Technical Document

GlobalTop Technology Inc.

GNSS Module Application Notes (MT3333 series)

Revision: A05

Reference layout, design tips, guides, and cautions for GlobalTop GNSS modules.

Version History

Title:	**GlobalTop GNSS Module Application Notes (MT3333 series)**
Subtitle:	**GNSS Module**
Doc Type:	**Technical Document**

Revision	Date	Editor	Description
A00	2013-01-23	Allen	First Release
A01	2013-08-14	Dylan	Add Gms-g6 module, Gms-b6 module
A02	2013-10-15	Dylan	Add Gmm-g3(B) module
A03	2014-10-28	Rock	Add PA6E-CAM module
A04	2014-11-12	Delano	Revised Gms-g6, Gms-b6 ground plane
A05	2016-05-13	Max	Add 1.8V boost to 3.3V application and switch

Table of Contents

Attention

Please read carefully before you start

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

➢ This application note provides the necessary guideline to successfully design a system using GNSS modules. For detailed module specification, please refer to the corresponding datasheet of GNSS module.

➢ GNSS Module is an electrostatic sensitive device, please don't touch GNSS module directly, please follow ESD safety rule when handling.

➢ 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 full ephemeris data received.

Technical Support

If you have any technical problems or any required information not found elsewhere in our documents, please feel free to contact us. When write to us please follow the form below to help us to serve you better.

   1. Your company name and website

   2. Description about application and system. Please attach illustration figures where applicable.

   3. GNSS module type

   4. GNSS firmware version

   5. Detail descriptions of problem encountered and the conditions by which the problem occurred. Please provide with pictures or videos files where applicable.

   — Test setup descriptions
   — The problem or issue addressable in the figure

Technical contact information: support@gtop-tech.com

1. Introduction

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

Stand Alone Module
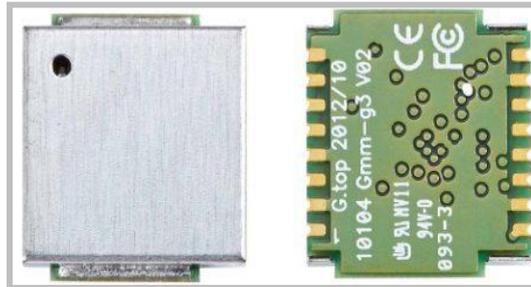
Gmm-g3, Gmm-g3(B)
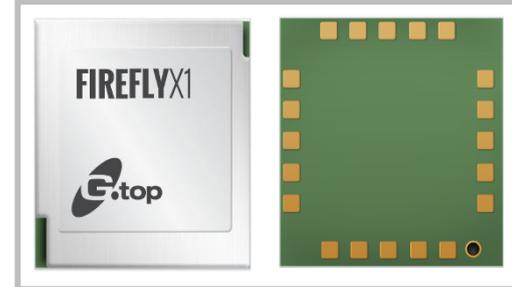
Gmm-3301

Built-in Chip Antenna Module
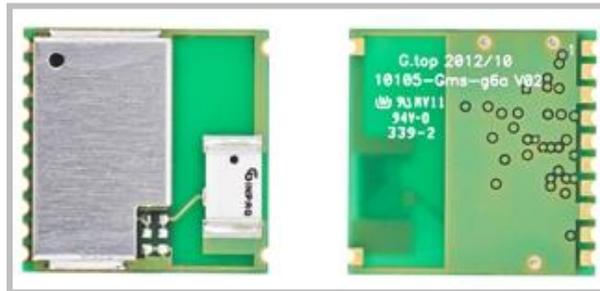
Gms-g6a

Built-in Patch Antenna Module


Gms-g6

Gms-b6
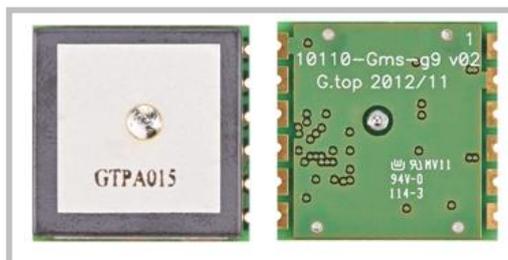
Gms-g9
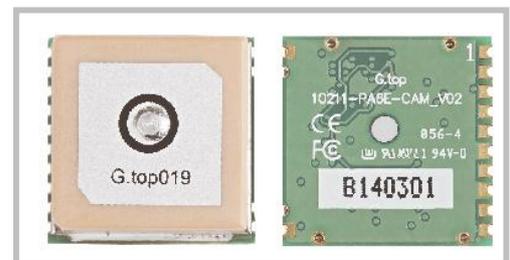
PA6E-CAM

2. General Rules for Design-in

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

2.1 Circuit Design

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

2.1.1 Power supply VCC

It is necessary to provide a clean and stable power supply for the GNSS 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
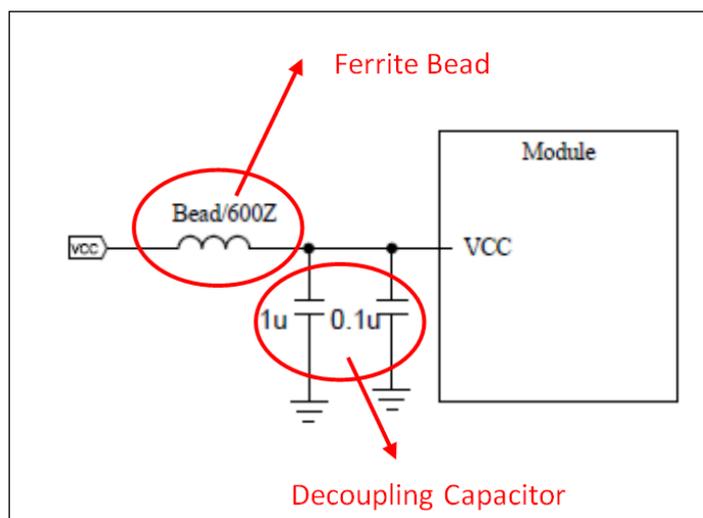
Power design for GNSS module

2.1.2 VBACKUP backup battery

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

There are 2 applications in applying VBACKUP to the GNSS module.

Application 1:

Backup power can be used when the main power is not supplied to keep RTC running and navigation data. For a short term VCC off, with this backup power, the GNSS module is capable of a faster TTFF, shorter Time-to-First-Fix, and short hot start time at next power on.

For most system, it is recommended to provide the module with backup power (e.g. Li-Ion rechargeable coin battery, super cap) See figure for reference design.

About super capacitor reference design, please refer to Appendix I: Super Capacitor Design.

The module has a built-in battery recharge circuit. It charges the rechargeable coin battery while the module is power on.
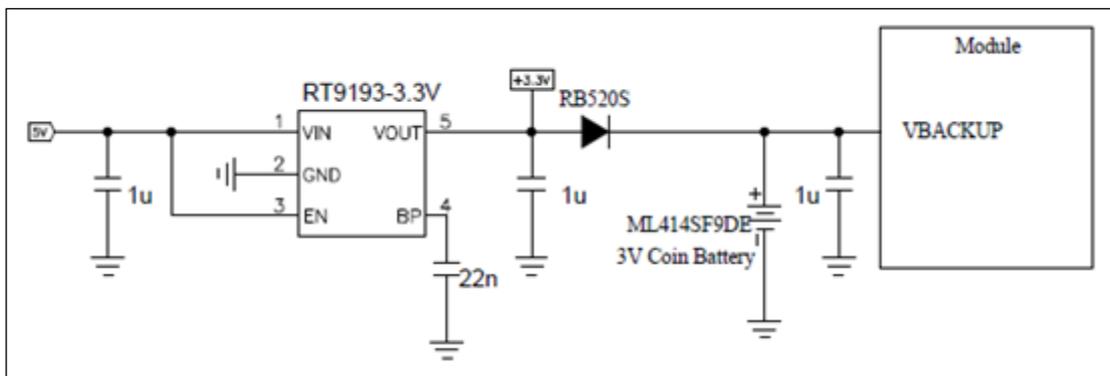
The rechargeable coin battery to the VBACKUP of the

