Hardware Design Guide

AirPrime XA1100/XM1100

41111082
Rev 2

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

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.
2	January 19, 2018	Updates throughout.

Contents

1: Introduction

Sierra Wireless has various GPS modules designed for many different applications. This application note is for the models listed below that use the MT3337 chip:

- Stand Alone GPS module: XM1100

Figure 1-1: XM1100 GPS module

- Integrated Antenna GPS Module: XA1100

Figure 1-2: Integrated Antenna GPS Module

Precautions

Please read carefully before you start

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

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

The GPS 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 that you test the device outdoors with open sky for at least 10 to 15 minutes to ensure that full ephemeris data is received.

2: General Rules for Design

This section provides some rules to obtain the best performance when using the GPS module.

Circuit Design

Power Supply

A clean and stable power supply is required for our GPS modules to perform optimally. An unstable power source will significantly impact the GPS performance. To achieve high-quality performance, the Vcc ripple must be controlled under 50mVpp. Additional considerations to stability power supply include:

1. Adding ferrite bead, power choke, or low pass filter for power noise reduction.
2. Adding a linear regulator compared to a switched DC/DC power supplier in the ripple.
3. Using enough decoupling capacitors with the VCC for stable voltage.

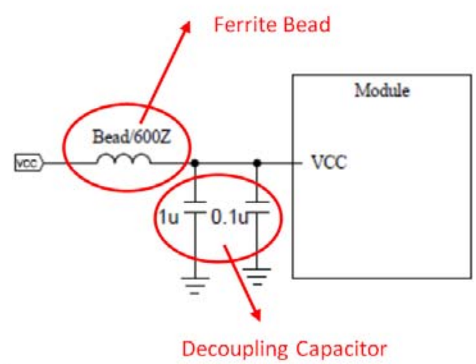


Figure 2-1: Power Design for the GPS Module

VBACKUP Backup Battery

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

For most systems, it is recommended that the module be provided 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 needed to maintain RTC operation and retain Ephemeris data in flash memory which helps the module get quicker 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 each time its restarted.

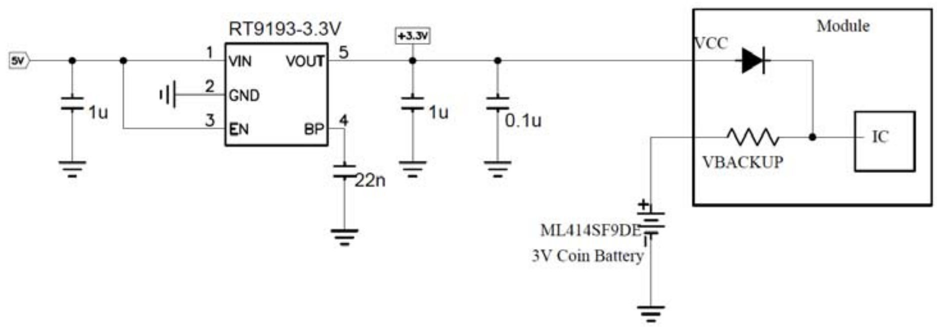


Figure 2-2: Rechargeable coin battery with VBACKUP

UART 0 (RX/TX) –Serial Interface

1. UART is the TTL level interface that carries the baud rate ranging from 9600 bps to 115200 bps.
2. Placing a damping resistor on the RX and TX of the GPS 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. 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 a level shifter should be used. For more information please refer to UART to RS232 Interface.
5. If USB logic-level is needed for any particular application please refer to the UART to USB Interface design guideline.

Antenna Compliance Design

The GPS antenna is a receiving part of the device that acquires weak GPS signals from the sky. A common solution is 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 is a standalone component without a signal amplifier such as LNAPatch antennas and chip antennas are the most commonly used passive antennas with GPS modules. When using an external patch antenna, ensure that it is correctly fine-tuned for use with the specific module to ensure best possible signal strength.

An *Active* antenna is a standalone external antenna that comes with an integrated an LNA. These antennas provide higher gain and better performance than that of a passive antenna. An example of an active antenna includes Patch Antenna with RF Cable and integrated LNA.

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

Designing an External Passive Patch Antenna with GPS Module

1. In general, a 50Ω patch antenna will work well with the GPS 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 GPS 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.

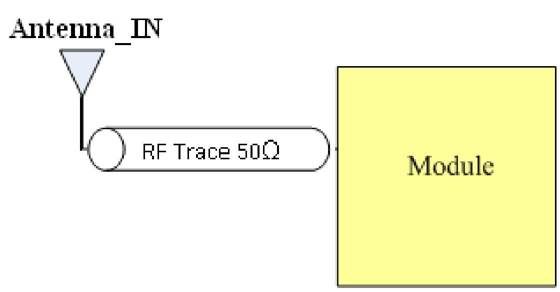


Figure 2-3: PCB trace design for antenna impedance matching

Selecting an Active Antenna Architecture for the GPS Module

An external active antenna requires DC power 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 choke coil couples the DC power to the RF trace to perform this method.

Sierra Wireless supports two architectures for this.

- **Mode1**: an external power supply needs to be provided and connected directly to the external antenna via the choke coil. See Table 2-1 Mode 1.

Note: Choke Coil for reference: LQG15HS33NJ02D (Murata)

- **Mode2**: the power supply comes from *VCC*. The circuit also auto switches the RF signal from the internal patch antenna to the external antenna when connecting an external antenna. See Table 2-1 Mode 2.

