AN4819
Application note

PCB design guidelines for the BlueNRG-1 device

Introduction

The BlueNRG1 is a very low power Bluetooth low energy (BLE) single-mode system-on-chip compliant with Bluetooth® specification v4.1. The BlueNRG-1 can act as master or slave. It embeds a 2.4 GHz RF transceiver, Cortex M0 microcontroller and a by-passable DC-DC step-down converter to achieve optimum power consumption.

The BlueNRG-1 device is provided in QFN32 pins package.

ST provides all necessary source files (reference designs) for customers that want to speed-up their developing.

This application note is intended to accompany the reference designs of the QFN32 pins application board and provide detailed information regarding the design decisions employed within STMicroelectronics designs. In addition, it details the design guidelines for developing a generic radio frequency application using a BlueNRG-1 device.

The RF performance and the critical maximum peak voltage, spurious and harmonic emission, receiver matching strongly depend on the PCB layout as well as the selection of the matching network components.

For optimal performance, STMicroelectronics recommends the use of the PCB layout design hints described in the following sections. Also, but not less important, STMicroelectronics strongly suggest to use the BOM defined in the reference design, BOM that guarantee, with a good PCB design, the correct RF performance.

For further information, visit the STMicroelectronics web site at *www.st.com*.

1 Reference schematics

Different application boards were developed to show the BlueNRG-1 device functionality. The schematics of the different application boards are reported in the next pictures and refer to the different possible combinations:

1. SMD discrete balun, DC-DC converter ON ();
2. SMD discrete balun, DC-DC converter OFF ();
3. Integrated balun, DC-DC converter ON ();

All the layout guidelines described in the next paragraphs have to be applied to all these application boards.

