WT11I

DESIGN GUIDE

Monday, 28 November 2011

Version 1.1

Contents:

Version history

Version:	Author:	Comments:
1.1	PRa	WT11i-E additions
1.0	PRa	

1 INTRODUCTION

This document describes the basic principles for EMC and RF performance with Bluetooth modules and basic techniques to prevent facing problems with EMC and RF with WT11i. WT11i evaluation kit is shown as an example design. Evaluation kit reference includes the connection diagrams for all the interfaces available with WT11i. Basic layout recommendations are given for WT11i.

2 TYPICAL EMC PROBLEMS WITH BLUETOOTH

2.1 Radiated Emissions

CE and FCC regulations define certain limit for unintended radiated emissions from a device. WT11i is design and verified to meet these regulations with the evaluation board. Typical emission peaks with Bluetooth are at 1.6 GHz and 4.8 GHz (second harmonic). The primary modes of suppressing radiated emissions at 1.6 GHz are to use proper band pass filtering and EMC shielding in the module. Thus emissions at 1.6 GHz can not be effected by the mother board design. The same methods apply also for 4.8 GHz emissions but in this case the layout to which the module is mounted and the antenna design when using WT11i-E can cause increased level of radiated emissions.

The key method to avoid any increase in the emissions is to avoid any antenna structures in the layout. The simplest way is to use one solid ground plane at inner layer of the PCB and route all the signals at top and bottom layers. Following figure shows typical construction of 4-layer design.

Figure 1: Typical 4-layer PCB construction

Quite often it is not possible, due to lack of space or due to PCB manufacturability, to dedicate whole layer for GND and instead overlapping GND layers are used on all layers. In this case it is very important to avoid RF radiating from edges of the PCB. Overlapping GND planes can easily create a patch antenna and the RF energy travelling between GND layers will radiate from the edges unless using techniques to prevent unintentional radiation.

To prevent radiated emissions from the edges of the PCB one should use stitching GND vias separated by max 3 mm at all edges. These GND vias will operate as a shield preventing RF energy travelling between the PCB layers to radiate from the PCB edge. See following figure.

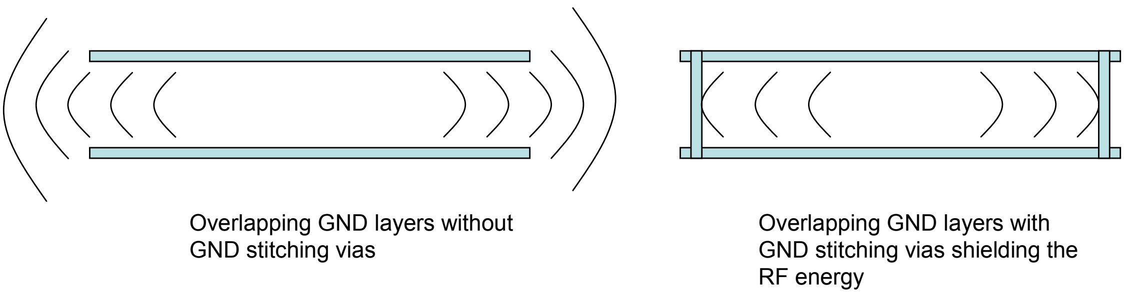


Figure 2: Using GND stitching vias to prevent unintentional radiation from the edges of the PCB

Also the module can create a patch antenna structure if not mounted properly. All the GND pins of the module should be connected directly to a solid GND plane in the mother board.

When using an external antenna with a cable there is a risk that the antenna cable creates an antenna. RF energy travelling at the GND areas of the module travels through the shield of the cable and radiates to surrounding space since it has no return path. The antenna cable should be as short as possible and one should avoid using cable that has length of λ, $\lambda/2$ or $\lambda/4$, where λ is the wave length c/f.

2.2 RF Noise in Signal Lines

Digital signal lines are usually very insensitive to RF power used with Bluetooth devices. However one should use good consideration when designing a layout for supply voltages and analog signal lines such as audio signals. Excessive RF noise coupled to supply voltage lines can have an impact on RF performance of the module and RF noise that couples to audio signal lines usually demodulates down to audio band causing very unpleasant whining noise.

Noise couples to signals lines either through a parasitic capacitance or by coupling to a loop. The noise that couples to a loop is proportional to the area of the loop and to the electromagnetic field flowing through the loop. Thus the noise can be minimized in two ways. Minimizing the field strength flowing through the loop by placing the signal lines far from the RF source or most importantly minimize the size of the loop by keeping the trace as short as possible and making sure that the path for the return current (usually GND) is low impedance and follows the forward current all the way as close as possible. When using fully differential signals they should be routed as differential pairs, parallel and symmetrically.

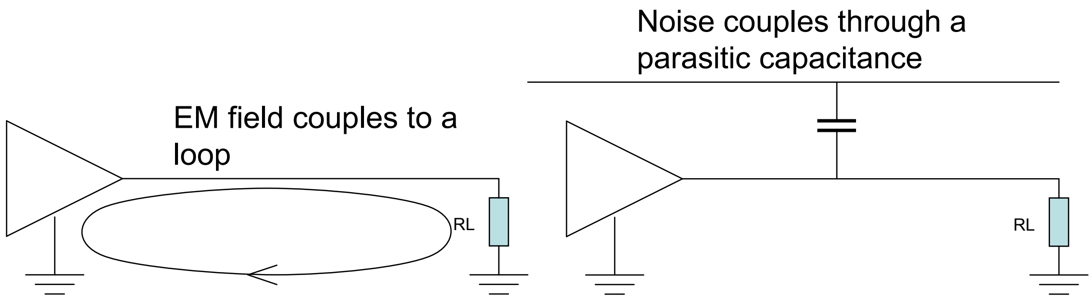


Figure 3: Noise coupling schemes

Following figure shows how to use LC filtering to filter RF noise from the signal lines. The placement of the filtering components is critical and usually they should be placed as close as possible to critical pins, such as power supply or audio input/output. Once the RF noise enters for example to an operational amplifier from certain pin, the frequency is way beyond the band width of the amplifier and thus the noise travels without any attenuation to the input of the amplifier and takes it out from it's linear region causing demodulation of RF down to audio frequencies ~1kHz.

