

GPS Module Application Notes (UBX-G7020 series)



Table of Contents

<u>Table of Contents</u>	2
<u>1. Introduction</u>	3
<u>2. General Rules for Design-in</u>	3
<u>2.1 Circuit Design</u>	3
<u>2.1.1 Power supply VCC</u>	3
<u>2.1.2 VBACKUP backup battery</u>	4
<u>2.1.3 UART 0 (RX/TX) –Serial Interface</u>	4
<u>2.1.4 USB(DP/DM) Interface</u>	4
<u>2.1.5 GND-Ground</u>	5
<u>2.1.6 1PPS</u>	5
<u>2.2 Layout Guideline</u>	7
<u>2.2.1 Layout underneath the GPS module</u>	7
<u>2.2.2 Keep far away from high profile or metal canned components</u>	8
<u>2.2.3 Placement</u>	9
<u>2.2.4 Trace</u>	9
<u>2.2.5 Ground Segmentation</u>	10
<u>2.2.6 Ground Plane</u>	10
<u>3. Thermal Profile for SMD Modules</u>	11
<u>SMT Reflow Soldering Temperature Profile</u>	11
<u>SMT Solder Mask:</u>	11
<u>Manual Soldering:</u>	11
<u>4. Troubleshooting</u>	12
<u>Appendix I: Super Capacitor Design</u>	14
<u>Appendix II: 50 Ω Antenna Matching</u>	15
<u>Appendix III: UART to RS232 Interface</u>	16
<u>Appendix IV: UART to USB Interface</u>	17
<u>Appendix V: How to efficiently transfer 1PPS through extended distances</u>	18
<u>Appendix VI: Caution for Reflow Soldering Process</u>	22

1. Introduction

The purpose of the application note is to assist system developer and integrator to get familiar with UBLOX's UBX-G7020 chipset based POXA receiver modules during the process of implementation.

2. General Rules for Design-in

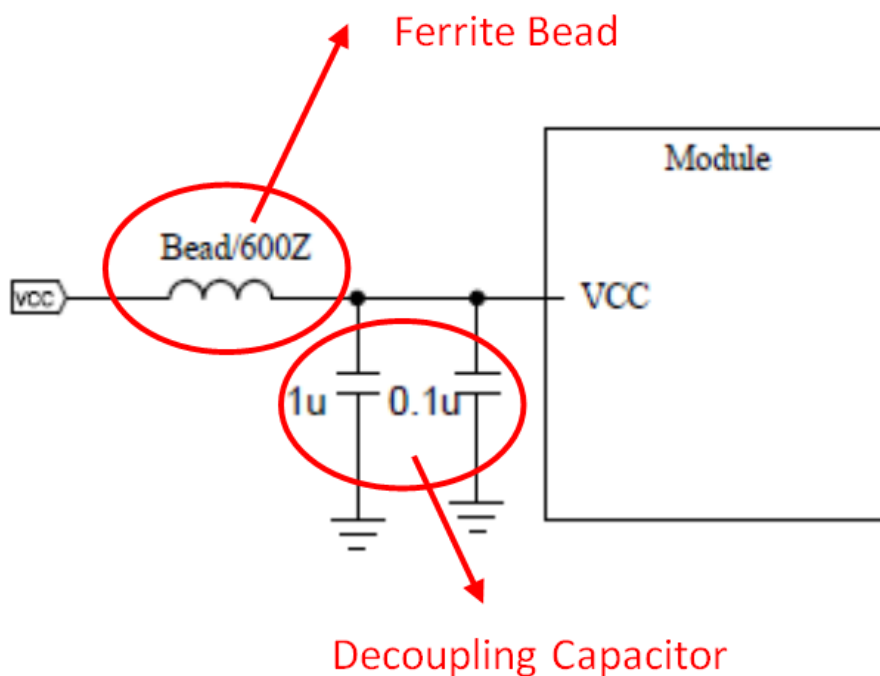
In order to obtain good GPS performances, there are some rules which require attentions for using GPS module

2.1 Circuit Design

2.1.1 Power supply VCC

It is necessary to provide a clean and stable power supply for the GPS module to obtain good performance. Unstable power source will have a significant negative impact on the performance. To achieve this, the Vcc ripple must be controlled under 50mV_{pp}. In addition, there are also some important suggestions for main power circuit design:

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



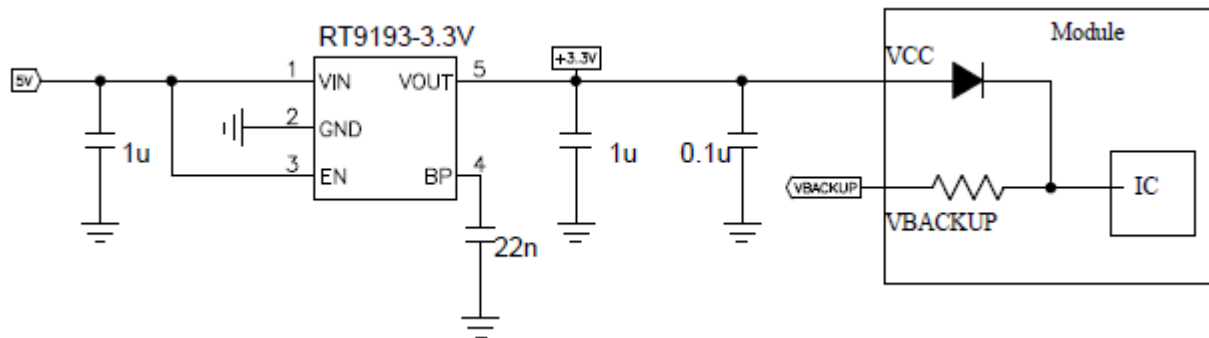
Power design for GPS module

2.1.2 VBACKUP backup battery

There is application for applied VBACKUP to GPS module.

Application:

GNSS module has a built-in charging circuit which charges the rechargeable coin battery. If the coin battery is not installed at the VBACKUP, the module will still function but only that the module will perform cold start at each power on event.



