Installation Manual

Track Group Inc.

Wireless Module

Model Number: OTD41LE910B1

FCC ID and IC Number for this product are as follows.

FCC ID: TPO-LE910B1
IC: 6512A-LE910B1

Antenna Requirements

The antenna connection and board layout design are the most important aspect in the full product design as they strongly affect the product overall performances, hence read carefully and follow the requirements and the guidelines for a proper design.

The antenna and antenna transmission line on PCB shall fulfil the following requirements:

Item	Value
Frequency range	Depending by frequency band(s) provided by the network operator, the customer shall use the most suitable antenna for that/those band(s)
Bandwidth	140 MHz in LTE/WCDMA Band 2 445 MHz in LTE Band 4 70 MHz in LTE/WCDMA Band 5 80 MHz in LTE Band 8 / GSM900 47 MHz in LTE Band 12 41 MHz in LTE Band 13 42 MHz in LTE band 17
Impedance	50 ohm
Input power	> 24dBm Average power
VSWR absolute max	≤ 10:1 (limit to avoid permanent damage)
VSWR recommended	≤ 2:1 (limit to fulfill all regulatory requirements)

PCB design guidelines

Since there is no antenna connector on the module, the antenna must be connected to the module's antenna pad (K1) by means of a transmission line implemented on the PCB.

This transmission line shall fulfil the following requirements:

Item	Value
Characteristic Impedance	50 ohm (+-10%)
Max Attenuation	0,3 dB
Coupling	Coupling with other signals shall be avoided
Ground Plane	Cold End (Ground Plane) of antenna shall be equipotential to the module ground pins

The transmission line should be designed according to the following guidelines:
• Make sure that the transmission line's characteristic impedance is 50ohm;

• Keep line on the PCB as short as possible since the antenna line loss shall be less than about 0.3 dB;

• Line geometry should have uniform characteristics, constant cross section, avoid meanders and abrupt curves;

• Any kind of suitable geometry / structure (Microstrip, Stripline, Coplanar, Grounded Coplanar Waveguide...) can be used for implementing the printed transmission line afferent the antenna;

• If a Ground plane is required in line geometry, that plane has to be continuous and sufficiently extended, so the geometry can be as similar as possible to the related canonical model;

• Keep, if possible, at least one layer of the PCB used only for the Ground plane; If possible, use this layer as reference Ground plane for the transmission line;

• It is wise to surround (on both sides) the PCB transmission line with Ground, avoid having other signal tracks facing directly the antenna line track.

• Avoid crossing any un-shielded transmission line footprint with other signal tracks on different layers;

• The ground surrounding the antenna line on PCB has to be strictly connected to the main Ground Plane by means of via holes (once per 2mm at least), placed close to the ground edges facing line track;

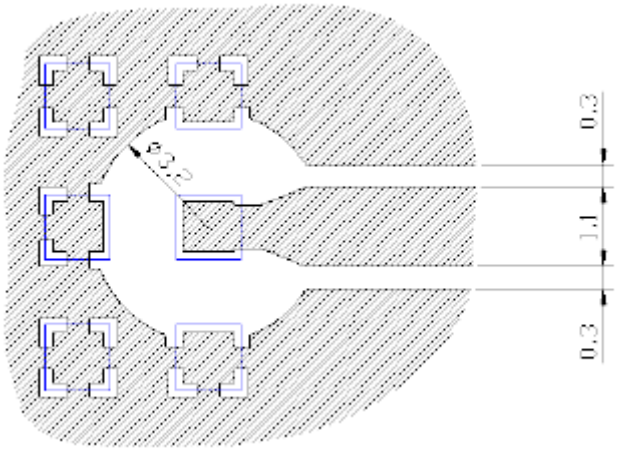
• Place EM noisy devices as far as possible from the module antenna line;

• Keep the antenna line far away from the module power supply lines;

• If EM noisy devices (such as fast switching ICs, LCD and so on) are present on the PCB hosting the module, take care of the shielding of the antenna line by burying it in an inner layer of PCB and surround it with Ground planes, or shield it with a metal frame cover.