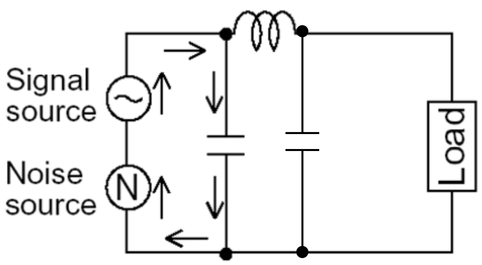


Figure 4: Filtering RF noise

Following figure show an ideal capacitor and an equivalent circuit of an actual capacitor. The capacitor has certain serial inductance which depends on the physical package of the capacitor. The high frequency characteristics are strongly depended on the serial inductance and at certain frequency the inductance becomes dominant from the capacitance and the impedance begins to increase. Thus to effectively filter noise at 2.5 GHz one should choose a capacitor which has minimum impedance at 2.5 GHz, typically 15 pF NP0 0402.

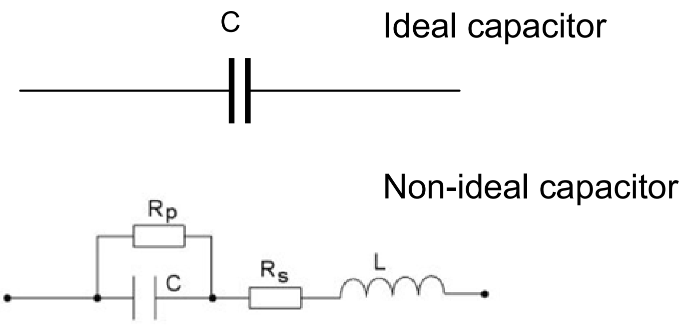


Figure 5: Equivalent circuit of a capacitor

3 USING LEVEL SHIFTERS TO INTERFACE WITH 1V8 or 5V0 DEVICES

There are three common way to interface between two different logic level regions. The simplest way is to use a logic buffer which operates at certain logic levels but is designed to accept higher logic levels. An example is 74LVC244. It operates with 3V3 supply voltage but it can accept up to 5V5 levels to inputs. Thus it can be used as a level translator from 5V to 3V3. If the upper threshold voltage of the 5V device is less than 3V there aren't usually any problems in interfacing 3V3 output directly to 5V input. One should however take into account that the output voltage level from CMOS output depend on the load current that it drives.

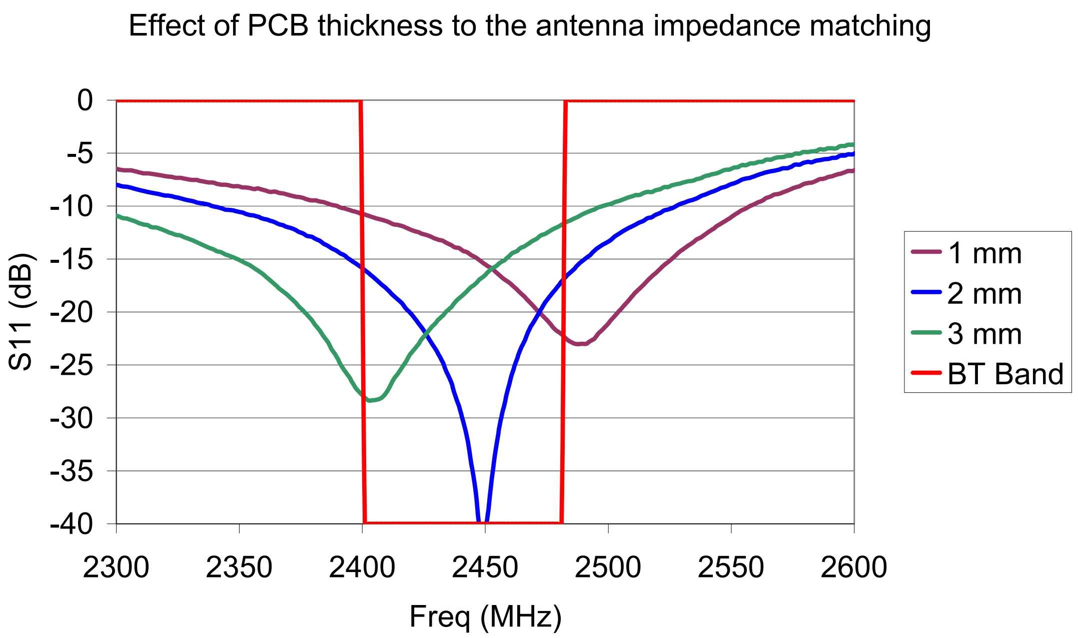
Second way is to use a translating transceiver such as 74LVC4245. The advantage of this kind of transceiver is that it can operate as a level translator in both ways, from high voltage to low voltage and vice versa. The draw back is that it requires a control signal to define the direction of the translation.