The built-in charging circuit

2.1.3 UART 0 (RX/TX) –Serial Interface

- (1) UART is the TTL level interface that carries the baud rate ranging from 4800 bps to 115200 bps.
- (2) Placing damping resistor on the RX and TX of the GPS module could limit the interference from host MCU or high speed digital logics. Fine tuning the damping resistor is required to efficiently suppress interference. Since the damping resistor is wire wound component and may function as chock coil.
- (3) Please leave RX floating if it is not used as there is an internal pull-up to VCC.
- (4) Please don't connect diode(s) to RX/TX as it will decrease signal driving capability which might adversely affect RX/TX signal level. In some cases no data output will occur.
- (5) 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" in Appendix III.
- (6) If USB logic-level is needed for any particular application. For more information please refer to " UART to USB Interface" in Appendix IV.

2.1.4 USB(DP/DM) Interface

- (1) Placing damping resistor on the USB_DP and USB_DM of the GPS module could limit the interference from host MCU or high speed digital logics. Fine tuning the damping resistor is required to suppress interference.
- (2) If USB interface is used for connection, please install u-blox USB driver before you connect the module to the external USB host (i.e. PC)

2.1.5 GND-Ground

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

Please refer to “2.2.5 Ground Segmentation”.

2.1.6 1PPS

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

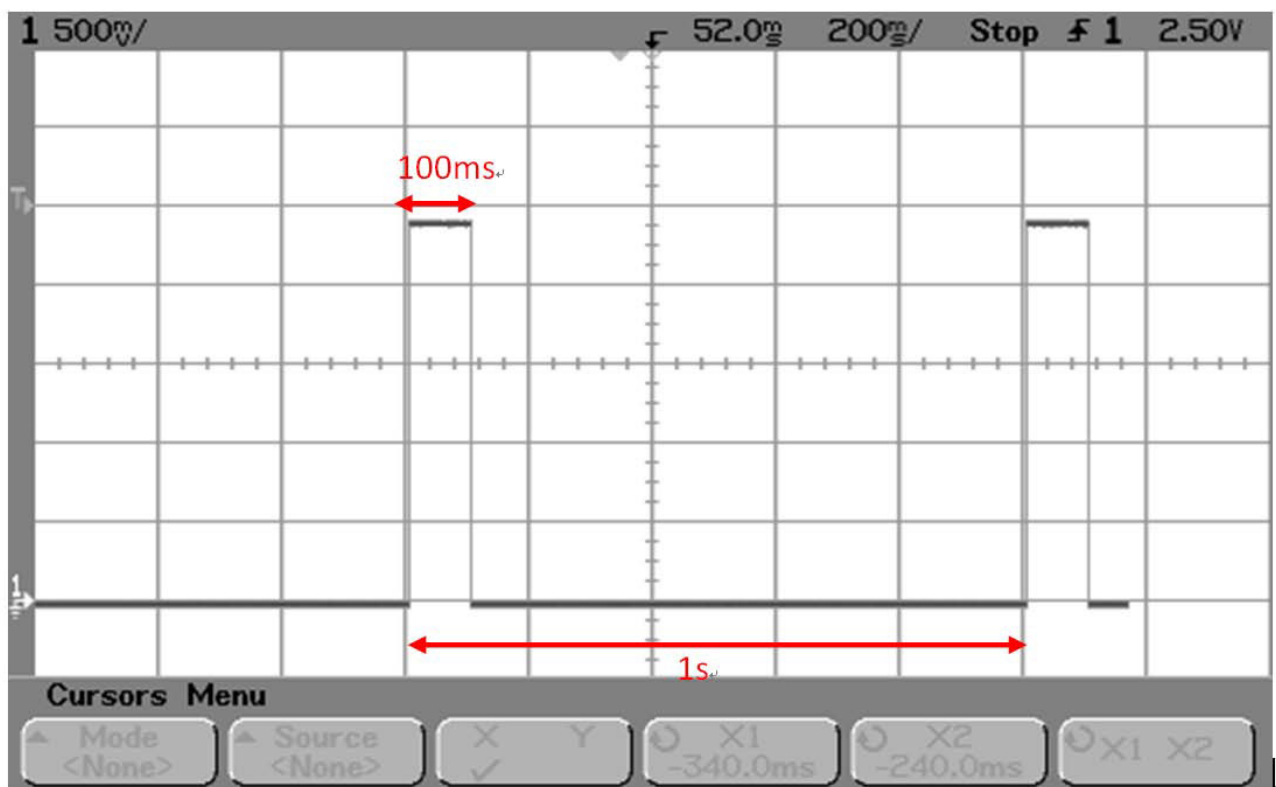
Low Voltage level: 0~0.4V

High Voltage level: 2.8~3.3V

Period: 1s

Accuracy (jitter): $\pm 30\text{ns}$

Duration: 100ms (duration can be configured via commands)



1PPS signal and its pulse width with 100ms duration

Free run 1PPS output after 3D_FIX

The module outputs the 1pps after 3D_FIX is obtained.

Cable delay compensation

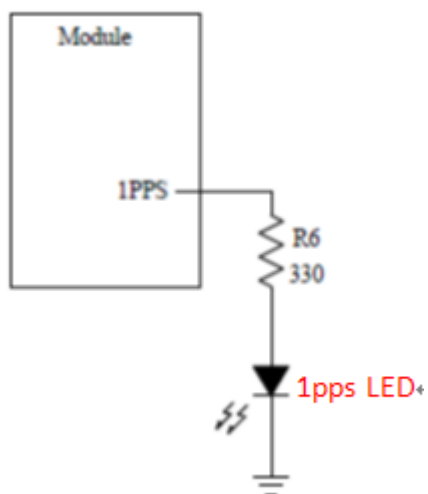
In some cases a long distance connection (~300m) may be needed. For a timing application, the cable length is critical.

For more information on 1PPS signal transmission delay please refer to Appendix V.

Or contact us for more assistant.

LED indicator for 1PPS signal

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



1PPS signal design for IO

2.2 Layout Guideline

Please follow the PCB layout rules covered in this section specifically for better GPS performance while minimizing noise effect.

