HORNET Modules
LAYOUT RECOMMENDATIONS
and INTEGRATION

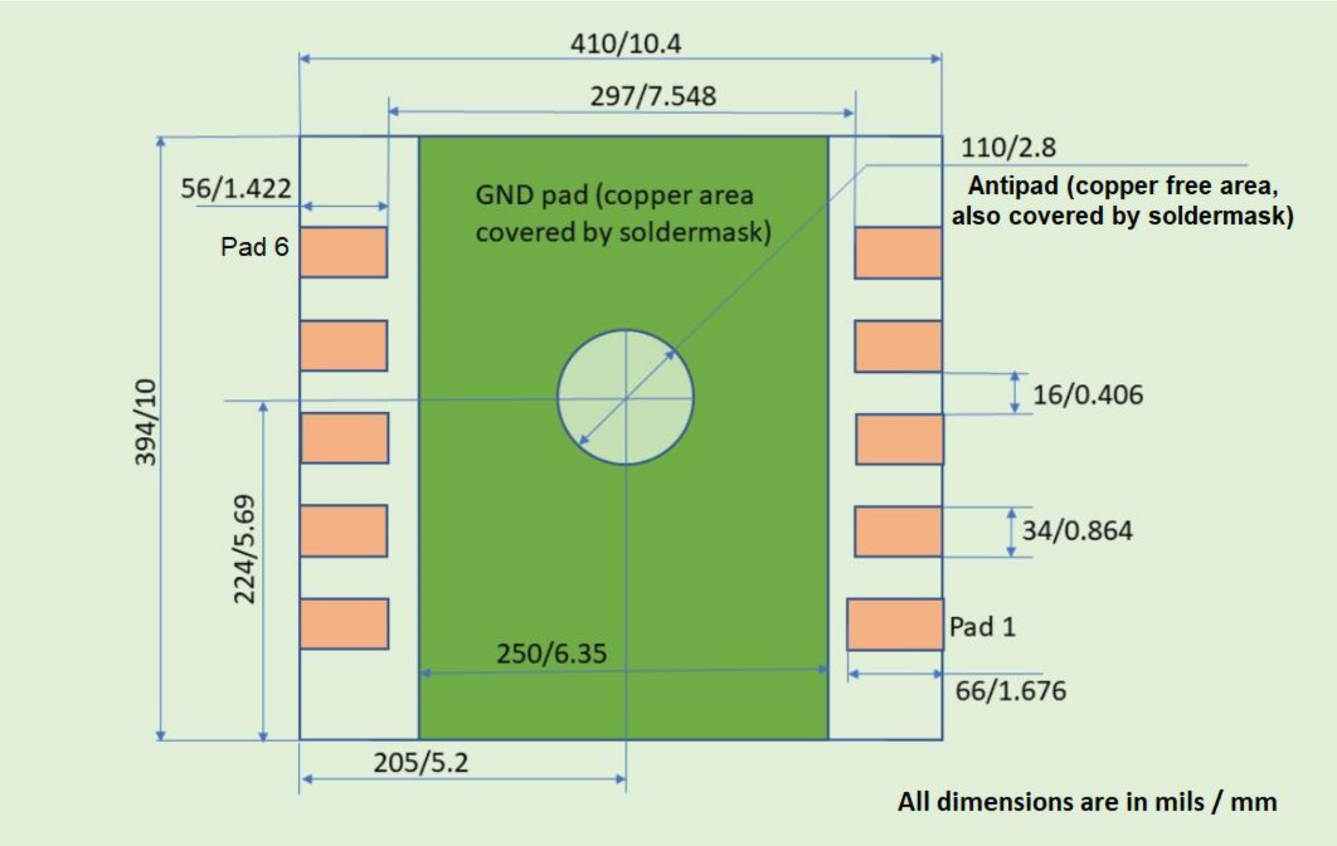
ORG1410 / ORG1411 / ORG1510 / ORG1518

Application Note

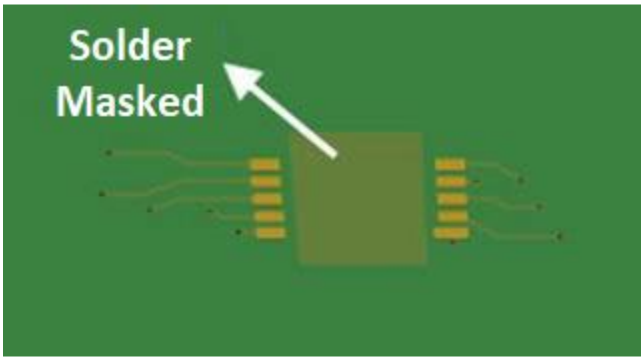
1. FOOTPRINT

ORG1410 / ORG1510