Table 2-1: Modes

	XM1100	XA1100
Mode 1 (via the VANT) (External Choke Coil)	✓	
Mode 2 (from the VCC) (Internal Choke Coil)		✓

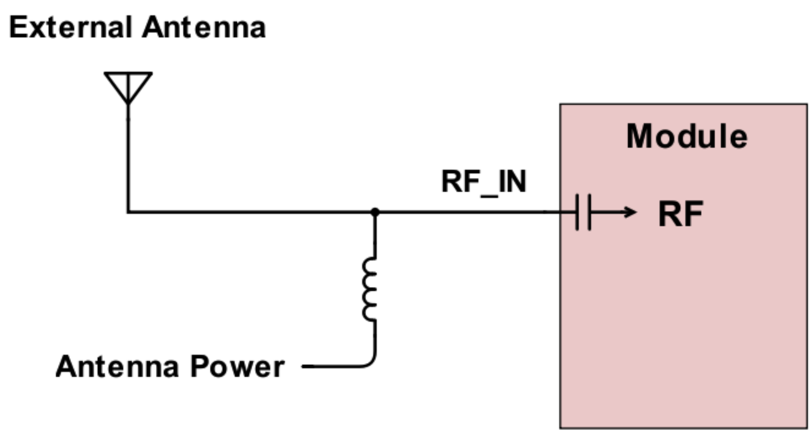


Figure 2-4: Mode 1: External Choke Coil

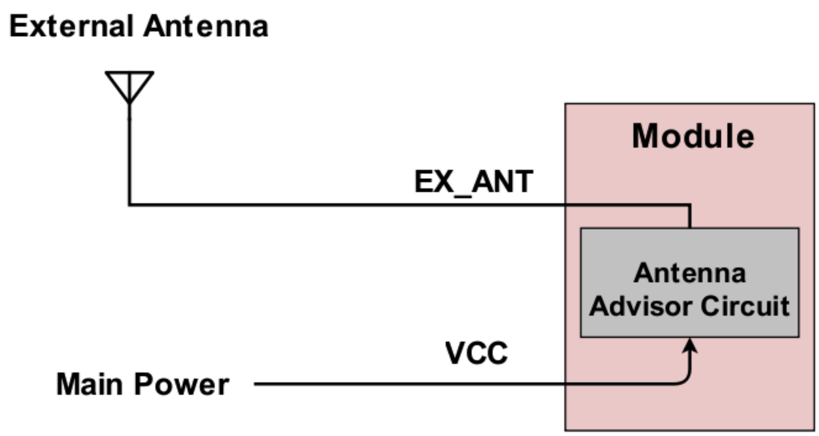


Figure 2-5: Mode 2: Internal Choke Coil

Designing a Chip Antenna with GNSS Module

It's recommended to consult chip antenna vendors for more specific design guidelines. Below are references for the chip antennas from Pulse and Unictron.

1. Pulse Antenna

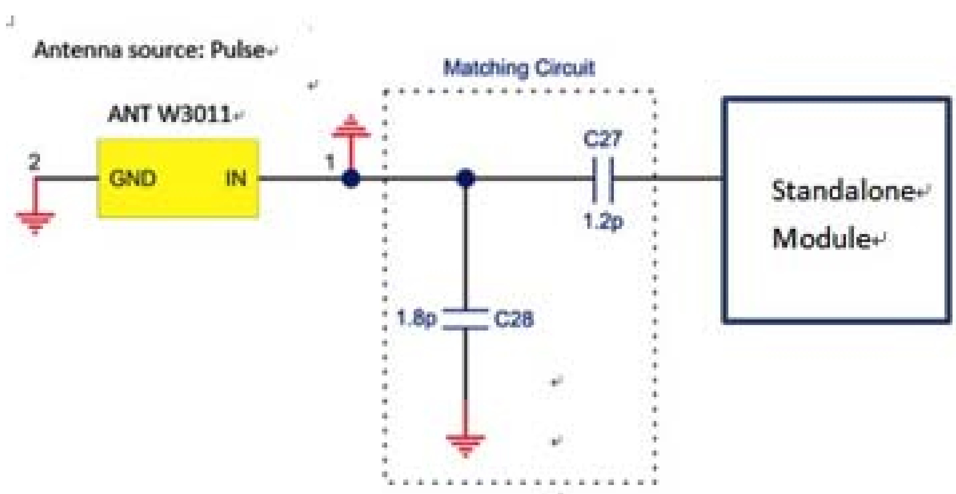


Figure 2-6: Pulse Schematic Design

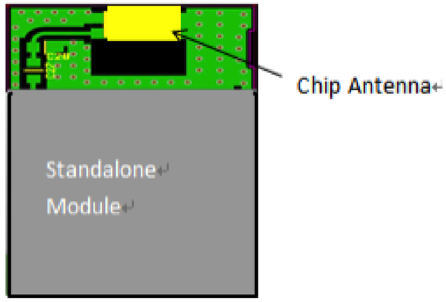


Figure 2-7: Pulse PCB Layout

Check the antenna datasheet before tuning the RF to match the component footprint (such as: C27, C28). You can base this on the PCB size and housing to tune for an optimal value and meet the GNSS's frequency to get good reception.