Figure 1: SMD discrete balun, DC-DC converter ON

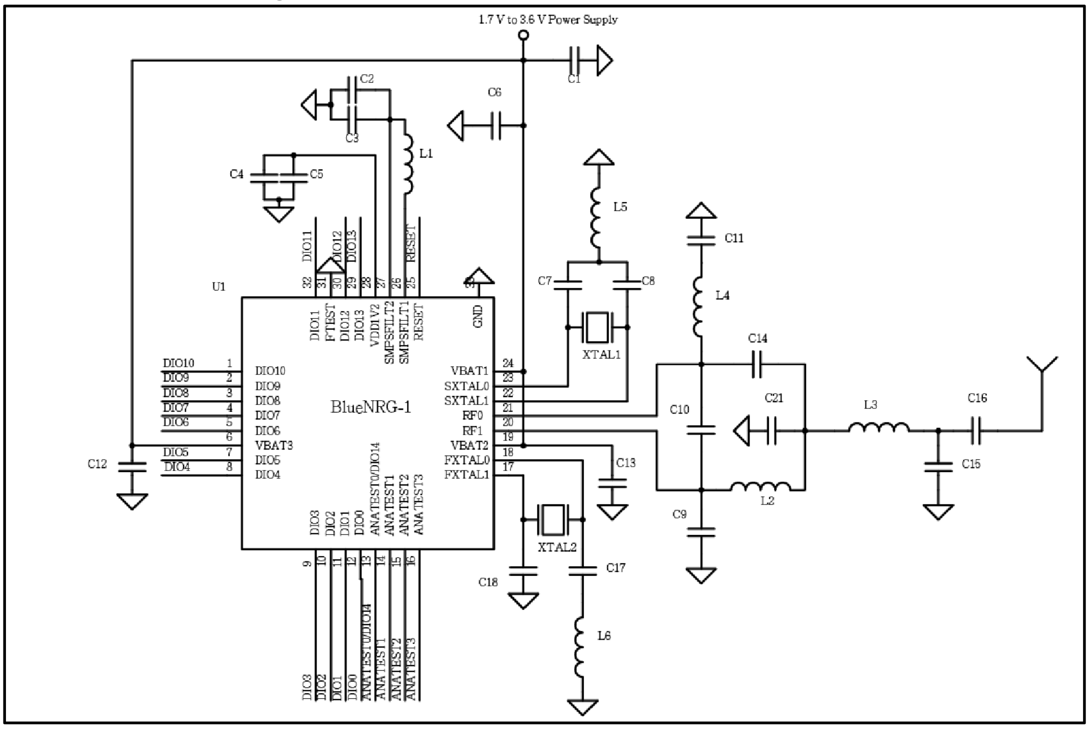


Figure 2: SMD discrete balun, DC-DC converter OFF

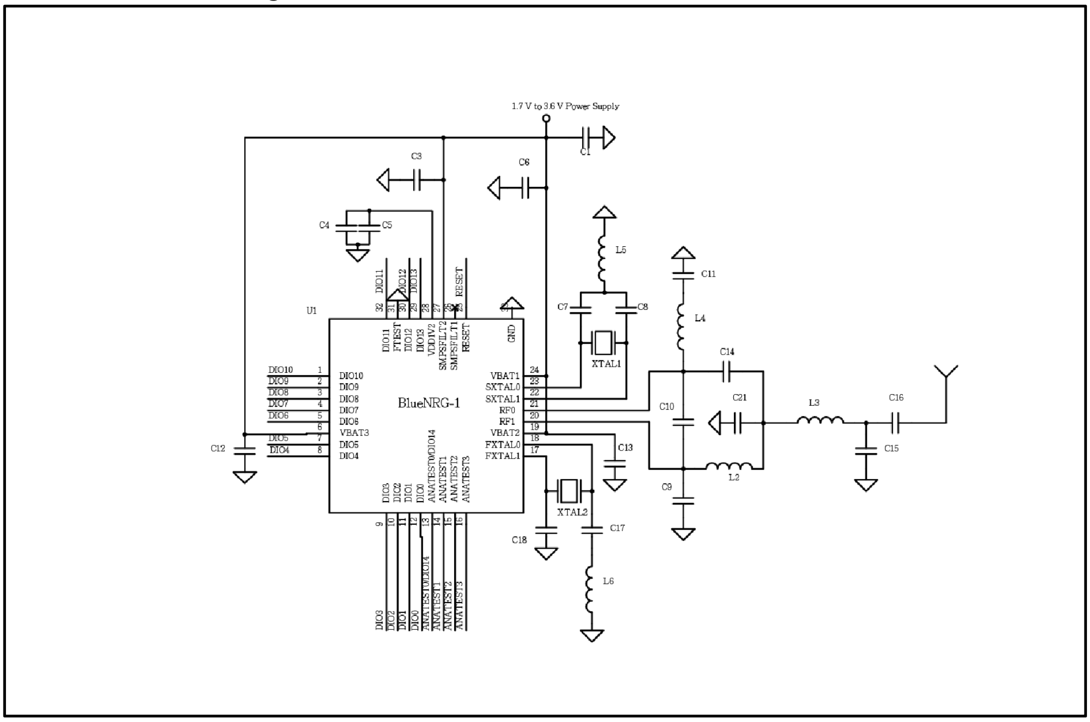


Figure 3: Integrated balun, DC-DC converter ON

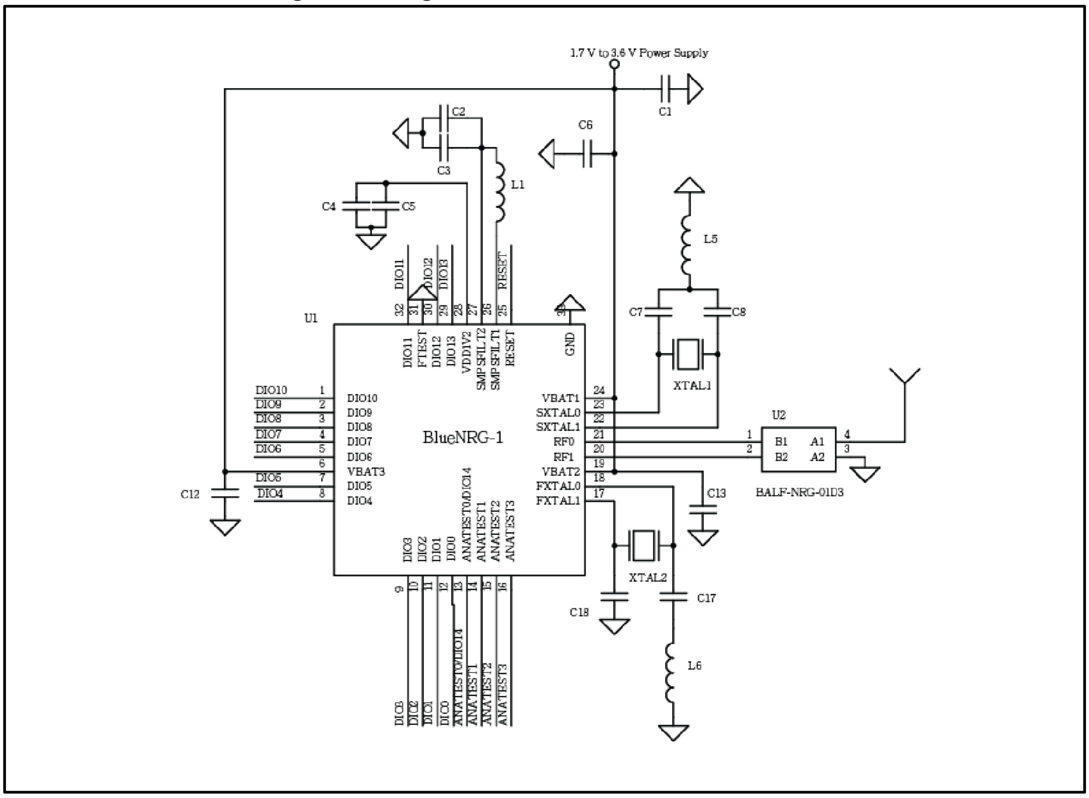


Table 1: BlueNRG-1 application board external components description

Components	Description
C1, C6, C12	Decoupling capacitors for battery voltage
C2, C3	DC-DC converter filtering capacitors
C4, C5	Decoupling capacitor for on-chip 1.2 V voltage regulator
C9, C10, C11, C14, C15, C16, C21	RF discrete balun filter/matching capacitors
C7, C8	XTAL1 capacitors
C17, C18	XTAL2 capacitors
L1	DC-DC converter inductor
L2, L3, L4	RF discrete balun filter/matching inductors
L5	XTAL1 filtering inductor
L6	XTAL2 filtering inductor
XTAL1	Low frequency crystal
XTAL2	High frequency crystal
U1	BlueNRG-1 device
U2	Integrated balun

2 Components dimensioning

The chosen of the external components is very important for correct application functionality. In the next paragraph the description of the main components, their functionality and how to choose them is described.

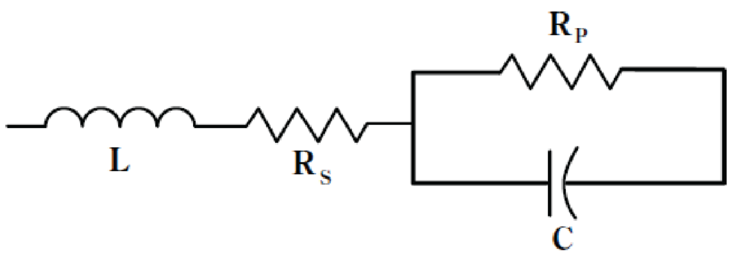
2.1 Capacitors

A capacitor is a passive electrical component used to store energy in an electrical field. The forms of practical capacitors vary widely, but all contain at least two electrical conductors separated by a dielectric.

Capacitors differ from each other for construction techniques and materials used to manufacture. A lot of different types of capacitors exist (double-layer, polyester, polypropylene and so on), but this document will focus on the surface mount versions of ceramics only. The other types of capacitors are not indicated for characteristic or cost for the application targeted in this document.

A capacitor, as a practical device, exhibits not only capacitance but also resistance and inductance. A simplified schematic for the equivalent circuit is shown in the figure 4.

Figure 4: Capacitor equivalent circuit