Application 2:

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.
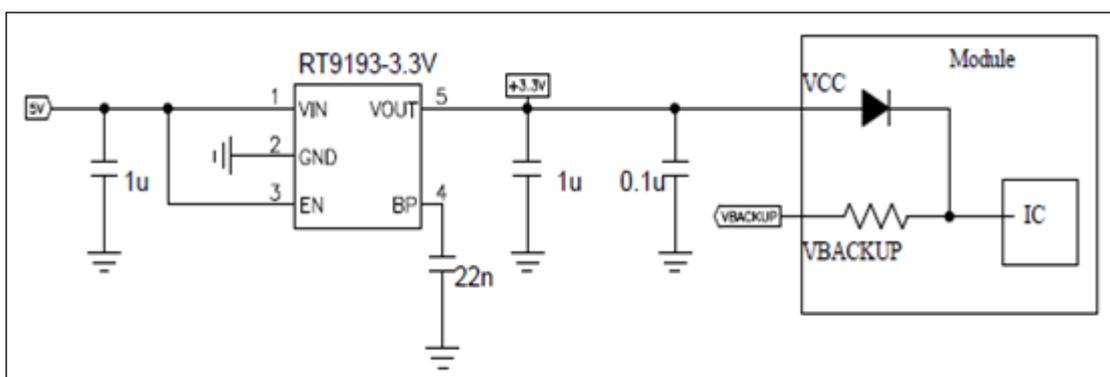
Built-in a charging circuit for GNSS module

GlobalTop Technology
GNSS Module Application Notes (MT3333 series)

9

Ver. A05

2.1.3 UART 0 (RX/TX) –Serial Interface

 Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

(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 GNSS 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 choke coil.

(3) 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.

(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" in Appendix III.

(5) 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 GND-Ground

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

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

The separation of ground between 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 in general (such as GNSS 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 connected to one point only.

Another way 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 preferable located at the border of the board practically.

2.1.5 Antenna Compliance Design

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B)

GNSS antenna is a receiving part of the device to acquire weak GNSS signal from sky. Popular solution would be ceramic patch antenna because of its small form factor with low cost. There are two types of antennas, passive and active.

Passive antenna is solely antenna by itself without signal amplifier such as LNA. Examples are patch antenna and chip antenna. If you have your own patch antenna, it is required to get it fine tuned for the specific module model to obtain the best possible signal strength.

External antenna is a standalone device integrates with LNA and patch antenna with a RF cable for higher gain and better performance than that of passive antenna.

Antenna can 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 operate with good enough performance.

Design Patch antenna with GNSS module

(1) In general, a 50Ω patch antenna will work well with GNSS module. The antenna can be connected to the Antenna_IN pin with a 50Ω impedance trace.

(2) Pleases keep the patch antenna far away from noise source such as switching power supply and high speed digital logics and radar wave guide.

(3) The 50Ω trace should be kept as short as possible to reduce the chance of being picking up the noise from air and PCB. A simple direct-line trace is most recommended.

(4) If needed, a matching circuit could be placed between patch antenna and GNSS module. The matching circuit design should be discussed with the module and patch antenna maker.

(5) For 50 Ω matching, please refer to ""50 Ω Antenna Matching" in Appendix II.
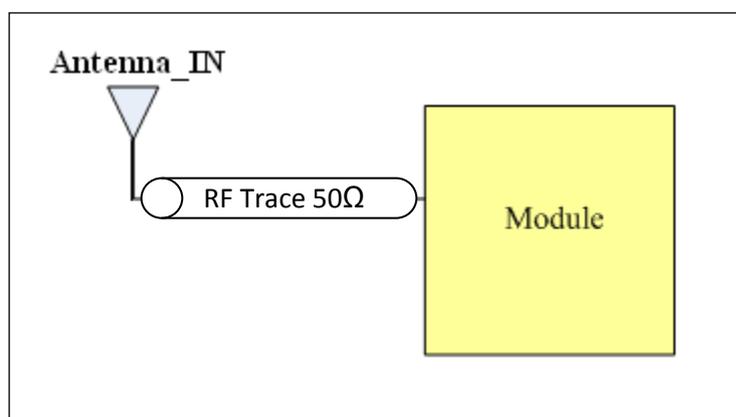
PCB trace design for antenna impedance matching

Select External Antennal architecture with GNSS module

An external antenna needs DC power to function. A typical method is to feed DC power into the RF trace. The external antenna can extract the power from the RF trace. Thus we can use the RF trace to transport both RF signal and DC voltage. To fully achieve this, we use a RF chock coil to couple the DC power to RF trace.

For your reference, in general, GlobalTop modules provide the circuit connecting external antenna and power feeding method as depict in the figure below. Please select the module carefully while applying the design with an external antenna.

The power supply for the external antenna needs to be externally feed into the module via the VANT pin and connect antenna to Antenna_IN pin. (i.e. **Gmm-g3, Gmm-g3(B)**)

Power Feeding Configuration
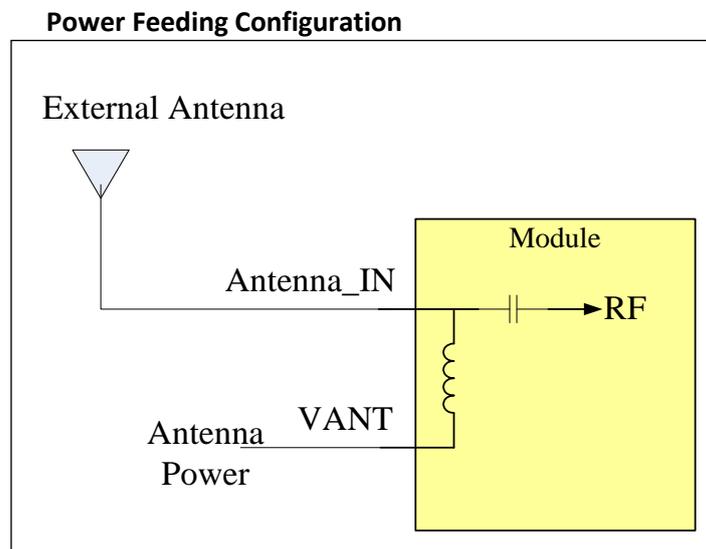
External Antenna of Specification

When select your external antenna, reference the specification shown in the table below.

For Gmm-3301, Gmm-g3 module user, you will need 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
Impedance	50Ω
Noise Figure	2dB

For Gmm-g3(B) module user, you will need 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
Impedance	50Ω
Noise Figure	1.5dB

2.1.6 1PPS

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

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

Low Voltage level: 0~0.4V

High Voltage level: 2.8~3.1V

Period: 1s

Accuracy (jitter):±10ns

100ms pulse width duration (Firmware customization for pulse width duration is available)
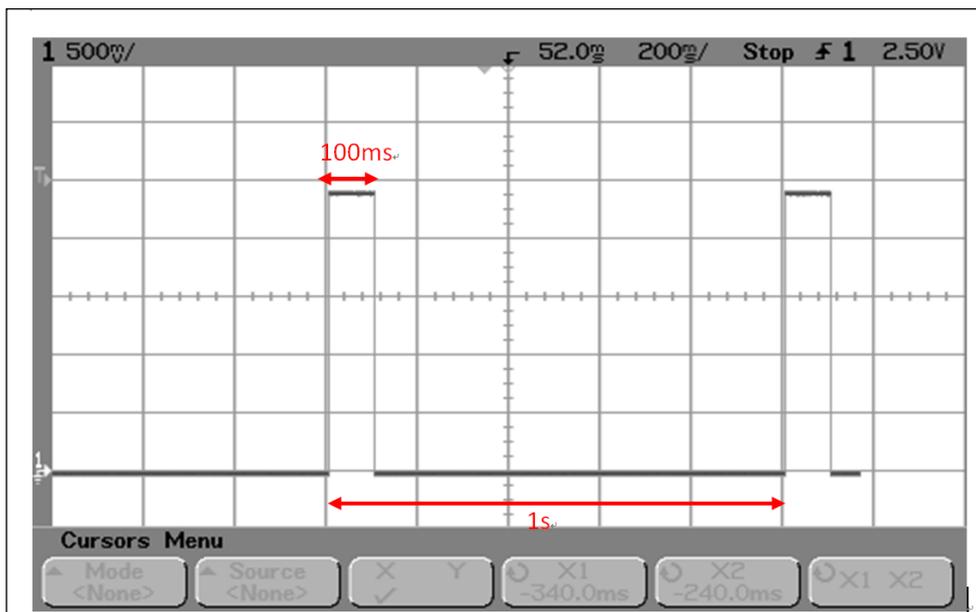
1PPS signal and its pulse width with 100ms duration

Free run 1PPS output before 3D_FIX

GlobalTop standard GNSS module outputs 1PPS signal after the module obtained 3D_FIX. This is a factory default setting.