Pulse web site: www.pulseeng.com/antennas

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

2. Unictron Antenna

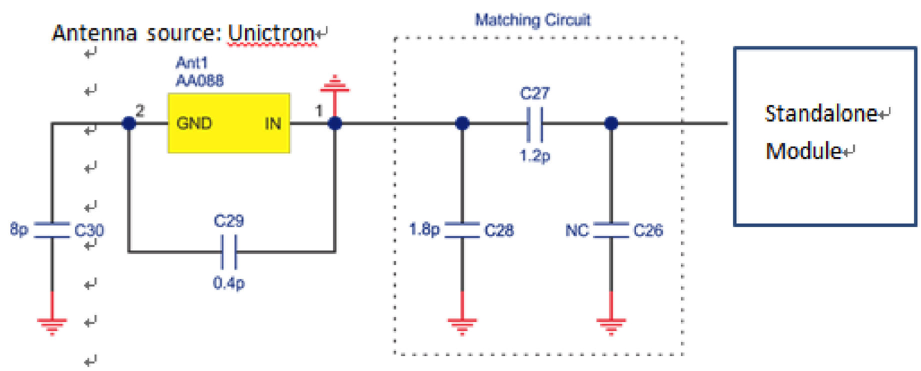


Figure 2-8: Unictron Schematic

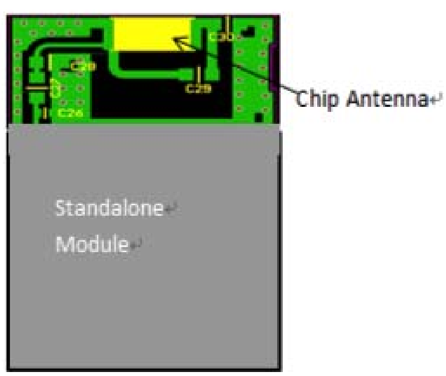


Figure 2-9: Unictron PC

Check the antenna datasheet before tuning RF to match the component footprint (such as: C26, C27, C28, C29, C30). You can tune based on PCB size and housing to tune for an optimal value in order to meet the GNSS's frequency and for good reception.

Unictron web site: http://www.unictron.com/index/

Note: The values for C26,C27,C28,C29, and C30 are tuned based on your trace.

1PPS

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):±20ns
- 100ms pulse width duration

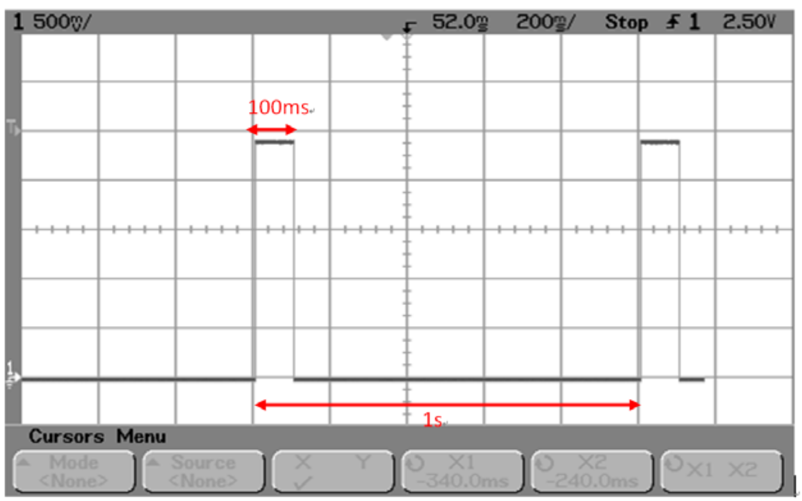


Figure 2-10: 1PPS signal and its pulse width with 100ms duration

Free run 1PPS Output Before 3D_FIX

The Sierra Wireless standard GPS 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.

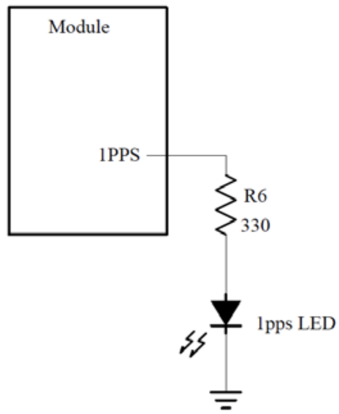


Figure 2-11: 1PPS Signal Design for IO

1.8V Boost to 3.3V Application

If you want to use the 3.3V's GNSS module in a 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 2-12 below).

TPS61097A-33 is the TI's boost IC which can support boost functionality. In its application C1 and C2 need to use 10uF and L1 is 10uH. 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

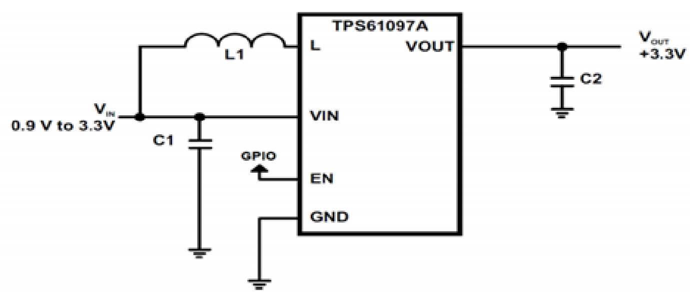


Figure 2-12: Application Schematic for Ti Boost IC

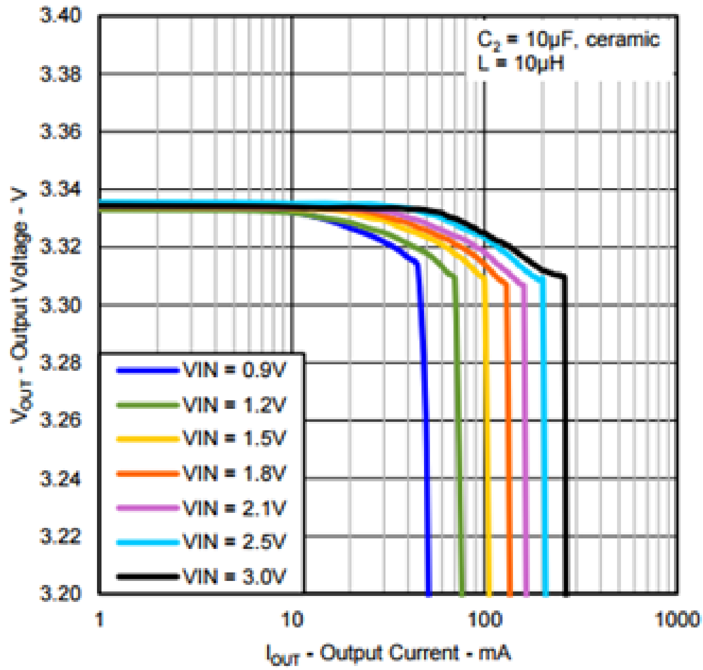


Figure 2-13: Output Voltage vs Output Current

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 Figure 2-14 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.