Third commonly used way is to use bi-directional level translators such as ST2378. The advantage of this kind of level translators is that it works bi-directionally without any control signals. The draw back is that it can not drive resistive loads so it should always be connected to high impedance node.

4 LAYOUT GUIDE FOR WT11i-A

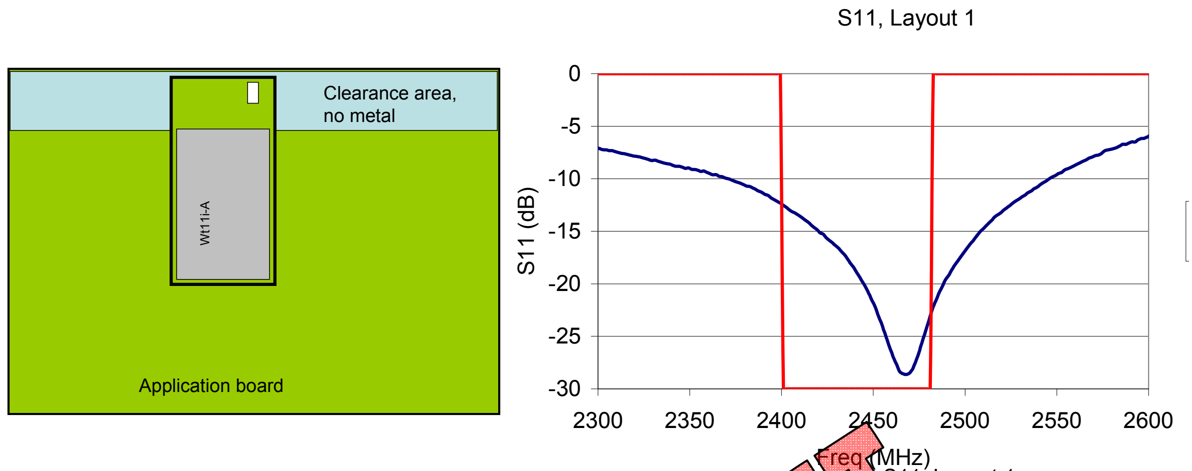
4.1 Effect of the PCB thickness to the impedance matching

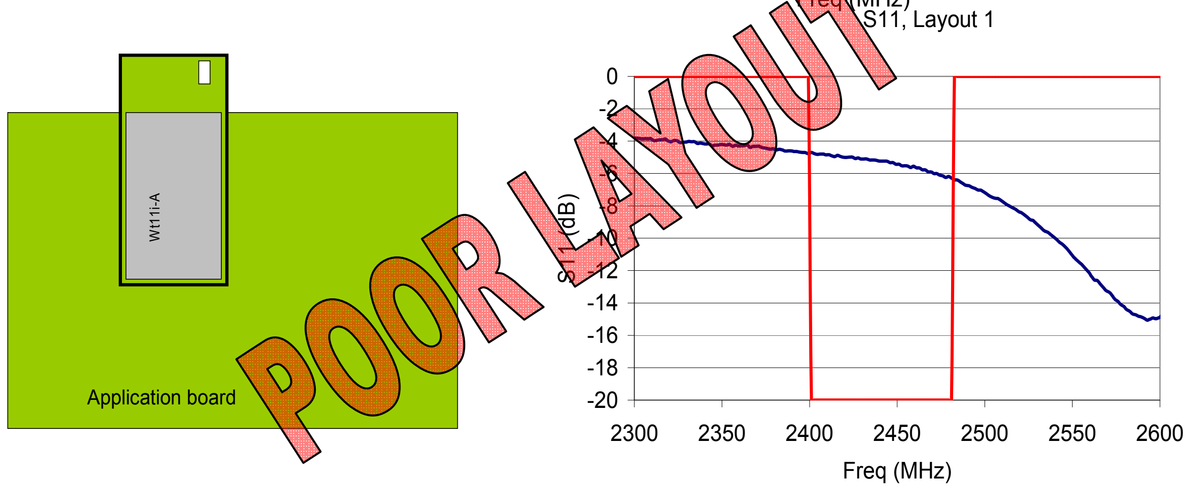
Any dielectric material in close proximity to the antenna will effect on the impedance matching of the antenna by lowering the resonance frequency. Following figure shows how different FR4 thickness under the antenna effect on the resonance frequency. Recommended PCB thickness for the PCB is 1.6 mm – 2.8 mm. Avoid placing plastic cover closer than 3 mm from the antenna as this will also tune the resonance frequency downwards.



4.2 Effect of the layout to the impedance matching

For the reasons described in chapter 4.1 the layout will also effect on the resonance frequency of the chip antenna.

