

Two DeLorme Drive  
Yarmouth, ME 04096  
(Phone) 800-293-2389  
(Fax) 207-846-7054  
www.delorme.com

## GPS Module Antenna and RF Design Guidelines

December 2007

You can eliminate much of the requirement for complex RF design by using a GPS receiver module, but the signal path from the antenna to the module RF-input is still a critical part of the board design that can have a big impact on the system performance. A simple RF-signal path is not difficult to design provided you follow some basic guidelines and use a software design tool for the geometry calculations. The following is intended to summarize the basic RF design guidelines and give examples for proper layout and antenna-to-module RF signal path design.

### Choosing an Antenna

As a general rule of thumb, when the antenna is located within 6" of the Module RF-input, you can use a passive antenna. When the antenna is located beyond 6" from the module input, you should use an active antenna to overcome the cable loss and maximize the signal-to-noise ratio. There are many choices of antenna configurations for GPS; the best choice is often a balance between size, gain, bandwidth, noise, and cost. The best advice is to test several antennas in the configuration of the final system to determine which provides the best overall performance.

### Antenna Configuration

- **Passive Ceramic Patch** – A ceramic patch antenna is a great low-cost choice that provides good sensitivity and good omni-directionality. Its small size allows you to mount it within the same enclosure as the module. It is also possible to mount the patch antenna on the same PCB as the module, but to reduce the possibility of digital noise, we recommend that you mount the antenna on the opposite side of the board to the module. In addition, to improve the performance of the patch, use the largest possible ground plane under the antenna.
- **External Active Antenna** – An active antenna is essentially a passive antenna with a built-in LNA and a coaxial cable to connect the antenna to the module. It may be located remotely from the module and requires antenna power, normally provided via the coaxial cable. To attach the coaxial cable to the PCB, a coaxial connector is required. The active antenna usually costs more than a passive patch and consumes more power, but performance in low signal environments is typically much better.
- **Helix** – The Helix antenna can also be passive or active. The advantage of a Helix antenna is that it is small and easy to embed and it does not detune in proximity of people, which is why it is used in mobile devices.

### Antenna Connections

The connection between the antenna and the RF-input of the module is the most critical part of the PCB design for GPS. The goal is to provide a perfectly matched 50 $\Omega$  transmission line environment between a 50 $\Omega$  antenna and the module RF-input to ensure maximum power transfer to the RF front-end. Any discontinuities due to unmatched impedances in the signal trace, excessive vias, poor layout design, or energy coupling because of poor grounding can reduce the GPS performance significantly or even render it non-functional. To reiterate, this signal path is the most critical part of the GPS system design. Follow the guidelines below to maximize performance.

### PCB Layout – 50 $\Omega$ Connection

To move the RF signals to the modules, we suggest a 50 $\Omega$  grounded co-planar waveguide. In general form, this consists of the RF input signal with RF ground on either side and RF ground below. Theoretically, for a given RF signal trace width, the surrounding RF grounds should be at least twice as wide. In addition, the gap between the RF grounds and the RF signal is important. The coplanar waveguide is the lowest loss transmission line configuration for connecting the antenna to the module. You can use other methods such as microstrip or stripline, but we believe the coplanar waveguide is the most efficient.

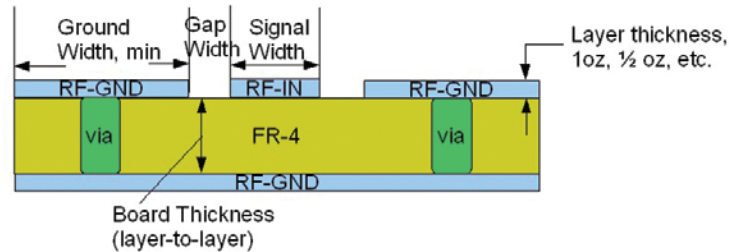


Figure 1) Cross section of coplanar waveguide

*NOTE: While we can provide guidance and an example for your reference on how to construct and layout a coplanar waveguide, we strongly recommend using transmission-line design software to ensure the design is correct for your specific PCB type and system design. We suggest you use a freeware program, such as AppCAD.*

For a two-layer board design, typically 0.062" thick board made with FR-4 material (Dielectric = 4.6) and 1 oz copper (1.2-1.4 mils thick), the RF-input should be 30-32mils wide, the gap to the adjacent grounds should be 6 mils, and each of the RF grounds should be at least twice the width of the input signal (60-64mils).