Typically for the capacitors are defined the ESR (equivalent series resistance) and the ESL (equivalent series inductance). The term ESR combines all losses both series and parallel in a capacitor at a given frequency so that the equivalent circuit is reduced to a simple R-C series connection. Same considerations for the ESL that is the equivalent series inductor comprised of three components: pad layout, capacitor height and power plane spreading inductance.

The main differences between ceramic dielectric types are the temperature coefficient of capacitance and the dielectric loss. COG and NP0 (negative-positive-zero, i.e ±0) dielectrics have the lowest losses and are used in filtering, matching and so on.

For RF parts it is generally recommended that multilayer (or monolithic) ceramic capacitors with a COG dielectric material, which is a highly stable class I dielectric offering a linear temperature coefficient, low loss and stable electrical properties over time, voltage and frequency.

For RF decoupling purposes select a capacitor value such that for the frequency to be decoupled is close to or just above the series resonant frequency (SRF) of the capacitor. At SRF the parasitic impedance resonates with the device capacitance to form a series tuned circuit and the impedance presented by the capacitor is the effective series resistance (ESR).

For DC blocking or coupling applications at RF, typically a capacitor with low insertion loss and a good quality factor is required. Since a capacitor's quality factor is inversely proportional to its ESR, select a capacitor with a low ESR and ensure that the SRF of the capacitor is greater than the frequency of operation. If the working frequency is above the SRF of the capacitor, it will appear inductive.

All the capacitors of the BlueNRG-1 application board used for the matching network and for the crystals have to be COG.

2.2 Inductors

An inductor is a passive electrical component used to store energy in its magnetic field. Any conductor has inductance. An inductor is typically made of a wire or other conductor wound into a coil, to increase the magnetic field.

