TELIT MODEM INTEGRATION DESIGN GUIDE

1vv0301189 Rev.0 2014-12-19

APPLICABILITY TABLE

PRODUCTS

- GE864-QUAD V2
- GE864-QUAD AUTO V2
- GE864-GPS
- GE865-QUAD
- GE866-QUAD
- GE910-QUAD
- GE910-GNSS
- GE910-QUAD AUTO
- GE910-QUAD V3
- GL865-DUAL
- GL865-DUAL V3
- GL865-QUAD
- GL865-QUAD V3
- GL868-DUAL
- GL868-DUAL V3
- GC864-QUAD V2
- UE910 V2 SERIES
- UE910-EU V2 AUTO
- UC864-E
- UC864-K
- UC864-AK
- HE863 SERIES
- UL865 SERIES
- UL865-N3G
- UE910 SERIES
- HE920 AUTO SERIES
- HE910 V2 SERIES
- HE910 SERIES
- HE910 MINI PCIE
- HN930 SERIES
- CC864-DUAL

- CC864-SINGLE
- CC864-SR
- CC864-K
- CC864-KPS
- CL865-DUAL
- CE910-DUAL
- CE910-SC
- DE910-DUAL
- DE910-SC
- DE910 MINI PCIE
- LE910 SERIES
- LE920 AUTO SERIES
- LN930 SERIES

SPECIFICATIONS SUBJECT TO CHANGE WITHOUT NOTICE

LEGAL NOTICE

These Specifications are general guidelines pertaining to product selection and application and may not be appropriate for your particular project. Telit (which hereinafter shall include, its agents, licensors and affiliated companies) makes no representation as to the particular products identified in this document and makes no endorsement of any product. Telit disclaims any warranties, expressed or implied, relating to these specifications, including without limitation, warranties or merchantability, fitness for a particular purpose or satisfactory quality. Without limitation, Telit reserves the right to make changes to any products described herein and to remove any product, without notice.

It is possible that this document may contain references to, or information about Telit products, services and programs, that are not available in your region. Such references or information must not be construed to mean that Telit intends to make available such products, services and programs in your area.

USE AND INTELLECTUAL PROPERTY RIGHTS

These Specifications (and the products and services contained herein) are proprietary to Telit and its licensors and constitute the intellectual property of Telit (and its licensors). All title and intellectual property rights in and to the Specifications (and the products and services contained herein) is owned exclusively by Telit and its licensors. Other than as expressly set forth herein, no license or other rights in or to the Specifications and intellectual property rights related thereto are granted to you. Nothing in these Specifications shall, or shall be deemed to, convey license or any other right under Telit's patents, copyright, mask work or other intellectual property rights or the rights of others.

You may not, without the express written permission of Telit: (i) copy, reproduce, create derivative works of, reverse engineer, disassemble, decompile, distribute, merge or modify in any manner these Specifications or the products and components described herein; (ii) separate any component part of the products described herein, or separately use any component part thereof on any equipment, machinery, hardware or system; (iii) remove or destroy any proprietary marking or legends placed upon or contained within the products or their components or these Specifications; (iv) develop methods to enable unauthorized parties to use the products or their components; and (v) attempt to reconstruct or discover any source code, underlying ideas, algorithms, file formats or programming or interoperability interfaces of the products or their components by any means whatsoever. No part of these Specifications or any products or components described herein may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language or computer language, in any form or by any means, without the prior express written permission of Telit.

HIGH RISK MATERIALS

Components, units, or third-party products contained or used with the products described herein are NOT fault-tolerant and are NOT designed, manufactured, or intended for use as on-line control equipment in the following hazardous environments requiring fail-safe controls: the operation of Nuclear Facilities, Aircraft Navigation or Aircraft Communication Systems, Air Traffic Control, Life Support, or Weapons Systems (“High Risk Activities"). Telit, its licensors and its supplier(s) specifically disclaim any expressed or implied warranty of fitness for such High Risk Activities.

TRADEMARKS

You may not and may not allow others to use Telit or its third party licensors’ trademarks. To the extent that any portion of the products, components and any accompanying documents contain proprietary and confidential notices or legends, you will not remove such notices or legends.

CONTENTS

1 INTRODUCTION

1.1 Scope

Scope of this document is to give an overview of the good rules to integrate a 2G, 3G, 4G modem with its antenna into a board, posing particular attention on the common pitfalls and best practices.

1.2 Audience

This document is intended for design engineers who are familiar with PCB design and layout and electronic devices integration, but are not experienced at integrating radio devices into complex systems.

1.3 Contact Information, Support

For general contact, technical support services, technical questions and report documentation errors contact Telit Technical Support at:

TS-EMEA@telit.com

TS-AMERICAS@telit.com

TS-APAC@telit.com

Alternatively, use:

http://www.telit.com/support

For detailed information about where you can buy the Telit modules or for recommendations on accessories and components visit:

http://www.telit.com

Our aim is to make this guide as helpful as possible. Keep us informed of your comments and suggestions for improvements.

Telit appreciates feedback from the users of our information.

1.4 Text Conventions

Danger – This information MUST be followed or catastrophic equipment failure or bodily injury may occur.

Caution or Warning – Alerts the user to important points about integrating the module, if these points are not followed, the module and end user equipment may fail or malfunction.

Tip or Information – Provides advice and suggestions that may be useful when integrating the module.

All dates are in ISO 8601 format, i.e. YYYY-MM-DD.

1.5 Related Documents

- LE920 Hardware user guide, 1vv03701089
- LE910 Hardware User Guide, 1vv0301089
- HE910 Hardware User Guide, 1vv0300925
- HE920 Hardware user Guide, 1vv0301014
- xE910 miniPCI Hardware User Guide, 1vv0301006
- LN930 Hardware User Guide, 1vv0301078
- UE910 Hardware User Guide, 1vv0301012
- UE910 V2 Hardware User Guide, 1vv0301065
- UE910-EU V2 AUTO Hardware User Guide, 1vv0301072
- UE866 Hardware User Guide, 1vv0301157
- UL865 Hardware User Guide, 1vv0301050
- GE910 Hardware User Guide, 1vv0300962
- GE866-QUAD Hardware User Guide, 1vv0301051
- GL865-DUAL/QUAD Hardware User Guide, 1vv0300910
- GL865-DUAL/QUAD V3 Hardware User Guide, 1vv0301018
- GE865-QUAD Hardware User Guide, 1vv0300799
- GE/GC864-QUAD V2 and GE864-GPS Hardware User Guide, 1vv0300915

2 GENERAL ANTENNA OVERVIEW

2.1 Antenna fundamentals

2.1.1 Wavelength and Antenna electrical length

An antenna is a device that couples electromagnetic energy between some form of transmission line and free space. The function of a transmitting antenna is to radiate the energy fed into it, while a receiving antenna gathers energy from a free space electromagnetic wave and passes it to a receiver.
Most forms of antenna will function both as transmitting and receiving antennas, and many properties, for example its radiation pattern, are the same whether the antenna is transmitting or receiving. It is often easier to think of the transmitting properties of an antenna, so most of the explanations that follow regard the antenna as being in transmit mode.

The electromagnetic wave generated or received by the antenna, propagates through the free space at the speed of light *c*.
The speed of light in the free space is *c0* – 300000 Km/s - and if the wave propagates in a dielectric medium with relative dielectric constant ε_r, the speed of light is reduced to:

$$c = \frac{c_0}{\sqrt{\varepsilon_r}}$$

A common reference unit used when managing electromagnetic waves at a specific frequency is the wavelength λ defined as:

$$\lambda = \frac{c}{f} = \frac{c_0}{f \cdot \sqrt{\varepsilon_r}}$$

And using common units it can be simply calculated as:

$$\lambda = \frac{300}{f \cdot \sqrt{\varepsilon_r}}$$

With λ expressed in [m] and f in [MHz].

In the same way we can calculate the dimension of the antenna "as seen" by the electromagnetic wave with the following:

$$\Lambda = \frac{L}{\lambda} = \frac{f}{c_0} \cdot \sqrt{\varepsilon_r} \cdot L = \frac{L}{\lambda_0} \cdot \sqrt{\varepsilon_r}$$

This length is defined as *Λ*, the electrical length of the antenna and it is a key parameter in the antenna design, usually it is also expressed as a fraction of λ_0.
For antennas laying for example partially on a PCB FR4 ($\varepsilon_r \approx 4$) and partially on air (ε_r=1), such as the typical PCB printed antenna; the electrical dimensions of the antenna are bigger than the real ones (ε_{reff} >1) and therefore the antenna can be smaller than the same one in the free space.

2.1.2 Antenna field zones

When an electromagnetic wave is generated by an antenna, we can distinguish three zones around the antenna, with different properties of the wave.

- *The reactive near field zone*: is the space around the antenna in its close proximity where the reactive field components predominate over the radiated field. In this zone any variation in electrical/magnetic properties has a strong impact on the antenna's properties. Typically the boundary of the reactive near field is placed at a distance from the antenna of about $\lambda/2\pi$.