• If EM noisy devices are not present around the line, the use of geometries like Microstrip or Grounded Coplanar Waveguide has to be preferred, since they typically ensure less attenuation if compared to a Stripline having same length;

The following image is showing the suggested layout for the Antenna pad connection:



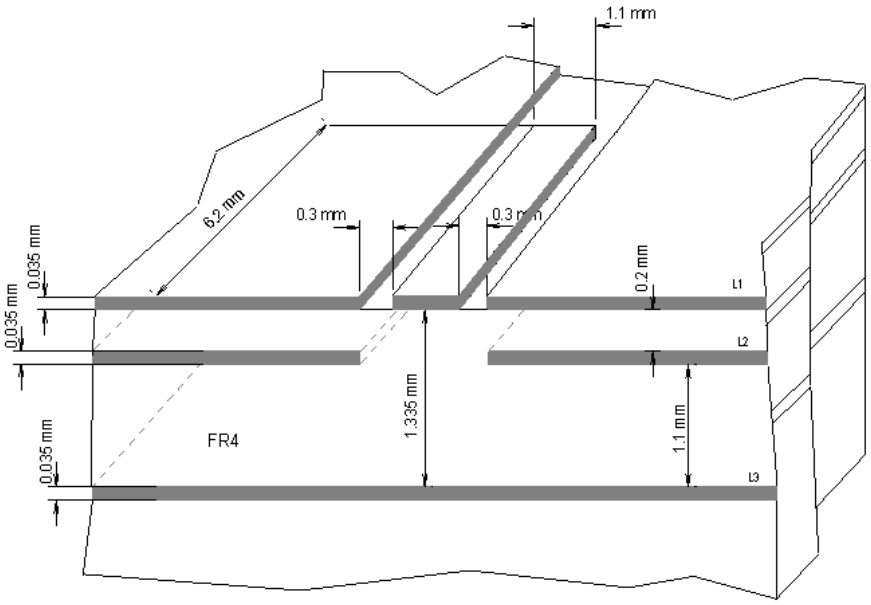
PCB Guidelines in case of FCC Certification

In the case FCC certification is required for an application using the module, according to FCC KDB 996369 for modular approval requirements, the transmission line has to be similar to that implemented on the module interface board and described in the following chapter.

Transmission line design

During the design of the module interface board, the placement of components has been chosen properly, in order to keep the line length as short as possible, thus leading to lowest power losses possible. A Grounded Coplanar Waveguide (G-CPW) line has been chosen, since this kind of

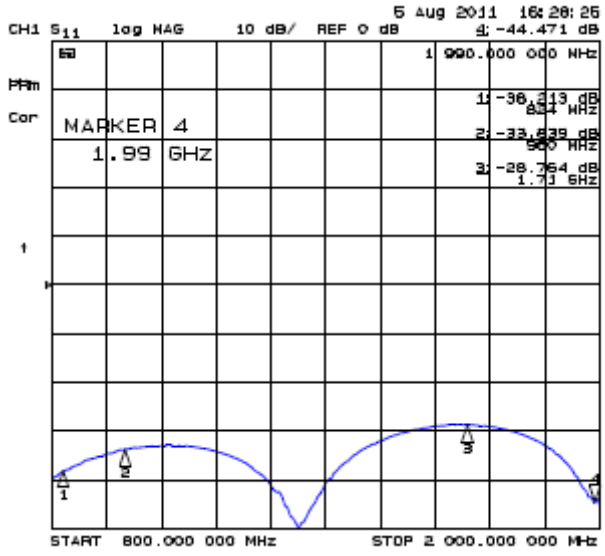
transmission line ensures good impedance control and can be implemented in an outer PCB layer as needed in this case. A SMA female connector has been used to feed the line.
The interface board is realized on a FR4, 4-layers PCB. Substrate material is characterized by relative permittivity εr = 4.6 ± 0.4 @ 1 GHz, TanD= 0.019 ÷ 0.026 @ 1 GHz.
A characteristic impedance of nearly 50 Ω is achieved using trace width = 1.1 mm, clearance from coplanar ground plane = 0.3 mm each side. The line uses reference ground plane on layer 3, while copper is removed from layer 2 underneath the line. Height of trace above ground plane is 1.335 mm. Calculated characteristic impedance is 51.6 Ω, estimated line loss is less than 0.1 dB. The line geometry is shown below:



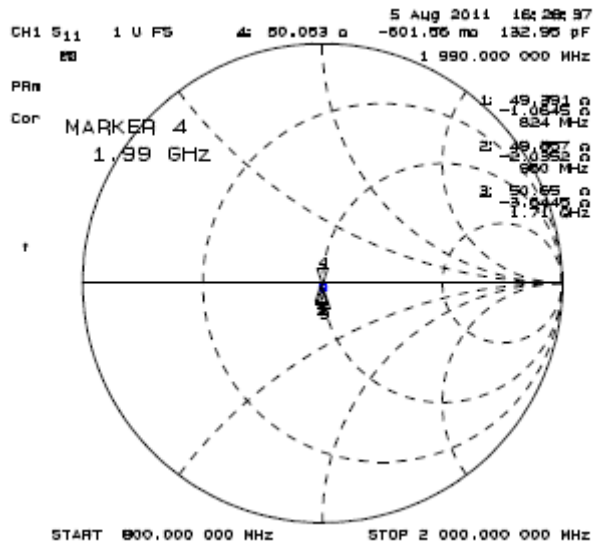
Transmission Line Measurements

An HP8753E VNA (Full-2-port calibration) has been used in this measurement session.
A calibrated coaxial cable has been soldered at the pad corresponding to RF output; a SMA connector has been soldered to the board in order to characterize the losses of the transmission line including the connector itself. During Return Loss / impedance measurements, the transmission line has been terminated to 50 Ω load.

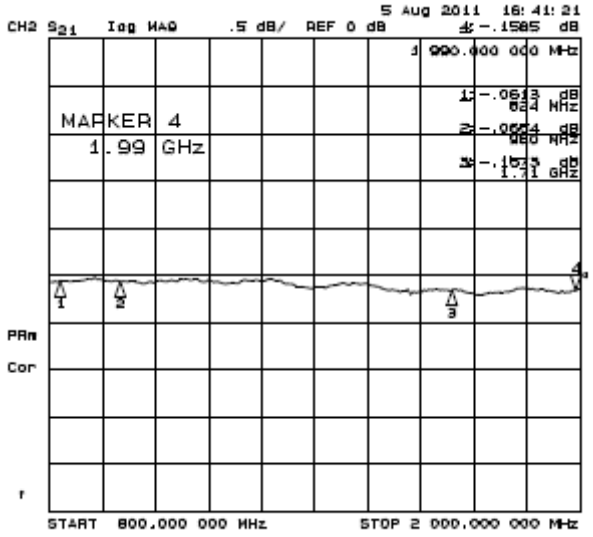
Return Loss plot of line under test is shown below:



Line input impedance (in Smith Chart format, once the line has been terminated to 50 Ω load) is shown in the following figure:



Insertion Loss of G-CPW line plus SMA connector is shown below:



Antenna Installation Guidelines

- Install the antenna in a place covered by the LTE signal.
- Antenna must not be installed inside metal cases
- Antenna shall also be installed according Antenna manufacturer instructions
- Antenna integration should optimize the Radiation Efficiency. Efficiency values > 50% are recommended on all frequency bands
- Antenna integration should not dramatically perturb the radiation pattern. It is preferable to get, after antenna installation, an omnidirectional radiation pattern, at least in one pattern cut
- Antenna Gain must not exceed values indicated in regulatory requirements, where applicable, in order to meet related EIRP limitations. Typical antenna Gain in most M2M applications does not exceed 2dBi
- If the device antenna is located farther than 20cm from the human body and there are no co-located transmitters, then the module FCC/IC approvals can be re-used by the end product
- If the device antenna is located closer than 20cm from the human body or there are co-located transmitters, then additional FCC/IC testing may be required for the end product (the module FCC/IC approvals cannot be reused)

Antenna Diversity Requirements

This product is including an input for a second Rx antenna to improve radio sensitivity. This function is named Antenna Diversity. The diversity antenna shall fulfil the following requirements:

Item	Value
Frequency range	Depending by frequency band(s) provided by the network operator, the customer shall use the most suitable antenna for that/those band(s)
Bandwidth	60 MHz in LTE/WCDMA Band 2 45 MHz in LTE Band 4 25 MHz in LTE/WCDMA Band 5 35 MHz in LTE Band 8 / GSM900 15 MHz in LTE Band 12 10 MHz in LTE Band 13 12 MHz in LTE band 17
Impedance	50 ohm
VSWR recommended	≤ 2:1 (limit to obtain max sensitivity)

Since there is no antenna connector on the module, the diversity antenna must be connected to the module's Diversity Antenna pad (F1) by means of a transmission line implemented on the PCB.

The second Rx antenna should not be located in the close vicinity of main antenna. In order to improve Diversity Gain, Isolation and reduce mutual interaction, the two antennas should be located at the maximum reciprocal distance possible, taking into consideration the available space into the application. For the same reason, the Rx antenna should also be cross-polarized with respect to the main antenna.

Isolation between main antenna and Rx antenna must be at least 10 dB in all uplink frequency bands. Envelope Correlation Coefficient (ECC) value should be as close as possible to zero, for best diversity performance. ECC values below 0.5 on all frequency bands are recommended.