Inductors differ from each other for construction techniques and materials used to manufacture. A lot of different types of inductors exist (air core inductor, ferromagnetic core inductor and variable inductor), but this document will focus on the inductors useful for RF only. Usually in RF the air core inductors are used. The term air core describes an inductor that does not use a magnetic core made of ferromagnetic material, but coil wound on plastic, ceramic, or other nonmagnetic form. They are lower inductance than ferromagnetic core coils, but are used at high frequencies because they are free from energy losses called core losses.

Usually the real circuit of an inductor is composed of a series resistance and a parallel capacitor. The parallel capacitor is considered to be the inter-winding capacitance that exists the turns of the inductor. If the inductor is placed over a ground plane then this capacitance will also include the capacitance that exists between the inductor and the ground plane. The series resistor can be considered as the resistance of the inductor winding.

In term of circuit performance, as already mentioned for the capacitors, the self-resonant frequency and the quality factor are the main inductor parameters, especially for the circuit where the losses need to be minimized. At the self-resonant frequency, the inductor impedance is at maximum. For frequency above the self-resonance the inductor behavior change and it will appear capacitive.

In general wirewound inductors have a higher quality factor than a multilayer equivalent. They will also reflect and radiate more energy which can give rise to higher emission levels, especially in term of self-coupling. Inductive coupling can give rise to undesired circuit operation: to minimize coupling mount the inductors in sensitive circuit areas at 90 degrees to one another.

In the BlueNRG-1 application board two different inductor types are used:

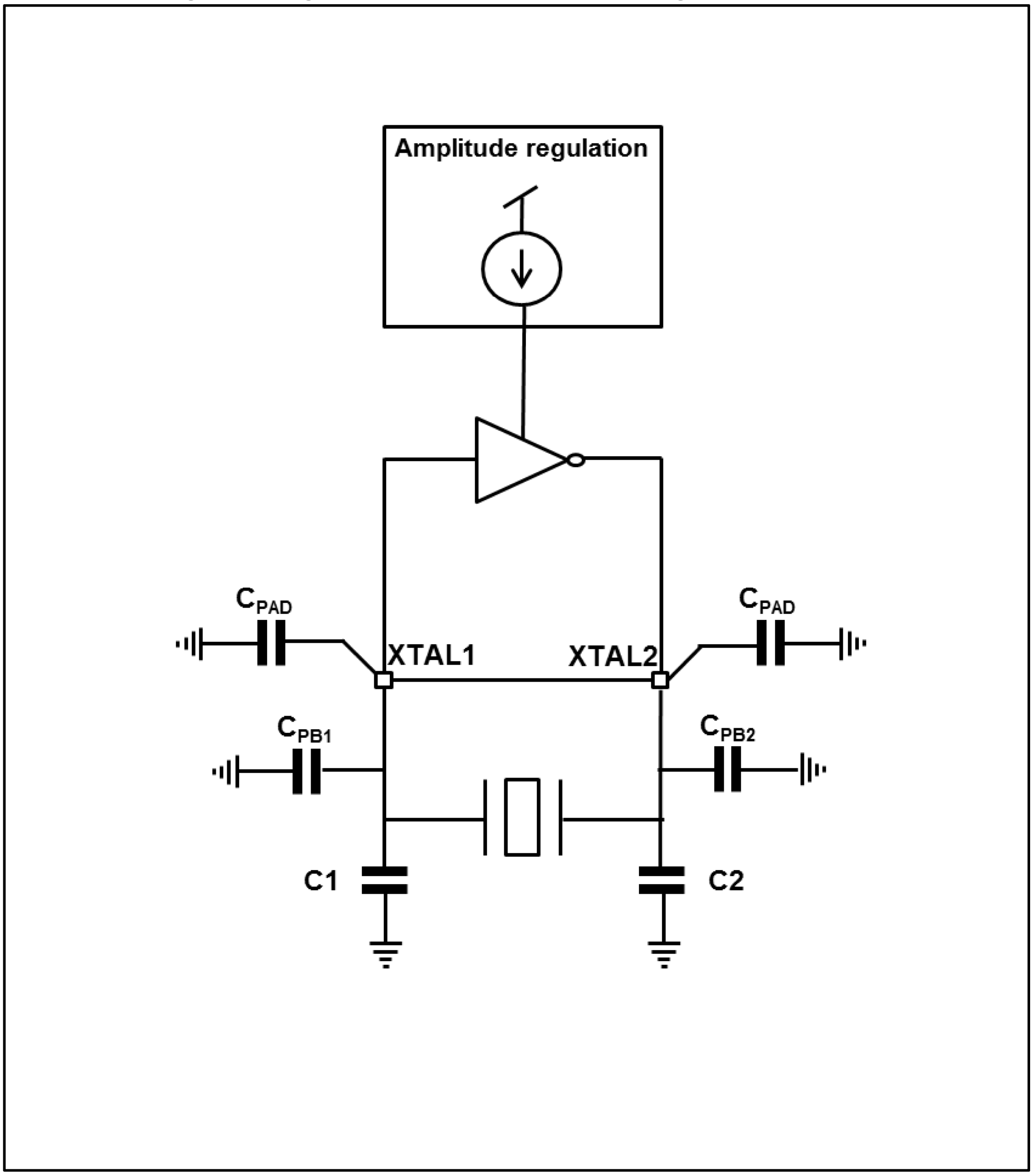
1. DC-DC converter coil: the nominal value is 10 uH, a 4.7 uH can be used. The DCR has to be less than 1 ohm, the rated current has to be higher than 100 mA.
2. RF matching and filtering coil: in this case the best solutions are the high Q coils, but a good compromise between application cost versus RF performances is to choose an inductor with a medium Q.