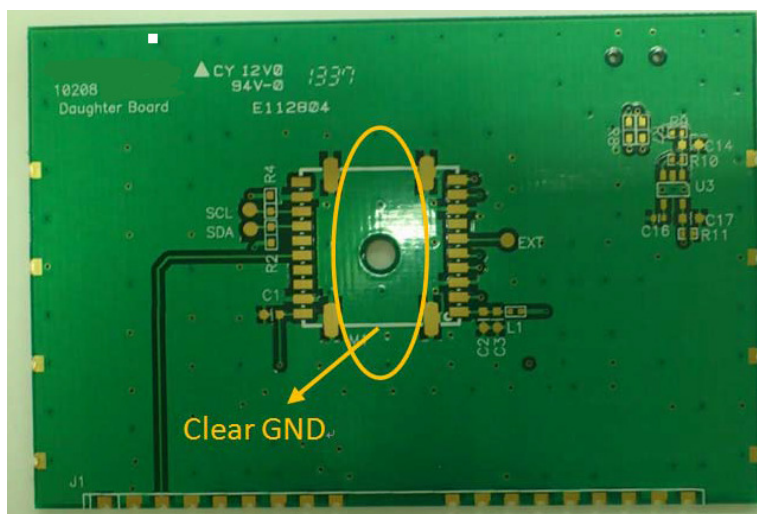
2.2.1 Layout underneath the GPS module

In general, the received GPS signal is normally weak at a signal level around -160dBm to 130dBm. Any noise or harmonics will degrade the quality of GPS reception.

In modern GPS positioning products, most host system contains noise generating components such as LCD screen, MCU, high speed digital and RF system (i.e. GSM, BT, Wi-Fi, DVB-T). In order to minimize the noise effect please design with noises source in mind and avoid it is highly recommended.

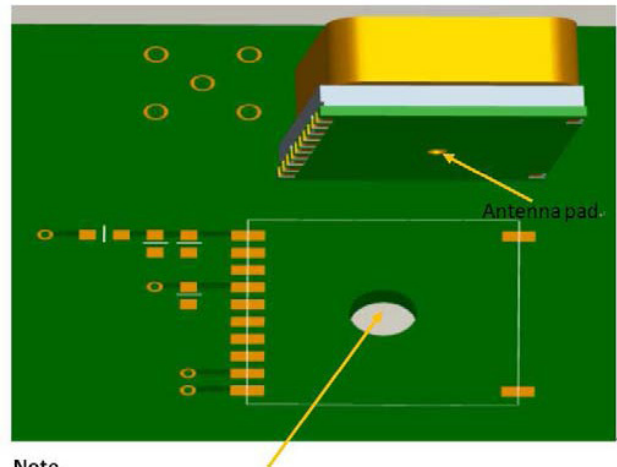
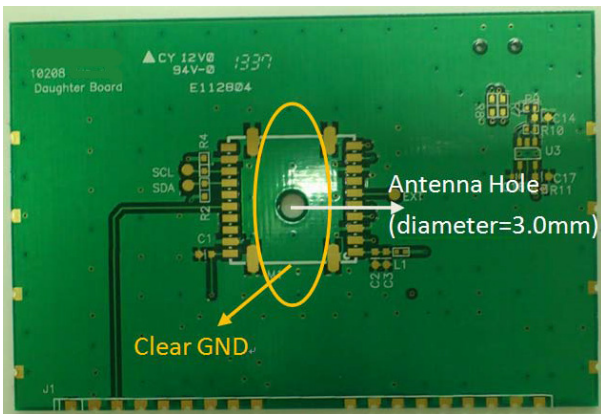
Proper consideration on placing traces can minimize the effect of noise to GPS module.

Do not place any trace underneath the GPS module but give the GPS module a **clean GND** plane is highly recommended.



GPS on a clean GND plane

Place an appropriate hole size according to the location of the extended RF feeding pin of POXA1328-015. The hole provides an area to fit the extended pin on your system PCB so that the coupling of antenna signal to GND plane can be avoided.



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.

GPS receiver module on a clean GND plane and antenna hole

2.2.2 Keep far away from high profile or metal canned components

It is good practice to place GNSS module far away from any high profile component especially with metal case enclosed components such as E-CAP. The Antenna field pattern can be affected and pattern distortion can happen. At the worst case it will degrade the GNSS signal up to ~10dB.



2.2.3 Placement

Place the decoupling capacitors for VCC close to GNSS module.

Place the damping resistors for TX/RX close to GNSS module.

Do not place:

GNSS module in proximity to high-speed digital processing circuitry

GNSS module in proximity to high-current switching power circuitry

GNSS module in proximity to clock sources circuitry

GNSS module in proximity to the high profile metal case component as **section 2.2.2** suggested



2.2.4 Trace

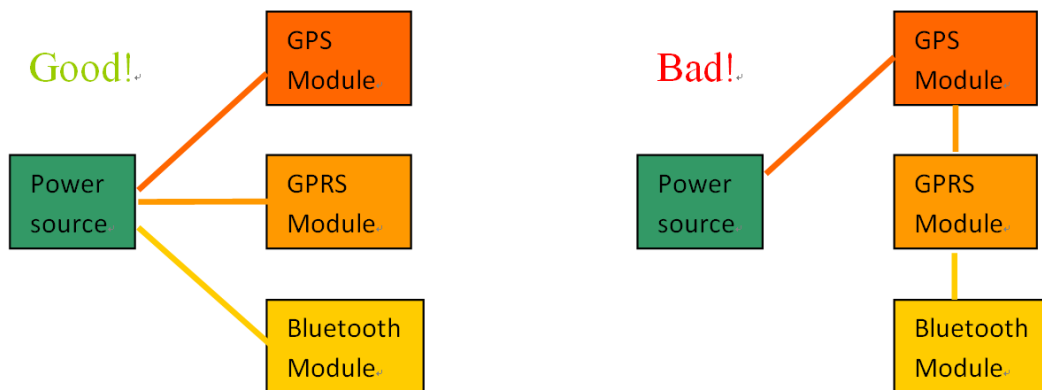
(1) The USB differential signals should be traced closely and of equal-length for better noise immunity and minimum radiation.

(2) Apply 50 ohm impedance RF trace for correct impedance matching.

(3) Any right angle turn in a trace routing should be done with two 135 degree turn or an arc turn



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



2.2.5 Ground Segmentation

In general, the separation of ground between GPS 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 GPS and GPRS.

