

# ISA.12

## Transmission Line Design and Gerber Design Review



Service name:

### ISA.12 Transmission Line Design and Gerber Design Review

#### Deliverables:

PCB Transmission Line Design

#### Duration:

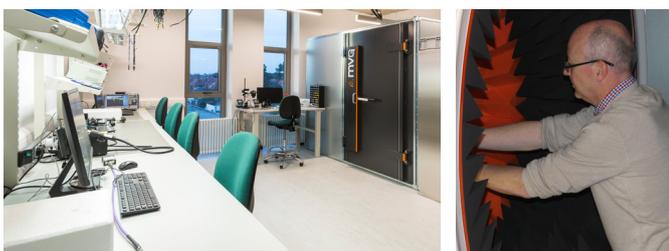
2 days

#### Items:

- A PCB Transmission Line Design
- B Gerber File Review

#### What is the problem or concern we are addressing?

Your product has 1 or more radios in it that have antennas. You've implemented the antenna on the PCB or you need to place an RF connector on the PCB. You now need to connect the radio to the antenna or RF connector 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 or wireless. 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

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 engineering will then design an appropriate transmission line for you that meets your needs, and work with you to make sure it's implemented correctly in your PCB design before fabricating boards.

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. After the transmission line design has been incorporated into the PCB layout, Taoglas will review the transmission line and Taoglas antenna implementation.

#### What does Taoglas need?

PDF format copies of your schematics for all PCBs which need transmission line assistance. The best format is native Altium files. Design files for all PCBs which need transmission line assistance. The best format is native Altium files or gerber files. Please include a document defining the PCB stackup, layer thicknesses, materials and finishes for the PCB. A spreadsheet of the bill of material for each PCB in the design.

#### Deliverables

Taoglas will compile a report or email with dimensioned transmission line designs for all the applicable connections, which your PCB layout resource can implement in your native electrical CAD design tool. After the design has been incorporated into the PCB layout, the report will be updated with any further comments from Taoglas Engineering.

Taoglas makes no warranties based on the accuracy or completeness of the contents of this document and reserves the right to make changes to specifications and product descriptions at any time without notice. Taoglas reserves all rights to this document and the information contained herein. Reproduction, use or disclosure to third parties without express permission is strictly prohibited. Copyright © Taoglas Ltd.