$$



Capacitor's voltage



Example:

$$V_0 = 2.5V, \quad V_1 = 1.8V, \quad I = 10\mu A, \quad C = 0.2F$$

$$\begin{aligned} T &= 0.2 (2.5 - 1.8) / (10 \text{ e-}6 + 0.2 \times 10^{-6}) \\ &= 0.14 / 10.2 \times 10^{-6} \\ &= 13,725 \text{ seconds} \\ &= 3.8 \text{ hours} \end{aligned}$$

T: Backup time (second)
 C: Capacitance of Gold Capacitor (F)
 V_0 : Applied voltage(V)
 V_1 : Lower limit voltage for memory backup(V)
 I: Current during backup(A)
 I_L : Leakage current(A) about 0.2uA

Figure A-1: Calculating backup time

Appendix B: 50 Ω Antenna Matching

We used the *AppCAD* tool to simulate 50 Ω impedance for the RF PCB layout.

Table B-1: Antenna Matching

RF Line Width (W)	PCB FR4 Thickness	Dielectric Parameters	Copper Thickness per ounce
1.8mm	1mm	4.6	0.035mm

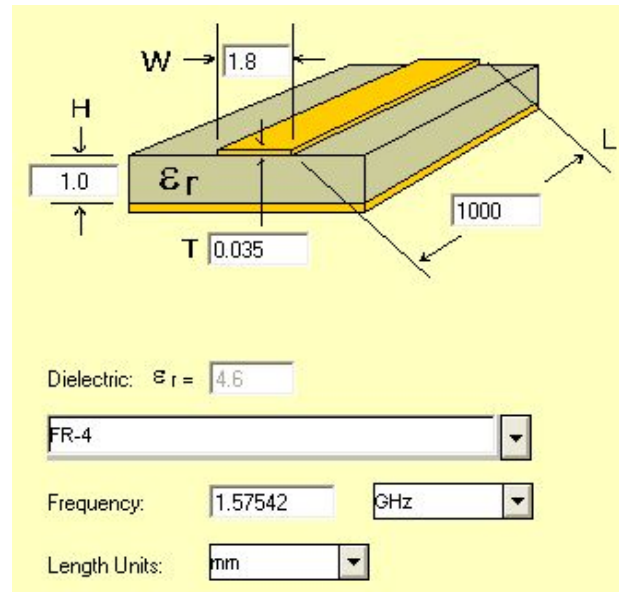


Figure B-1: Dimensions

Notice:

For multi-layer layouts, you could place a ground layer in the second layer to minimize the trace width in a specific PCB (such as FR4) and impedance.

For impedance calculation, there is free software available to calculate the trace width or impedance. Once such software package is:

<http://www.awrcorp.com/products/additional-products/tx-line-transmission-line-calculator>

Appendix C: UART to RS232 Interface

Typically an RS232 or USB interface is required to connect the PC to the Sierra Wireless GNSS module for communication. The majority of Sierra Wireless modules use a set of communication ports in TTL-logic. Some newer ones support direct USB connection. A bridge IC is needed for RS232 signal conversion.

The supported baud-rates are 4800, 9600, 14400, 19200, 38400, 57600, and 115200 bps.

Please refer to the reference circuit in [Figure C-1](#) below for RS232 signal conversion. An SP3320 IC is used here as an example.