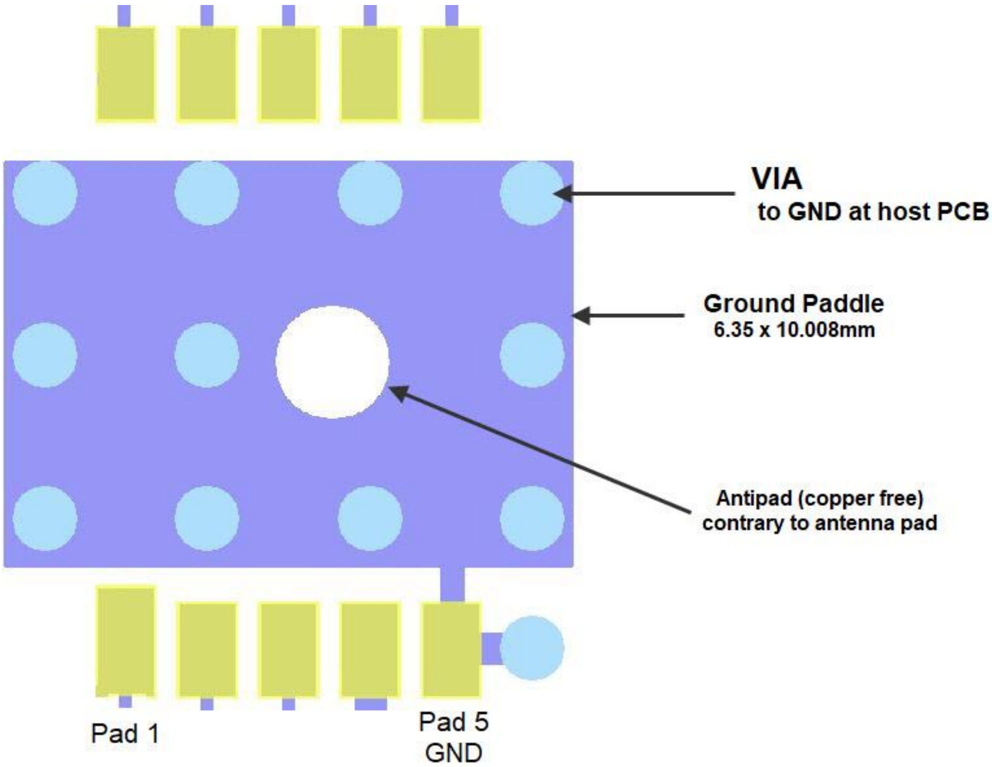
Top View – PCB side:



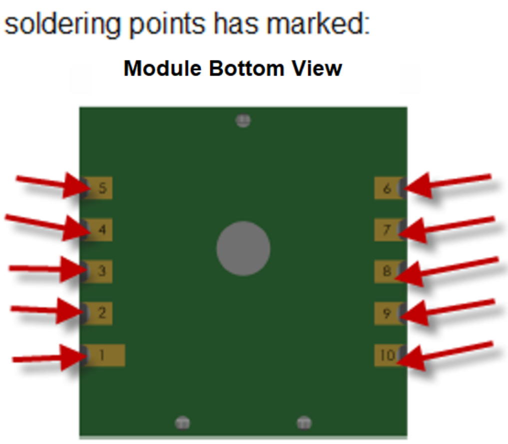
At the top layer where ORG1410/1510 placed, the Ground Paddle must be solder masked. Host PCB as shown on picture below:

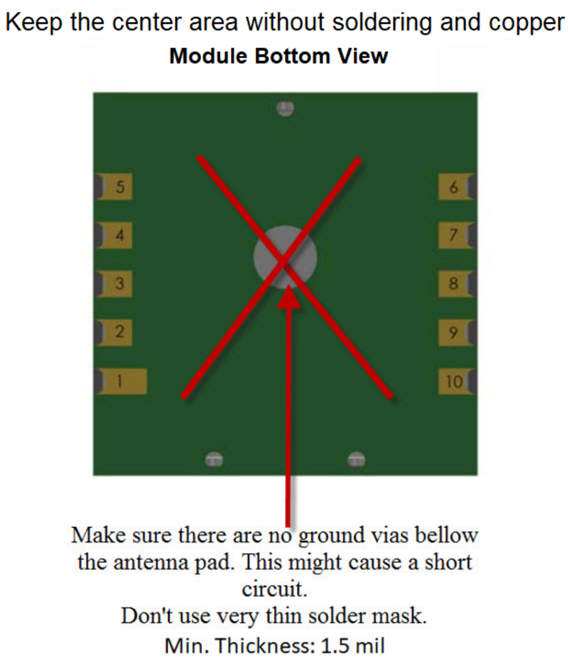


The Ground paddle should be connected to the main Ground plane on Host PCB by multiple VIAs as shown on pictures below:



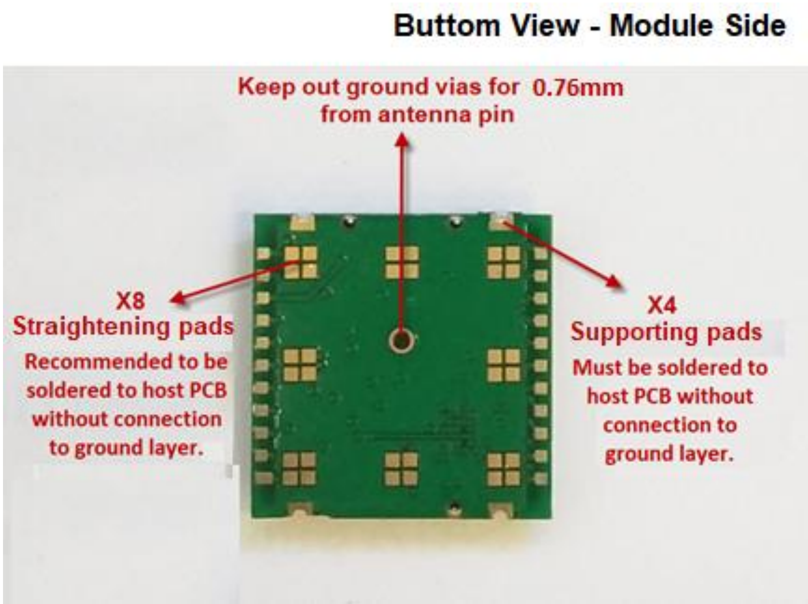
- Keep on the copper keep-out area of 3mm from each side as shown on picture above.tt
- Recommended Ground plane size below the module should be at least 13x13mm.
- Keep antipad area contrary to the antenna pad. The diameter of the Antipad should be 0.76mm bigger then the antenna pad.