2.3 External quartz

The BlueNRG-1 includes a high frequency and a low frequency integrated oscillators that required two external crystals.

The BlueNRG-1 includes a fully integrated, low power 16/32 MHz Xtal oscillator with an embedded amplitude regulation loop. In order to achieve low power operation and good frequency stability of the Xtal oscillator, certain considerations with respect to the quartz load capacitance C0 need to be taken into account. *Figure 5: "Diagram of the BlueNRG amplitude regulated oscillator"* shows a simplified block diagram of the amplitude regulated oscillator used on the BlueNRG.

Figure 5: Diagram of the BlueNRG amplitude regulated oscillator



Low power consumption and fast startup time is achieved by choosing a quartz crystal with a low load capacitance C0. A reasonable choice for capacitor C0 is 15 pF. To achieve good frequency stability, the following equation needs to be satisfied:

$$C_0 = \frac{C_1' * C_2'}{C_1' + C_2'}$$

Where C1'=C1+C_{PCB1}+C_{PAD}, C_2'= C_2+C_{PCB2}+C_{PAD}, where C_1 and C_2 are external (SMD) components, C_{PCB1} and C_{PCB2} are PCB routing parasites and C_{PAD} is the equivalent small-signal pad-capacitance. The value of C_{PAD} is around 0.5 pF for each pad. The routing parasites should be minimized by placing quartz and C_1/C_2 capacitors close to the chip, not only for an easier matching of the load capacitance C_0, but also to ensure robustness against noise injection. Connect each capacitor of the Xtal oscillator to ground by a separate via.

Regarding the low frequency crystal oscillator the same consideration has to be done.

It is important to underline that the BlueNRG-1 integrates an internal low frequency RC oscillator that can be used without external quartz. The customer can choose to use the internal or the external one. The BlueNRG integrates also an internal high frequency RC oscillator, but it is disabling after an initial system bootstrap and it is necessary to use external quartz for radio operations.

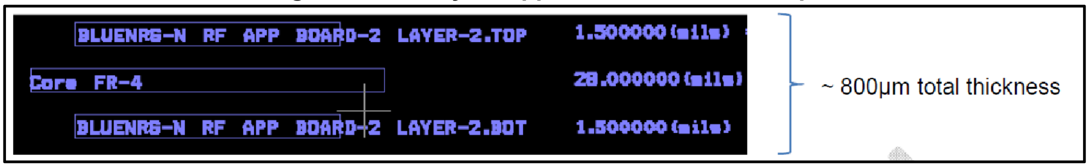
3 Two or four layers application board

Different approach can be taken when an application board is designed.

3.1 Two layers solution

When it is possible to route all the tracks on two layers and a cheaper solution is requested, a two layers application board can be designed.

Figure 6: Two layers application board stack-up



The suggested thickness of the board is about 800 µm.

