YIC 裕中企業股份有限公司
Yuechung International Corp. (YIC)

H/Q：新北市汐止區新台五路一段 79 號 20 樓之 4 (遠東世界中心 **C** 棟)　台灣汐止公司
20F-4, No.79, Sec.1, Xintai 5th Rd., Xizhi Dist., New Taipei City 22101, Taiwan
TEL: 886-2-2698-2191 (REP.)　**FAX:** 886-2-2698-2192 / 2698-2193
E-MAIL: service@yic.com.tw　**URL:** www.yic.com.tw

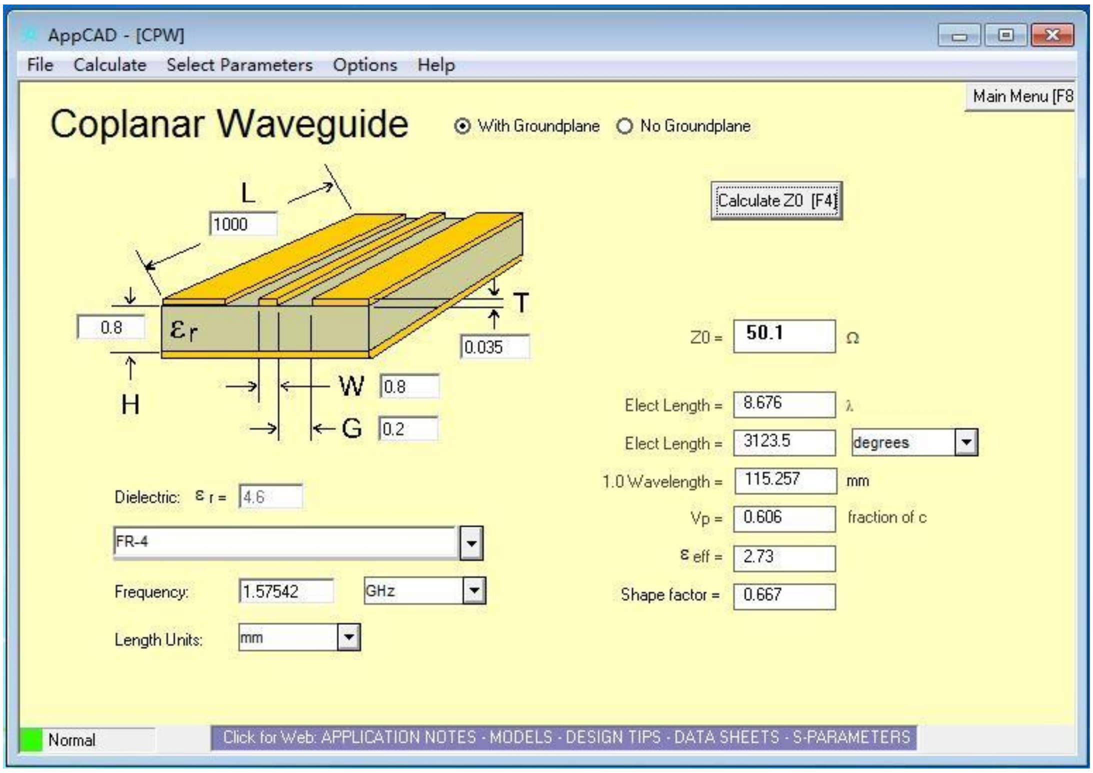
PCB Layout Guidelines for YIC GNSS Modules

1. Separate RF and digital circuits into different PCB regions.

It is necessary to maintain 50-ohm impedance throughout the entire RF signal path. Try keeping the RF signal path as short as possible, do not route the RF signal line near noisy sources such as digital signals, oscillators, switching power supplies, or other RF transmitting circuit. Do not route the RF signal under or over any other components , or other signal traces. Do not route the RF signal path on an inner layer of a multi-layer PCB to minimize signal loss. Avoid sharp bends for RF signal path. Make two 45-deg bends or a circular bend instead of a single 90-degree bend if needed.