If the board is thinner, such as a 0.031" thickness FR-4, then the RF-input width can be reduced to 25-26 mils and the RF grounds reduced to 50-52mils wide, still with a 6mil gap between.

If 1/2 oz copper is used for the RF-input and surrounding grounds, add 3-5 mils in width.

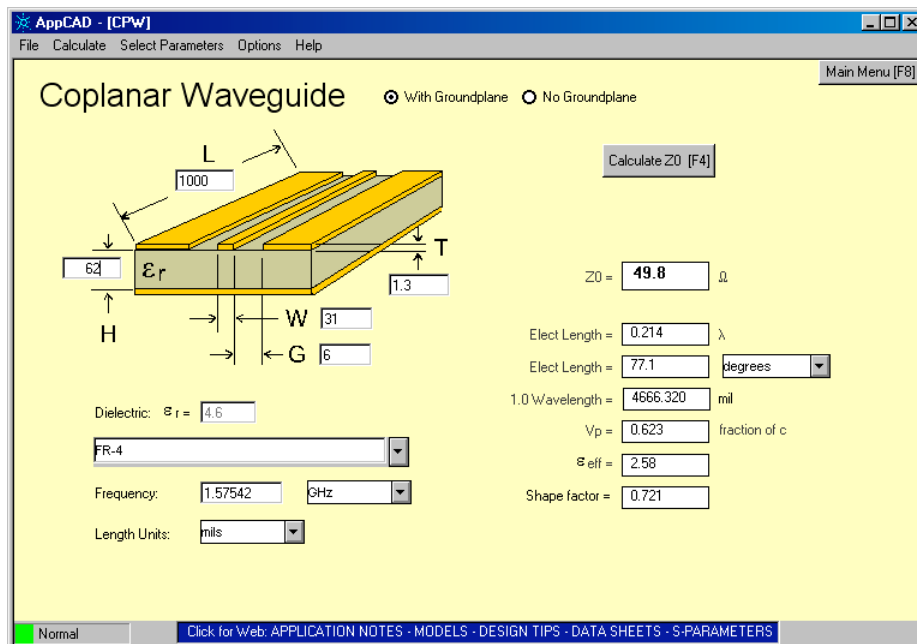


Figure 2) Screenshot of AppCAD freeware with example geometries.

## General RF Layout Guidelines:

- Maintain a characteristic impedance of  $50\Omega$  throughout the entire RF signal path. Keep the RF signal path as short as possible, and do not route near noise sources such as digital signals, oscillators, switching power supplies, or other RF transmitters, such as Bluetooth.
- Do not route the RF signal under or over any other components (including the module) or other signal paths.
- Avoid sharp bends. If a bend is necessary make two  $45^\circ$  bends or a radius bend instead of a single  $90^\circ$  bend.
- Avoid vias whenever possible. Every via adds inductive impedance to the signal path. Vias are acceptable for coupling the RF grounds between layers.
- Do not route the RF signal path on an inner layer of a multi-layer PCB (if possible) to minimize signal loss and minimize the need for interlayer vias.

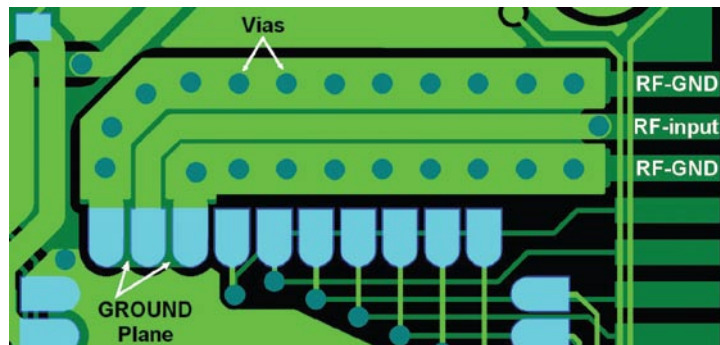


Figure 3) Sample RF input layout with bend.

## Guidelines for Isolating Digital and RF Grounds

- Give careful consideration to the floor plan of the design. The ideal floor plan will partition digital and RF circuitry into clearly different regions.
- Keep RF and digital signal paths as short as possible.
- Do not route digital signals long distance across the board as they may pick up or couple noise into/from the RF circuitry.
- Locate bypass capacitors as close as possible to the supply pin they are bypassing.