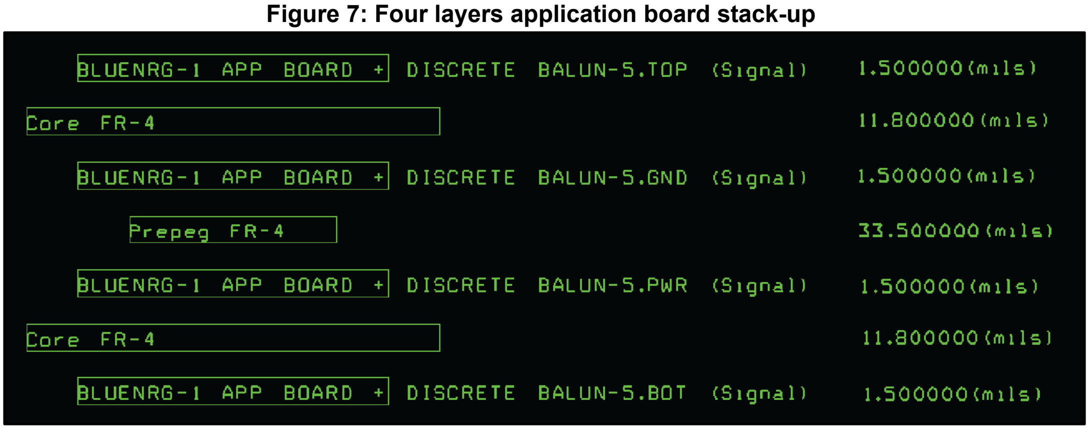
The two layers have to be so distributed:

1. TOP layer: used for RF signal and routing.
2. BOTTOM layer: used for grounding under the RF zones and for routing in the other part.

3.2 Four layers solution

When it is not possible to route all the tracks on two layers and/or a cheaper solution is not requested, a four layers application board can be designed, see *Figure 7: "Four layers application board stack-up"*.

Figure 7: Four layers application board stack-up



When a four layers solution is used the suggested thickness between the TOP layer (RF part) and the INNER1 layer is 300 µm.

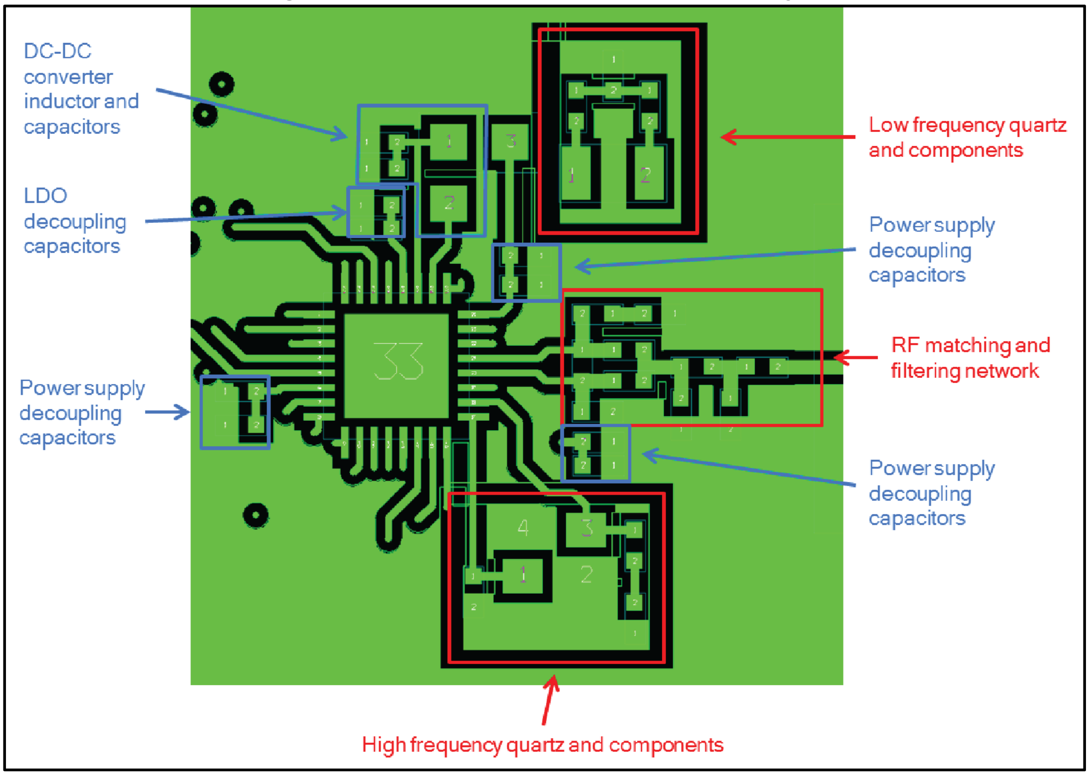
The four layers have to be so distributed:

1. TOP layer: used for RF signal and routing.
2. INNER1 layer (GND): used for grounding under the RF zones and for routing in the other part.
3. INNER2 layer (PWR): used for power and low frequency routing
4. BOTTOM layer: used for low frequency routing