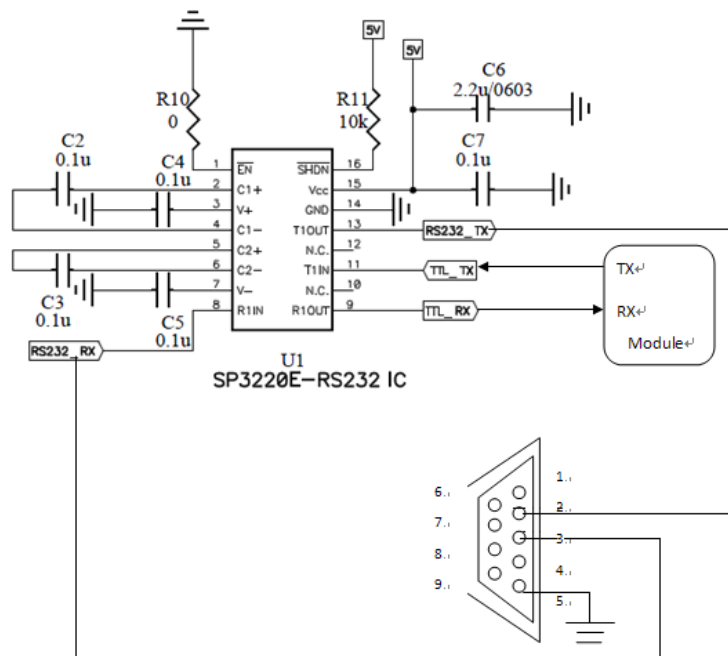


Figure C-1: RS232 Signal Conversion Example

Appendix D: UART to USB Interface

If the Sierra Wireless module you have purchased does not come with a USB interface, it is possible to connect the module to an external USB IC. To further enhance the transferring speed, use one that is capable of USB version 2.0. Once the driver for the chosen USB Bridge IC is successfully installed onto Windows or another operating system, the USB Bridge IC will automatically be recognized as a COM port.

Note: a proper driver must be installed or else the operating system will not be able to recognize the device!

Please refer to the reference circuits in [Figure D-1](#) below for the conversion. A CP2102 IC is used here as an example.

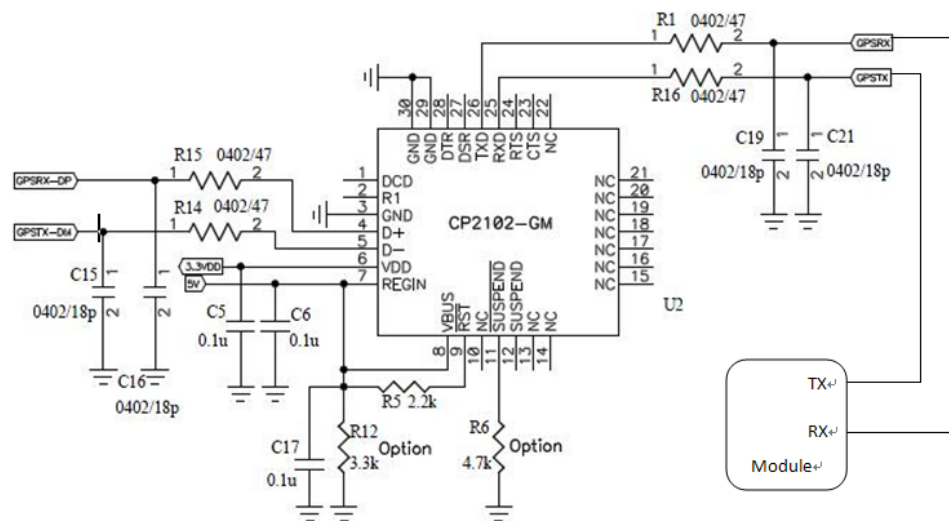


Figure D-1: UART to USB

Pin29 and Pin30 are the bottom ground pads which are not documented in the IC datasheet. You may ignore these two pins in the drawing.

Appendix E: How to Efficiently Transfer 1PPS Through Extended Distances

There are several aspects to note when transferring a 1PPS signal through a long cable which will result in a 1PPS signal degradation, an increase in noise, and an increase in signal delay. As the 1PPS signal is traveling through the communication cable, impedance matching must be implemented to prevent waveform distortion. Depending on your application, an OP-amp can be added to effectively control the accuracy of the voltage level for the 1PPS signal.

When using a 1PPS signal for time synchronization, the OP-amp chosen should have a high slew rate property to prevent a large delay in the 1PPS. The accuracy level of 1PPS for Sierra Wireless GNSS modules is within 100ns. This applies to most of Sierra Wireless' MTK GNSS modules. The reference design in [Figure E-1](#) below can be used to improve accuracy to within 80ns.