- *The radiating near field zone*, where the radiated field predominates but still the antenna dimensions are not negligible with respect to the distance from it. This means that the radiated pattern will be changing with the distance from the antenna. Typically the boundary of the radiating near field is between the end of the reactive near field and a distance from the antenna of about $\lambda/2\pi$ and the beginning of the far field radiating zone, conventionally placed at a distance of $2 \cdot D^2/\lambda$ where D is the dimension of the antenna.

- *The radiating far field zone*, where radiated field predominates and the antenna dimensions are negligible with respect to the distance from it. In this zone the radiation pattern does not change with distance, and field changes in amplitude with the 1/r law, hence its energy decreases with $1/r^2$ law. This region begins where the radiating near field ends and continues to infinity and is the distance at which receiving and transmitting antennas are usually placed in real life applications.

From these simple facts we can understand that the proximity area around the antenna (the reactive near field zone) is the most important for its characteristics and everything that's inside this region will strongly affect the way that the antenna behaves and its properties. For this reason embedded antennas are always a challenge, since the antenna must be designed together with the surrounding system and cannot be handled as a stand-alone device; in the same way, taking an antenna designed to work in free space and placing it directly inside an embedded system will usually result in poor performances.

2.1.3 Main antenna related parameters

If our radio device is to operate effectively, we need to make sure that the antenna is chosen to have the right performance parameters. Its radio performance also strongly depends on the way the antenna is integrated into the device. In selecting and integrating antennas we need to consider the following parameters:

Parameter	Description
Efficiency	This is the ratio of the power radiated by the antenna to the power fed to it by the connected transmitter. (This definition is sometimes known as the Total Efficiency or Realised Efficiency, because it includes the reflection loss caused by the mismatch of the antenna.) *An efficiency of 50% is usually considered good for a mobile device measured in free space. M2M devices are often less efficient than this*

Input impedance	If an antenna has input impedance different from that of the line feeding it, some of the power fed to it will be reflected at its terminals, and therefore cannot be radiated. *A transmitter is usually designed to feed a load with an impedance of 50 ohms (resistive) and its performance will decrease if the connected impedance is far from 50 ohms*.
Mismatch loss VSWR	The ratio of the power fed into the antenna (forward power minus reflected power) to the forward power in the transmission line that feeds it. As an example, an antenna with VSWR of 3:1 (return loss of 6dB) reflects ¼ of the power fed to it, so the mismatch loss = ¾ = 1.25dB.
Bandwidth	Many characteristics of an antenna change with frequency. They will all fall within stated limits over the frequency range defined by the bandwidth *The bandwidth for an antenna will normally be selected to cover the standard mobile radio bands in which we wish to provide operation. If an antenna covers multiple frequency bands it should not be assumed that it will operate at frequencies in between them.*
Polarisation	The electric field of an EM wave has a characteristic orientation in space, known as the polarisation plane. *For many mobile radio applications the plane of polarisation is not important and is not specified, because the orientation of the platform may be varied quite randomly by reflections and scattering. GPS signals are transmitted and received using right-hand circular polarisation, in which the plane of the electric field rotates clockwise (seen looking in the direction of propagation). This is a special case in which the receiving antenna needs to be correctly polarized*
Radiation pattern	The radiation pattern represents the way in which energy is radiated into the surrounding 3-D space. It may be seen as a solid shape, or more commonly as sections through the 3-D shape, usually in planes at right angles, related to orientation of the antenna or radio platform. *The radiation pattern of antennas on small platforms usually looks like a doughnut with the hole aligned with the long axis of the platform, and it will change in the presence of the user. For GPS applications it is important that the pattern has a broad upward-pointing shape to allow reception of signals from the sky.*
Gain	Gain is a measure of the ability of an antenna to focus the power fed to it in some directions rather than others. In the direction(s) of maximum radiation the gain is typically greater than 1; the average gain in all directions in space is equal to the efficiency Gain is often not specified for mobile devices as it is a function of the efficiency. Handsets are rather characterized by parameters known as the Total Radiated Power (TRP) for transmission and Total Isotropic Sensitivity (TIS) for reception
SAR (specific absorption rate)	This is a measure of the amount of energy absorbed by the body when exposed to electromagnetic fields. Its unit is watts/kilogram. Maximum limits are specified in national and international standards. The SAR depends strongly on the antenna position with respect to the body; therefore it is important for applications used by humans to accurately study the antenna placement.

Dimensions	When designing mobile radio devices the dimensions of the antenna are usually very important. The usual dimensions of an antenna are a function of their operating frequency and its wavelength. Antenna dimensions are usually referred to wavelength and calculated as electrical dimensions as shown before. In small portable user devices they must be compressed to a small fraction of their usual length. The extent to which this can be done is subject to well-known limits which relate bandwidth and efficiency to volume (measured in cubic wave-lengths). Small antennas – particularly when mounted on small platforms – cannot provide the performance we can obtain from a larger ones **The dependence of efficiency and bandwidth on the design of the whole platform, not just on that of the antenna, is referred to many times in this Application Note.**

2.1.4 Antenna types

One of the most familiar forms of antenna is a dipole, comprising a thin element one half-wavelength long, fed across a gap at the centre (Fig 1).

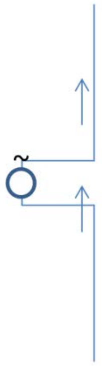


Figure 1: dipole antenna

This antenna has the familiar doughnut-shaped radiation pattern, an input impedance of around 70 ohms and a gain of 2.1dBi in the plane at right angles to its axis. Its bandwidth is increased (and its impedance and centre frequency are reduced) if the element is made fatter.

A dipole is a symmetrical (balanced) antenna. Its geometry and electrical properties are symmetrical about its centre and it should be fed from a balanced source, ie one which has equal and opposite voltages from each terminal to ground. A half-wave dipole is inconveniently large inside a mobile device.
The unbalanced monopole is obtained replacing one of the two branches of the half-wave dipole by an infinitely large ground plane (Fig 2).

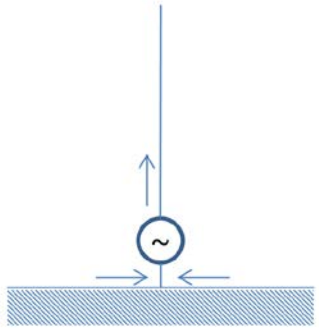


Figure 2: unbalanced monopole antenna

Due to the effect of mirroring, the radiation pattern above the ground plane remains unaffected. Also an infinite ground plane is not feasible, therefore usually the ground plane dimensions are reduced, but ideally it needs to be at least a quarter wavelength in radius; having a ground plane smaller than that will strongly impact on antenna efficiency.
The unbalanced monopole (Fig 2) is smaller in size than the dipole, but it is still too large (75mm at 900MHz) and the ground plane itself ideally needs to be at least a quarter wavelength in radius.

A practical internal antenna is created by folding the radiating element so it lies parallel to the ground plane (an inverted-L, Fig 3).

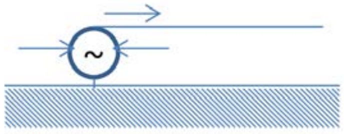


Figure 3: Inverted L monopole antenna

Better control of the input impedance is obtained by grounding the end of the element and feeding it part way along (an inverted-F antenna, Fig 4).

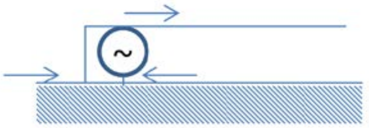


Figure 4: Inverted F monopole antenna

Further reduction in size is obtained by meandering the element (Fig 5).

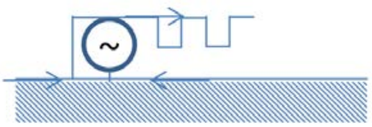


Figure 5: Meandered Inverted F monopole antenna

Not only does this much smaller antenna still require a ground plane, which provides the other terminal to complete the current path, but most of the effective radiation will come from currents in the ground plane rather than from those in the antenna element.

It is the presence of the ground current which both allows unbalanced antennas to operate, and also creates some of the difficulties which many people experience in using them. We must provide the ground current with a well-defined low-loss path which is isolated as far as possible from other circuits on the platform. If we fail to do this, RF energy is likely to be coupled into DC supplies, audio and digital circuits, and will not be radiated, introducing harmonics spurious emissions and other unwanted effects. When receiving, this unwanted coupling can introduce processor and TDM burst noise which may desensitize the receiver and impair performance.

Multi-band antennas are usually created by providing two (or more) radiating elements with different lengths. In the case of the 2G mobile phone bands the approximate 2:1 frequency ratio between upper and lower bands makes this technique very effective.

3 CONNECTING THE ANTENNA TO A MODULE

3.1 Connection types

Three main methods are available to connect the antenna to a module:

- Direct connection via a circuit trace
- Connection via a circuit trace and spring contacts
- Connection via coaxial cable

The connection must be always a controlled impedance transmission line with its characteristic impedance as closer as possible to the antenna/source impedance, which usually is 50 ohm.