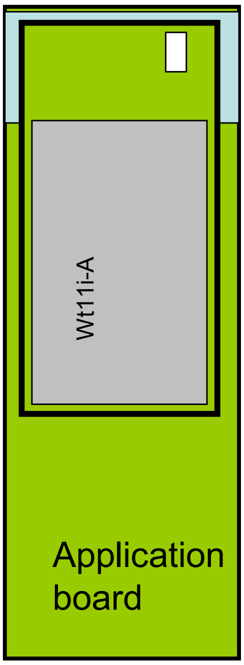


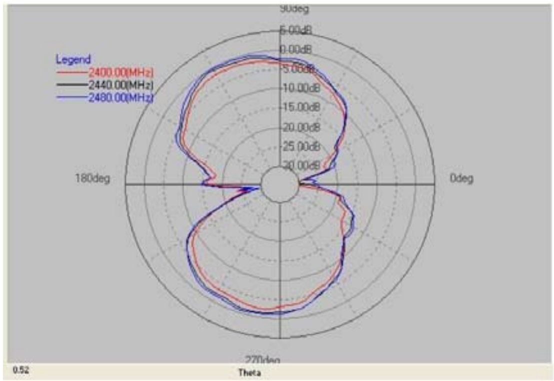


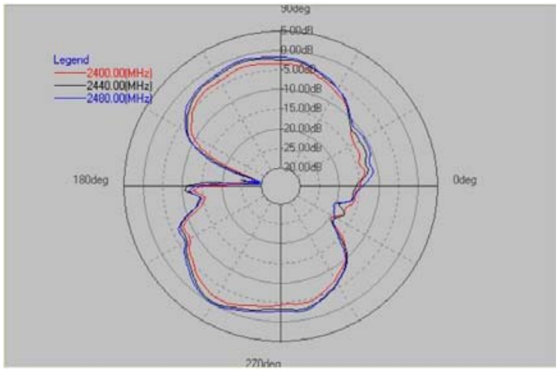
4.3 Effect of the layout to the antenna radiation pattern