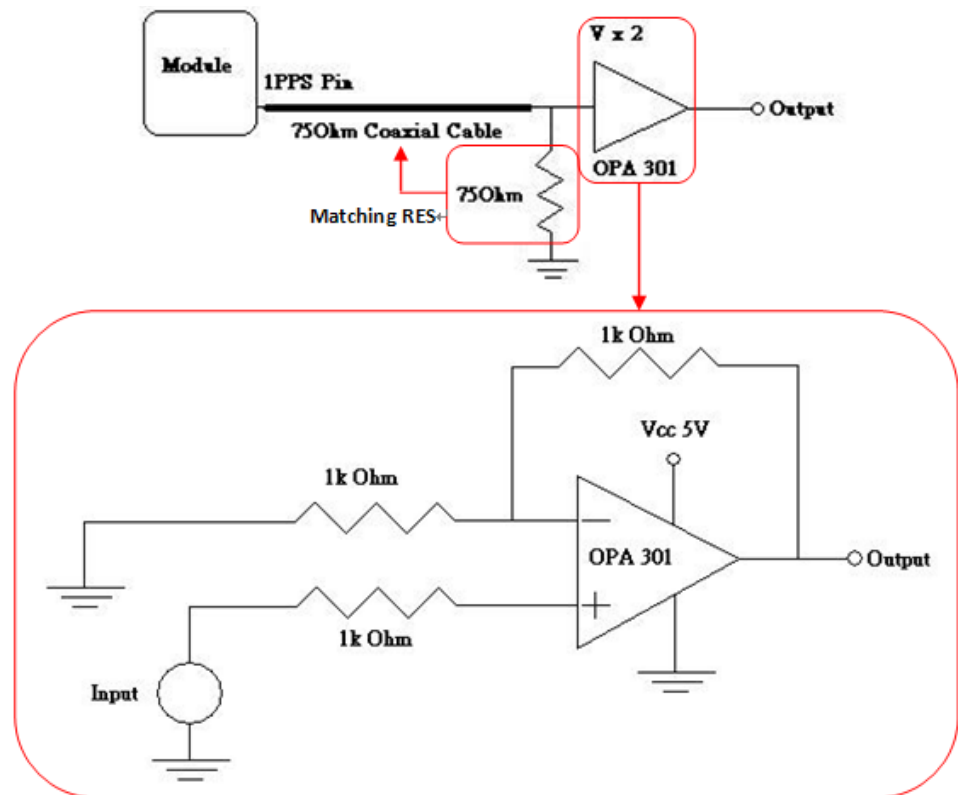


Figure E-1: Design to improve accuracy to within 80ns

1pps Delay Time

When using coaxial cable for long distance communication using an OP-amp for amplification, pay close attention to the coaxial cable time delay and the rising time of the wave form due to the OP-amp's Slew Rate. A brief explanation for this is given below.

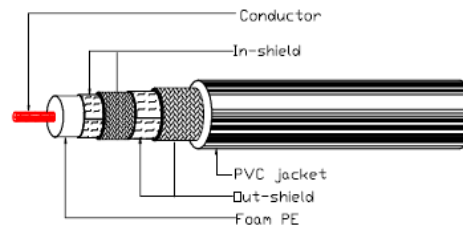
Figure 5-2 shows the specification of the coaxial cable with the model number RG6U.

CONSTRUCTION

Item	A.Q.L.	Description
Conductor	No./mm	1.02 (± 0.025)
	1.02 CCS	
Insulation	No./mm	4.57 (± 0.20)
	4.57	
Shield	B-APA	Bonded double aluminium foil shield
	64/0.16(AL)	64 (±2); 0.16 (± 0.01)
	APA	Aluminium wire braid shield
	48/0.16(AL)	48 (±2); 0.16 (± 0.01)
Jacket	No./mm	7.20 (± 0.30)
	7.20 PVC	

Figure E-2: RG60 Specifications

DESIGN



CHARACTERISTICS

ITEM	Description	Standards	
Conductor Resistance	20°C, conductor DC resistance	<10Ω/100M	
Capacitance		53.2 ± 5pF/M	
Velocity Ratio		83 ± 3%	
Characteristic Impedance	20°C, 200MHz	75Ω ± 3Ω	
Return Loss	20°C, 5 --- 200MHz	>20dB	
Attenuation	20°C -dB/100M (± 10%)	55MHz	4.94
		211MHz	9.43
		400MHz	13.12
		870MHz	19.99
		1000MHz	21.46
		2000MHz	31.82

Figure E-3: RG6U Design and Characteristics

The Delay Time Caused by the Cable

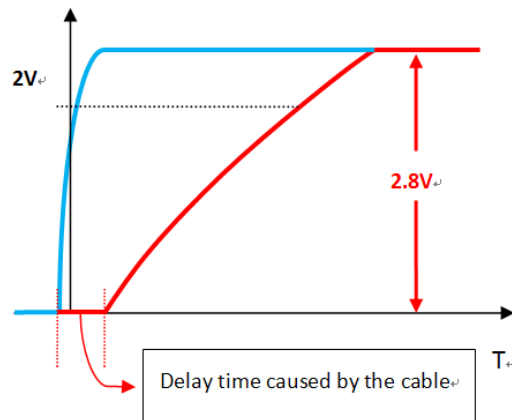


Figure E-4: The delay time caused by the cable

Calculating the Delay Time in Respect to the Communication Cable Length

The simplest method to calculate this is to have a good grasp of the relationship of the three parameters:

1. The speed of the traveling waveform.
2. The distance of the wave form traveled.
3. The delay time.

$$\frac{\ell}{\mu_p} = T$$

ℓ = Distance of the wave form traveled (m)

μ_p = Speed of the waveform (m/s)

T = Delay Time (ns)

If the speed of the waveform (in the coaxial cable) is unknown, it is possible to use a known coaxial cable length to test. For example, the signal travel in one meter of coaxial cable may have a delay time measured by an oscilloscope as 5ns and consequently the μ_p will be $2 * 10^8$ (m/s).