Avoid vias with RF signal path whenever possible. Every via adds inductive impedance. Vias are acceptable for connecting the RF grounds between different layers. Each of the module's ground pins should have short trace tying immediately to the ground plane below through a via.

The bypass capacitors should be low ESR ceramic types and located directly adjacent to the pin they are for.



Antenna Design

1.1 The RF Front End part generally needs to be placed in a different mask cover (separated) from the Transceiver chip. If other PAs are used, the above requirements must also be followed.

1.2 Inductors should be placed to avoid mutual coupling as much as possible

1.3 The complementary paths of each group of I/O differential pairs can be as symmetrical as possible to ensure excellent phase balance and common mode rejection.

1.4 If necessary, it is recommended to clear the metal layers under all devices and traces to ensure smaller parasitic capacitance.

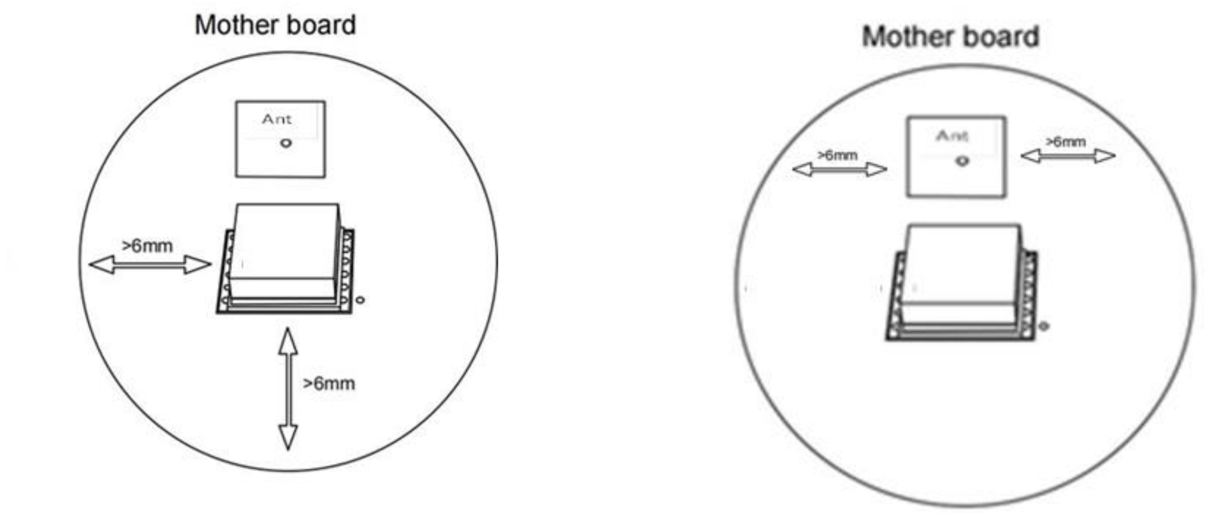
1.5 RF matching components and traces should be isolated from other circuits and traces as much as possible using GND copper.

1.6 In addition to ensuring 50 ohm impedance control, the layout of the TX and RX traces should also be as short as possible. This can minimize trace losses, which is also very important. In addition (for example), in terms of Layer distribution, the TX traces can go through the TOP layer, refer to the GND of Layer 2. The RX traces can go through Layer 4, refer to the GND plane of Layer 3 and Layer 5

2. Antenna Placement on PCB Design Guide (If the design uses patch antenna)

The radiation characteristic of antenna depends on various factors, such as the size, shape of PCB and dielectric constant of components nearby. It is recommended to follow the rules listed below.