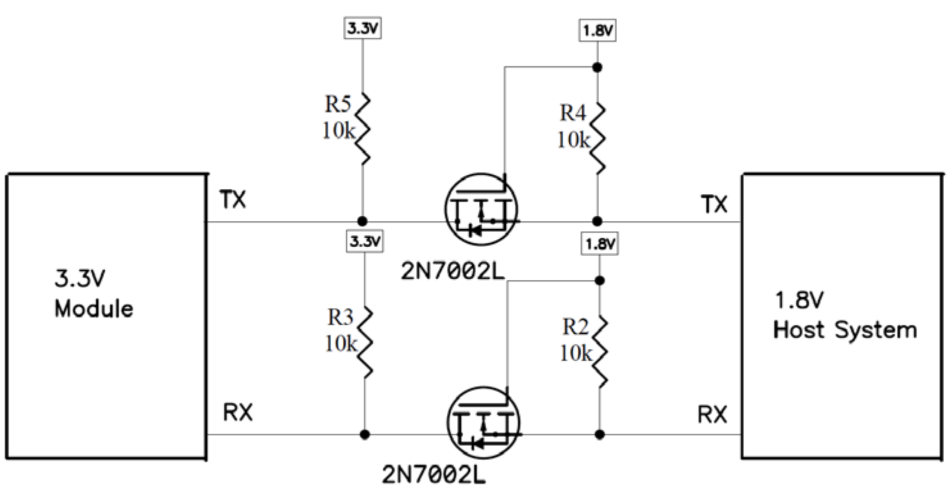


Figure 2-14: Signal Level Shift Circuit

Layout Guidelines

Please follow the layout guidelines during the design process.

Layout Underneath the GPS Module

A GPS signal has a very weak signal level at around -160dBm to 130dBm. Any noise or harmonic will further decrease the quality of the GPS signal. Many modern GPS products include an LCD, MCU, High Speed digital, and other RF systems such as Bluetooth, Wi-Fi, DVB-T, Cellular etc.…).

For Module XM1100

To minimize noise interference, please don't place any trace underneath the GPS module.It is recommended to provide the GPS module with a clean GND plane.

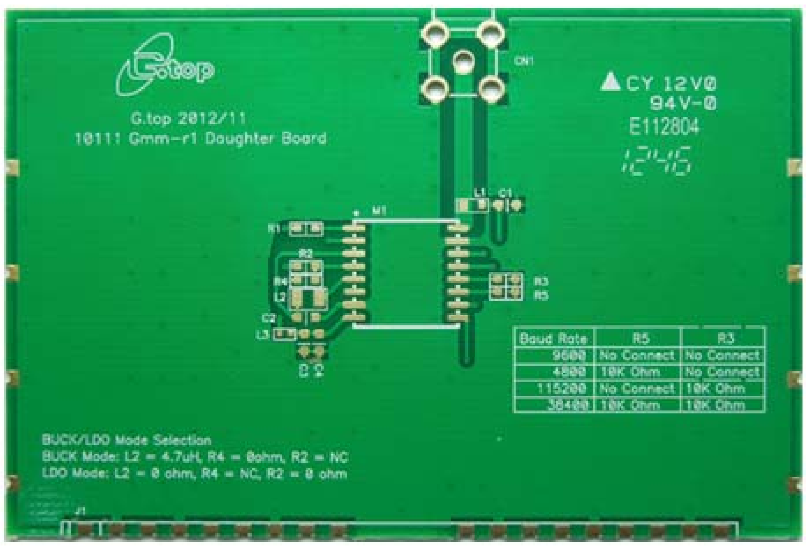


Figure 2-15: GPS Module on a clean GND Plane

For Module XA1100

To avoid the interference, place many vias on the two sides of the RF trace which goes from the module to the SMA/RF connector on your system PCB as illustrated in Figure 2-16 below.

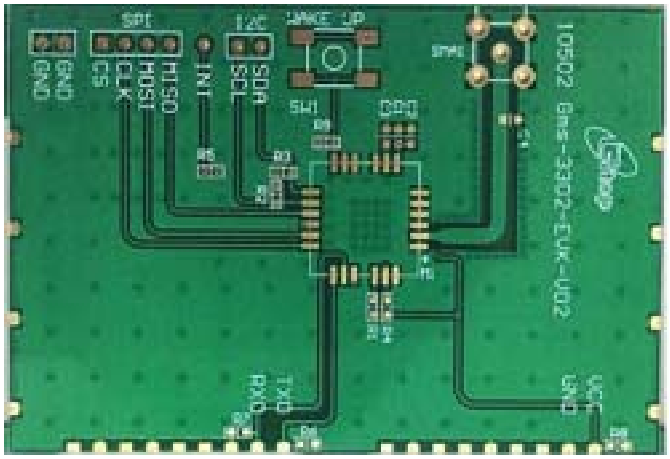


Figure 2-16: GNSS Antenna module with clear GND plane.

Keep Far Away from High Profile or Metal-Canned Components - For XA1100

It is good practice to place the GPS module far away from any high-profile components especially those enclosed in metal cases 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 GPS signal up to ~10dB.