If the value of capacitance and inductance for the coaxial cable is known, then μ_p (m/s) can also be calculated.

$$\frac{1}{\sqrt{LC}}$$

Other coaxial cable electromagnetic fundamental parameters along with the inner and outer diameter of the physical cable can be used in combination to perform theoretical calculations.

Waveform Rising Time Caused by OP-amp

Figure E-5 shows Slew Rate which is the time needed for a rising waveform. When the SR ($V/\mu\text{sec}$) of OPA is higher, the rising time will be shortened as well.

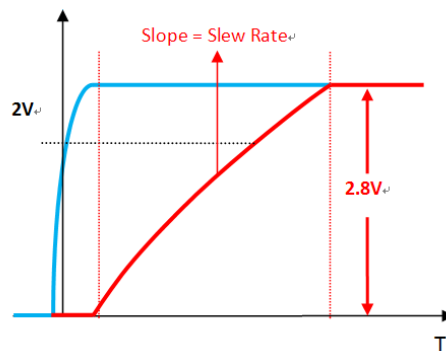


Figure E-5: Slew Rate

Recommended OP-amp

If an inadequate OP-amp is chosen (i.e. with a low slew rate), the delay at the rising time introduced to the waveform can be significantly larger than that of the coaxial wire.

We recommend using OP-amp 301 to decrease the waveform rising time.

Specification of OP-amp 301:

- Low Power: 9.5mA (Typ) on 5.5V
- Single Power: 2.7V ~ 5.5V
- High Slew Rate: 80 $V/\mu\text{sec}$
- Tiny Packages: MSOP and SOT23

Voltage degradation of Communication cable

We tested a 10m long coaxial cable and observed that only insignificant amount of voltage degradation occurred. Users should not be concerned with this issue.

Appendix F: Reflow Soldering Precautions

Table F-1: Precautions

	Details	Suggestions	Notes
1	Before proceeding with the reflow-soldering process, the GPS module must be pre-baked.	Pre-bake Time: 6 Hours @ 60°±5°C or 4 Hours @ 70°±5°C	The maximum tolerated temperature for the tray is 100°C. After the pre-baking process, please make sure the temperature is sufficiently cooled down to 35°C or below in order to prevent any tape and reel deformation.
2	Because PCBA (along with the patch antenna) is highly endothermic during the reflow-soldering process, extra care must be paid to the GPS module's solder joint for any signs of cold welding or false welding.	The parameters of the reflow temperature must be set accordingly to module's reflow-soldering temperature profile.	Double check to see if the surrounding components around the GPS module are displaying symptoms of cold welding or false welding.
3	Special attention is needed for the PCBA board during reflow-soldering to see if there are any symptoms of bending or deformation to the PCBA board, possibly due to the weight of the module. If so, this will cause concerns at the latter half of the production process.	A loading carrier fixture must be used with PCBA if the reflow soldering process is using rail conveyors for the production.	If there is any bending or deformation to the PCBA board, this might cause the PCBA to collide into one another during the unloading process.
4	Before the PCBA undergoes a reflow-soldering process, the production operators must check with their own eyes to see if there are positional offsets to the module, because it will be difficult to readjust after the module has gone through the reflow-soldering process.	The operators must check with their own eyes and readjust the position before the reflow-soldering process.	If the operator is planning to readjust the module position, please do not touch the patch antenna while the module is hot in order to prevent a rotational offset between the patch antenna and module.