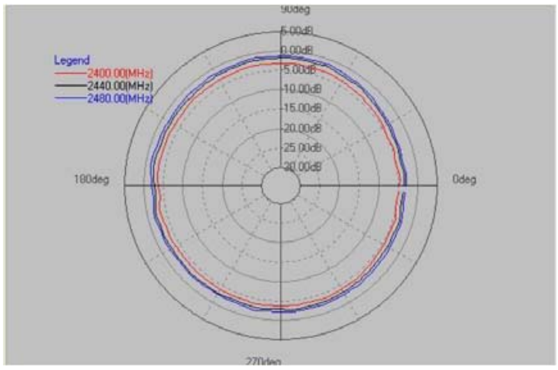
TOTAL EFFICIENCY ~45 %

PEAK GAIN 0dBi



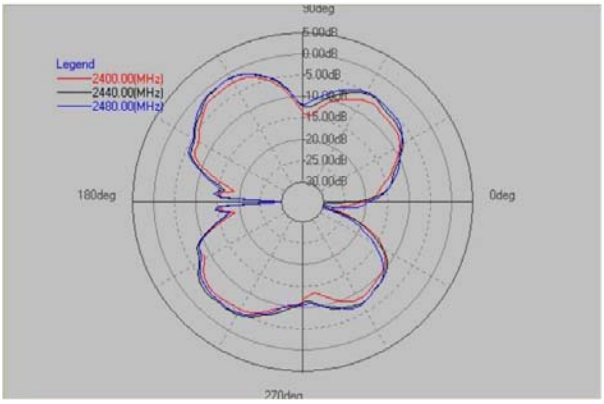


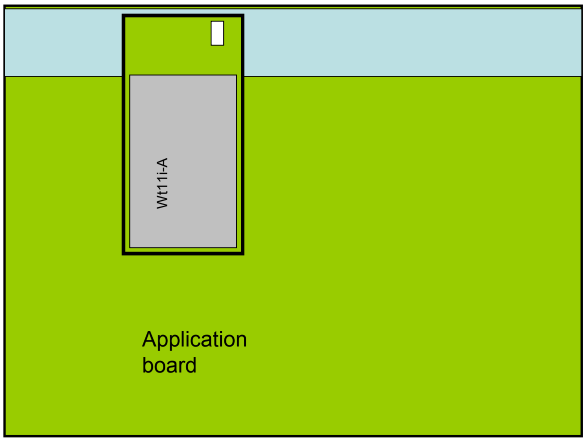


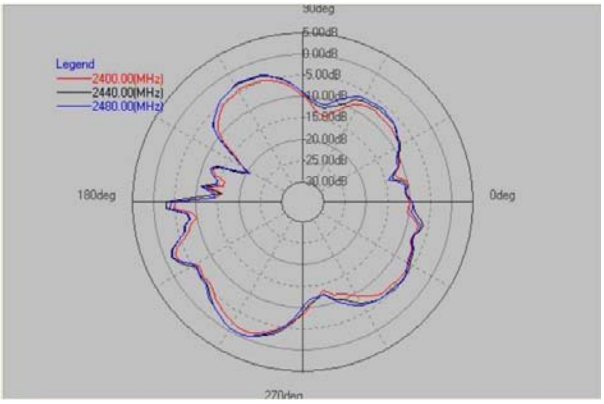


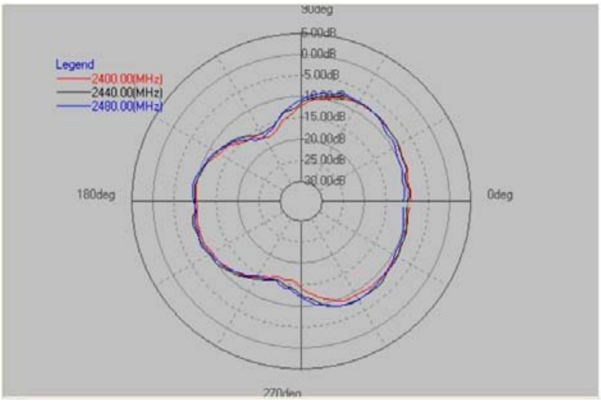
TOTAL EFFICIENCY ~35%

PEAK GAIN 0.5 dBi

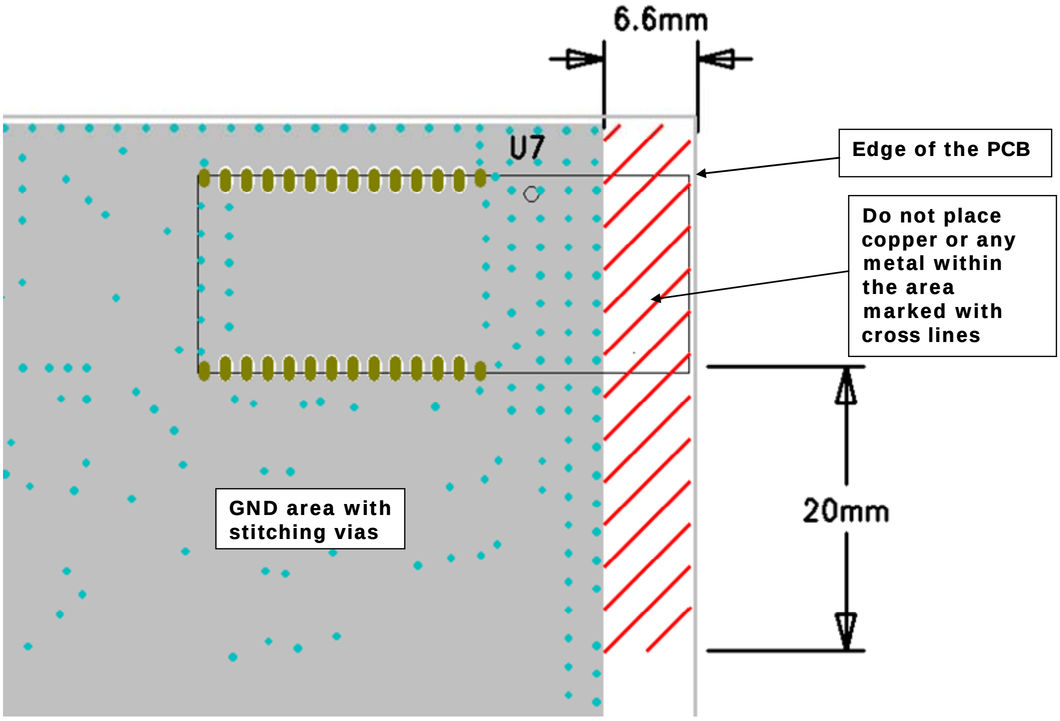








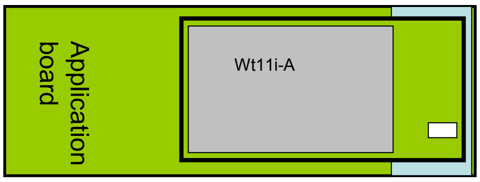
4.4 Recommended layout for WT11i-A

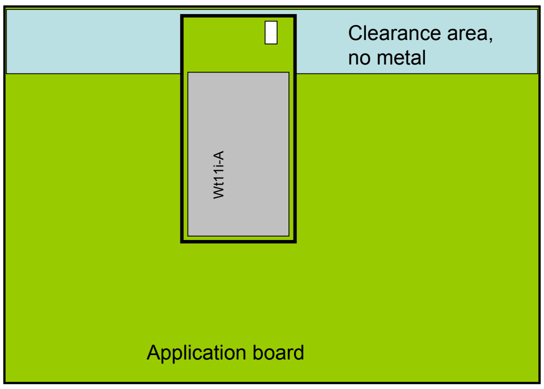


- DO not place any metal within the clearance area marked to figure above
- Connect all the GND pins to a solid GND plane
- If using overlapping GND planes use GND stitching vias separated by max 3 mm to avoid emissions from the edge of the PCB
- Make sure that the return current follows the forward current all the way for all the signals as close as possible. Make sure that the path for the return current (GND) is low impedance.