For an always 1PPS signal output configuration, regardless of 3D_Fix or not, this is available to our customer via firmware customization service. Please be noted that the accuracy before 3D_Fix on the 1PPS is a free run and will lock until GPS 3D_Fix.

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.

Delay caused by the cable length could be compensated with firmware setting. Please contact us if that is useful for your implementation.

LED indicator for 1PPS signal

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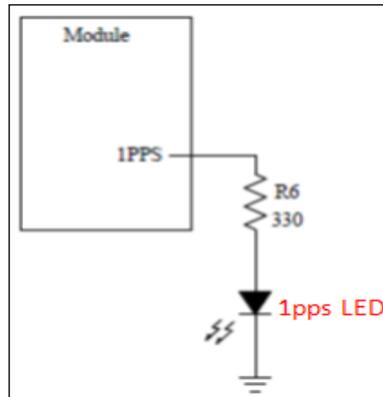
1PPS signal design for IO

2.1.7 3D_Fix

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

3D_Fix signal is an output pulse signal used for GNSS fix application. 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)

Signal output after 2D or 3D Fix:

Duration: Constant low level output

(Firmware customizable)
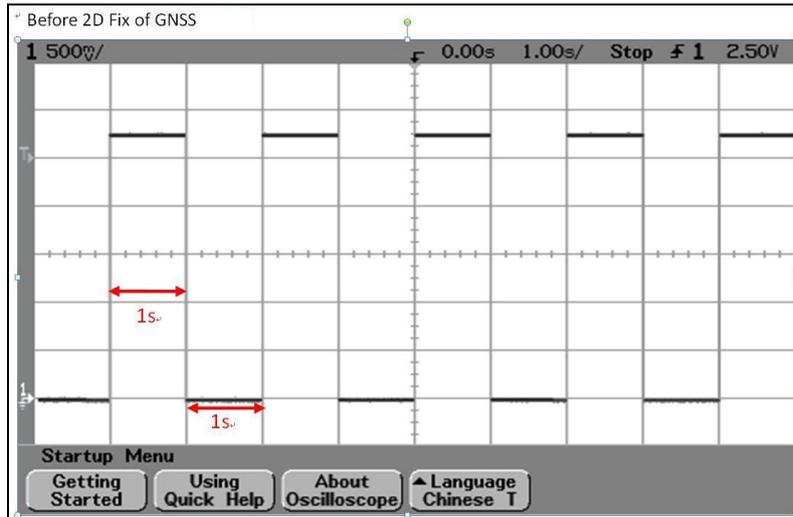
For 3D_Fix indication, user may connect a LED with a 330ohm resister in series in two ways showing in the two figures below.
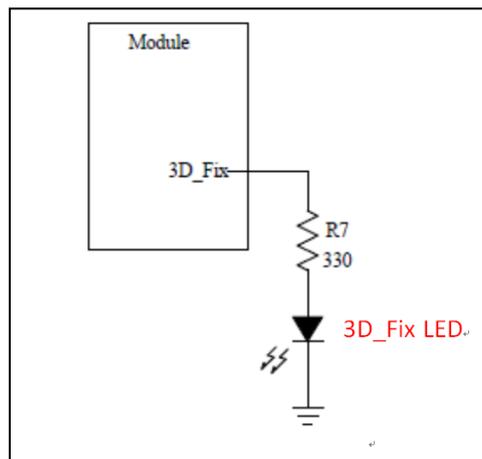
3D_Fix signal design for IO

(LED ON when high, LED OFF when low)
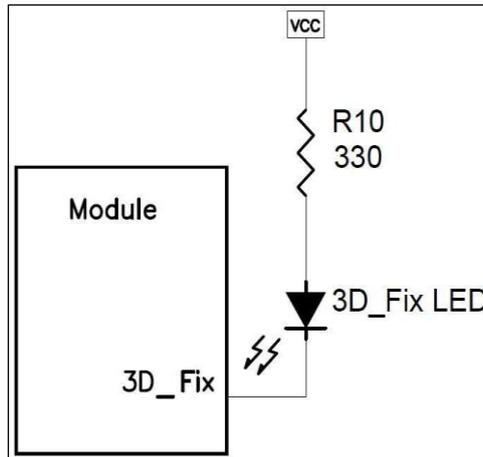
3D_Fix signal design for IO

(LED ON when low, LED OFF when high)

2.1.8 1.8V boost to 3.3V application

In the 1.8Voltage ' system, if you want to use 3.3V's GNSS module, we need to consider two conditions, first one is power supply translation, second is signal level shift; About power supply translation, we can adopt boost circuit which can let 1.8V boost to 3.3V (refer to below figure), TPS61097A-33 is Ti's boost IC which can support boost function, in its application, C1 and C2 need to use 10uF and L1 is 10uH, it can support about 100mA output; about capacitor and inductor's placement can refer it's application note , you can go to Ti web site:
http://www.ti.com/lit/ds/symlink/tps61097a-33.pdf
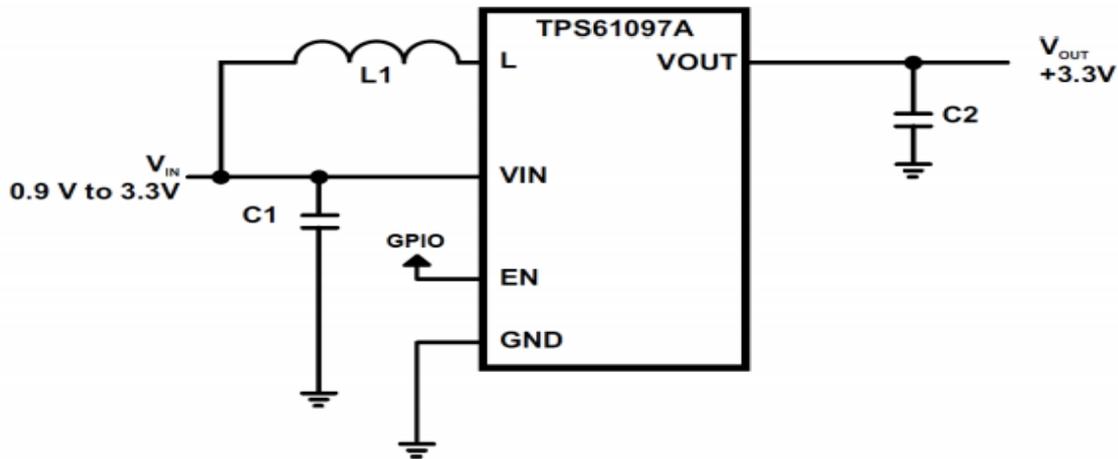
(Application schematic)
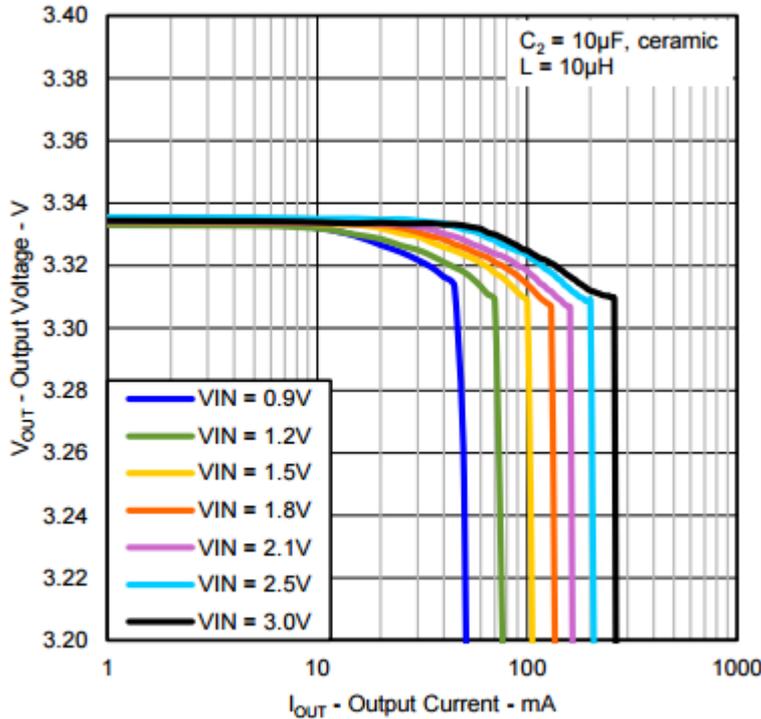
(Output Voltage vs Output Current)