Table F-1: Precautions

	Details	Suggestions	Notes
5	Before handling the PCBA, they must be cooled to 35°C or below after they have undergone a reflow-soldering process, in order to prevent a positional shift which may occur when the module is still hot.	<ol style="list-style-type: none"> 1. One may use an electric fan behind the reflow machine to cool them down. 2. Cooling the PCBA can prevent the module from shifting due to fluid effect. 	It is very easy to cause a positional offset to the module and its patch antenna when handling the PCBA under high temperatures.
6	<ol style="list-style-type: none"> 1. When separating the PCBA panel into individual pieces using the V-Cut process, special attention is needed to ensure that there are sufficient gaps between patch antennas. 2. If the V-Cut process is not available and the pieces must be separated manually, please make sure the operators are not using excess force which may cause a rotational offset to the patch antennas. 	<ol style="list-style-type: none"> 1. The blade and the patch antenna must have a gap greater than 0.6mm. 2. Do not use the patch antenna as the leverage point when separating the panels by hand. 	<ol style="list-style-type: none"> 1. Tests must first be performed to determine if the V-Cut process is going to be used. Ensure that there is enough space between the blade and patch antenna so that they do not touch one another. 2. An uneven amount of manual force applied to the separation will likely to cause a positional shift in the patch antenna and module.
7	When separating the panel into individual pieces during latter half of the production process, special attention is needed to ensure the patch antennas do not come in contact with one another in order to prevent chipped corners or positional shifts.	Use a tray to separate the individual pieces.	It is possible to chip a corner and/or cause a shift in position if the patch antennas come into contact with each other.

Note: for patch antenna users: patch antenna refers to the patch-on-top antenna which is built on the GPS module (i.e. PA/Gmm Module Series) and may not be applicable to all GPS modules.

Other Cautionary Notes on the Reflow-Soldering Process

1. Module must be pre-baked **before** going through the SMT solder reflow process.
2. The usage of solder paste should follow the “First-in-First-out” principle. Opened solder paste needs to be monitored and recorded in a timely manner (refer to IPQC standards for related documentation and examples).
3. Temperature and humidity must be controlled within an SMT production line and storage area. A temperature of 23°C, 60±5% RH humidity is recommended (please refer to IPQC standards for related documentation and examples).
4. When performing solder paste printing, check if the amount of solder paste is in excess or insufficient, as both conditions may lead to defects such as electrical shortage, empty solder, and etc.
5. Make sure the vacuum mouthpiece is able to bear the weight of the GPS module in order to prevent a positional shift during the loading process.
6. Before the PCBA goes through the reflow-soldering process, the operators should check by his/her own eyes to see if there is a positional offset to the module.
7. The reflow temperature and its profile data must be measured before the SMT process and match the levels and guidelines set by IPQC.
8. If an SMT protection line is running a double-sided process for PCBA, please process the GPS module during the second pass only to avoid repeated reflow exposures of the GPS module. Please contact Sierra Wireless beforehand if you must process the GPS module during the first pass of double-side process.

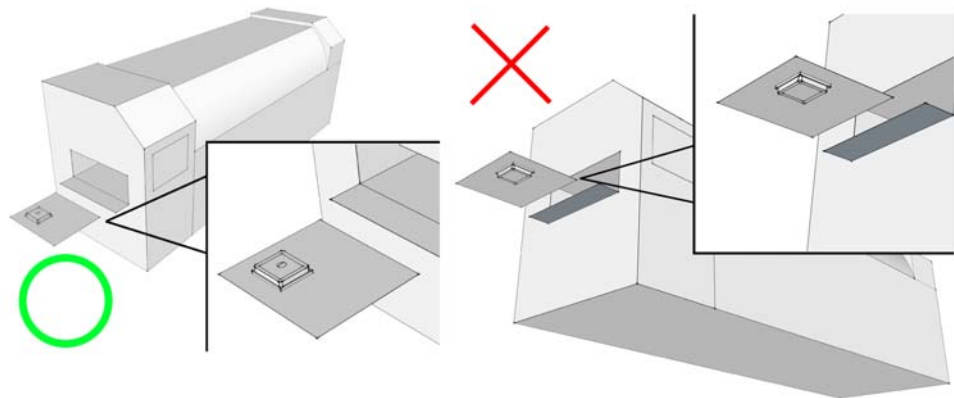


Figure F-1: Placing the GPS module right-side up when running the reflow-solder process; do not invert.

Manual Soldering

Soldering Iron

Heat Temperature: Under 380°C | Time: Under 3 sec.

Notes:

1. To prevent further oxidation, do not directly touch the soldering pads on the surface of the PCB board.
2. The solder paste must be defrosted to room temperature before use so that it can return to its optimal working temperature. The time required for this procedure is unique and dependent on the properties of the solder paste used.
3. The steel plate must be properly assessed before and after use, so its measurement stays strictly within the specification set by SOP.
4. Watch out for the spacing between the soldering joints, as excess solder may cause electrical shortages.
5. Exercise caution and do not use an extensive amount of flux due to possible siphon effects on neighboring components, which may lead to electrical shortages.
6. Please do not use a heat gun for long periods of time when removing the shielding or inner components of the GPS module, as it is very likely to cause a shift to the inner components which will lead to electrical shortages.