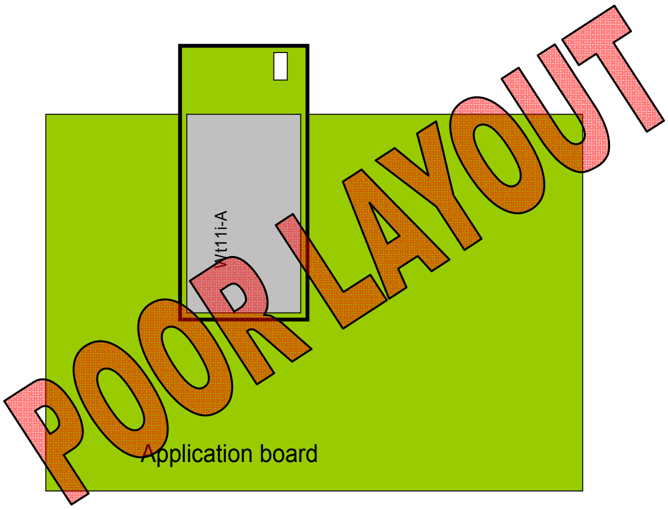
4.5 Layout Examples

4.5.1 Good Layouts for RF

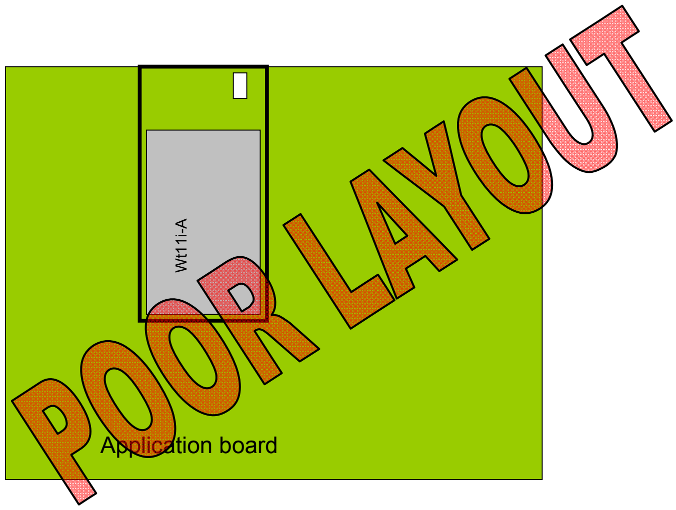




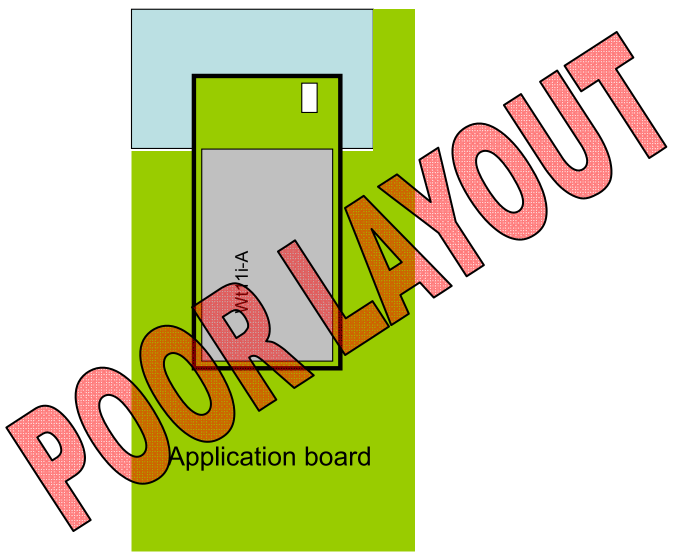
4.5.2 Poor Layouts for RF



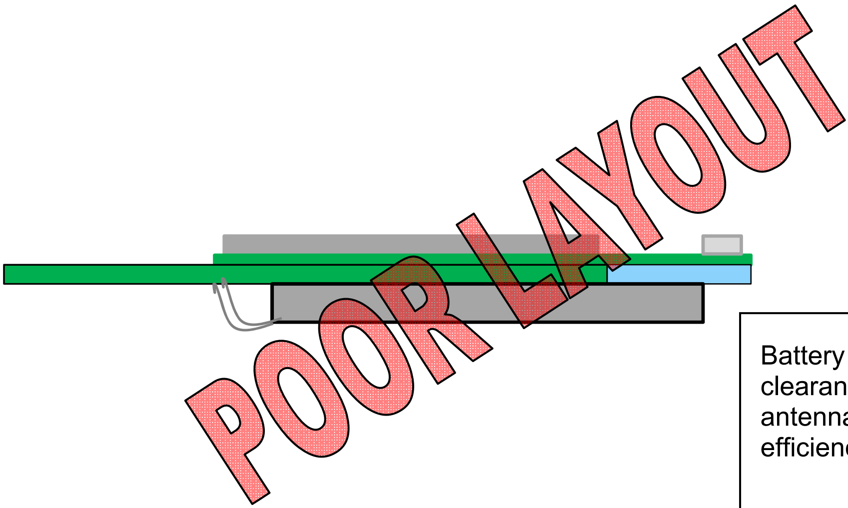
Poor impedance matching for the antenna



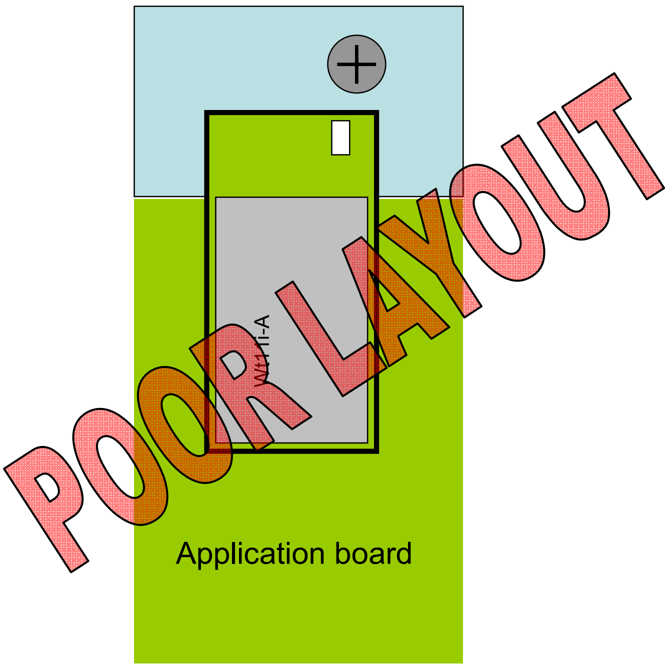
GND plane under the antenna → poor radiation efficiency