One way to segment the ground is to place digital and noise component 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 way to do this is to place the two different sets at different layers of the board, while the ground of each layer is connect in one point only which is preferable at the border of the board practically.

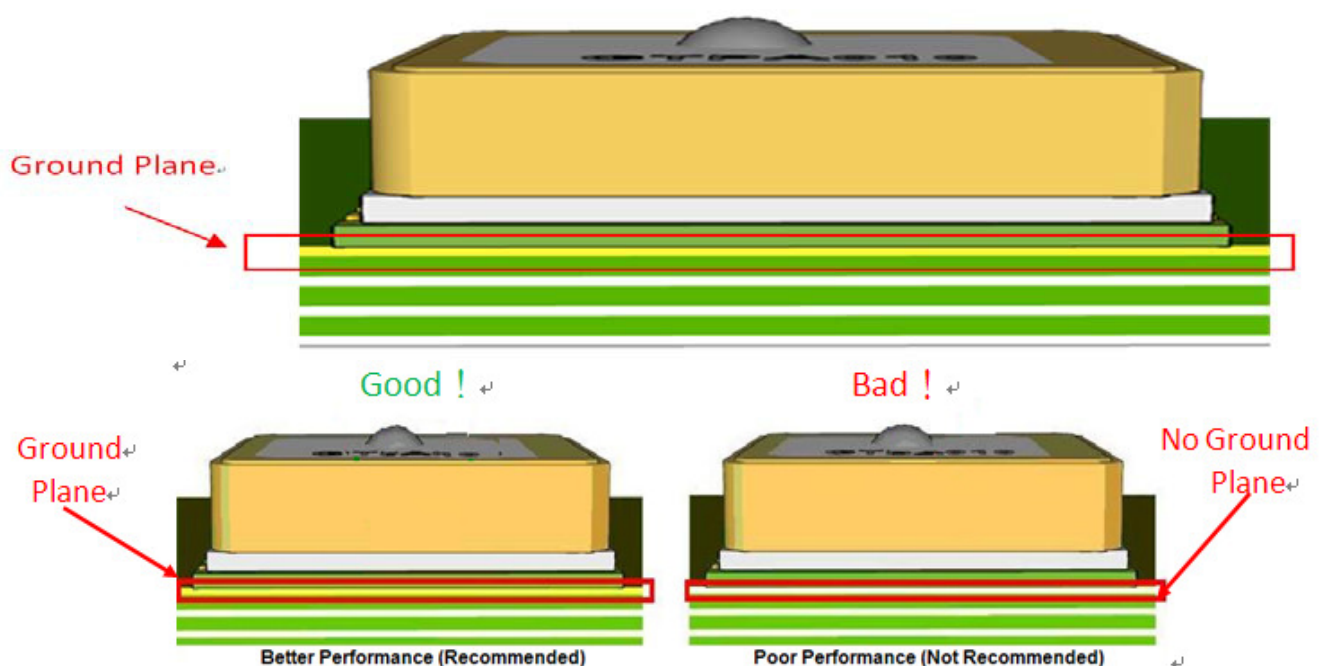
2.2.6 Ground Plane

For Poxa1328-015, a large GND plane directly underneath the module could enhance the magnetic-field line of the antenna for better GPS signal reception. Typically it will improve ~2dB maximum.

It is strongly recommended to have a ground plane designed underneath the GPS 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 GPS module as shown in the figure below.



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.

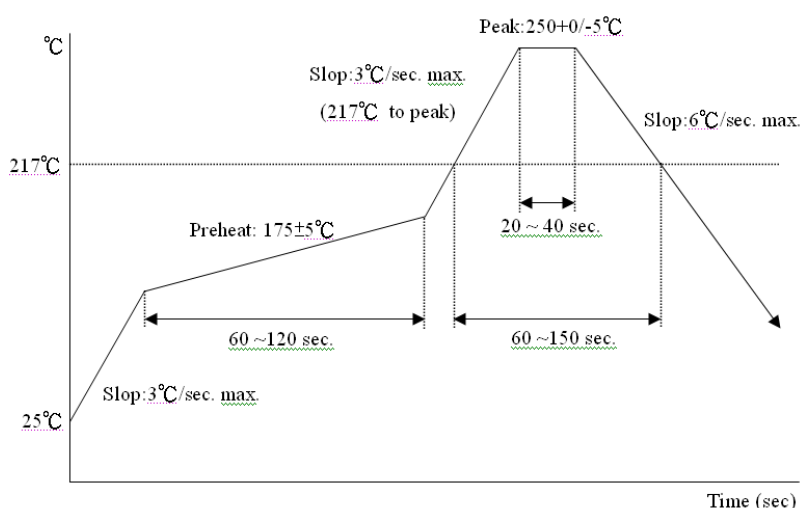
Preheat: 175 ± 25°C, 60 ~ 120 seconds

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.



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 individual GPS module data sheet)

Manual Soldering

Soldering iron: Heat temperature: Under 380°C | Time: Under 3 second.

4. Troubleshooting

How to check for the working status of GPS module?

The first thing to check for is the NMEA sentence output through TX using various application tools. For example: you may use windows default tool – Hyperterminal in WinXP or you may use other 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. Down below are a go through list of possible items to check.