3.1.1 Connection through a circuit trace

This method of connection is simple and has low manufacturing costs, but requires the designer to understand how an RF transmission line operates, since the connection track must be designed as a waveguide with 50 ohm characteristic impedance.
Most monopole antennas for M2M applications are surface-mount devices; these are usually fed by a grounded coplanar waveguide (Figure 6)

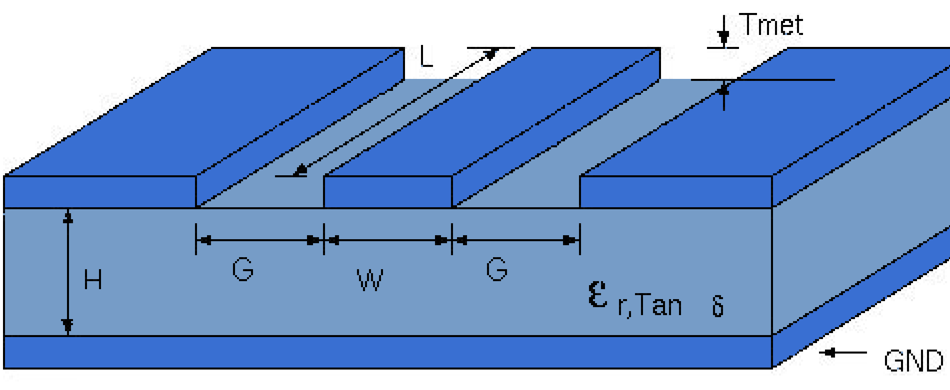


Figure 6:

Figure 6: Coplanar waveguide with ground (CPWG)

This line comprises a feed line of width *w* having slots of width *G* on each side and the surface conductor thickness *t*. Under the track and slots there is a uniform Ground reference plane, so that between the feed line and the ground surface there is a thickness H of dielectric with dielectric permittivity ε_r .

While on the model there is no direct connection between the upper layer ground surface and the bottom layer ground surface, we suggest placing vias in the ground surfaces all along the ground edges of the slot, creating a fence of vias, placed one each 2mm, that bonds together the upper ground surface with the bottom ground surface and avoids that the RF field couples with the tracks in the inner layers of the PCB.

3.1.2 CPW with spring connections to the antenna

If the antenna is designed to be assembled on top of the ground plane, SMD spring connections can be fitted onto the main PCB which will contact with pads provided on the antenna. These are available at low cost in many different patterns. Pogo-pins can also be used, but may not perform so reliably in the field. Some antennas have integral springs, in which case only contact pads are needed on the PCB.

3.1.3 Coaxial cables

The antenna and module should be generally placed as close together as possible. If it is essential that the antenna is located far away from the module, or on another PCB, you may need to use a coaxial cable between the module and the antenna. Take care to avoid the outer conductor of the cable becoming excited – keep it close to a ground plane wherever possible – or the result will be that the observed antenna impedance and its performances varies according to the exact position of the cable. Cable currents can be suppressed or at least reduced using ground independent antennas or, if this is not applicable, ferrite rings or sleeving on the coax. Unfortunately at high frequencies these devices are lossy and simply absorb the energy travelling along the cable, reducing the RF efficiency of the platform. It is often better, if possible, terminating the cable several centimeters from the antenna and run a CPW to the antenna.

3.1.4 Matching network

The antenna feed impedance depends on its geometry, the geometry of the platform, the presence of other components and the case. If the feed impedance of the antenna is different from the 50 ohm characteristic impedance of the transmission line feeding it, then a certain portion of RF power is reflected back to the RF source and is not available for radiation. This reduces the performances and may introduce other criticalities such as RF power amplifier compression and harmonics generation.

An optimum wideband impedance match between the antenna and the feed line will reduce the amount of reflected power and ensure that the full RF output power from the module is delivered to the antenna. It is wise to provide means for a matching network to be fitted as close as possible to the antenna terminals. Matching one band will usually require a simple L-C network in the form of an L, Pi or Tee. More components are needed if the matching is done in more bands. Allow for two components in parallel, one series and two in parallel with the transmission line and fit 0R on the series components until the required values are determined by test.

3.2 Designing a transmission line

The transmission line connecting any GSM/GPRS/UMTS/GPS module to the antenna must meet following requirements:

- line's characteristic impedance must match the Antenna Input impedance and the module's Front-End Impedance, in order to ensure proper impedance matching. Typical characteristic impedance value in such systems is 50 Ω;

- line's characteristic impedance must be uniform. Generally all mechanical aspects: track width, ground gap and eventually reference ground plane shall be constant all along the transmission line length, avoiding discontinuities, abrupt curves and meanders;

- line's power losses have to be kept to a minimum in order to ensure best RF power transfer between module and antenna, and vice-versa;

Theoretical value of 50 Ω cannot always be exactly achieved in practice; anyway, line's characteristic impedance must be uniform and as close as possible to this value. For this reason, the transmission line, regardless of its structure, has to be designed accordingly, in order to keep its characteristic impedance close to the required value.

A Transmission Line model can be derived from Maxwell's Equations. This model allows, for certain transmission line geometries (Coaxial cable, Microstrip, Stripline, CPWG…), to define formulas to calculate characteristic impedance with respect to line dimensions or viceversa.
These calculations are typically performed with dedicated software tools that allow synthesizing line geometry for given characteristic impedance.
There are several software programs available for the calculation of the characteristic impedance of different waveguides (CPWG, microstrip, stripline), some are freely available and on-line calculators such as:

I-Laboratory tools [www1.sphere.ne.jp/i-lab/ilab/index_e.htm]
EEWeb App. [www.eeweb.com/toolbox/asymmetric-stripline-impedance]

Chemandy Calculators [chemandy.com/calculators/calculator-index.htm]
Mantaro [www.mantaro.com/resources/impedance_calculator.htm]

Some are licensed such as the POLAR PCB field solver, some are free such as the Transmission Line Calculator program freely available from AWR:
TX-LINE [http://www.awrcorp.com/products/optional-products/tx-line-transmission-line-calculator]

You can use whatever you prefer, but when implementing the waveguide in the PCB proper care has to be taken to the following aspects:

- line geometry must be uniform and as similar as possible to canonical model, in order to ensure that calculated characteristic impedance value and control are achieved;

- in the typical case of PCB, printed transmission lines requiring ground planes (Microstrip, Stripline, Grounded Coplanar Waveguides…), the extension of ground layer cannot be infinite as in the canonical model. GND layer must be extended enough (say, at least 1 cm each side at GSM frequencies, or at least 10 times the width of transmission line track) around transmission line in order to ensure that field lines path is similar to that in the model and the impedance calculus are accurate. A single, well extended, ground plane for referencing both RF transmission lines and digital/baseband lines is typically the best solution;

- reference ground plane under RF transmission line must be continuous, without interruptions both in transverse and longitudinal directions, in order to allow RF current paths to close properly;

- when a Grounded Coplanar Waveguide is used, it is wise to distribute equally spaced via holes close to CPWG coplanar ground edges (about 2mm spacing, about 1mm distance from the edge) in order to improve line grounding and reduce signal coupling inside the PCB from adjacent tracks;

- if possible, avoid vias or abrupt curves/angles on the transmission line track and keep ground plane geometry as uniform as possible. Right-angle bending should include metering at the corner. This helps keeping characteristic impedance controlled;

- keep RF transmission line far from known noise sources on the application board, unless the line is well shielded (Coaxial cable, Stripline..). Anyway, it has to be noted that the use of unbalanced transmission lines (a coaxial cable, for example) for feeding balanced systems (a printed dipole antenna, for example) without the use of a proper Bal-Un device typically leads to common mode currents on the line's external shield, which can radiate or, reciprocally, pick up RF Interferences. For this reason, it's always wise to keep transmission lines route as far as possible from noisy or RF susceptible areas on the application board;

- branching of transmission lines has to be avoided. Unused transmission line branch act as an open section of line, a stub which loads the used transmission line branch, thus altering the impedance matching;

- If two line sections have to be switched, a GaAs SPDT switch has to be used. Some SMT connectors allow switching between line branches with internal mechanical switches. When selecting the GaAs RF Switch check that it can handle without compression the full +33dBm transmission of the modem, usually this corresponds to a 1dB compression point set to +36dBm typ, otherwise distortion will be introduced with harmonics generation.