4 Layout recommendation

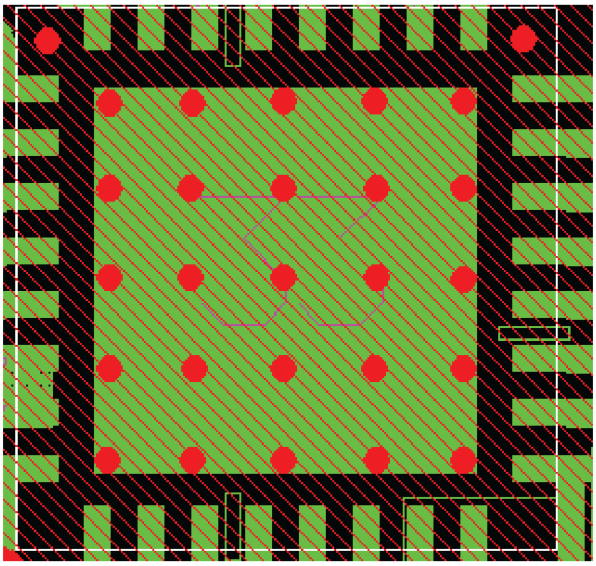
The application board TOP layer layout using the BLueNRG-1 is shown in *Figure 8: "BlueNRG-1 application board TOP layer"*.

Figure 8: BlueNRG-1 application board TOP layer



It is very important to connect very well the ground of the exposed pad of the QFN32 to the ground of the application board. So a lot of vias are necessary to be sure that the parasitic inductor introduced from each via is negligible.

Figure 9: Vias on the exposed pad of the QFN32 package



The ground of the two external crystals has to be isolated from the ground of the RF part of the board. This is because the RF ground is “dirty” and this signal can disturbs the correctly functionality of the two crystals.

Also to reduce the coupling effects some cunning have to be taken:

1. In the high frequency crystal (XTAL2) the load capacitor of the FXTAL0, pin 18, has to be connected to ground in series with an inductor (see *Figure 10: "High frequency crystal inductor"*);
2. In the low frequency crystal (XTAL1) the ground parts of two load capacitors have to be connected together and, after, connected to the ground by an inductor.
3. The two tracks that connect the low frequency crystal to the SXTAL0 and SXTAL1, pins 23 and 22, have to be put in layer different from the TOP.

Figure 10: High frequency crystal inductor

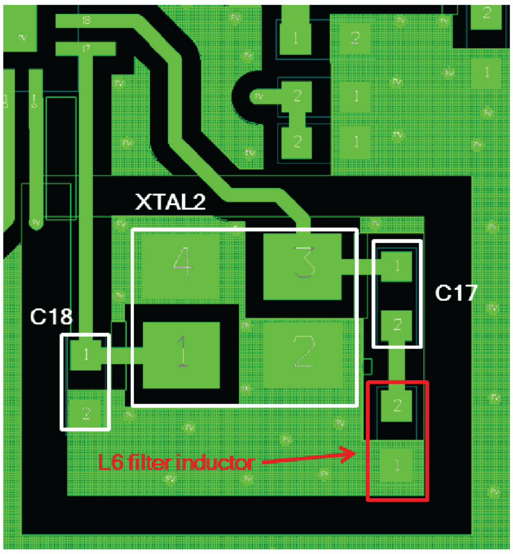
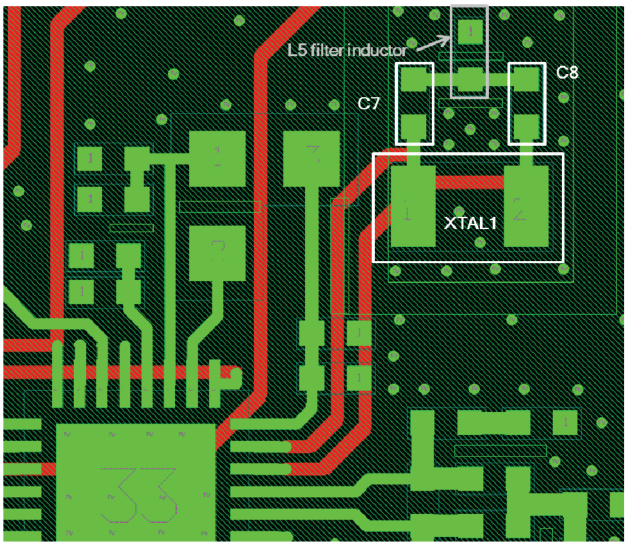


