ISA.12

PCB & Gerber Design RF Review

Outcomes and Deliverables

- PCB Transmission Line Review.
- Gerber File Review.
- Report and/or Interactive Technical Support.

Duration

2 days (this is a typical estimated duration – actual duration on quote may differ).

What We Need

- Mechanical drawings.
- PCB files and circuit schematics – preferably native Altium files.

What is the problem or concern we are addressing?

Your product has 1 or more radio modules in it that require antennas. The antennas, or appropriate RF connectors, have been integrated onto the PCB. This now needs to be connected to the module input with a transmission line of the appropriate impedance.

Designing and implementing an RF transmission line can be a daunting task for those new to RF. Selecting an appropriate impedance range, designing for stack-up and production tolerances, minimizing parasitic losses, and even selecting an appropriate transmission line type are all parameters that need to be understood and controlled. Taoglas has the expertise to help you implement a PCB transmission line the right way.

The Process

Part 1 – Gerber Review

Taoglas will review the applicable areas in your PCB design with you to identify the transmission line design constraints, such as PCB substrate material type and PCB stack-up.

Taoglas will evaluate the design and integration against known design rules. We will make recommendations to ensure appropriate transmission lines, that meet your needs, are implemented correctly onto your PCB. These recommendations could include shorter RF paths, ground pours, antenna positioning and trace dimension modifications.

This review and design only applies to 50-ohm transmission lines for antennas and antenna connectors; it does not cover other controlled-impedance traces such as memory data buses, USB data lines, etc.

What does Taoglas need?

We require any documents you have relating to the PCB of the device. These documents should define the PCB stack-up, later thicknesses, materials and finishes for the PCB. A bill of materials for each PCB is also recommended. Ideally these files should be native Altium files.

Circuit schematics of all the PCBs in your device are also required. This is to better understand the RF paths in your design. Once again, these files should ideally be native Altium files.

Part 2 – Next Steps

Taoglas offers a number of services which would typically follow on from this service. These services are intended to optimize the RF performance and maximize likelihood of certification for your design.

These services include:

- **CSA.10:** Antenna Feasibility Study
- **CSA.20:** Passive Antenna Testing, Matching and Fine Tuning

Visit Taoglas Website or contact Taoglas sales for further information.

Please note - devices, systems and equipment falling within the scope of Annex I of the EU Dual Use Regulation 821/2021 are not eligible for this service. For queries, please consult your legal department or contact exportcompliance@taoglas.com.