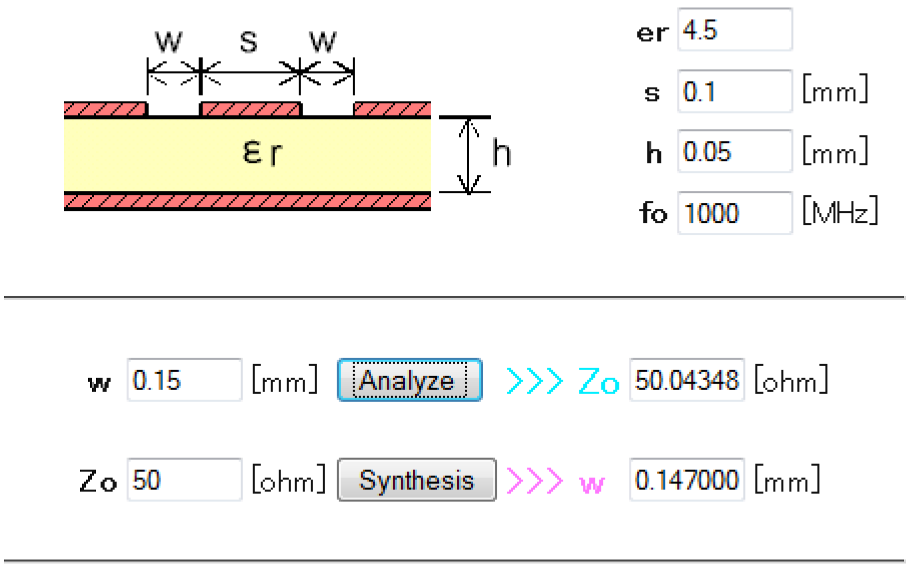
- take care in reducing line's power losses. Apart from Mismatch losses, which can be minimized with proper impedance matching, keep transmission line as short as possible and, when implementing printed transmission lines, use a low-loss dielectric material as substrate. Note that given the characteristic impedance, a stripline is typically thinner in comparison to a micro-strip track. This may lead to additional losses in some cases, therefore micro-strip or CPWG have to be preferred when PCB layup would lead to the synthesis of a very thin strip-line.

3.2.1 Example of Coplanar waveguide with Ground calculation

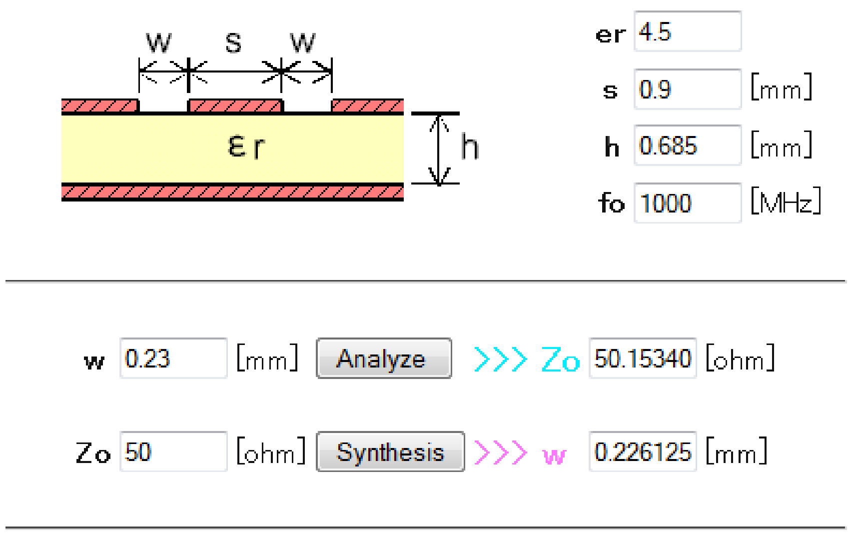
Let's suppose that the PCB build up of the application is as follows:

Layer	Material	Thickness
Top Layer	Copper	35µm
Dielectric	FR4	50µm
Layer 2	Copper	35 µm
Dielectric	FR4	600 µm
Layer 3	Copper	35 µm
Dielectric	FR4	50 µm
Bottom Layer	Copper	35 µm

We want to route a Coplanar with Ground waveguide on the TOP layer with 50 ohm characteristic impedance, considering an $\varepsilon_r = 4.5$ for FR4 standard material.
The impedance calculus gives a track thickness and a gap width that can be tricky for normal PCB manufacturing



It is therefore necessary to increase the dielectric height by removing copper under the antenna track in layer 2, keeping the ground plane on layer 3. With such arrangement the dielectric thickness under the track becomes 685 µm.



The new calculation gives good manufacturing values for the PCB track and can be implemented.

3.3 Maximizing the antenna performance of your device

3.3.1 Platform design

Once you have mounted a Telit module on your platform, connected it to an antenna and powered your hardware, the whole device has become a radio transmitter/receiver. With a maximum 2W peak burst power (in the 2G 850/900MHz bands) RF currents will flow through every exposed conductor and interconnection, since strong RF field surrounds all the application.

The ideal platform PCB, from an RF point of view, has continuous copper over both external surfaces, connected together with conducting vias around all edges. RF currents flow unimpeded on the outside surface of this "shielded box" PCB, while all internal tracks are isolated inside it, thanks to the faraday cage effect. Additional grounds are permitted inside the "box" if required to separate analog and digital circuits. This ideal design is not always easy to obtain, but wherever there is the need for cutting the external copper surface, then attention must be paid in providing an alternative acceptable path for the external RF currents.

Wires to external devices – batteries, speakers, alarms, sensors, data links and other hardware – are all exposed to the external RF fields. Flying wires and long PCB exposed tracks will act as monopole antennas and they will pick up RF energy and feed it both back into the connected devices and to the internal circuits in the main PCB. The RF energy collected will mostly be lost, reducing the TRP of the system, and furthermore, it is likely to cause problems – such as unwanted spurious harmonic emissions, buzzing on the audio, malfunctions of digital parts and power supplies. For these reasons you should:

- Keep external connections and exposed long PCB tracks to a minimum;
- Connect them around the center of the main PCB (with the modem and antenna) and run the connections away at right angles as far as possible;
- Decouple from RF and/or shield external connections as close as possible to the connector;
- Keep battery connections very short (use spring pins, not wires if possible);
- Add filtering capacitors to GND in close proximity of the wire connector to the PCB, so that the RF energy picked up can be shunted to GND and will not flow to the electronic components.

3.3.2 Antenna location

For devices with one PCB the best antenna position is typically at one end in a corner, with the feed close to a corner. Typical arrangements are shown in Figure 7.

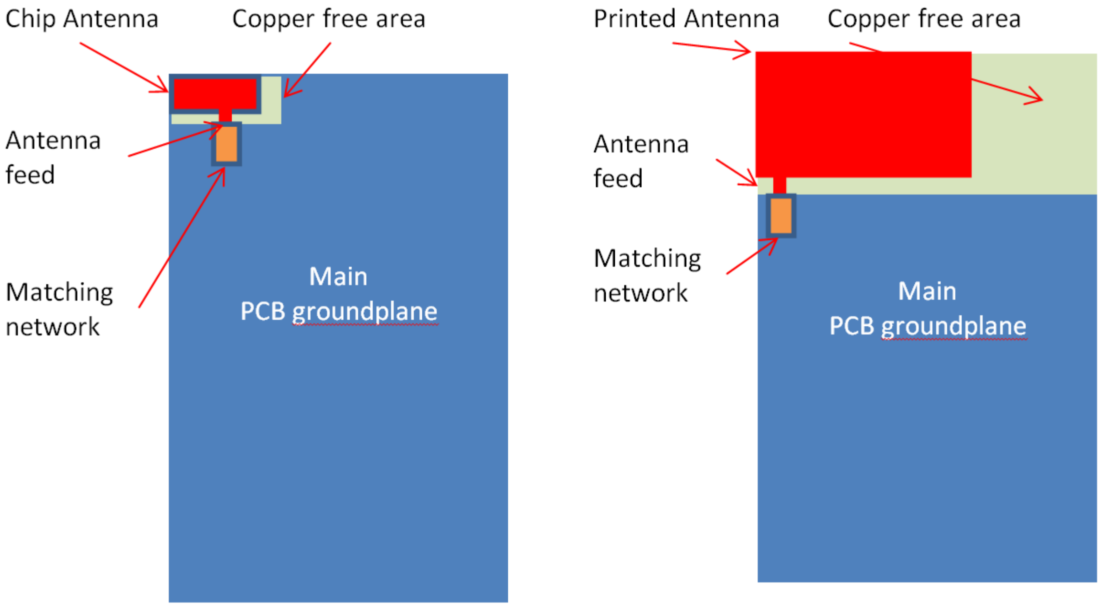


Figure 7: Typical antenna positions for Chip antenna and Printed antennas

Note that, unless you are using a ground independent antenna, the Main PCB ground plane is a relevant part of the antenna and its dimensions will strongly impact on antenna efficiency and bandwidth.

If your platform is very large and complex, you may wish to put the radio function on a separate RF PCB plugged into the motherboard. This configuration poses many challenges, including the potential proximity of the antenna to the motherboard circuits and ground plane, and the method and position of the ground connections between the RF board and the main board. It must be kept in mind that around the antenna the RF field is particularly strong and therefore all PCBs of the system will be exposed to this field and should be "RF proof", not only the modem and antenna PCB.
Having ground surfaces, apart from its reference plane, in the proximity of the antenna (the motherboard or other PCBs of the system) is usually deleterious for the antenna efficiency and performances, since the RF field closes directly on these surfaces instead of being radiated efficiently.
If you really need to have multiple PCB stacks in the proximity of the antenna, then you should think about removing the copper from the zones that are in close proximity of the antenna or stack the PCBs under the reference ground plane of the antenna.

It would be wise to make a simple trial assembly using almost blank (all-copper) PCBs and any major components such as displays, keypads or other hardware that will lie close to the assembly. Make the inter-connections you intend, make all ground connections and terminate signal and battery lines with the impedances and bypass arrangements you are intending to use in the final circuit. Place the battery in position and use a preliminary antenna design to explore in a variety of possible positions on your assembly the likely antenna input impedance with a Vector Network Analyser or its power output by feeding it with an external modem EVK.