About signal level shift, when user's host system is 1.8 Voltage and Gtop module is 3.3 Voltage ,so host system can control Gtop module by adopting signal level shift circuit (refer to below), R1~R4's values are default value, In actual design, user can adjust their values to achieve control ; 2N7002L can select low $RDS_{(On)}$, it will be reduced power consumption and voltage's drop.

2.1.9 Antenna switch reference

To implement antenna switch design for standalone module, you can adopt a SPDT (single pole, double throw) IC to achieve this function, RF input1 and RF input2 can be different antenna source, host can select desired antenna source by controlling Vcont Pin .
If you want more detail information, please refer to
http://www.njr.com/semicon/PDF/NJG1635AHB6_E.pdf
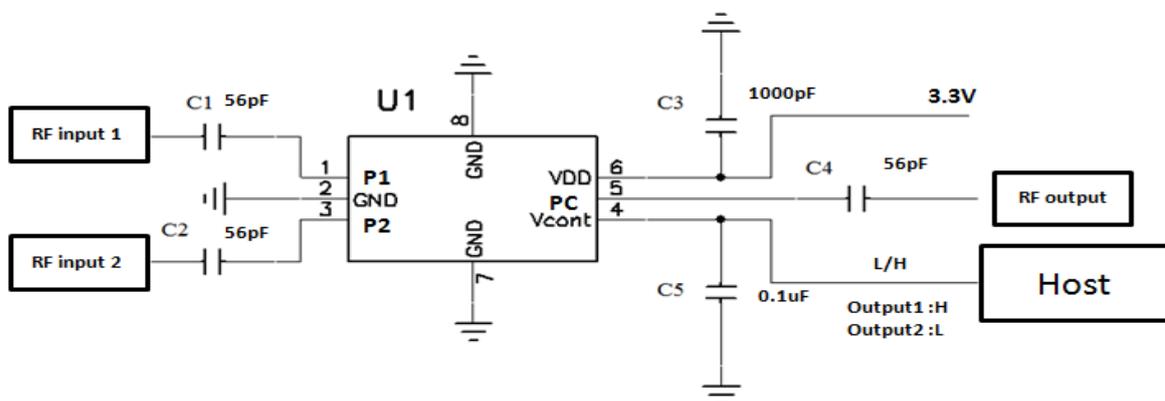
2.2 Layout Guideline

Please follow the layout guideline for the right GNSS module when design.

2.2.1 Layout underneath the GNSS module

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

In general, GNSS modules have high receiving sensitive at around -165dBm. During the hardware integration, try to avoid noise or harmonic in these bands to prevent unnecessary reception degradation: Beidou 1561.098MHz ±2.046MHz and GPS 1572.42MHz±2MHz or GLONASS 1598.0625~1605.375MHz.

In modern GNSS 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.

For module Gmm-3301, Gmm-g3, Gmm-g3(B)

Don't place any trace underneath the GNSS module where the circle area is pointed in the figure. In other word, give GNSS module a clean GND plane.
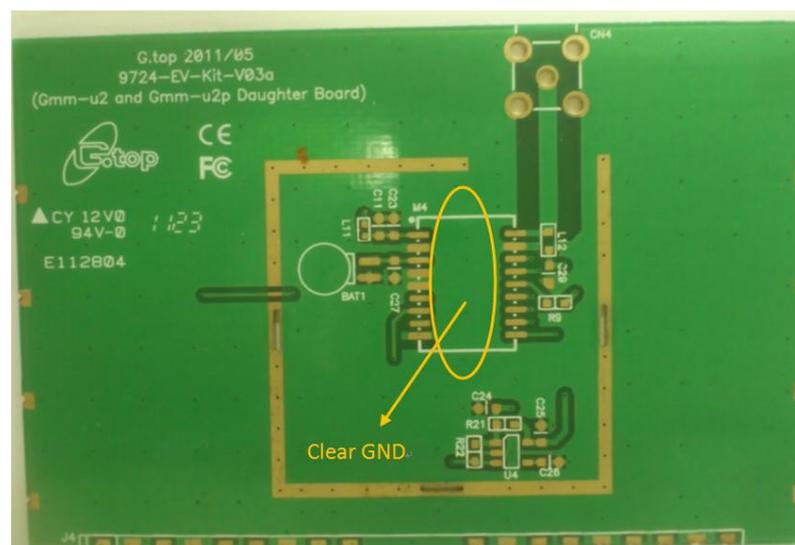
GNSS module on a clean GND plane

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

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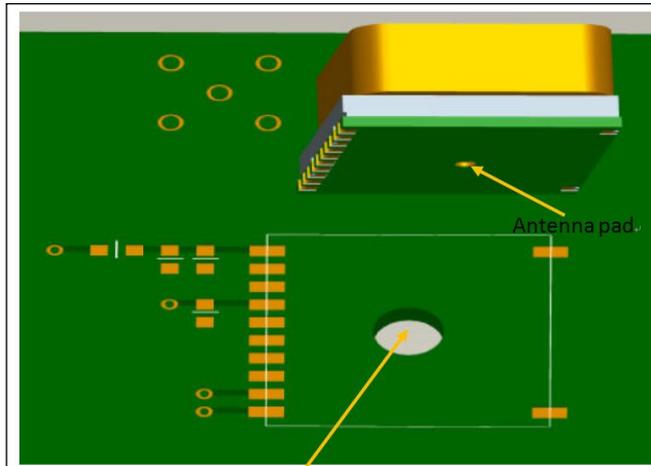
Note.