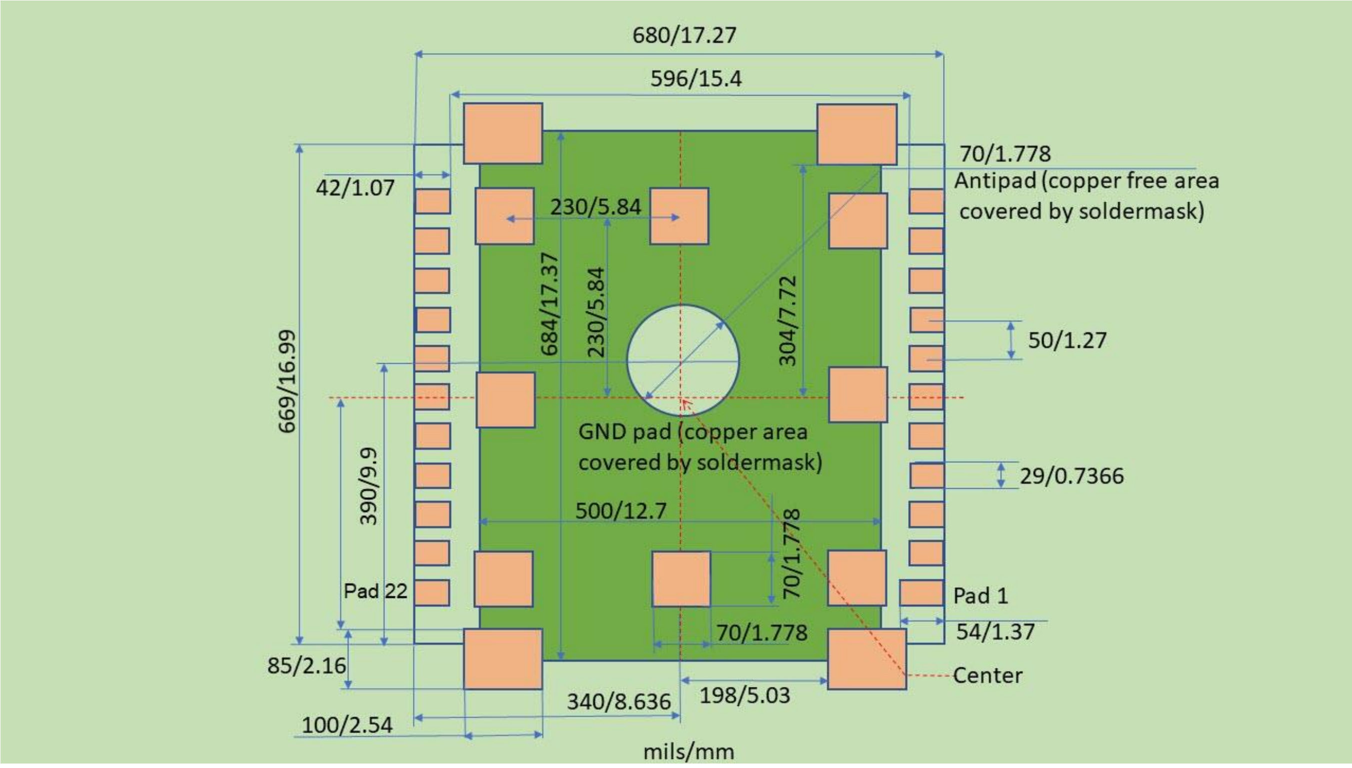


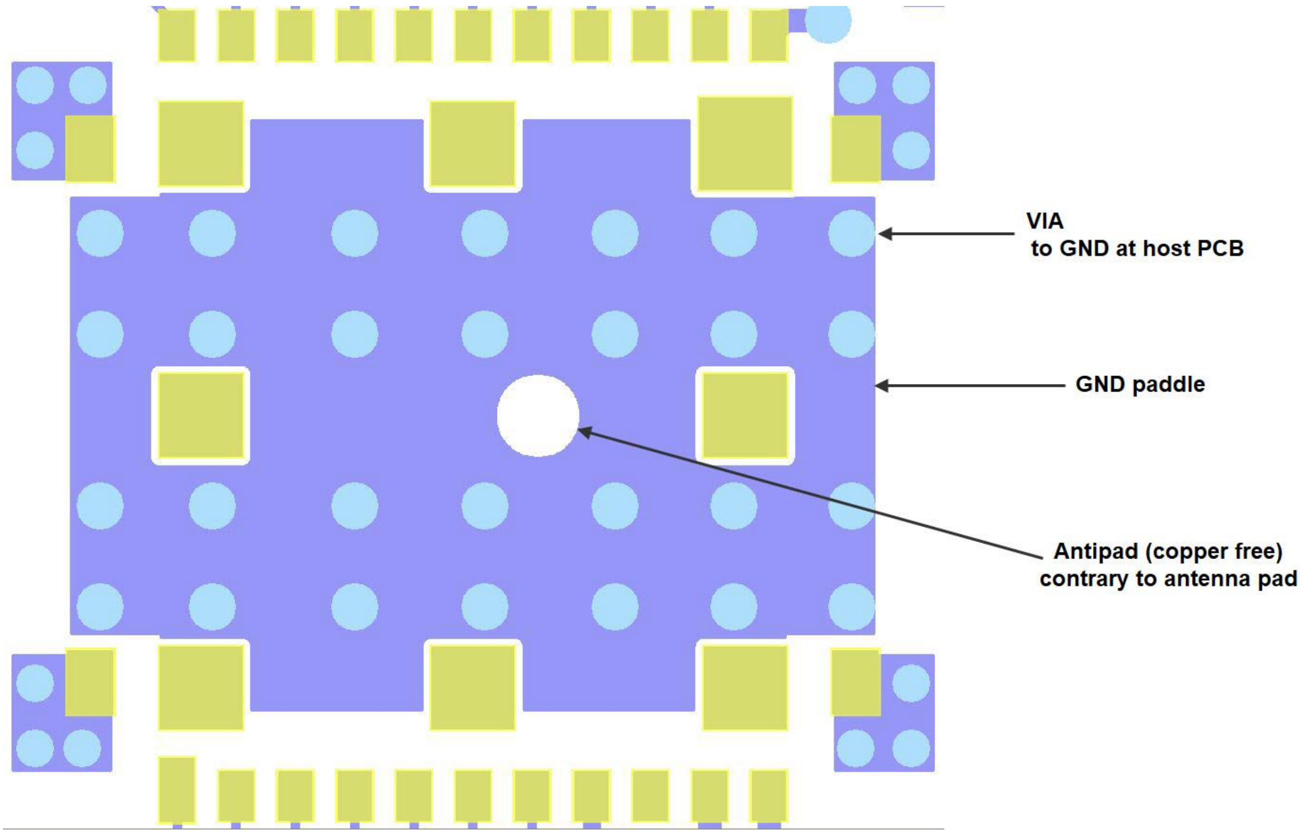
ORG1518

4 supporting pads on the surrounding of the footprint must be soldered to host PCB, without connecting to ground layer. They are floating inside the module.
8 straightening pads, each one containing 4 squares - are recommended to be also soldered to host PCB, without connecting to ground layer.
The purposed of all the 12 straightening and supporting pads is to add mechanical strength to the module.



- Ground paddle in the middle must be solder masked.
  Silk print of module's outline is highly recommended for SMT visual inspection.
- Keep on the copper pour areas and external components keep-out area (besides decoupling capacitors to pad4) of 3mm from each side of the module on the host PCB.
- Recommended Ground plane size below the module should be at least 21x21mm.
- ~~Keep antipad area contrary to the antenna pad. The diameter of the Antipad should be 0.76mm bigger then the antenna pad.~~


680/17.27
596/15.4
70/1.778
Antipad (copper free area
covered by soldermask)
42/1.07
230/5.84
684/17.37
230/5.84
304/7.72
50/1.27
669/16.99
390/9.9
GND pad (copper area
covered by soldermask)
29/0.7366
500/12.7
70/1.778
Pad 22
Pad 1
70/1.778
54/1.37
85/2.16
Center
340/8.636
198/5.03
100/2.54
mils/mm