This simple and raw procedure will not report the exact final antenna performances, but it will alert you of potential problems and should help you in finding a satisfactory arrangement. You will find issues you would not otherwise know about, such as a critical grounding position on a display, or an external connection that affects antenna performance. It helps you in avoiding later expensive and time-consuming redesign.

3.3.3 Placement of the antenna on the PCB

The antenna should always be placed on the edge of a PCB, preferably on one where it will have a 'view' out of the equipment in which it is installed. If the PCB is small, position the antenna along one of the short edges of the board, as this position will generally produce a larger impedance bandwidth than when the antenna is on the long edge.

If you choose an on-groundplane antenna; the space on the reverse side of the board might be used for components and the intermediate board layers can accommodate tracks for other circuit functions. A small clearance is required between the antenna and other components on the same side of the board, but if the components are only of low height (small SMD R, L or C), the clearance can be only 2mm. Taller components, or components belonging to circuits which could interact with strong RF fields, should be kept further away, with a clearance up to 5mm. A reasonable rule of thumb for components not sensitive to RF fields and not radiating RF noise is that they should lie below the line inclined at 30° shown in Fig 8.

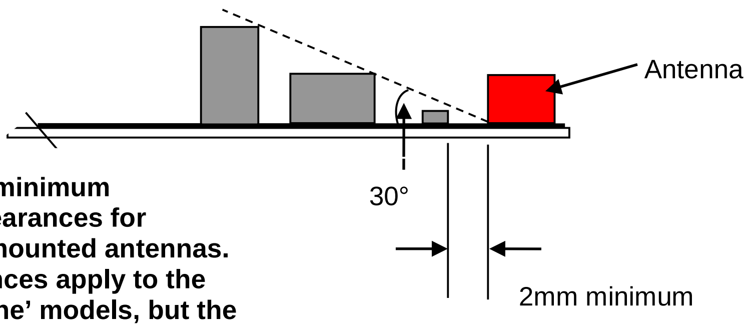


Fig 8: Typical minimum component clearances for groundplane-mounted antennas. Similar clearances apply to the 'off groundplane' models, but the distances are measured from the groundplane edge.

The effect of infringing the suggested limit will be a progressive reduction of the impedance bandwidth of the antenna and an increased risk of loss of RF energy by coupling into other circuits. It is very difficult to provide hard quantitative guidelines because of interactions between the antenna, the pcb dimensions and the particular layout, component types and circuit functions of the relevant components. Detailed electromagnetic modelling is possible, but it is very expensive to model a complex circuit in the necessary detail, especially when some of the interactions involve complex digital ICs.

3.3.4 Choice of antenna type

The first decision to be taken, when designing an application with an antenna, is how much space is assigned to the antenna and the second is whether using an off-the-shelf antenna or designing a custom one.

The first choice is fundamental, the antenna needs a certain amount of clearance and free space around it; you shall consider the antenna as a tridimensional object even if the antenna is a thin PCB.

You cannot fit a small antenna in the small slot available between the battery and the plastic cover and expect to have a good efficiency. The right antenna volume shall be predisposed since the beginning in the design of the device: generally the smaller the volume, the lower the performances.

The second choice is also relevant, and depending on it all the design will follow.

If you can afford the design of the custom antenna, in particular a printed antenna in the PCB, then this will be probably the lowest cost solution and can give good performances, but it will take time and some expense to correctly complete its design, furthermore you should remember that the antenna is designed together with the application and is strongly affected from dielectrics and conductors in its reactive field zone and therefore it is not possible to simply move a good antenna design into another board and expect that everything will work the same.

If you can accommodate an off-the-shelf antenna, respecting the constraints that come with it from its manufacturer, then this can be a good start, provided that the antenna is considered with its clearance, but it will give little chances of improvement if the performances don't reach the target.

In general, choose the largest antenna you can accommodate in the design, as they have wider impedance bandwidths than smaller ones and it will be easier to match and will produce higher efficiency.

4 USB LINES DESIGN

4.1 USB LINES: a particular type of waveguide

USB lines shall be designed as controlled impedance differential waveguides in order to guarantee a reflection-free transmission and correct high speed USB 2.0 operation.
In this case there are two signal lines and a Ground Plane reference; we can therefore define different characteristic impedances:

- Differential impedance: The impedance measured between the two lines when they are driven with opposite polarity signals.
- Odd impedance: The impedance measured testing only one of the differential traces referenced to the Ground plane when the two traces are driven with opposite polarity signals
- Common impedance: The impedance measured between the two traces referenced to the Ground plane when the two traces are driven with the same signal
- Even impedance: The impedance measured testing only one of the differential traces referenced to the Ground plane when the two traces are driven with the same signal

The Odd impedance is equal to half of the Differential one and the Even impedance is twice the Common impedance.
The characteristic impedance of the single uncoupled line is between the Odd Impedance and the Even impedance.

The USB line requirement is as follows:

- Common mode Characteristic impedance to GND : 30 Ω ± 30%
- Differential Characteristic impedance between lines : 90 Ω ± 15%

This can be also expressed as:

- Even Characteristic impedance to GND : 60 Ω ± 30%
- Odd Characteristic impedance between lines : 45 Ω ± 15%

All directives given for the transmission line apply, with a further requisite on differential characteristic impedance.
The differential characteristic impedance can be calculated for some simple topology of track routing, for example using on-line calculators and programs such as:

EEWeb App. [www.eeweb.com/toolbox/edge-coupled-microstrip-impedance/]

EEWeb App. [www.eeweb.com/toolbox/edge-coupled-stripline-impedance/]

Mantaro

[www.mantaro.com/resources/impedance_calculator.htm#differential_microstrip2_impedance]

[www.mantaro.com/resources/impedance_calculator.htm#differential_stripline2_impedance]

Colorado Microstrip [http://www.cepd.com/calculators/microstrip.htm]

Finetune [http://www.finetune.co.jp/~lyuka/technote/ustrip]

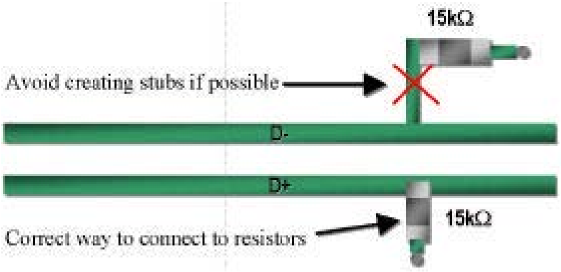
Wcalc [http://wcalc.sourceforge.net/cgi-wcalc.html]

But for this calculus we prefer the Transmission Line Calculator program freely available from AWR:

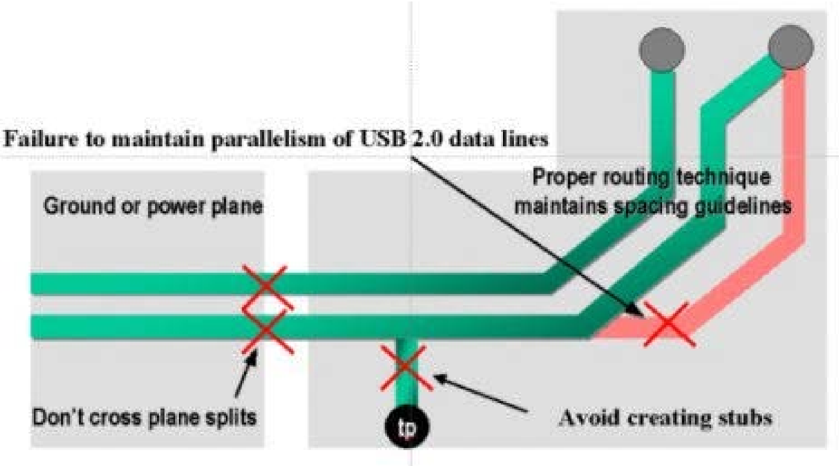
TX-LINE [http://www.awrcorp.com/products/optional-products/tx-line-transmission-line-calculator]

As specified in the High Speed USB Platform Design Guidelines, when routing the tracks you should avoid the following routing mistakes:

- Avoid creating unnecessary stubs:



- Crossing a plane split
- Create stubs with test points
- Failure to maintain the waveguide topology, in particular the parallelism between the two tracks