Item 1: VCC

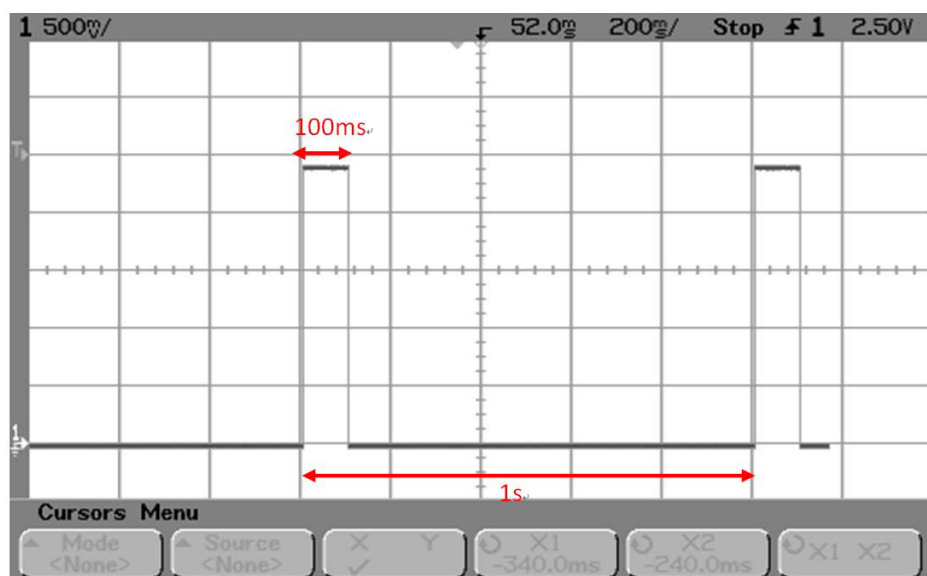
The voltage should be kept between 3V to 3.6V. **(Typical: 3.3V). Please double-check.**

Item 2: VBACKUP

The voltage should be kept between 1.4V~3.6V. **(Typical: 3.0V)**. If backup battery is not installed for the VBACKUP, the pin could still be measured as the voltage comes from the built-in battery recharging circuit. It is recommended to provide power to VBACUP pin as it is used to keep RTC time running and stored navigation data. **Please double-check**

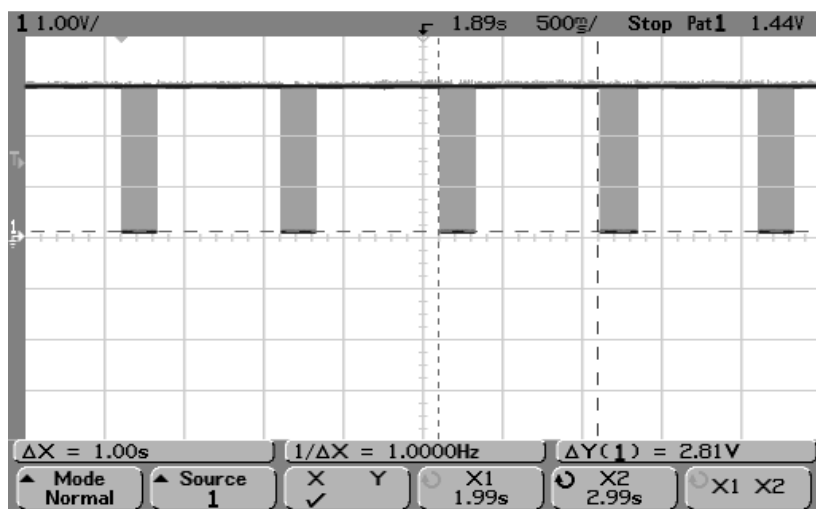
Item 3: 1PPS

If all the items above are within the specification then check 1PPS signal output. After 3D Fix, **the 1PPS pin should continuously output 100ms high for 1second period.**



Item 4: TX0

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



Appendix I: Super Capacitor Design

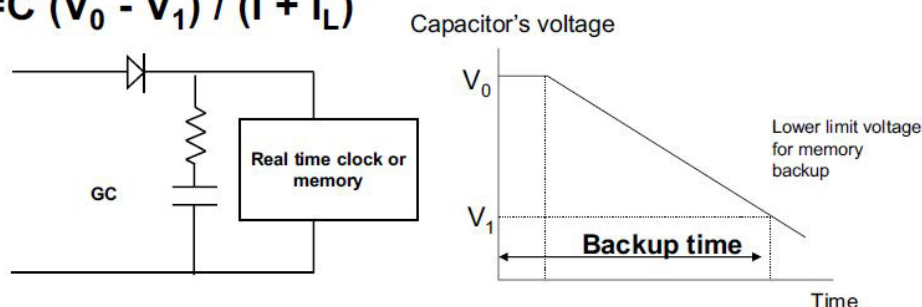
About Super Capacitors

Super capacitor has lifetime similar to that of aluminum electrolytic capacitor. 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 teaches how to calculate the backup time.

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



Example:

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

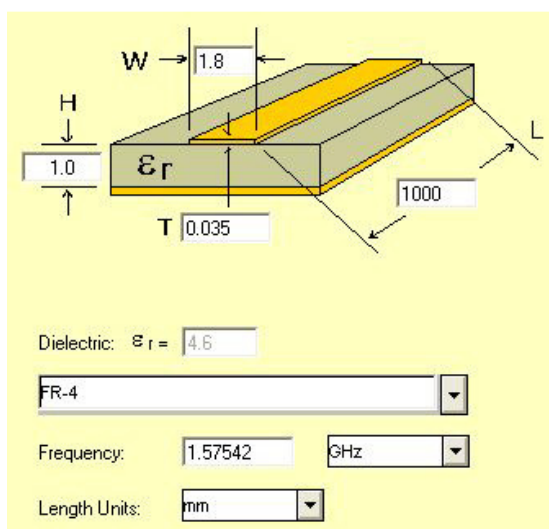
$$\begin{aligned} T &= 0.2 (2.5 - 1.8) / (10 \times 10^{-6} + 0.2 \times 10^{-6}) \\ &= 0.14 / 10.2 \times 10^{-6} \\ &= 13,725 \text{ seconds} \\ &= 3.8 \text{ hours} \end{aligned}$$

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 0.2 μ A

Appendix II: 50 Ω Antenna Matching

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

RF line width (W)	PCB FR4 Thickness (H)	Dielectric parameter	Copper Thickness an ounce
1.8mm	1mm	4.6	0.035mm



Notice:

For multiple-layer layout, designer could place a ground layer in the 2nd layer to minimize the trace width in specific PCB (such FR4) and impedance.

For impedance calculation, free software to calculate trace width or impedance is available.