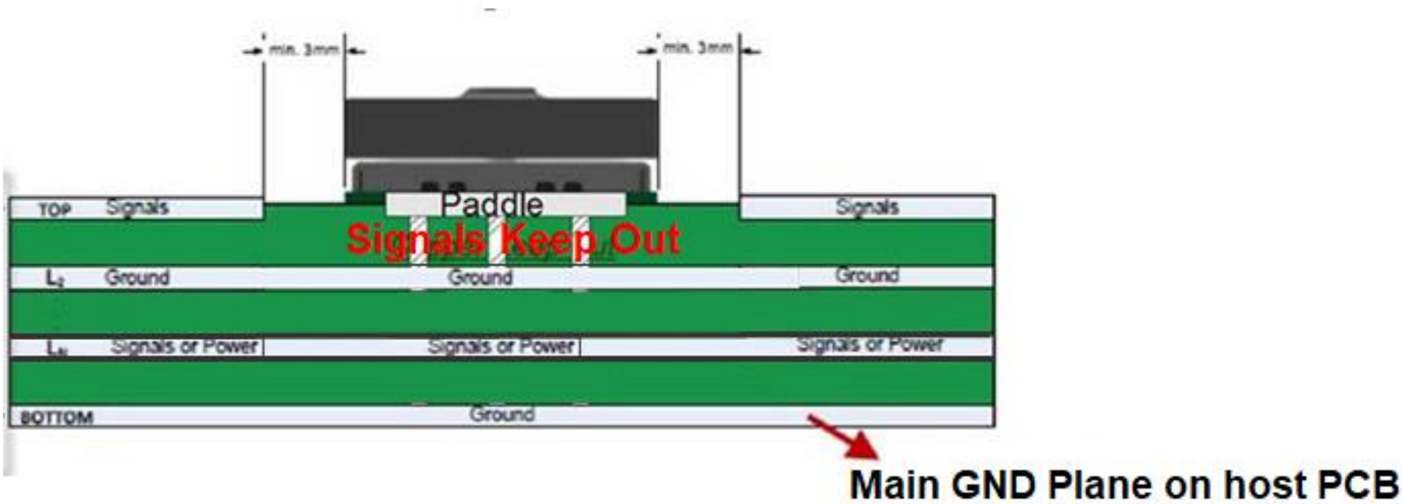

VIA
to GND at host PCB
GND paddle
Antipad (copper free)
contrary to antenna pad

2. DESIGN RESTRICTIONS

- Avoid current loops by connecting single Ground pad to main Ground.
- Route the selected Ground pad to main Ground with shortest possible trace or via.
- Avoid copper pour on the module side, keeping out the module minimum 3mm from the copper planes, metals planes or enclosures, connectors or LCD screens.
- Keep out of signal or switching power traces and vias under the module.
- Signal traces to/from the module should have minimum length.
- In case of adjacent high-speed components, like CPU or memory, high frequency components, like transmitters, clock resonators or oscillators, metal planes, like LCD or battery enclosures, please contact OriginGPS for more precise, application specific recommendations.
- Adding a dedicated LDO in order to power the module – place the LDO as close as possible to Vcc pad.
- Decoupling capacitors might be placed in the keep out area as close as possible to the VCC pad.
- Connect the GND pad with the closest GND via.

3. PCB STACK UP

The GND plane should be below the top layer where ORG141X/151X placed and connected with ground VIAs as shown on picture below:



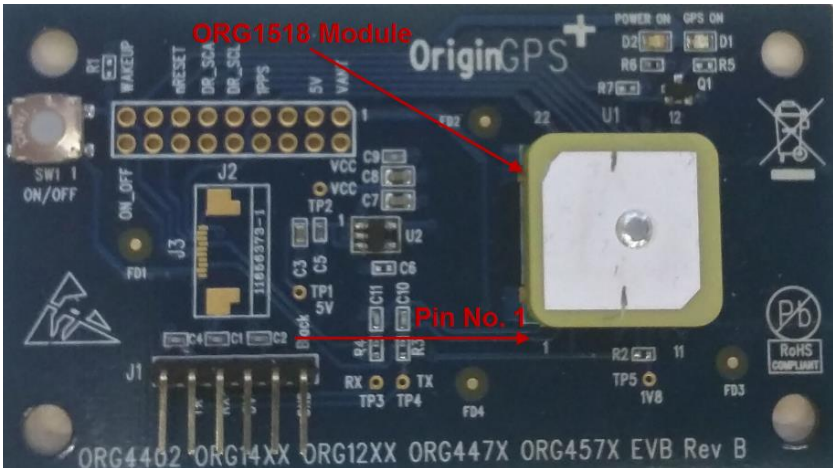
4. Module position on rectangular board

The host board serves as module's ground plane. The location and orientation of the module on the host board has a significant influence on module's performance.