4.1.1 Example of differential pair waveguide for USB calculation

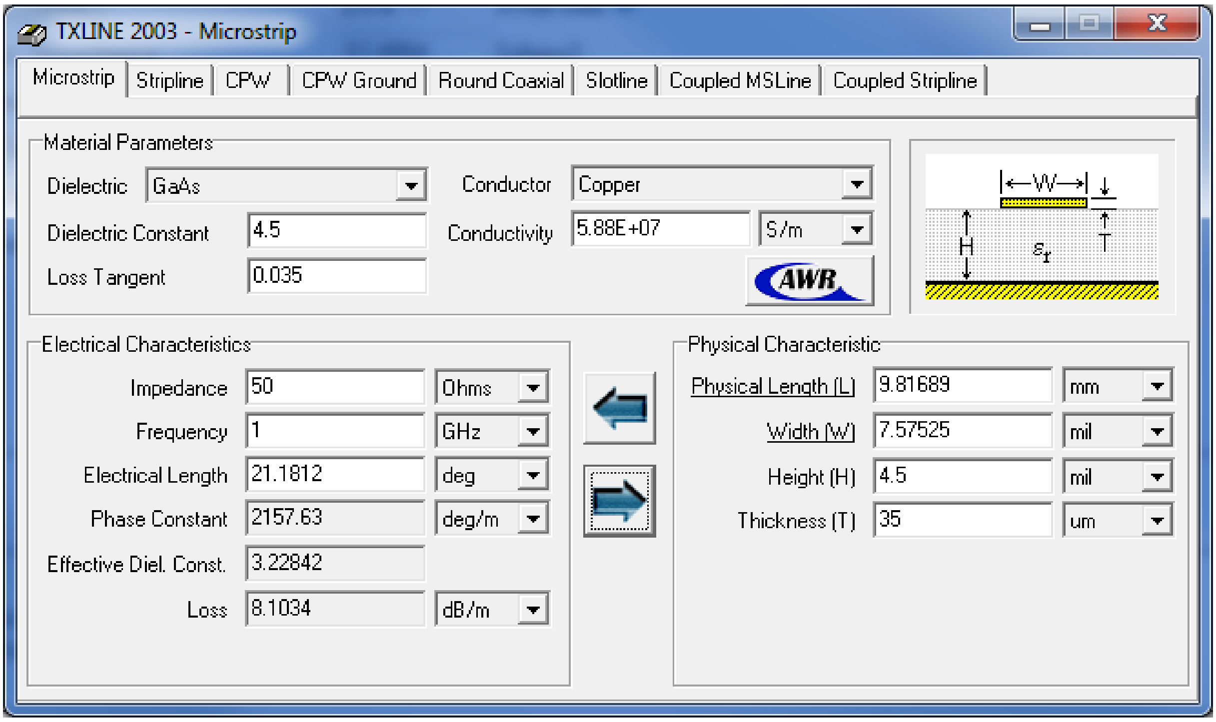
Let's suppose that the PCB build-up of the application is as follows:

Layer	Material	Thickness
Top Layer	Copper	35µm (1.4 mil)
Dielectric	FR4	115µm (4.5 mil)
Layer 2	Copper	35 µm (1.4 mil)
Dielectric	FR4	1350 µm (53.1 mil)
Layer 3	Copper	35 µm (1.4 mil)
Dielectric	FR4	115 µm (4.5 mil)
Bottom Layer	Copper	35 µm

We want to route a differential pair waveguide on the TOP layer for the USB 2.0, hence with about 30 ohm common mode characteristic impedance and 90 ohm differential characteristic impedance, considering an ε_r = 4.5 for the FR4 dielectric standard material, that means 45 ohm Odd impedance and 60 ohm Even Impedance.

There is no free synthesis tool for such differential microstrip line; therefore we need to proceed with a trial and error approach.

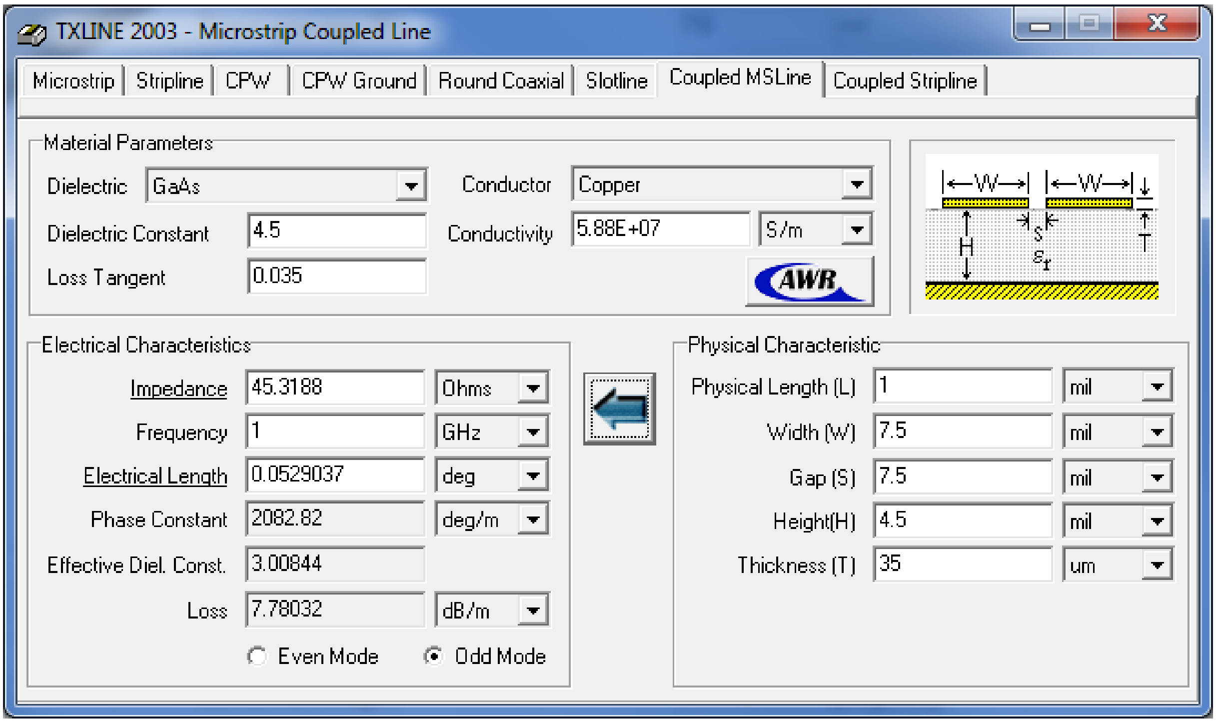
In order to have an initial reasonable value for the parameters we start considering a single microstrip track, setting its impedance to around 50 ohm (in between 45Ω Odd and 60Ω Even), using the TOP layer to route the track and the Layer 2 for the reference ground plane, therefore the dielectric height will be 4.5 mil.



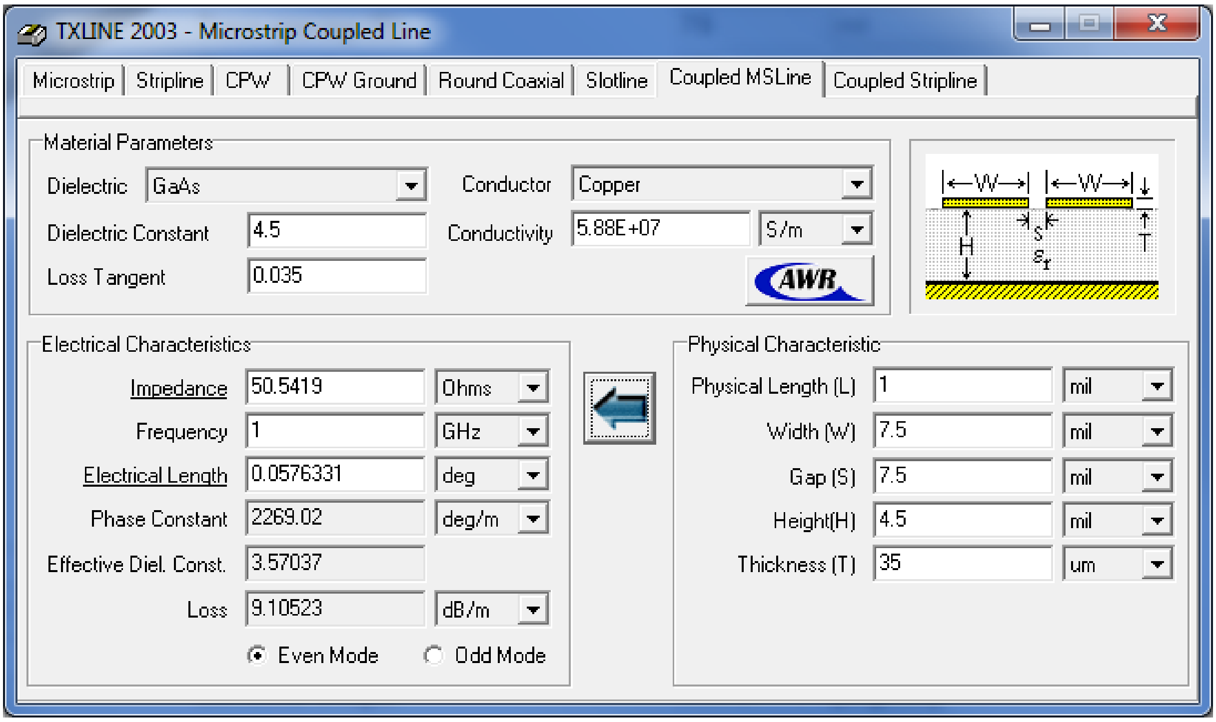
The resulting track width synthesized by the program is 7.5 mil, let’s take this as the starting value of the track width for the trial and error design of the coupled pair.