GND plane within the clearance area of the antenna→ poor radiation efficiency

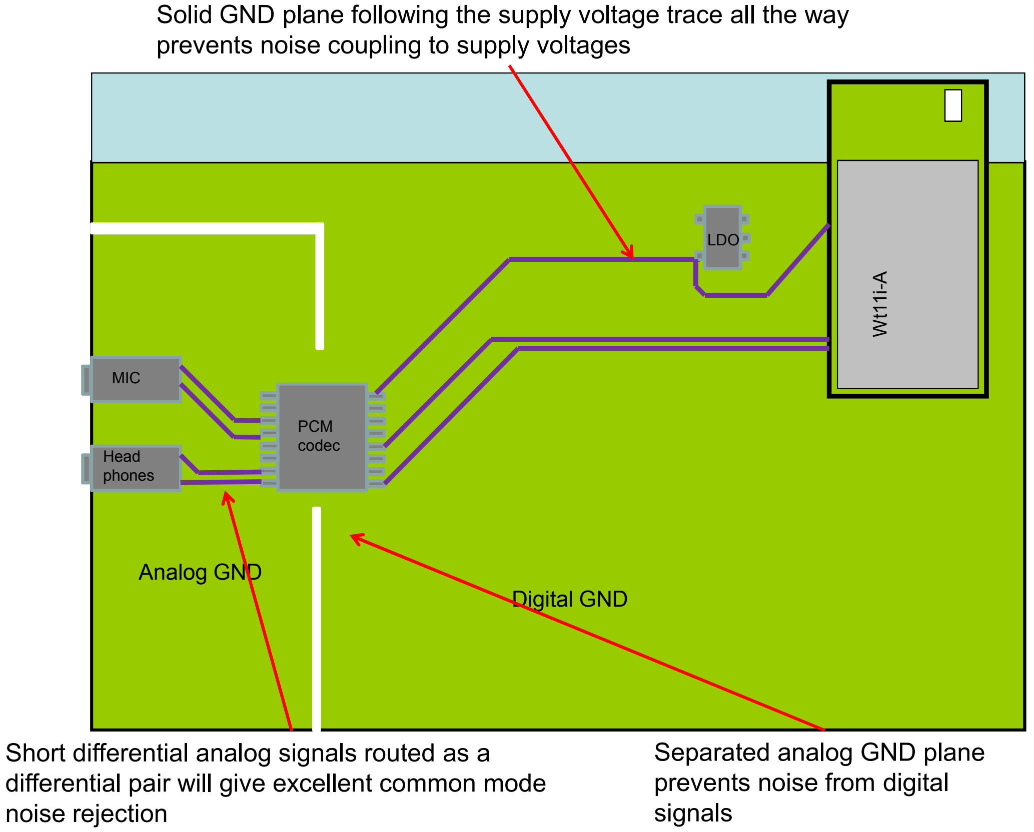


Battery within the clearance area under the antenna → poor radiation efficiency

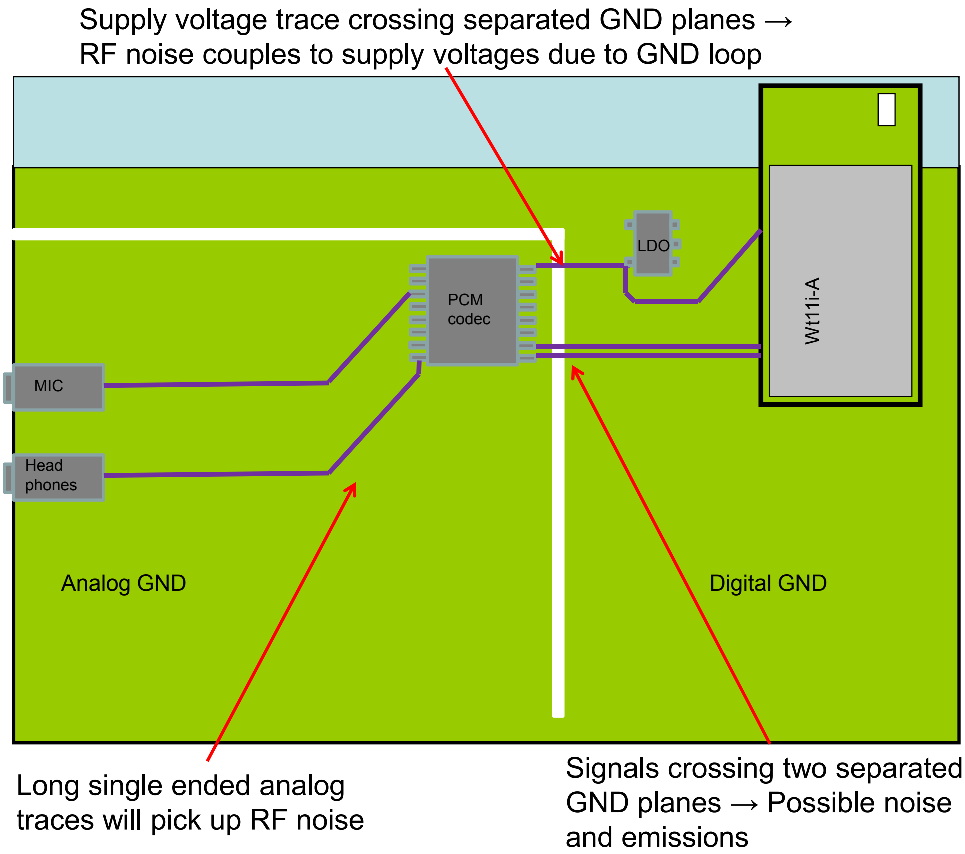


Metal object close to the antenna → poor radiation efficiency

4.5.3 Good Layout for Audio



4.5.4 Poor Layout for Audio



5 LAYOUT GUIDE FOR WT11i-E

With WT11i-E one has a freedom to place the module to place in the mother board as the placement of the module does not have any impact to the RF characteristics of the module or the performance of the antenna. The same rules for avoiding unintentional emissions and noise coupling in the layout apply for Wt11i-E as for WT11i-A. Use good layout practices to avoid loops. Make sure that the return current for any traces in the layout follow the forward current as close as possible and that the return current path is low impedance.