Figure 11: Low frequency crystal inductor and tracks



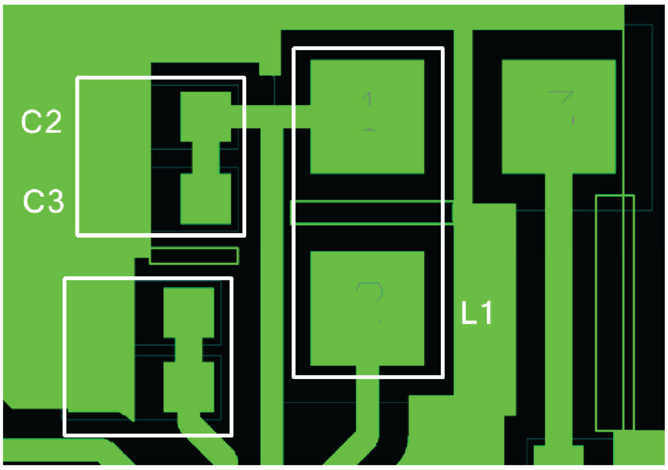
The DC-DC converter area is very sensitive and it is necessary to pay attention on the layout of this part. This is because the DC-DC converter generates ground noise that can get coupled on surrounding ground reducing the sensitivity and high frequency components can be coupled onto RF part.

So to ensure a correct layout it is necessary of:

1. Providing efficient filtering by placing capacitors as close as possible from the BlueNRG;
2. Reducing parasitic ensuring wide and short connections to BlueNRG.

In *Figure 12: "DC-DC converter layout zone"* the suggested layout is shown:

Figure 12: DC-DC converter layout zone



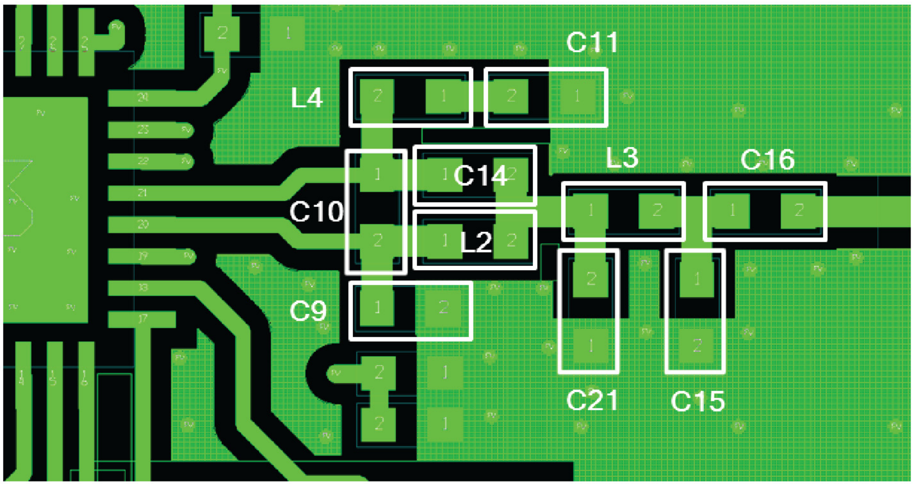
Special care has to be taken in the placement of the supply voltage filtering capacitors. It is in fact important to ensure efficient filtering placing these capacitors as close as possible from their dedicated pins on the BlueNRG.

The TX/RX part of the BlueNRG is a very sensitive part. The discrete balun has to be placed as close as possible to the TX/RX pins. The traces that connect the RF pins to the balun network (differential trace) should be of equal length. If the two differential signals are un-balanced, common-mode issues can be generated. The differential traces have to be routed closely together. Differential receivers are designed to be sensitive to the difference between a pair of inputs, but also to be insensitive to a common-mode shift of those input. Therefore, if any external noise is coupled equally into the differential traces, the receiver will be insensitive to this (common mode coupled) noise. More closely differential traces are routed together, more equal will any coupled noise be on each trace, therefore better will be the rejection of the noise in the circuit.

The parallel inductors in the balun (and in general) should be mutually perpendicular to avoid mutual couplings. If no perpendicular position is possible, turn away their interposing capacitors or resistors.

The interconnections between the elements are not considered transmission lines because their lengths are much shorter than the wavelength and, thus, their impedance is not critical. As results, their recommended width is smallest possible. In this way, the parasitic capacitances to ground can be minimized.

Figure 13: Discrete balun layout zone



An application board using an integrated balun was designed also. The integrated balun was developed internally to STMicroelectronics and can be used only with the BlueNRG-1 device. It is absolutely necessary to follow the layout rules described in the balun datasheet (BALF-NRG-01D3).

5 Reference

1. BlueNRG-1 Datasheet Rev 1
2. BALF-NRG-01D3 Datasheet Rev 3

6 Revision history

Table 2: Document revision history

Date	Version	Changes
30-Jun-2016	1	Initial release.