Please refer to the website for one such software:

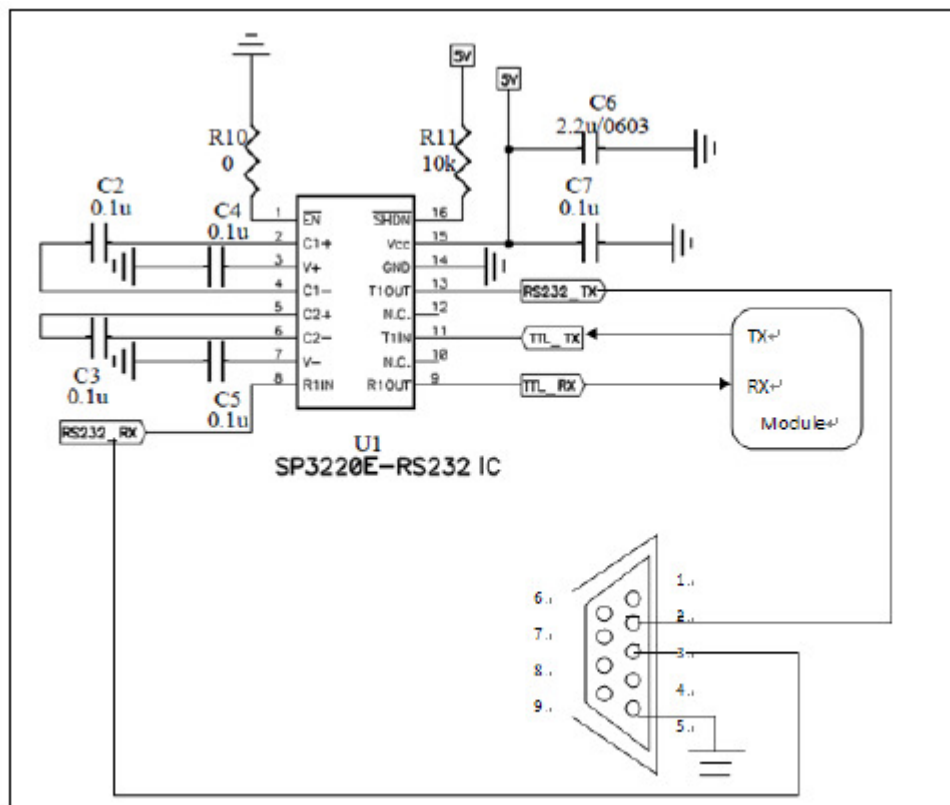
<http://web.awrcorp.com/Usa/Products/Optional-Products/TX-Line/>

Appendix III: UART to RS232 Interface

Typically RS232 or USB interface is required to connect the PC to the POXA GNSS module for communication. Majority of POXA modules use a set of communication port in TTL-logic. Some newer ones supports for direct USB connection. A bridge IC is needed for RS232 signal conversion.

Please note the supported baud-rates are: 4800, 9600, 14400, 19200, 38400, 57600, and 115200 bps.

Please refer to the reference circuit below for RS232 signal conversion. SP3220 IC is used here as an example.

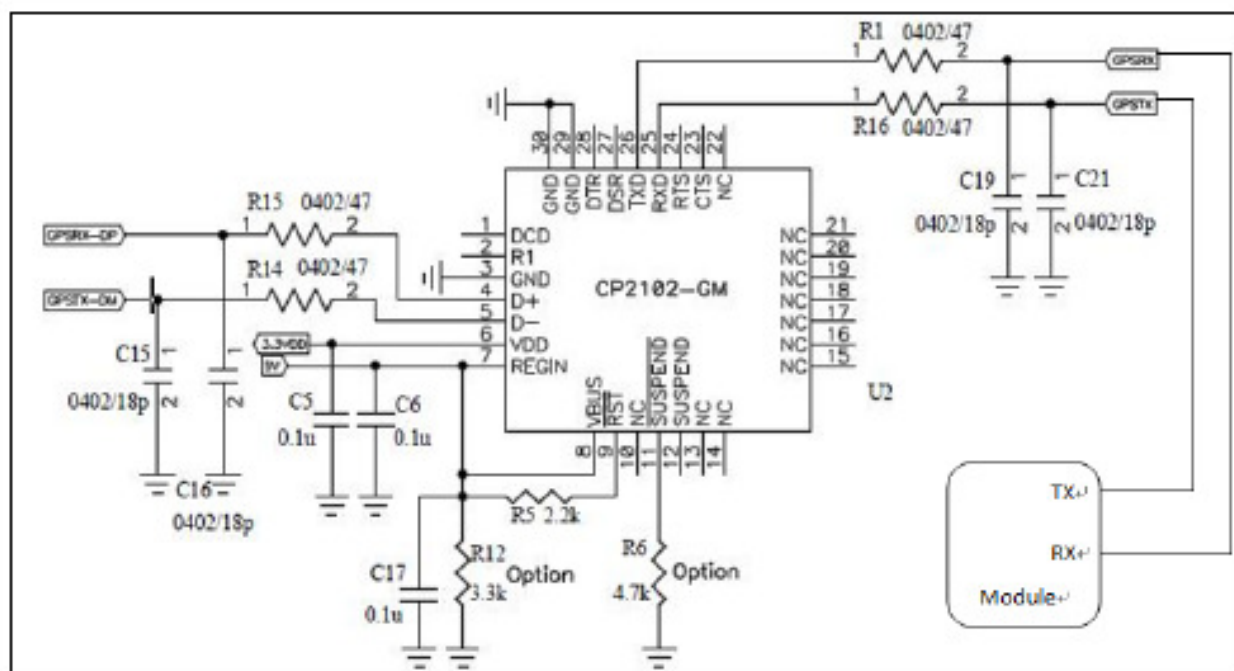


Appendix IV: UART to USB Interface

If the POXA module you have purchased do 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 other operating system, the USB Bridge IC will automatically be recognized as a COM port.

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

Please refer to the reference circuits below for the conversion: CP2102 IC is used here as an example.