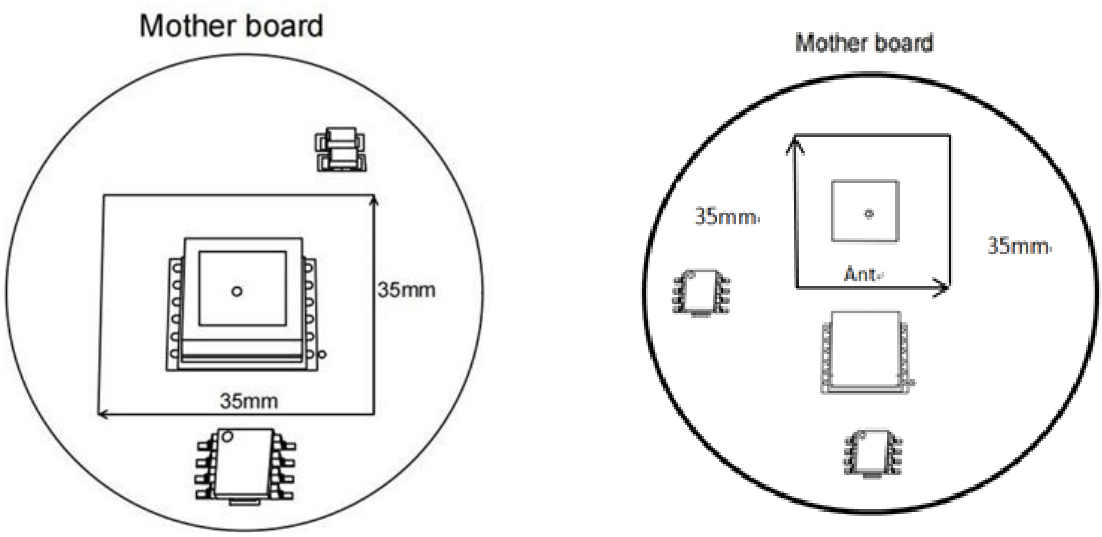
2.1 The module should be at least 6mm away from the nearest edge of the motherboard, that is, it is best to place it in the center of the motherboard. The antenna should be at least 6mm away from the edge of the motherboard.



Recommended Distance between Module and Mother Board Edges

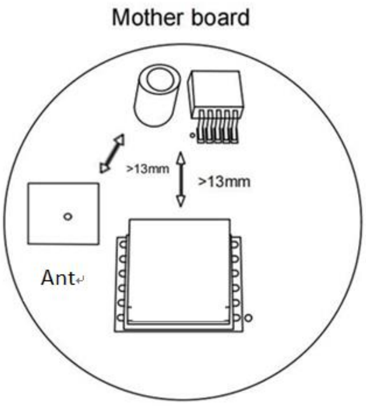
2.2 Make sure the antenna points to the sky.

2.3 The performance of embedded patch antenna depends on the actual size of the ground plane around the module. It is recommended to design a 35mm×35mm ground plane as shown below. Meanwhile, Do not put any component especially tall components in the areas whenever possible. (Interfering vias are not allowed either).