Figure 2-17: First example of bad and good GPS module placements

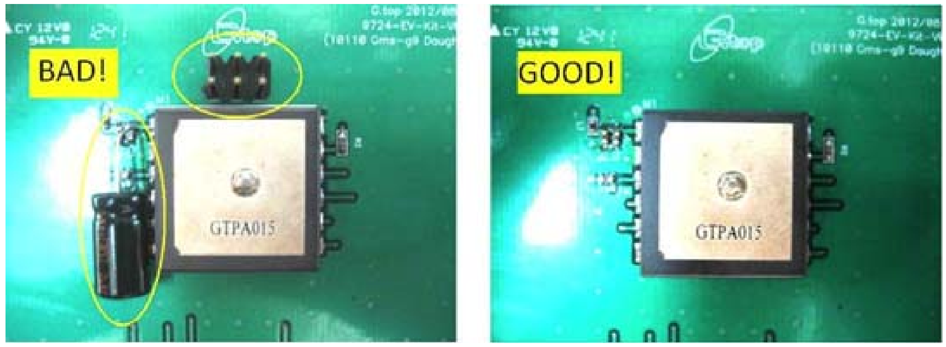


Figure 2-18: *Second Example of bad and good GNSS module placement*

Figure 2-19: *Third Example of bad and good GNSS module placement*

Figure 2-20: *Example of bad GPS module placement (Patch Antenna close to a high-profile metal case component) and good placement.*

Placement

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

Do not place the GPS module:

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

Trace

1. The USB differential signals should be traced closely and be 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.

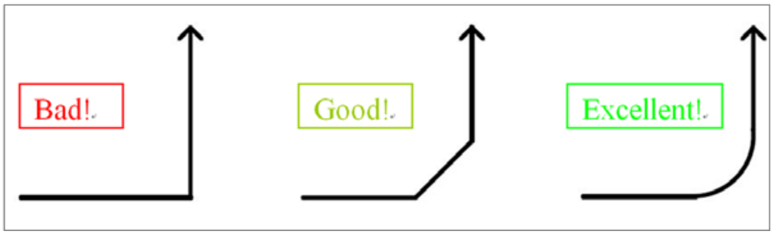


Figure 2-21: Examples of turns in trace routing

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

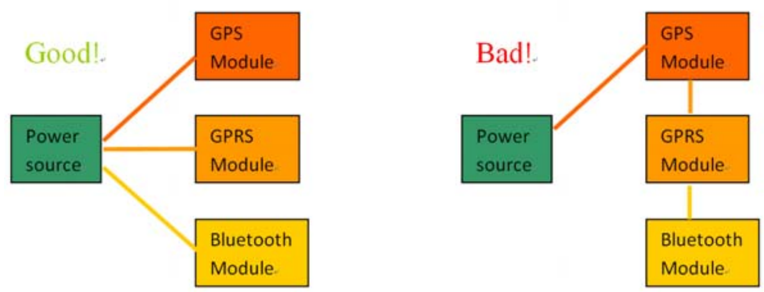


Figure 2-22: Examples of Independent Trace

Ground Segmentation

In general, the separation of ground between the GPS module and the rest of the system is recommended to avoid interference. If this is not possible, it is best to follow these rules: segmentation of ground between digital and analogue system, high current and low current system, and different radiation systems such as GPS 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; each set is 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 ideally located at the border of the board.

Ground Plane - For XA1100

A large GND plane directly underneath the module can enhance the magnetic-field line of the antenna for better GPS signal reception. Typically it will improve reception by up to ~2dB. It is strongly recommended that the ground plane underneath the GPS module be as large 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 GPS module as shown in Figure 2-23:

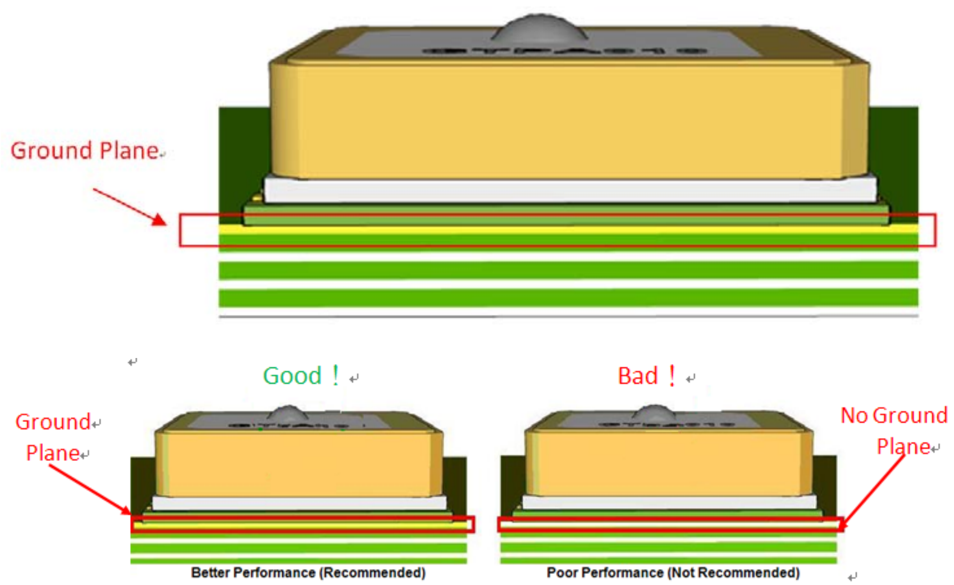


Figure 2-23: Ground Plane for FR4 Design

3: Thermal Profile for SMD Modules

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

SMT Reflow Soldering Temperature Profile

- Solder paste alloy: SAC305(Sn96.5/Ag3.0/Cu0.5).
- Pre-heat Temp: 150~200℃; Soak time(ts): 60~120sec.
- Peak temp(Tp) ≤ 250℃.
- Time above 220℃(tp): 40~90sec.
- Optimal cooling rate < 3℃/sec, from peak to 220℃.
- The oxygen concentration < 2000 ppm.