We change tab to the *Coupled MS Line* and insert the dimensions, calculating the Odd Mode characteristic impedance with a target of 45 ohm, changing only the Gap (S).

After a few trials we find that a Gap of 7.5 mils is suited to reach 45 ohm Odd impedance, hence 90 ohm differential impedance.

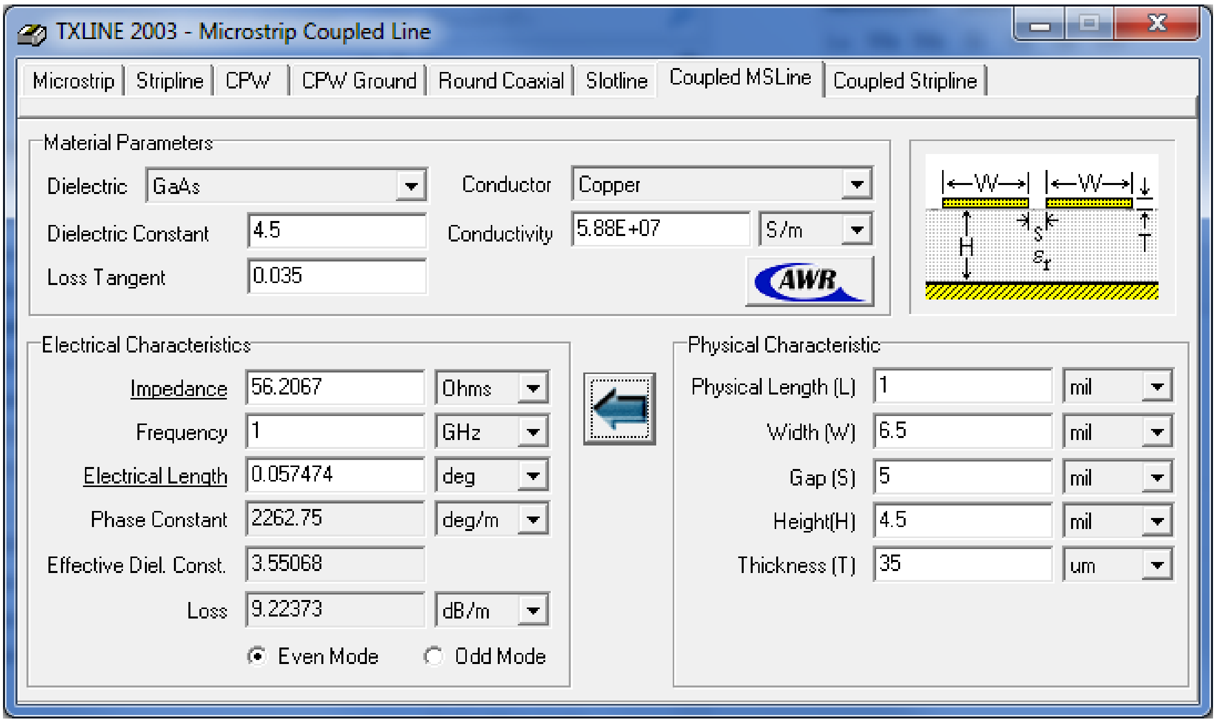


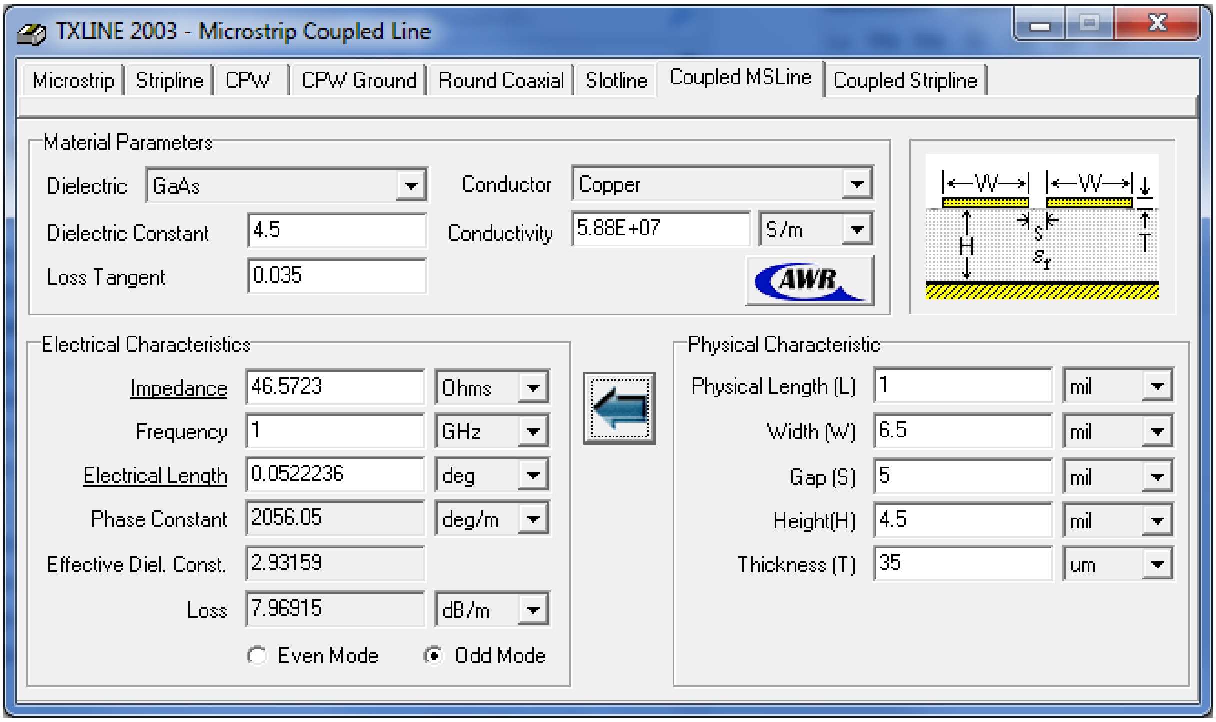
Let’s check now the Even Mode impedance (select Even Mode and run again the calculation):



The result is a 50 ohms Even Mode impedance, corresponding to 25 ohm common mode impedance, we need to rise it a bit closer to 30 ohm, meaning an Even mode characteristic impedance closer to 60 ohm. We can do this by increasing the track width and reducing the Gap to keep more or less constant the Odd mode characteristic impedance.

After a few trials, we reach an acceptable value:





Therefore our differential waveguide will be with track width W = 6.5 mil, Gap between tracks S = 5 mil and in order to avoid further coupling with the ground, the ground surfaces on the sides of the Top layer differential tracks will be kept 100 mil away from the tracks.

With the given measures the differential waveguide will have 93 ohm differential mode impedance and 28 ohm common mode impedance, fulfilling the USB 2.0 requirements.

5 EMI ASPECTS

In applications embedding a communication device, several issues become critical, due to the proximity of the antenna to the PCB of the application and in general of a RF device presence with the RF field emission coming from the antenna. The Electro Magnetic Interference (EMI) issues become more and more critical as the application and the antenna get tightly coupled and for small devices with embedded antennas this is unavoidable and should be managed in the right way.
Attention will now be posed on the EMI issues of the integration related to the presence of strong RF fields surrounding the application; these can be primarily divided into two kinds:

- receiver sensitivity issues, with low sensitivity of the receiver in some or all channels
- unwanted spurious emissions from the application

5.1 Receiver Sensitivity issues

Due to the extreme sensitivity of the current receivers, any unwanted emission on the receiver band can overlap with the signal to be received and easily cover it, resulting in a numb frequency spot on the receiver performance, even if receiver and its antenna are working perfectly.
So, while referred to EMI normative the emissions of the device can still be low and well inside regulatory limits, for the functionality of the receiver these emission pose severe limitations and must be avoided or removed. Usually these unwanted emissions come from harmonics of working frequencies of noisy devices; some examples can be:

- switching regulators
- microcontroller memory bus and generally CPU operations
- oscillators

The golden rules to limit these emissions are quite straightforward:

- use a good grounding system
- decouple with adequate capacitance the loads
- keep the noisy traces buried inside the inner layers of the PCB
- keep current loops as small as possible
- shield the noisy devices into a metallic shield if required
- use lower voltage range on digital devices
  [For example prefer 1.8V CPU rather than 3.3V ones]

Good grounding is the primary prerequisite for a successful PCB design with RF components; with a poor grounding any other action to limit unwanted emissions will be almost ineffective. We strongly suggest having a single ground approach, the multiple different grounds with a single interconnection approach is always critical in RF applications. While at low frequency, where PCB dimensions are negligible with respect to the wavelength, the return current through the different grounds flow conducted in a controlled manner through the different grounds avoiding multiple return paths and irradiation; at high frequency the PCB dimensions and hence the ground interconnections are comparable or greater than wavelength, they become antennas themselves; the return currents therefore can close through radiated paths in space, out of control and with multiple return paths, leading to interferences between the different groundings and furthermore they radiate energy that will generate unwanted emissions.