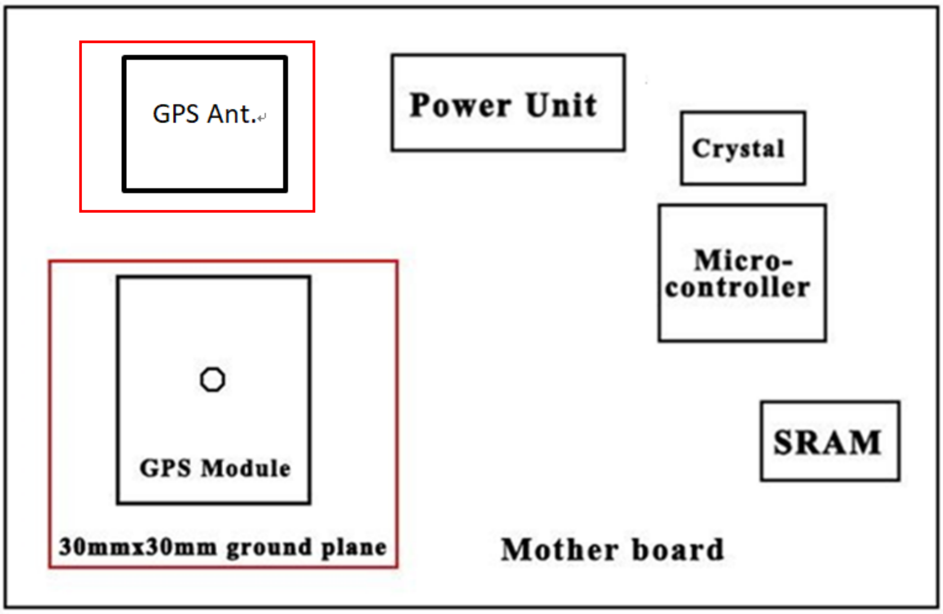
Recommended Ground Plane

2.4 Because antennas are easily affected by metal, the distance between patch antennas and modules and other tall metal parts should be at least 13mm. Otherwise, the antenna performance will be affected.



Recommended Distance between Module (Antenna) and Tall Metal Components

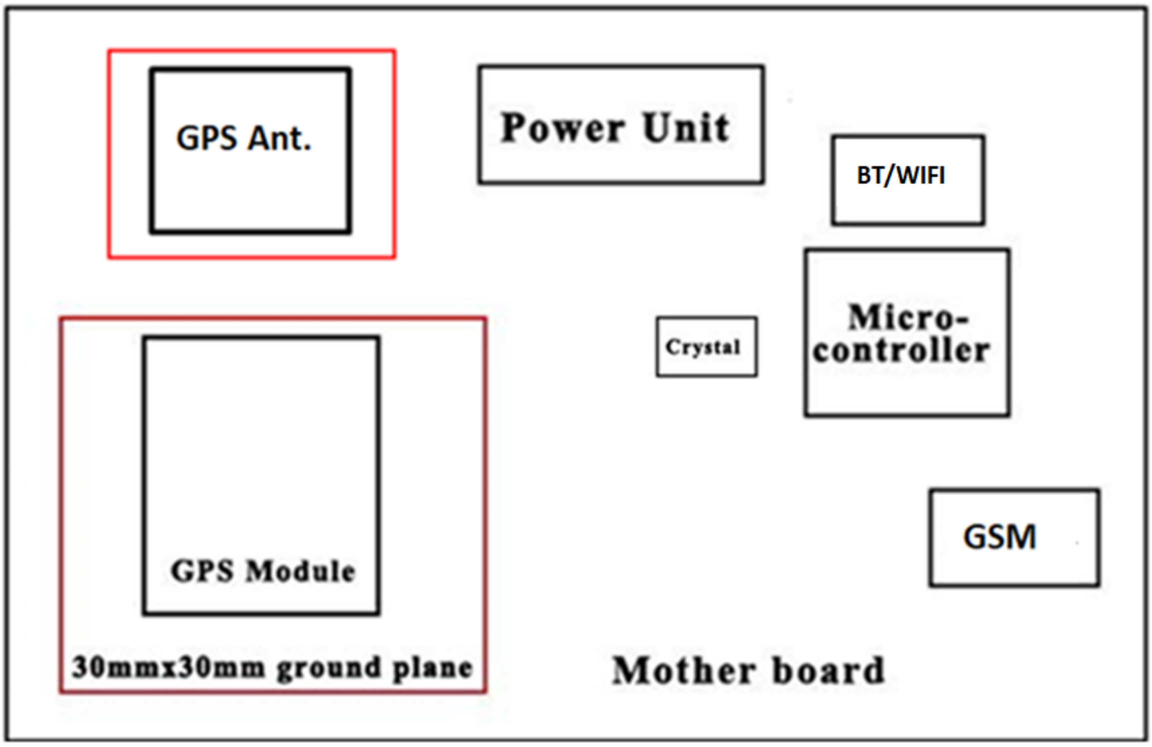
2.5 Make sure the microcontroller, crystal, and other high speed components and interfaces are placed on the opposite side of the module and GPS antenna, and keep them away from the module as far as possible, such as in diagonal position of the mother board.



Recommended Placement of GNSS Antenna and Module

2.6 Make sure interfering signals (USB, Crystal, etc.) are in inner layer and shielded by ground plane, and keep them and their vias far away from the module.

2.7 Make sure RF system such as BT/WIFI/GSM is on the opposite side of the module, and keep them away from the module & Antenna as far as possible, such as in diagonal position of the board.