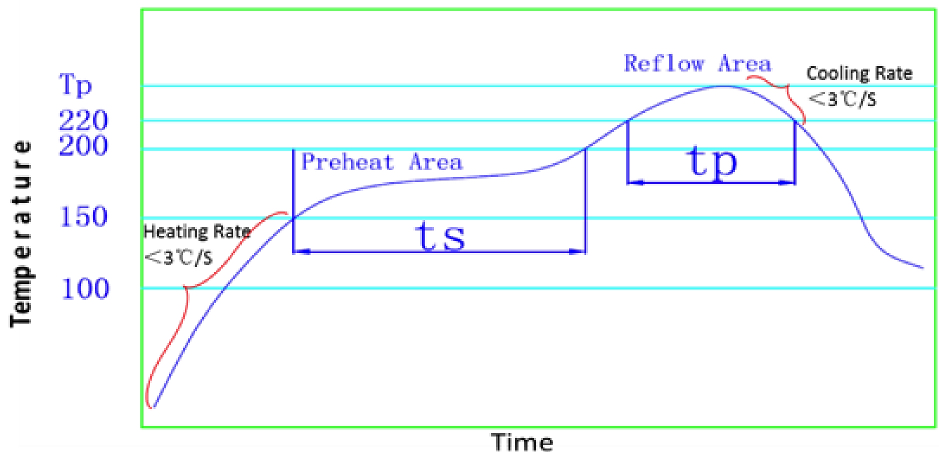


Figure 3-1: SMT Reflow Soldering Temperature Profile

SMT Solder Mask

Using the dimensions of the PCB pad as a reference, subtract the size by 0.1 to 0.2 mm and use that as a layout for thte paste mask (for a PCB pad layout, please see “Recommended PCB pad layout” on the individual GPS module PTS).

Manual Soldering

- Soldering iron: Bit Temperature: under 380°C | Time: under three seconds.

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 GNSS tool provided by Sierra Wireless.

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 to 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 that power be provided to the *VBACKUP* pin as it is used to keep RTC time running and preserve stored navigation data.

Item 3: TX0

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

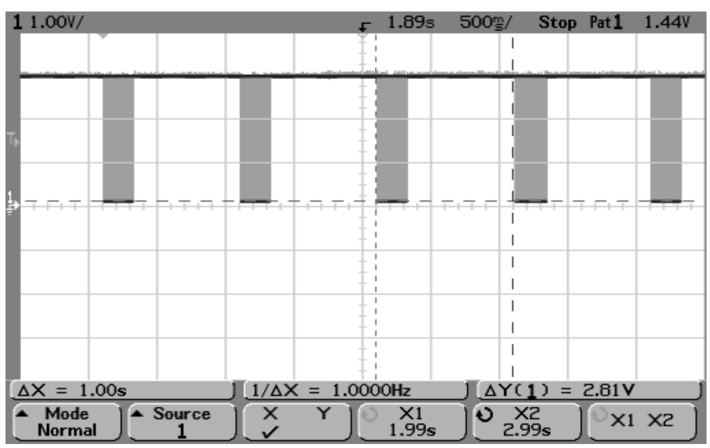


Figure 4-1: TX0

Appendix A: Super Capacitor Design

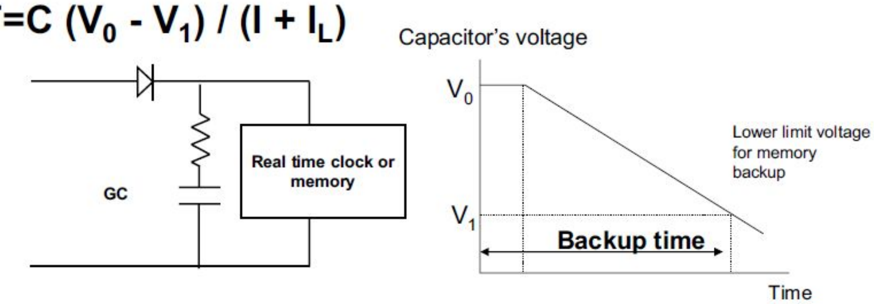
About Super Capacitors

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:

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



Example:

V_0 = 2.5V, V_1 = 1.8V, I = 10uA, C = 0.2F

T = 0.2 (2.5 –1.8) / (10 e-6 + 0.2x10^{-6})
= 0.14/10.2x10^{-6}
= 13,725 seconds
= 3.8hours

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 50Ω 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

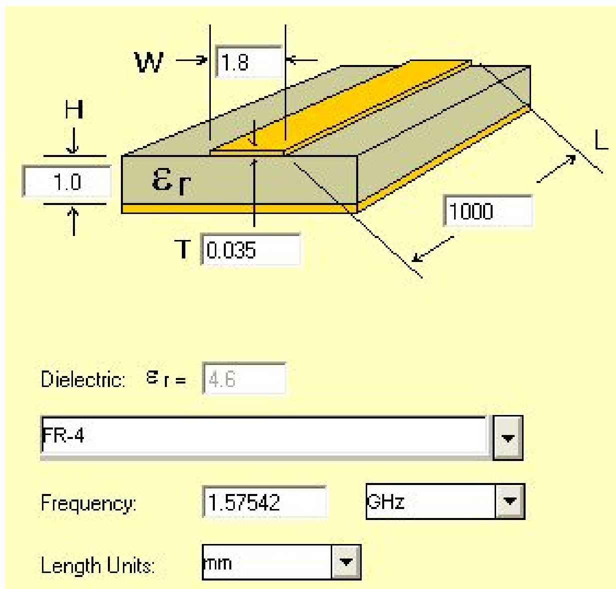


Figure B-1: Dimensions

Notice:

For multi-layer layouts, you can 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 GPS module for communication. Most 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.

