GSA.60

GNSS Antenna Circuit Design on Mainboard

Service name:

GSA.60 GNSS Antenna Circuit Design on Mainboard

Deliverables

Report, Schematics, Bill of Materials

Duration:

2 Weeks

Service Delivery Objectives:

A. Active receive front-end circuitry design

B. Low-Noise Amplifi er and Band-pass Filters

C. Integration onto customer main board

D. Transmission line layout

E. Circuit Test Report

What is the problem or concern we are addressing?

Your product has a GNSS receiver and a through-hole or SMT Taoglas antenna. You also want an active frontend for the GNSS receiver, similar to those available on Taoglas' active patch modules. But your company doesn't have the expertise to design, implement, and test such a circuit. Let Taoglas design and test this circuitry for you.

There are many factors to consider when it comes to designing front-end circuitry, such as size, cost, power consumption, power supplies, other radios in the system, and more. Taoglas has the expertise to take these factors into account, and design an optimum solution.

The Process

Taoglas will review the applicable areas in your design with you to identify the design constraints, such as PCB substrate material type, PCB stack-up, PCB size, other radios, and more. Taoglas engineering will then design an appropriate front-end circuit for you that meets your needs, and work with you to make sure it's implemented correctly in your schematic and PCB design before fabricating boards. After prototypes have been fabricated, Taoglas will take three samples for design verification testing. The gain, impedance match, filter rejection, and noise figure of the sample front-ends will be measured to verify performance meets your requirements. A test report will be provided to you.

What does Taoglas need?

PDF format copies of your schematics for all the PCB into which the front-end will be integrated. The best format is native Altium files, as these can be edited directly.

Design files for all the PCB into which the front-end will be integrated. The best formats are native Altium files or gerber files. Other formats for which viewers are available may be provided as well. Please include a document defining the PCB stackup, layer thicknesses, materials and finishes for the PCB. A spreadsheet of the bill of material for this PCB.

Deliverables

- Taoglas will provide a schematic and Bill of Material for the front-end design, which you can import or implement in your schematic capture software. Taoglas will then provide a transmission line design which your PCB layout resource can implement in your native electrical CAD design tool.
- Taoglas will review the PCB design prior to fabrication to verify correctness. An email or report will be created with findings from this review.
- After prototypes have been tested, a circuit test report will be provided detailing the performance of the circuit implemented on your PCB, and any changes required to optimize performance.