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.
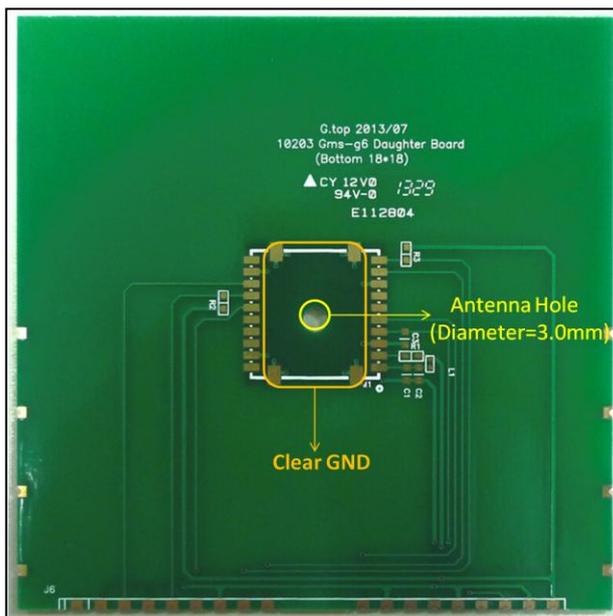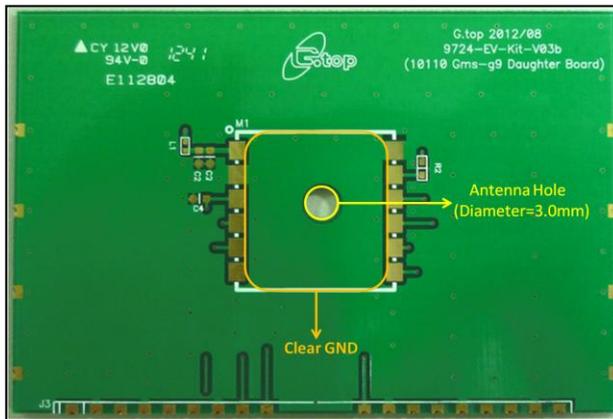
GNSS module on a clean GND plane and antenna hole

2.2.2 Keep far away from high profile or metal canned components

Compatible Model: Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

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, Coin battery, Pin Header. The Antenna field pattern can be affected and pattern distortion can happen. At the worst case it will decrease the GNSS signal up to ~10dB.

This document is the exclusive property of GlobalTop Tech Inc. and should not be distributed, reproduced, into any other format without prior permission of GlobalTop Tech Inc. Specifications subject to change without prior notice.

Copyright © 2013 GlobalTop Technology Inc. All Rights Reserved.

2.2.3 Placement

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

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

2.2.4 Trace

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

(1) The USB differential signals should be trace closely and of equal length for better noise immunity and minimum radiation.
(2) Apply 50 ohm impendence 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.
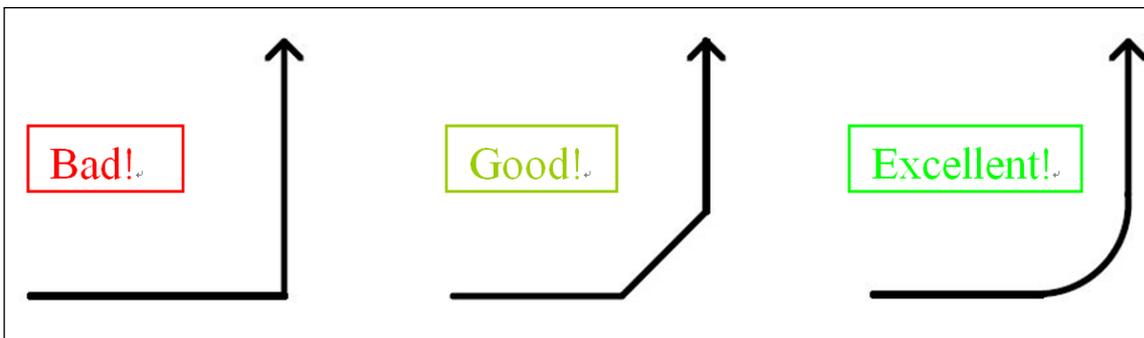
It is better to have independent trace of power source for any device
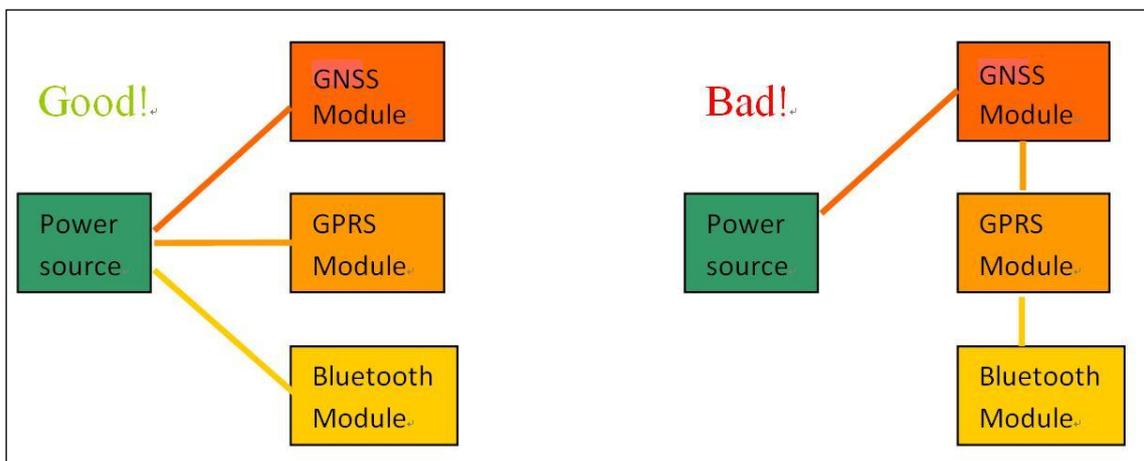
2.2.5 Ground Segmentation

Compatible Model: Gmm-3301, Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM

In general, the separation of ground between 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 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 the place the two different sets at different layers of the board, while the ground of each layer is contacted in one point only which is preferable located at the border of the board practically.

2.2.6 Ground Plane

(1) Compatible Model: Gms-g9,PA6E-CAM, Gms-g6, Gms-b6

For Gms-g9, PA6E-CAM, Gms-g6, Gms-b6 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 the figure below.
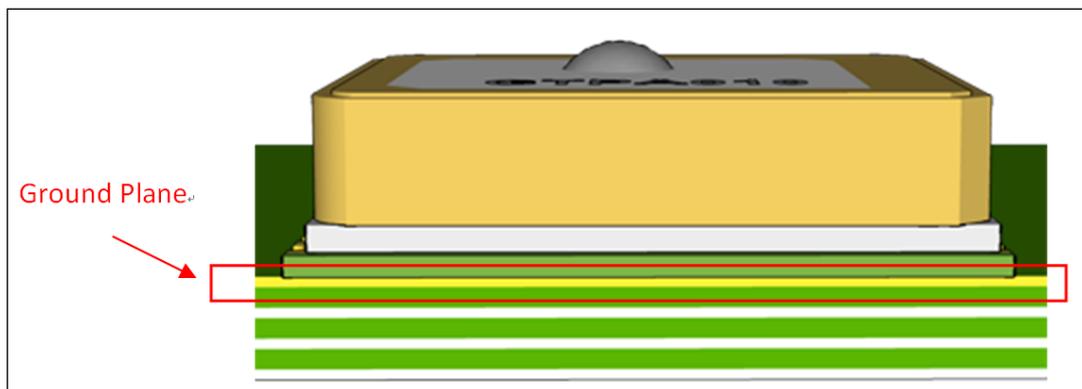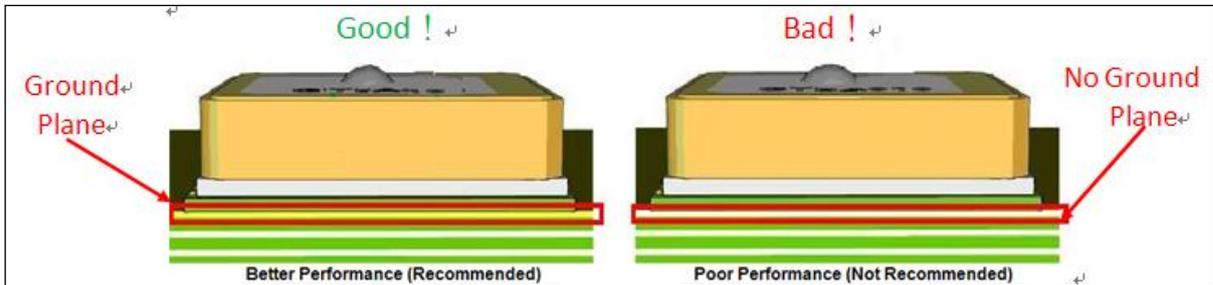
Ground Plane