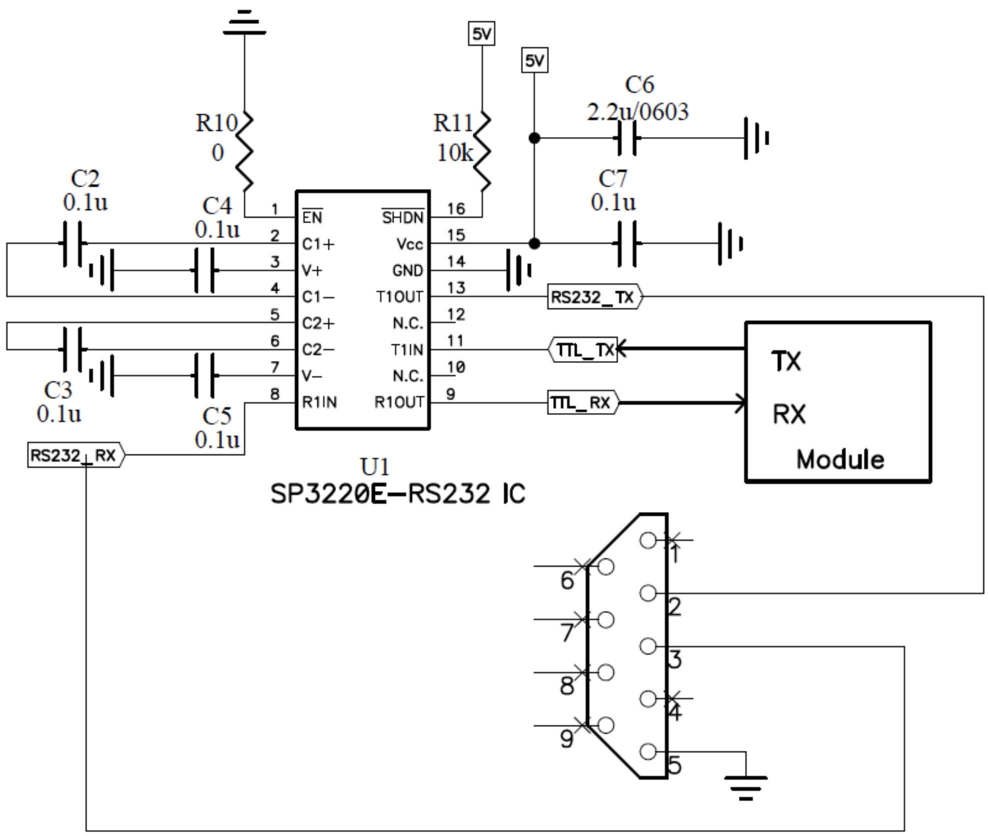


Figure C-1: RS232 Signal Conversion Example

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 for USB to be recognized by the operating system.

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

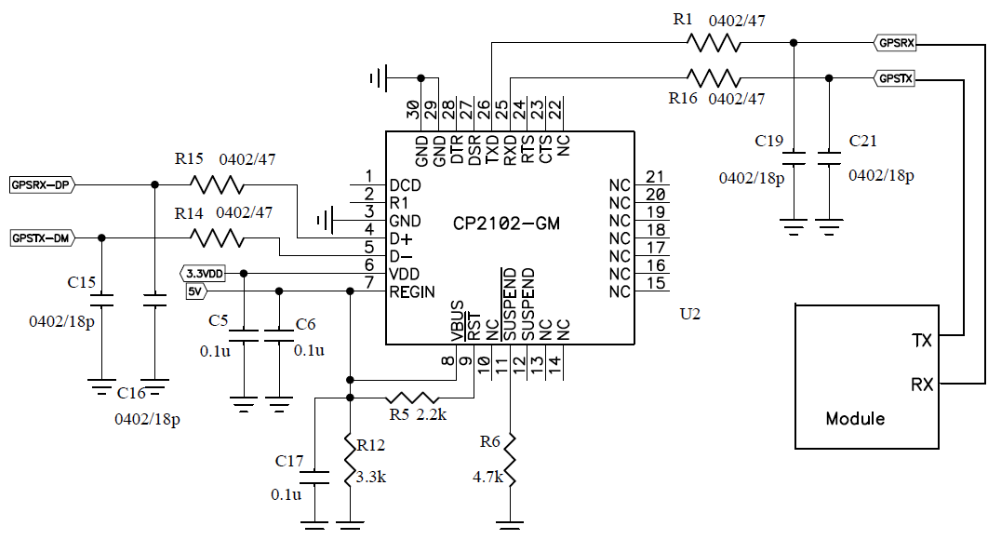


Figure D-1: UART to USB

Pin 29 and Pin 30 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 GPS modules is within 100ns. This applies to most of Sierra Wireless' MTK GPS modules. The reference design in Figure E-1 below can be used to improve accuracy to within 80ns.