Load decoupling is also another important aspect; it must be especially cured for switching regulators and fast microcontrollers that can easily generate high order harmonics with plenty of energy. In addition to the common decoupling with fairly big capacitance we suggest to add in parallel also a low impedance decoupling tuned at GSM frequencies; typically ceramic capacitors around 10pF and 33pF self resonate at a frequency around 1800 and 900 MHz, providing a very low impedance and effective decoupling at those frequencies.
These capacitors can be used either to provide locally the energy to for example a CPU, avoiding RF current flowing in long tracks, or to reflect energy away from sensitive components, but this will be analyzed in detail afterwards with the unwanted spurious emissions analysis.

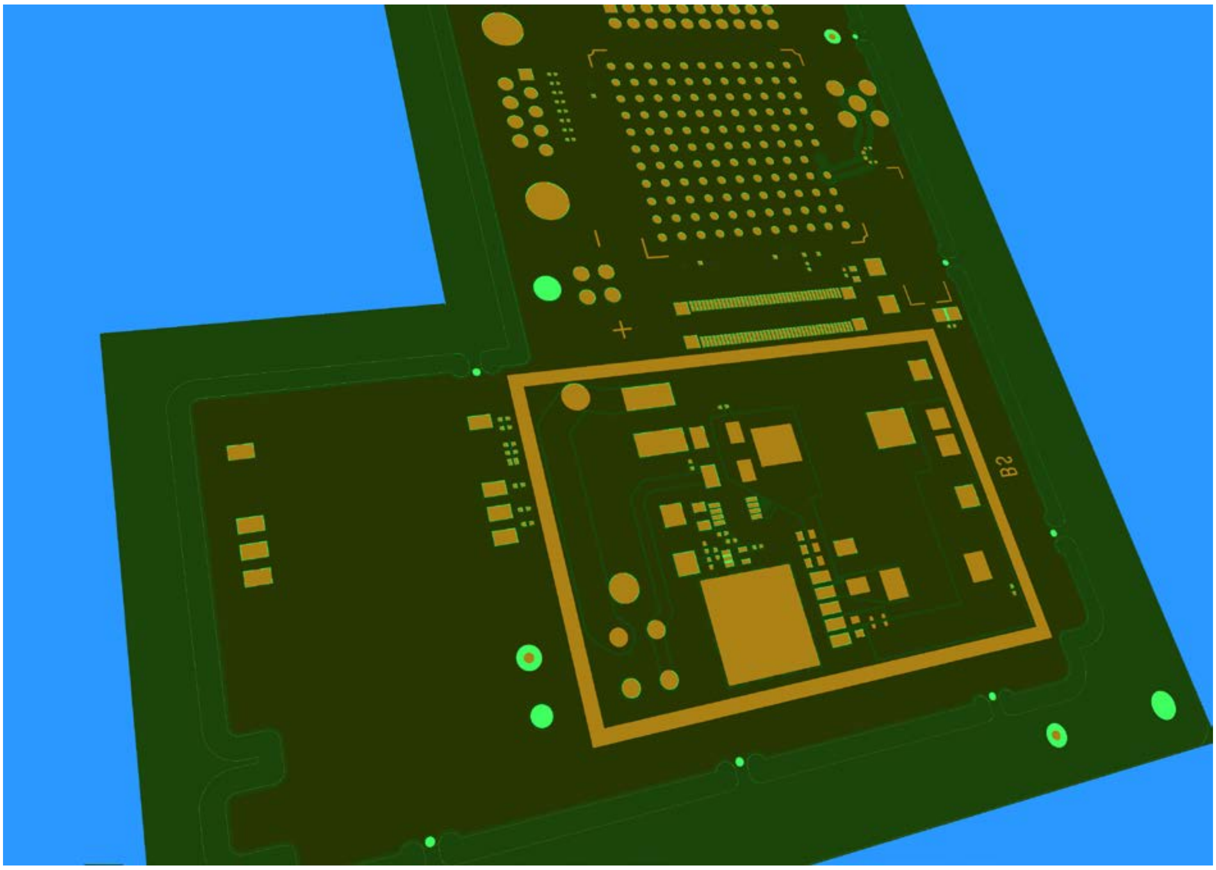
The two lay-out points regarding noisy traces and current loops are quite obvious; in order to reduce radiation from unwanted tracks either the track must be shielded inside the PCB between two GROUND planes or its current loop being reduced to a minimum. Here the PCB layer organization is important and we strongly suggest a PCB build-up of this kind:

- Top Layer: Ground Plane, with very little or no tracks at all, only components pads
- Inner Layer: Interconnections & power plane
- Inner Layer: Optionally but helpful a full Ground Layer without any discontinuity except from vias holes and holes and well connected to the top-bottom grounds.
- Inner Layer: Interconnections & power plane
- Bottom Layer: Ground Plane, with very little or no tracks at all, only components pads

This PCB layer organization requires usually at least a 4 layer PCB, but it will give the best results in terms of low unwanted emissions and immunity.

The last suggestion is usually the last viable solution, when all the preceding rules resulted in an unsatisfactory application or are not applicable, then a metallic shield covering the noisy components is applied; again, this is effective only if a suitable grounding has been implemented. Some components are so noisy that they need such a shielding and, in order to be able to shield the components, some minimal conditions must be implemented, all of them are anyhow helpful for the general PCB design:

- components must be packed into well-defined zones of the PCB and not randomly spread all over the PCB
- outside the shielded zones no exposed tracks shall be present on the PCB, only ground plane
- a ground ring shall be surrounding the components zones, where the shield will be soldered eventually



An example of a well done PCB

We always suggest forecasting the possibility to assembly a metallic shield over the components from the beginning, then usually this is not needed and shield cost can be saved; but in case this is really needed, there's no need to redesign the entire application and mechanics. In some cases, such as long flat interconnection cables, shielding is not practical and a ferrite coupled to the flat cable can give good results too.

5.2 Spurious emissions

The spurious emissions coming out of the application can be usually generated in two different ways:

- emission of harmonics of the system clocks or noise generated by the devices
- emission of harmonics and inter-modulation of the GSM transmission signal

The first emission can be seen exactly as the locked channel interference, therefore all the previous suggestions apply; on the second type of emission, few words must be spent to clarify the mechanism generating them.
When the GSM transmission occurs, a strong RF field is generated by the antenna close to the application PCB and components; this RF field couples with the PCB tracks and planes and is picked up by the tracks which can be seen as individual antennas [even if not so efficient, power here is so high that even a small coupling can result in enough RF energy flowing]. This RF energy is feed into the device that's on the PCB and if this is a non-linear component [basically any junction is non-linear] a full set of harmonics is generated. The harmonics generated in such a way are then re-radiated through the same track which acts both as receiving antenna at the fundamental frequency and transmitting antenna at the harmonic frequency. It is clear from this scheme that there must be three "actors" to generate such harmonics:

* a receiver: the receiving antenna track at the fundamental frequency
* an harmonic generator: the non-linear component [basically any active device]
* a transmitter: the transmitting antenna track at the harmonics frequency

In order to avoid such issues the main recommendations are:

* use a good grounding system, with ground on external layers of PCB
* keep the long traces buried inside the inner layers of the PCB
* protect from RF energy the sensible components with a 10-33pF capacitor to GND

The first two recommendations are quite straightforward, by having a good ground plane on external layers of the PCB and the tracks buried inside we shield the tracks from the RF field, killing both the receiver and transmitter “actors”.
The third recommendation is more difficult to be applied; who is the sensible component is not always easy to know in advance. In general, external cables and high impedance pins are more easily driven by RF energy than low impedance ones; therefore on all high impedance pins a 10-33pF capacitor shall be forecasted. The capacitor in this case reflects back the RF energy, avoiding the RF coupling that can activate the “generator”.

5.3 Multiple Transmitter-receivers co-location

The applications that integrate more than one transmitter/receiver are becoming more and more frequent in the M2M market; it is not uncommon having 3 or 4 different RF devices with 3-4 antennas in the same box, for example an application integrating a 3G modem, a WiFi radio and a GPS/Glonass receiver.
This kind of integration poses further challenges in the design and particular care shall be paid to reduce the coupling between the antennas and the devices front-ends.
Note that the separation is needed both for the correct operation of the receivers avoiding saturation and blocking phenomena and for the reduction of unwanted spurious emissions that may be generated by the intermodulation of the strong GSM carrier with the other transmitter.

Usually for Wifi & Bluetooth transceivers that operate at 2.4GHz it is not sufficient a low WIFI antenna efficiency at GSM frequencies to separate the signals and avoid spurious emissions or blocking. You usually need the insertion of a SAW filter immediately after the antenna.
Also on GPS front-end, if not already present, you need to insert a SAW filter immediately after the antenna, hence BEFORE the LNA, to reduce the signal strength feed to the LNA, avoiding its saturation and harmonics generation.

6 DOCUMENT HISTORY

Revision	Date	Changes
0	2014-12-19	First issue

Design Review
120°
180°
Process
Validation
240°
RF/EMC Pre-Certification
Module Selection
60°
300°
Approvals
Support
360°
Value Added Services & Connectivity
Enterprise Integration &
Application Enablement
m2mair