Due to ground plane considerations, to achieve best GPS performance on a rectangular board it is recommended to place the hornet module in the following ~~relative~~ location and orientation relative to board:

For ORG1410 / ORG1510:

For ORG1518:



5. Module integration with adjacent common components:

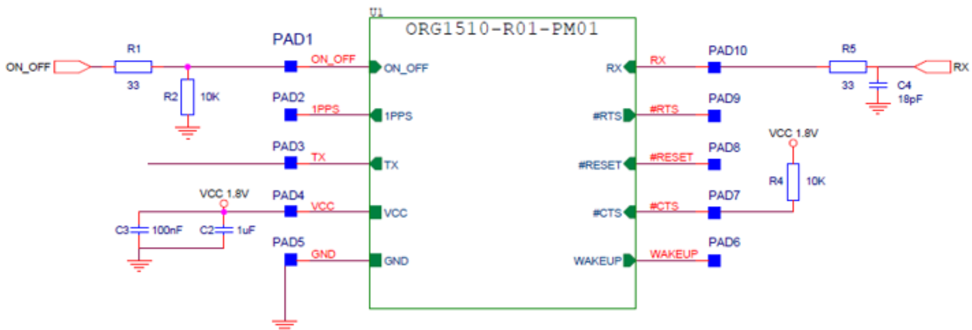
The Hornet module operates with received signal levels down to -167dBm and can be affected by high absolute levels of RF signals out of GNSS band, moderate levels of RF interference near GNSS band and by low-levels of RF noise in GNSS band.

RF interference from nearby electronic circuits or radio transmitters can contain enough energy to desensitize the module. This issue becomes more critical in tight placement, while maintaining industrial design constraints.

To prevent degraded performance, OriginGPS recommends taking in consideration these guidelines early in the design. In case of integration with adjacent high-speed components like CPU or memory, high frequency components like transmitters (GSM, Wi-Fi, Bluetooth, RFID, cellular), clock resonators or crystals, oscillators, LCD panels or CMOS image sensors it is recommended to do these actions:

1. Pre-design testing recommendation:
   Test Hornet EVB co-existence with your EVB or prototype board.
   Operate both EVBs as close as possible and measure SNR values on the Hornet module.
   This test is very important and can be a very good and early indication to possible interferences even before the circuit prototype production.
2. Keep distance from adjacent active components and keep on ground plane and multiple ground VIAs around the module.
3. Voltage supply ripple to the module should be max. 20mVp-p upon the following:
   - Up to 6 MHz for linear regulators that produce almost white noise.
   - Switching regulators that can produce harmonics up to 50MHz and these should be monitored.

   This measurement should be on pin 4 of the module on the Vcc DC line.
   For most cases decoupling capacitors of 1uF and 0.1uF should mounted as close as possible to Vcc pin4. See below C2 and C3 decoupling capacitors shown on picture below:



4. Adding EMI suppression noise components recommended by OriginGPS (after a design review).

6. Debugging procedure:

In case of degradation in module performance, one should identify the interference source and list of all fundamental frequencies of CPU, memory, transmitters and LO frequencies (GSM, Wi-Fi, Bluetooth, RFID, cellular), clocks, crystals, oscillators and voltage regulator switching frequency.

The fundamental frequencies, harmonics and intermodulation frequencies should be calculated and examine if there are in bands of GPS/GNSS or in close frequency.

7. Module enclosure choice

Module's enclosure material choice has a significant impact on module's performance. Any metallization or conductive materials as flat cables above module antenna or at close vicinity can degrade GPS signals significantly or eliminate it completely. Plastic materials with high dielectric constant and dielectric loss affect antenna frequency response and its return loss. Preferable materials are those used for RADOMs and special plastics for RF applications.