6 EXAMPLE DESIGN

Please refer to WT11 evaluation board

7 HOW TO APPROXIMATE THE RANGE

RF power propagates in free space within a virtual “pipe” which can be defined by so called Fresnel ellipsoid. Any obstacles within the area of this “pipe” will attenuate the RF power and thus decrease the actual range of the link. The radius of the “pipe” can be approximated by

$$R = \sqrt{\frac{D \times \lambda}{12}}$$

Where R is the radius, D is the distance between the antennas and lambda is the wave length.

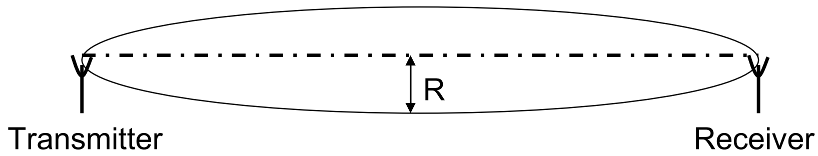


Figure 6: RF propagation area between TX and RX

The free space loss can be approximated by

$$L_P(dB) = 92{,}45 + 20\log F + 20\log D$$

Where F is frequency in GHz and D is Distance in kilometers. This approximation however does not apply to actual case where the signal is reflected from the ground. More realistic approximation can be calculated by

$$\frac{P_R}{P_T} = 2\left(\frac{\lambda}{4\pi r}\right)^2 \left[1 - \cos\left(k\frac{2h_1 h_2}{r}\right)\right]$$

Where h_1 and h_1 the height of the antennas respectively, k is the free space wavenumber and r is the distance between the antennas. The equation is expressed with the blue line in the figure XXX. From the figure one can see that at Bluetooth frequencies simple approximation -20dB/decade can be used in free space and -40dB/decade once the ground starts to dominate the power loss. The distance where the ground starts to effect can be calculated by

$$d_m = \frac{(12 \times h_1 \times h_2)}{\lambda}.$$

The total range can be approximated once the output power from the antenna (transmitter output power + antenna gain) and the receiver sensitivity (receiver sensitivity + antenna gain) is defined. As an example using antenna heights 1 m, 2 m and 3 m, TX power 16 dBm and receiver sensitivity -90dBm one can approximate the total ranges assuming an open field without obstacles within the RF path.

h = 1 m → R = 445 m

h = 2 m → R = 890 m

h = 3 m → R = 1300 m

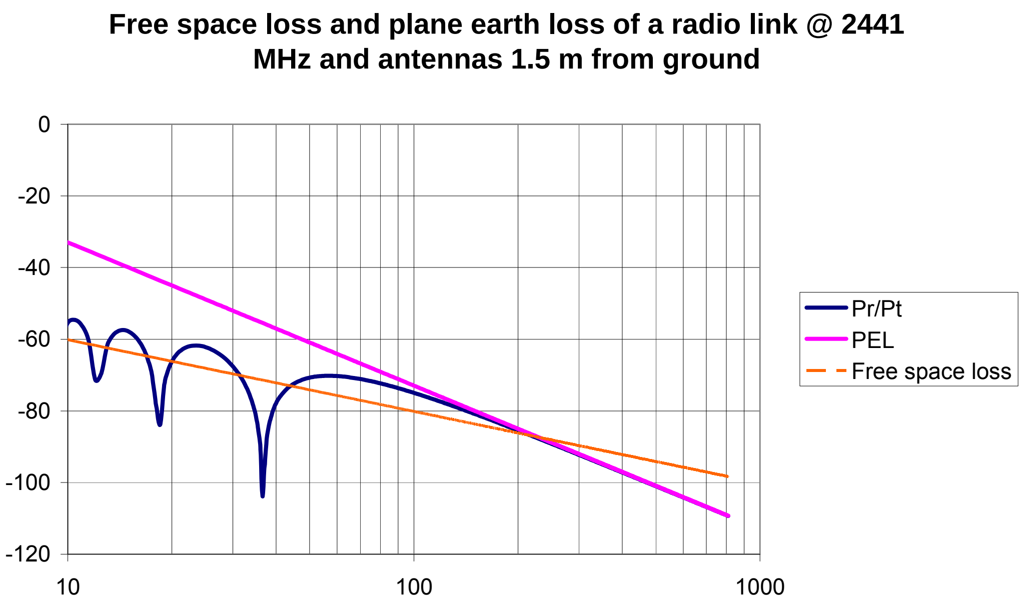


Figure 7: Calculated free space loss and plane earth loss at 2441 MHz with antennas at 1,5m from the ground

8 CONTACT INFORMATION

Sales: sales@bluegiga.com

Technical support: support@bluegiga.com
http://techforum.bluegiga.com

Orders: orders@bluegiga.com

Head Office / Finland:

Phone: +358-9-4355 060
Fax: +358-9-4355 0660

Street Address: Sinikalliontie 11
02630 ESPOO
FINLAND

Postal address: P.O. BOX 120
02631 ESPOO, FINLAND

Sales Office / USA:

Phone: (781) 556-1039

Street address: Bluegiga Technologies, Inc.
99 Derby Street, Suite 200
Hingham, MA 02043