Recommended Placement of GNSS Module with RF System

2.8 Keep DC/DC far away from the module

2.9 Device enclosure should be made of non-metal materials especially for those which are around antenna area. The minimum distance between antenna and enclosure is 3mm.

2.10 The RF part of GPS & GLONASS & Beidou module is sensitive to temperature. please keep them away from heat-emitting circuit.

2.11 It is recommended to reserve an integrate ground layer to isolate GPS & GLONASS & Beidou module from others.

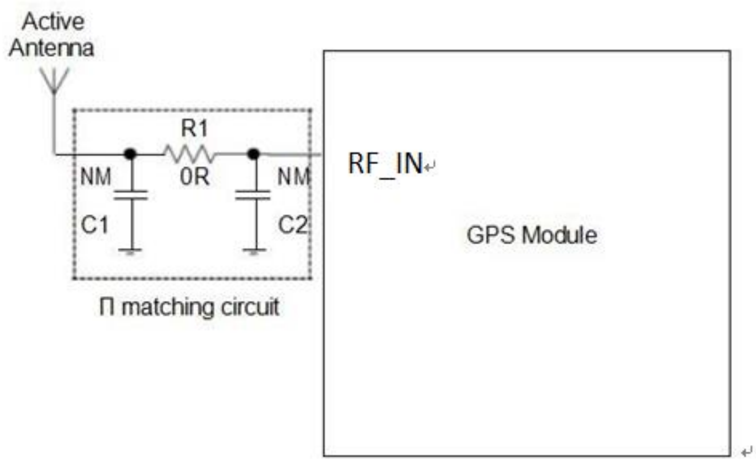
2.12 Module's enclosure material choice has a significant impact on module's performance. Any metallization or conductive materials as flat cables above module antenna or at close vicinity can degrade GPS signals significantly or eliminate it completely.

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.

3. External Active Antenna Circuit

The following figure is a typical reference design with active antenna. In this mode, DC on the RF_IN pin is powered by VCC_RF Pin and supplies power to the external active antenna.

3.1. π model match network :



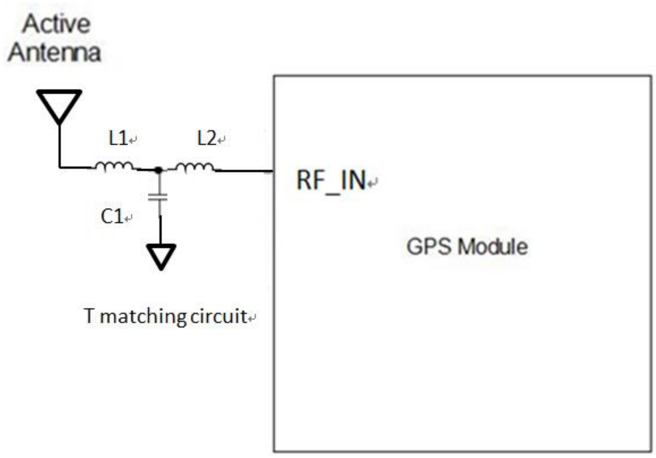
Reference Design for Active Antenna

C1, R1, C2 are reserved matching circuit for antenna impedance modification. By default, C1 and C2 are not mounted; R1 is 0 ohm. In this mode, R1 must not be capacitance, as current will stream through R1 to the active antenna. C1 and C2 must not be inductance or resistance to avoid short circuit.

The impedance of RF trace line in main PCB should be controlled as 50 Ohm, and the trace length should be kept as short as possible.

The following figure is a typical reference design with active antenna. In this mode, DC on the RF_IN pin is powered by VCC_RF Pin and supplies power to the external active antenna.

3.2. T model match network :



Reference Design for Active Antenna

L1, C1, L2 are reserved matching circuit for antenna impedance modification. By default, C1 is not mounted; L1 and L2 are 0 H.

The impedance of RF trace line in main PCB should be controlled as 50 Ohm, and the trace length should be kept as short as possible.