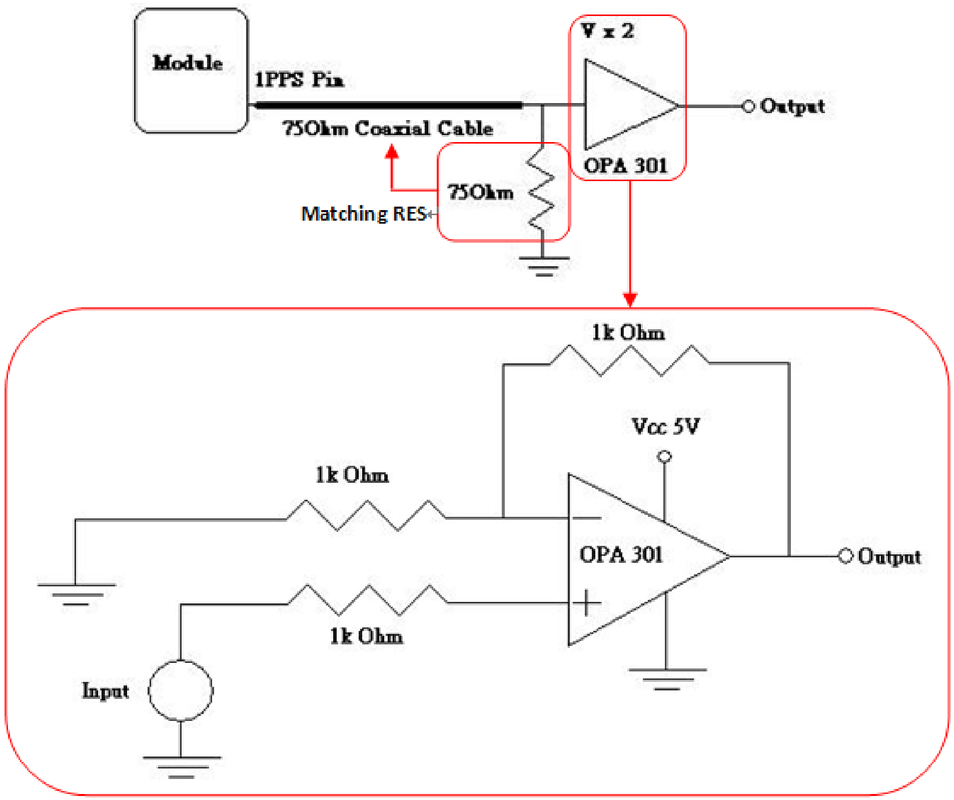


Figure E-1: Design to improve accuracy to within 80ns

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.

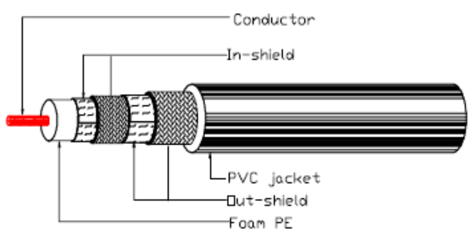
Figure 5-2 shows the specification of the coaxial cable with the model number RG6U.

CONSTRUCTION

Item		A.Q.L.	Description
Conductor	No./mm 1.02 CCS	1.02 (± 0.025)	Copper clad steel
Insulation	No./mm 4.57	4.57 (± 0.20)	Foamed PE insulation
Shield	B-APA		Bonded double aluminium foil shield
	64/0.16(AL)	64 (±2); 0.16 (± 0.01)	Aluminium wire braid shield
	APA		Double aluminium foil shield
	48/0.16(AL)	48 (±2); 0.16 (± 0.01)	Aluminium wire braid shield
Jacket	No./mm 7.20 PVC	7.20 (± 0.30)	Black flame retardant PVC jacket

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

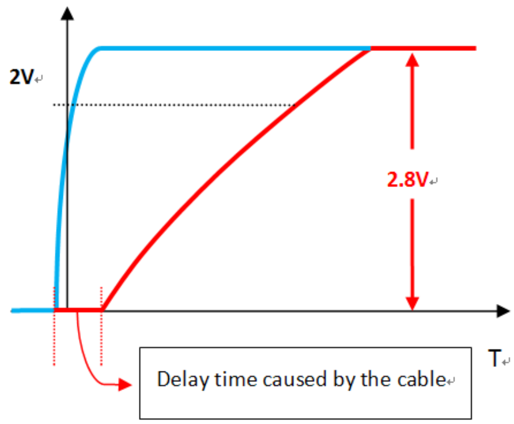


Figure E-4: The delay time caused by the cable

The waveform in the coaxial cable is traveling close to the speed of light but will show significant delay as the cable becomes longer.

Calculating the Delay Time in Respect to the Communication Cable Length

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/µsec) of OPA is higher, the rising time will be shortened as well.

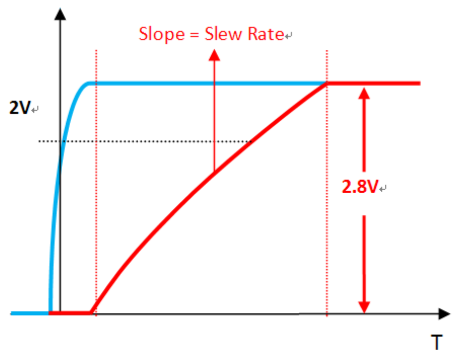


Figure E-5: Slew Rate

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/sec
- Tiny Packages: MSOP and SOT23

Voltage degradation of Communication cable

Tests with a 10m long coaxial cable showed an insignificant amount of voltage degradation. 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 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 boards, this might cause them to collide into one another during the unloading process.
4	Before the PCBA undergoes a reflow-soldering process, the production operators must visually check 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 visually check 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.	1. You 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	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.	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.	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.

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.

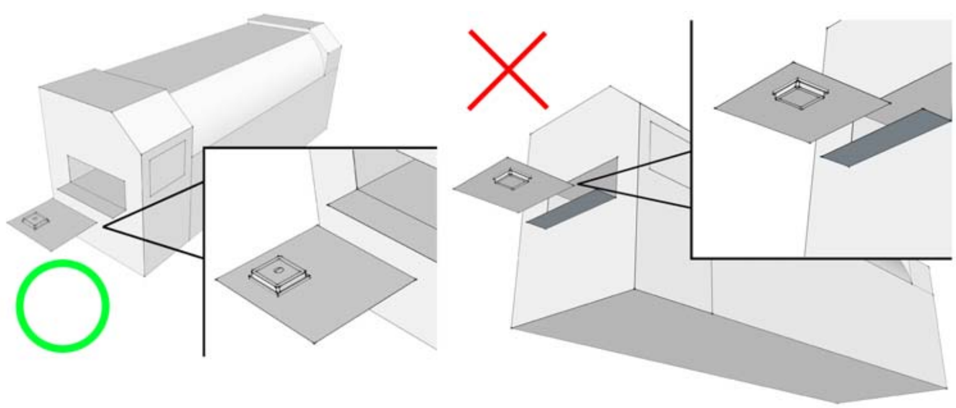


Figure F-1: Placing the GPS module right-side up when running the reflow-solder process; do not invert.

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 excessive 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.