Ground Plane for FR4 Design

(2)Compatible Model: Gms-g6a

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

As shown in the figure, the area circled where the chip antenna area is when placed on the PCB, no ground plan should be placed on this chip antenna area and also all other layers underneath.
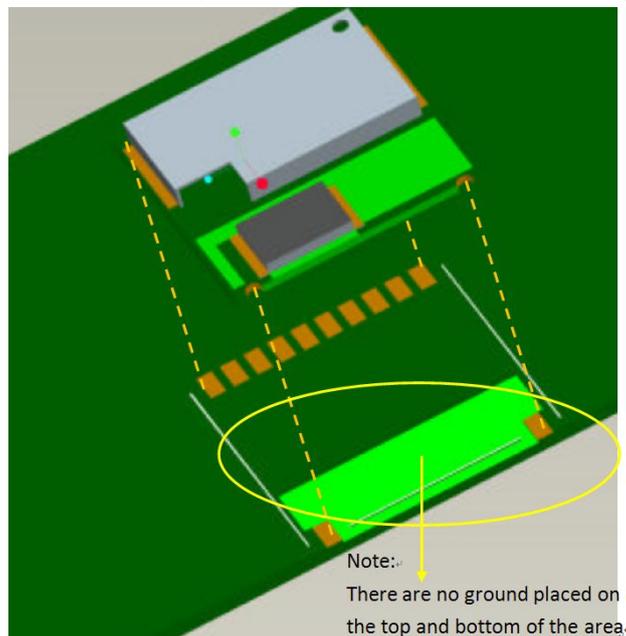
Note: The clearance area size at least 15x6.3mm.

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 below.
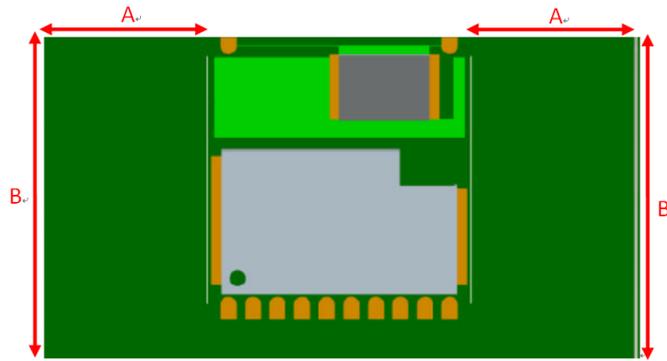
Variation in reception sensitivity according to the size of the GND plane is given in the table below.

Ground Plane Area (A x B) (mm)	Sensitivity increase in dB
No Ground Plane	0dB
2x 20 mm	+2dB~+4dB
7 x 20 mm	+4dB ~+5dB
12 x 20 mm	+5dB~+6dB
17 x 20 mm	+6dB~+7dB

2.2.7 Metal effect the GNSS module

Compatible Model: Gms-g6a

For Gms-g6a. Chip Antenna is a sensitive component with environment.
A consideration regarding to the housing and the environment where it was placed should be carefully taken.

(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 of a conductive material such as metal case, metal film or plastic housing coated in conductive material, which blocks GNSS signal from reaching the antenna; and 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.
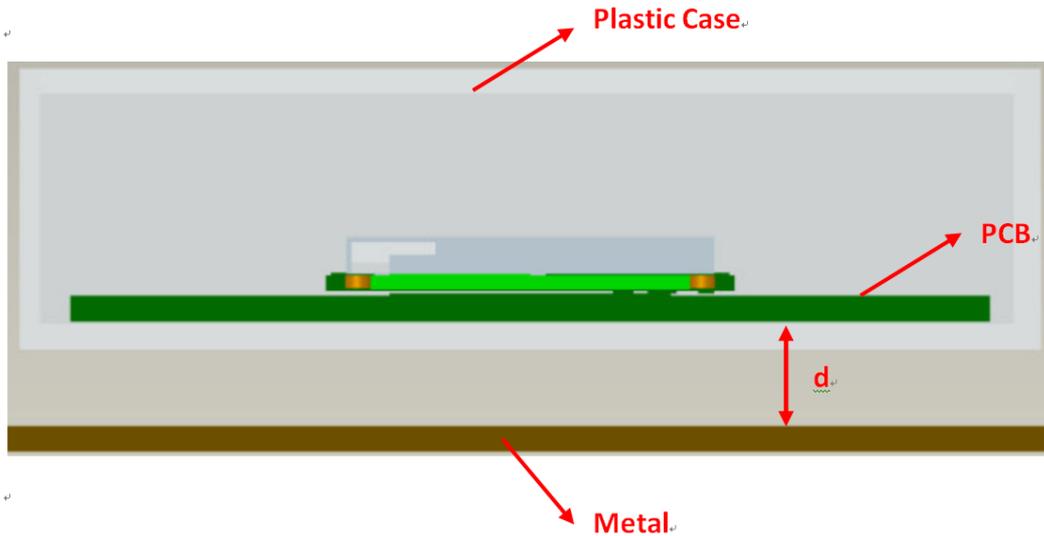
The table would give an insight on how reception performance can be negatively affected in regard to the distance between the metal and the mounted module.

Distance (mm)	Sensitivity decrease in dB
0mm	-14 dB ~-18dB
1mm	-7dB~-9dB
3mm	-6dB ~-7dB
5mm	-3dB~-5dB
7 mm	-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.
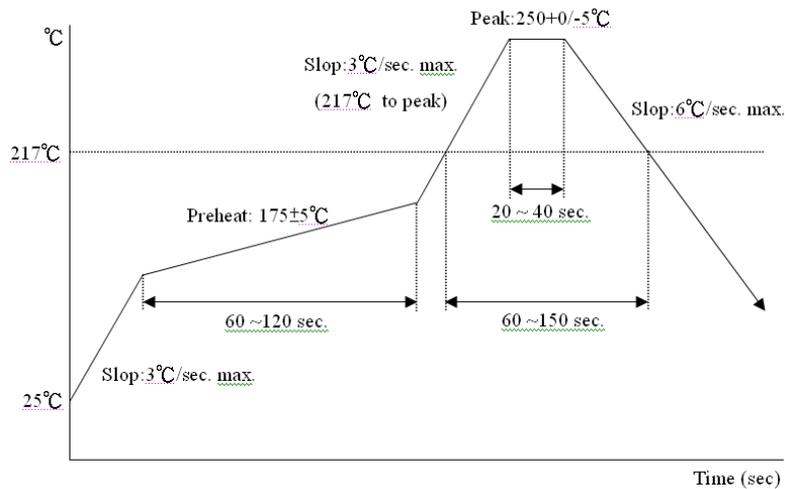
SMT Solder Mask

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 GNSS module data sheet)

Manual Soldering

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

4. Troubleshooting

How to check for the working status of GNSS 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 GNSS 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 4.3V. **(Typical: 3.3V)**. **Please double-check.**

Item 2: VBACKUP
The voltage should be kept between 2.0V~4.3V. **(Typical: 3.0V).** If 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 VBACUP pin as it is used to keep RTC time running and stored navigation data. **Please double-check**

Item 3: 3D-FIX
If all the measurements are within the specifications, please also measure 3D-FIX signal. Before 2D Fix, **the pin should output 1 second high followed by 1 second low signal.**

Notes: This is not applicable to those with customized 1PPS which has an always 1PPS signal configuration as described in "Free run 1PPS" in **section 2.1.6 1PPS**.
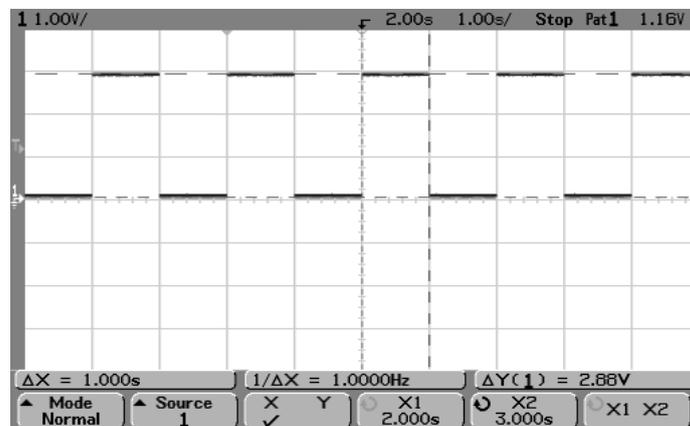
Item 4: TX0