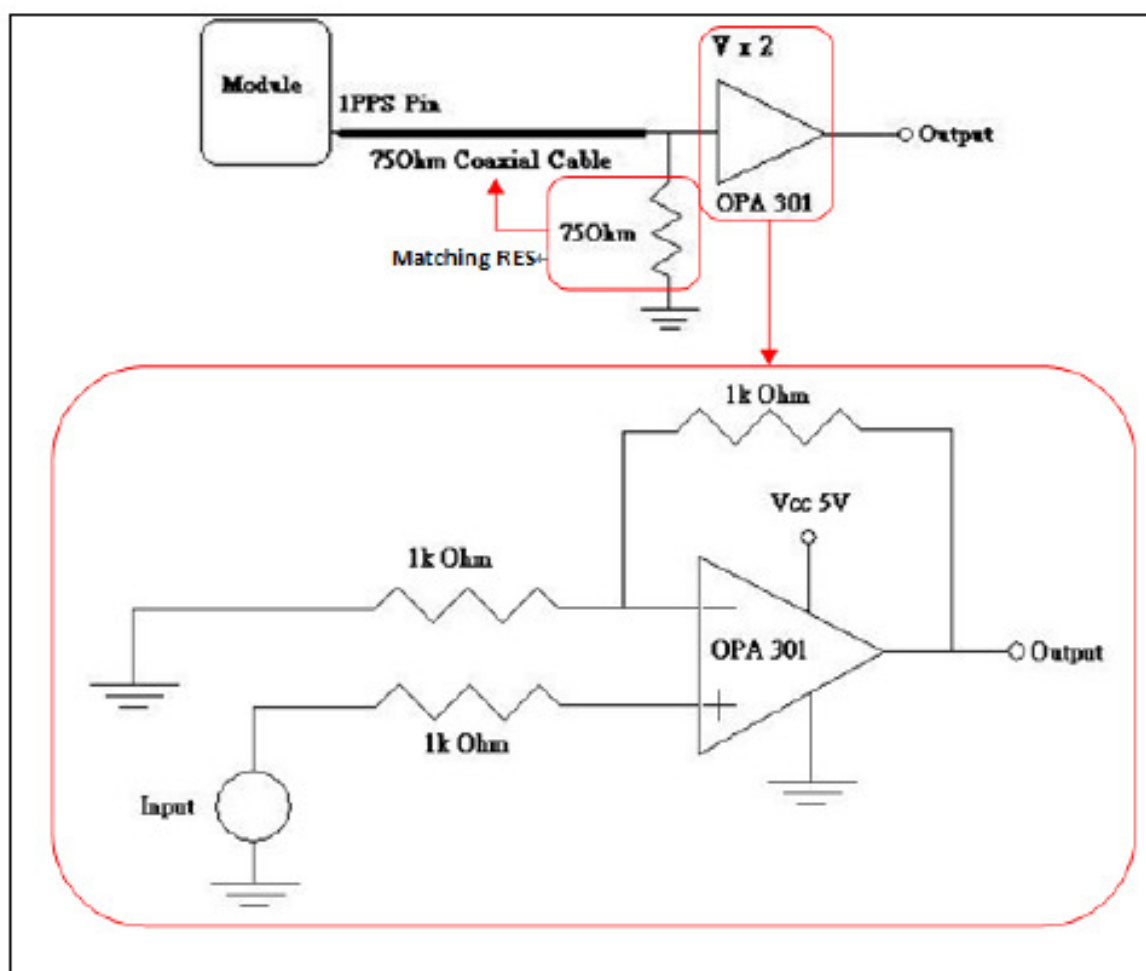


(The Pin29 and Pin30 are the bottom ground pads which are not documented in the IC datasheet. Customer may ignore these two pins in the drawing)

Appendix V: How to efficiently transfer 1PPS through extended distances

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

When using 1PPS signal for time synchronization, the OP-amp chosen should have a **high slew rate** property to prevent large delay in the 1PPS. The accuracy level of 1PPS for POXA GNSS module is within 100ns. This applies to most of POXA's MTK GNSS modules. The reference design below can be used to improve accuracy to within 80ns.



1PPS Delay Time:

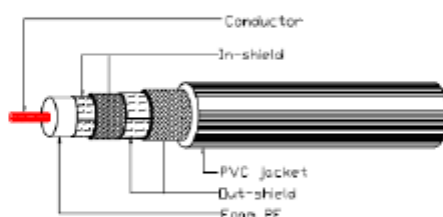
When using coaxial cable for long distances communication using OP-amp for amplification. Please pay close attention to the coaxial cable time delay and the rising time of the wave form due to OP-amp's Slew Rate. A brief explanation for this is given below.

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

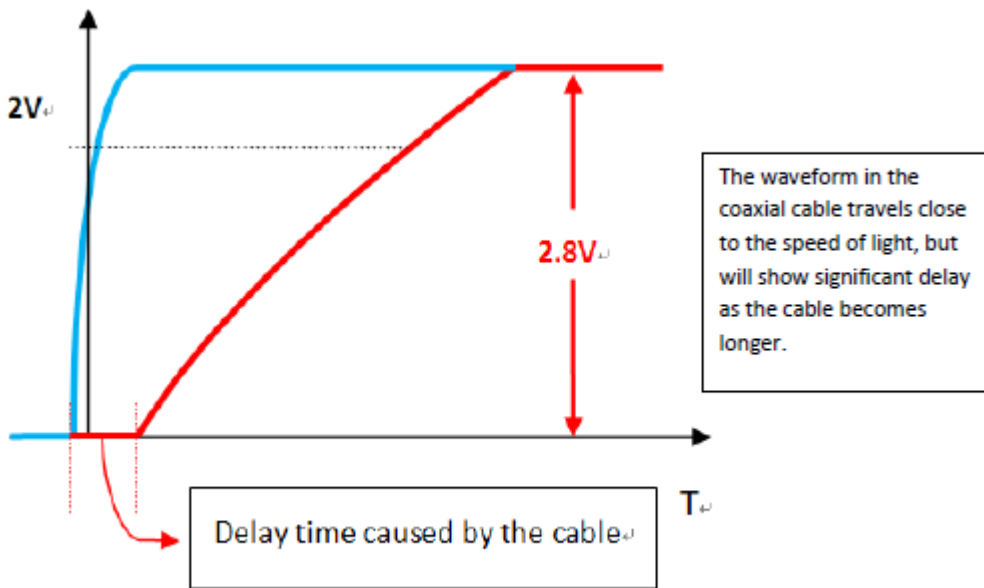
DESIGN



CHARACTERISTICS

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

The delay time cause 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 on the relationships 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)