The UART transmitter of the module, it outputs the GNSS NMEA information for application.
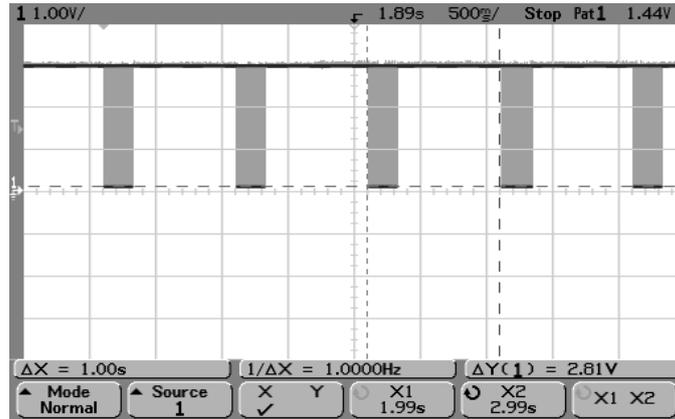
5. Contact

For help or further information, please contact us via the following methods:

GlobalTop Technology Inc.

Tainan Science-Based Industry Park Headquarter:

Address: No.16 Nan-ke 9th Road Science-based Industrial Park, Tainan 74147, Taiwan

Tel: +886-6-505-1268

Fax: +886-6-505-3381

Sales Email: sales@gtop-tech.com

Technical Support Email: support@gtop-tech.com

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.

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

Capacitor's voltage

Real time clock or memory

GC

V_0

Lower limit voltage for memory backup

V_1
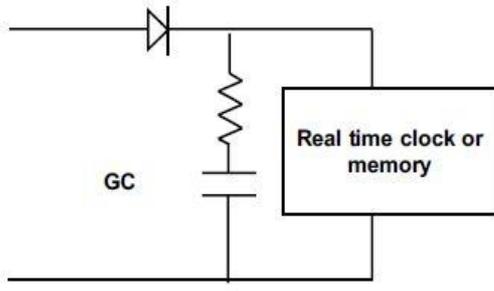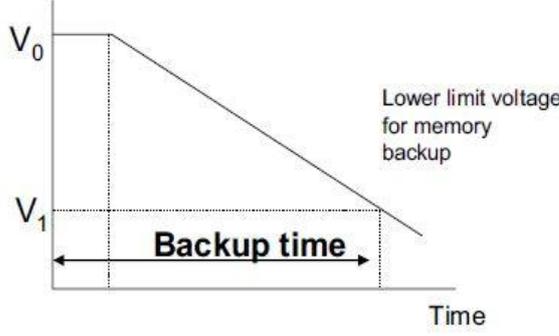
Backup time

Time

Example:

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

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

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

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
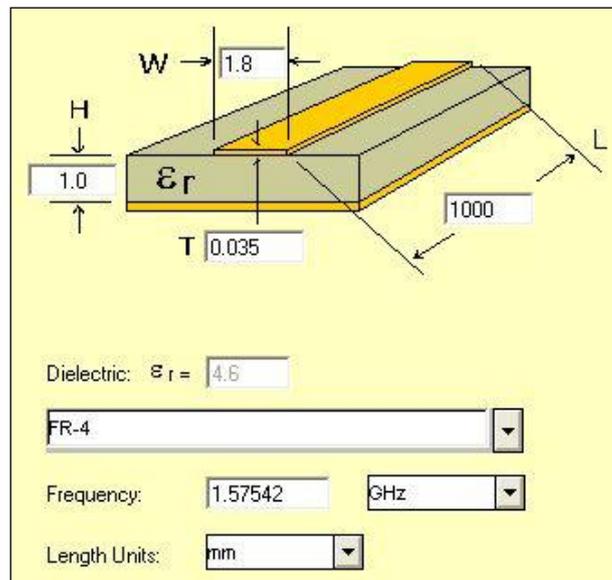
Notice:

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 GlobalTop GNSS module for communication. Majority of GlobalTop 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.
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. SP3320 IC is used here as an example.

Appendix IV: UART to USB Interface

If the GlobalTop 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 GlobalTop GNSS module is within 100ns. This applies to most of GlobalTop's MTK GNSS modules. The reference design below can be used to improve accuracy to within 80ns.
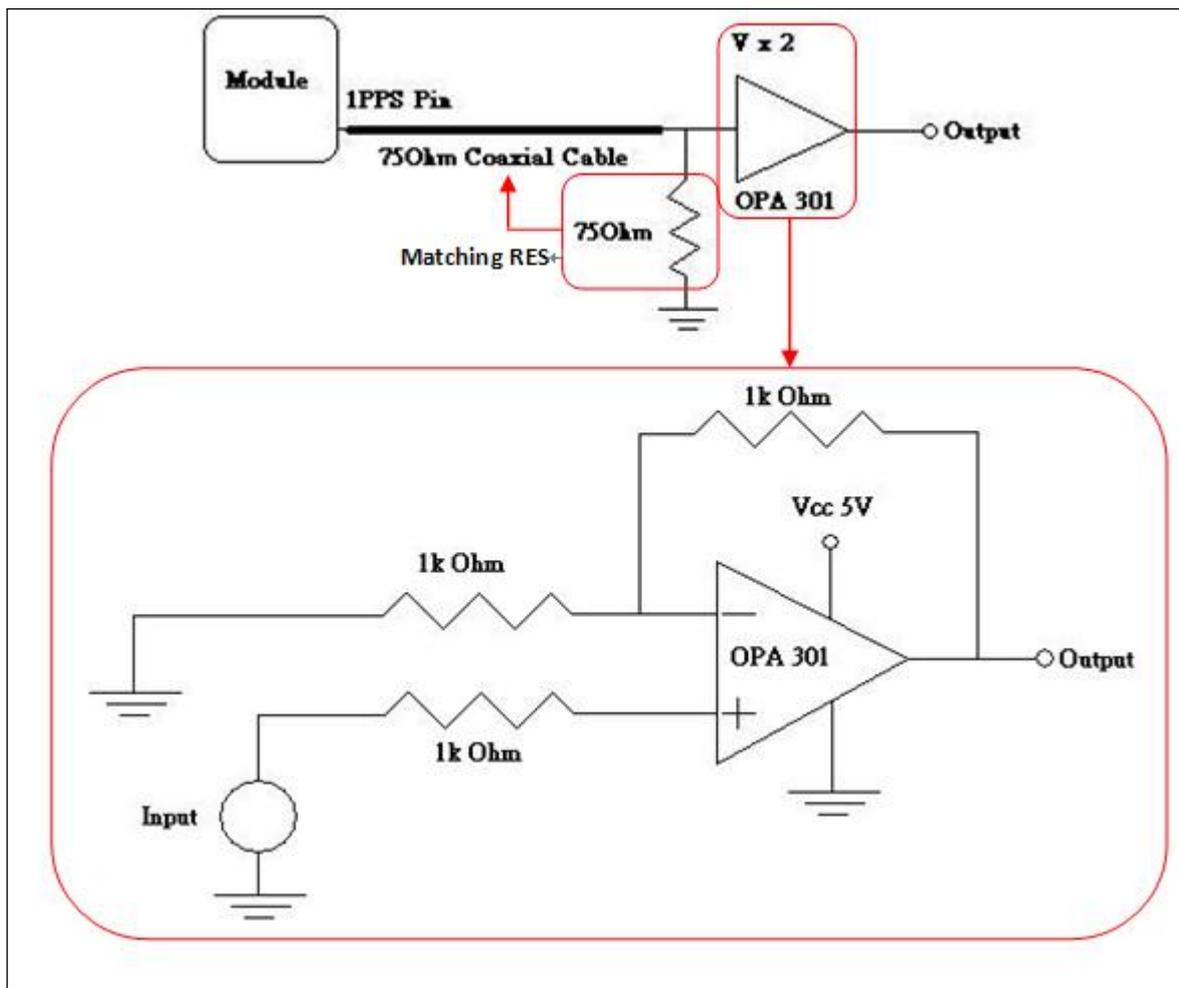
1PPS Delay Time:

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 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

DESIGN
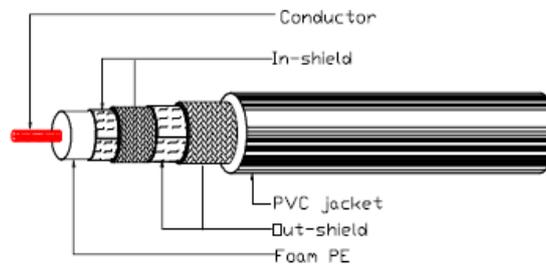
CHARACTERISTICS

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

The delay time cause by the cable:

The waveform in the coaxial cable is travelling close to the speed of light but will show significant delay as the cable becomes longer.
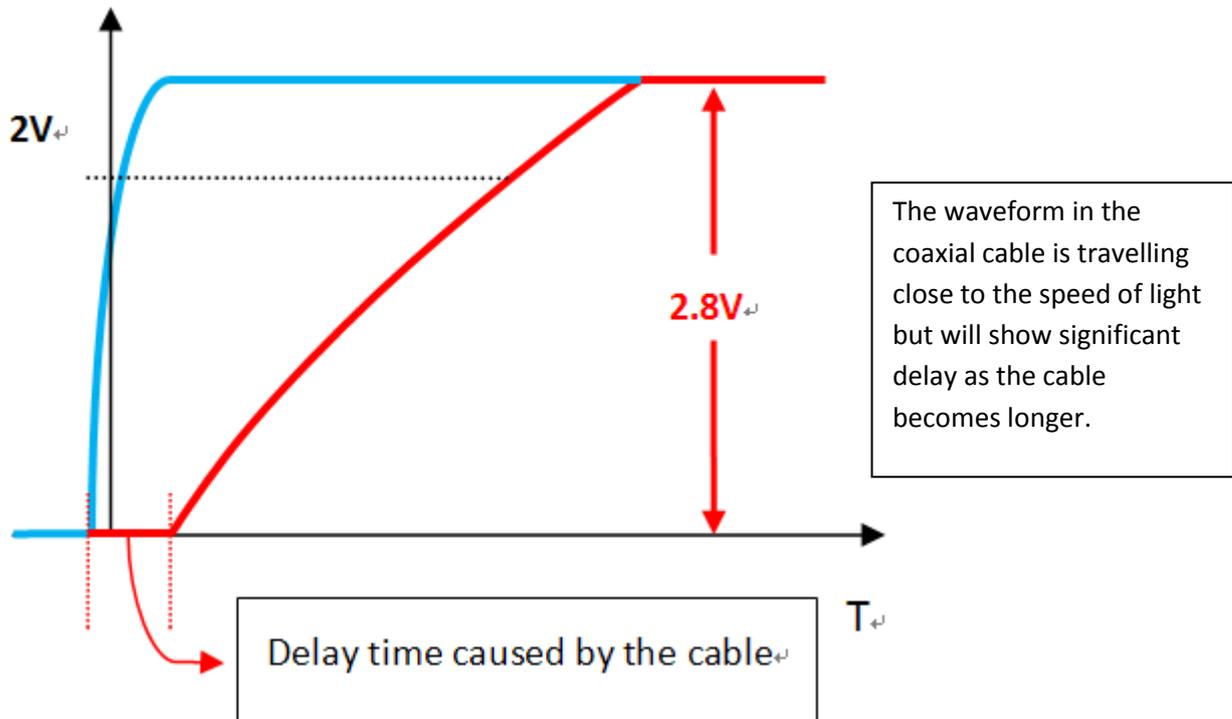
Calculating the delay time in respect to the communication cable length

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 $2 * 10^8$ (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
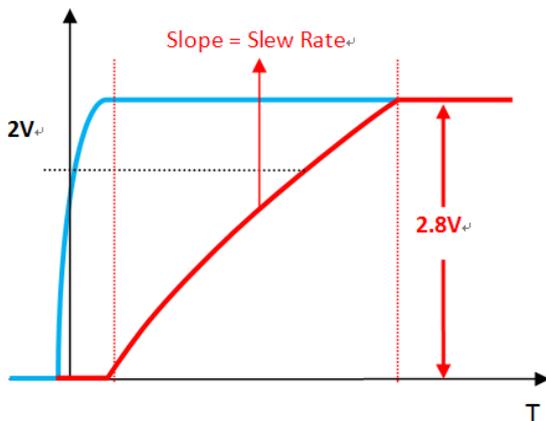
The left figure shows Slew Rate: which is the time needed for rising waveform. When the SR (V/μsec) of OPA is higher, the rising time will be shortened as well.

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°±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 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.	1. One may use 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 positional offset to the module and its patch antenna when handling the PCBA under high temperature.**
6	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. 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.	1. The blade and the patch antenna must have a distance gap greater than 0.6mm. 2. Do not use patch antenna as the leverage point when separating the panels by hand.	**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.** **2. An uneven amount of manual force applied to the separation will likely to cause positional shift in patch antenna and module.**
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 GlobalTop beforehand if you must process GPS module during the 1st pass of double-side process.
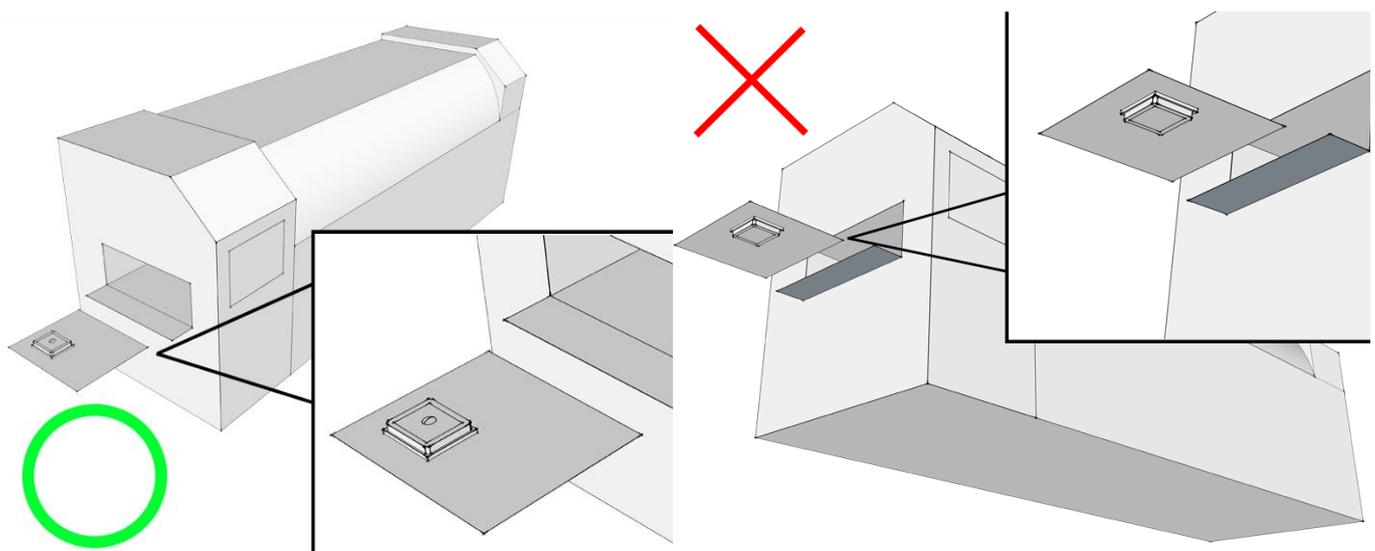

Figure 6.2: Place GPS module right-side up when running reflow-solder process, do not invert.

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.