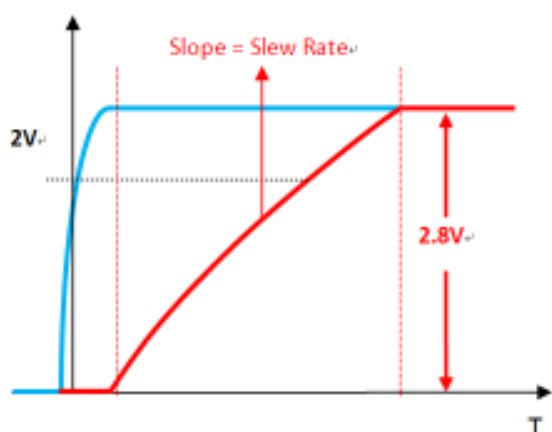
If the speed of the waveform (in coaxial cable) is unknown, it is possible to use a known coaxial cable length to test. For example, the signal travels in 1 meter of coaxial cable can have its delay time measured by the oscilloscope as 5ns and consequently the μ_p will be $2 \cdot 10^8$ (m/s).

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

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

Moreover, more 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



The left figure shows Slew Rate: which is the time needed for rising waveform. When the SR ($V/\mu\text{sec}$) of OPA is higher, the rising time will be shortened as well.

Recommended OP-amp

If inadequate OP-amp is chosen (i.e. with 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/ μsec
- Tiny Packages: MSOP and SOT23

Voltage degradation of communication cable

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

Appendix VI: Caution for Reflow Soldering Process

	Details	Suggestions	Notes
1	Before proceeding with the reflow-soldering process, the GPS module must be pre-baked.	Pre-bake Time: 6 Hours @ 60 \pm 5°C or 4 Hours @ 70 \pm 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 tray 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 if any signs of cold weld(ing) 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 weld(ing) or false welding.
3	Special attentions are needed for 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 causes the PCBA to collide into one another during the unloading process.
4	Before the PCBA is undergoing a reflow-soldering process, the production operators must check with 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 reflow-soldering process.	The operators must check with own eyes and readjust the position before 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 rotational offset between the patch antenna and module

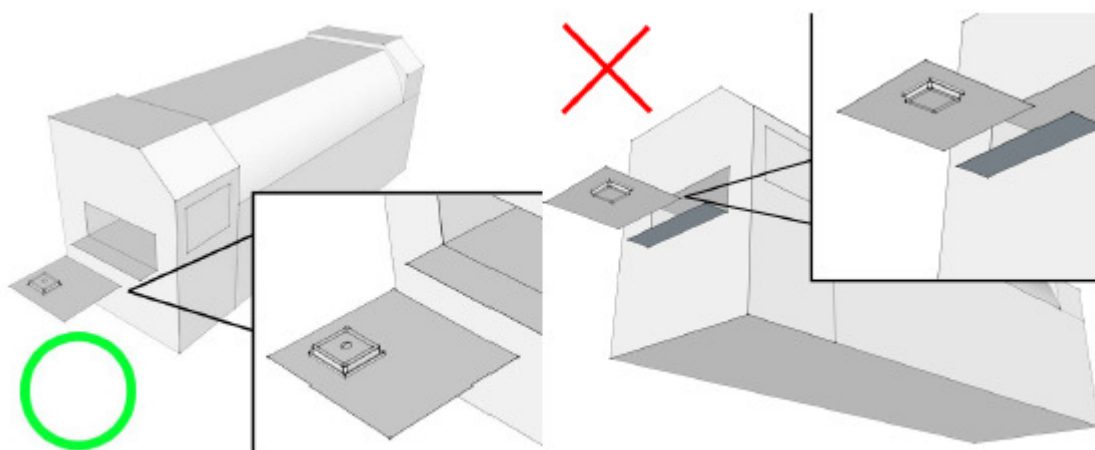
Note: For patch antenna users: patch antenna refers to the patch-on-top antenna which is built on the GPS module (i.e. PA/Gms Module Series) and may not be applicable to all GPS modules.

	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 positional shift which may occurred when the module is still hot.	<p>1. One may use electric fan behind the reflow machine to cool them down.</p> <p>2. Cooling the PCBA can prevent the module from shifting due to fluid effect.</p>	It is very easy to cause positional offset to the module and its patch antenna when handling the PCBA under high temperature.
6	<p>1. When separating the PCBA panel into individual pieces using the V-Cut process, special attentions are needed to ensure that there are sufficient gap between patch antennas.</p> <p>2. If 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 rotational offset to the patch antennas.</p>	<p>1. The blade and the patch antenna must have a distance gap greater than 0.6mm.</p> <p>2. Do not use patch antenna as the leverage point when separating the panels by hand.</p>	<p>1. Test must be performed first to determine if 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.</p> <p>2. An uneven amount of manual force applied to the separation will likely to cause positional shift in patch antenna and module.</p>
7	When separating panel into individual pieces during latter half of the production process, special attentions are needed to ensure the patch antennas do not come in contact with one another in order to prevent chipped corners or positional shifts.	Use tray to separate individual pieces.	It is possible to chip corner and/or cause a shift in position if patch antennas come in 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/Gms Module Series) and may not be applicable to all GPS modules.

Other Cautionary Notes on Reflow-Soldering Process:

1. Module must be pre-baked **before** going through SMT solder reflow process.
2. The usage of solder paste should follow “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 SMT production line and storage area. 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, please notice 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 to prevent positional shift during the loading process.
6. Before the PCBA is going through the reflow-soldering process, the operators should check by his/her own eyes to see if there are 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 SMT protection line is running a double-sided process for PCBA, please process GPS module during the second pass only to avoid repeated reflow exposures of the GPS module. Please contact ACTE- beforehand if you must process GPS module during the 1st pass of double-side process.



Place GPS module right-side up when running reflow-solder process, do not invert.

Manual Soldering

Soldering iron:

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

Notes:

1. Please do not directly touch the soldering pads on the surface of the PCB board to prevent further oxidation
2. The solder paste must be defrosted to room temperature before use so 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. Please watch out for the spacing between soldering joint, as excess solder may cause electrical shortage
5. Please exercise with caution and do not use extensive amount of flux due to possible siphon effects on neighboring components, which may lead to electrical shortage.
6. Please do not use the 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 and will leads to electrical shortage.