AN5948

Application note

How to develop RF hardware using STM32WBA MCUs

Introduction

STM32WBA microcontrollers integrate a high quality RF transceiver for Bluetooth® LE and 802.15.4 radio solution.

Special care is required for the layout of an RF board compared to a conventional circuit.

At high frequencies, copper interconnections (traces) behave as functional circuit elements introducing disturbances that can degrade RF performance. Parasitic components created by traces and pads contribute significantly to the overall circuit behavior. Layout rules must be carefully followed to mitigate these effects and achieve the requested performance.

This document describes the precautions to be taken to achieve the best performance from the MCU. The guidelines in this document are reflected in the implementation of the ST reference designs for these products.

Table 1. Evaluation boards with STM32WBA microcontrollers

Reference design (1)(2)	Microcontroller	Package
NUCLEO-WBA55CG	STM32WBA55CG	UFQFPN48
NUCLEO-WBA65RI	STM32WBA65RI	VFQFPN68
NUCLEO-WBA52CG	STM32WBA52CG	UFQFPN48

1. *Check www.st.com for availability.*
2. *Refer to www.st.com for datasheets of the microcontrollers and user manuals of the evaluation boards.*

The guidelines in this document are generic and must be adapted to the specific application. Unless specified otherwise, the proposed components for an STM32WBA5 device also apply to STM32WBA6.

Table 2. Applicable products

Type	Product lines
Microcontroller	STM32WBA52, STM32WBA62 (essential features)
	STM32WBA54/55, STM32WBA64/65 (advanced features)
	STM32WBA55, STM32WBA63 (cross-over)

1 General information

The STM32WBA microcontrollers are based on Arm® cores.

Note: Arm is a registered trademark of Arm Limited (or its subsidiaries) in the US and/or elsewhere.

Reference documents

[1] Datasheet *Multiprotocol wireless 32-bit MCU Arm®-based Cortex®-M33 with TrustZone®, FPU, Bluetooth® 5.4 and IEEE 802.15.4 radio solution* (DS14127)

[2] Datasheet *Multiprotocol wireless 32-bit MCU Arm®-based Cortex®-M33 with TrustZone®, FPU, Bluetooth® LE, IEEE802.15.4 radio solution* (DS14736)

[3] Application note *Guidelines for oscillator design on STM8AF/AL/S and STM32 MCUs/MPUs* (AN2867)

[4] Technical Note *Antenna radiation patterns of module STM32WBA5MMG* (TN1565)

[5] Datasheet *Matched low-pass filter for STM32WBA MLPF-WB-04D3* (DS14446)

2 RF basics

This section covers generic terms and definitions used in RF board design.

Refer to the following references for additional information.

1	Paul Horowitz and Winfield Hill	The art of electronics (3^{rd} edition)	Cambridge University Press
2	Roger C. Palmer	An introduction to RF circuit design for communication systems (2^{nd} edition)	Newnes
3	Christopher Bowick	RF circuit design (2^{nd} edition)	Newnes
4	Joseph J. Carr and George W. Hippisley	Practical antenna handbook (5^{th} edition)	McGraw-Hill Education
5	Smith chart (free SW)		http://www.fritz.dellsperger.net
6	Coplanar waveguide calculator (free SW)		http://wcalc.sourceforge.net

2.1 Terminology

2.1.1 Power

This unit, expressed in dBm, is the measure of the RF signal strength: **dBm = 10 Log P**, where P is the power in mW:

- 1 pW = -90 dBm
- 10 µW = -20 dBm
- 1 mW = 0 dBm
- 2 mW = 3 dBm
- 10 mW = 10 dBm

2.1.2 Gain

The gain (expressed in dB) is the ratio between the output and the input power of an RF device. Negative values correspond to attenuation.

2.1.3 Loss

If there is impedance mismatch, incorrect transmission line design or incorrect PCB material selection between two stages of a circuit, signal power losses appear and not all the power is transmitted from one stage to the following one. There are also inherent losses, for example, the dielectric loss, which depends upon the laminate and materials used to manufacture the board.

2.1.4 Reflection coefficient, voltage standing wave ratio, and return loss

When a signal flows from a source to a load via a transmission line, a mismatch between the characteristic impedance of the transmission line and the load results in a portion of the signal being reflected back to the source. The polarity and the magnitude of the reflected signal depend on whether the load impedance is higher or lower than that of the line.

The reflection coefficient (Γ) is the measure of the amplitude of the reflected wave versus the amplitude of the incident wave, namely $\mathbf{\Gamma = (Z - Z_0) / (Z + Z_0) = (z} - 1) / (z + 1)$.

The voltage standing wave ratio (VSWR) is the measure of the accuracy of the impedance matching at the point of connection. It is a function of the reflection coefficient and is expressed as $\mathbf{VSWR = (1 + |\Gamma|) / (1 - |\Gamma|)}$. If VSWR is 1, there is no reflected power.

The return loss (RL) is a function of the reflection coefficient, but expressed in dB:

$\mathbf{RL = 20 \log |\Gamma|}$.

2.1.5 Harmonics

The harmonics are the integer unwanted multiples of input frequency (fundamental frequency).

2.1.6 Spurious

The spurious are the noninteger multiples of input frequency (unwanted frequencies).

2.1.7 Intermodulation

When two RF signals are mixed together, intermodulation products are the signals composed by an integer multiple of the sum and the difference between the two signals.

2.2 Impedance matching

To optimize the RF performance, it is imperative to adapt the impedance matching from the antenna to the input of the chip, as well as that from the chip output to the antenna.

A poor adaptation introduces losses in the RX/TX chain. These losses immediately translate in lower sensitivity and in lower signal amplitude of the transmitted signal. These disadaptations, if high enough, increase the level of TX harmonics.

As a consequence it is very important to spend efforts to adapt as best as possible the RF chain. In the Bluetooth® LE bandwidth of the STM32WBA, and more generally in RF frequencies, spurious elements (such as PCB track inductances and layer capacitors, trace length) have a significant impact on the impedance matching. To achieve the best TX/RX budget (optimum transfer of signal and energy) between the load and the STM32WBA, a dedicated matching network is needed between the two blocks.

The maximum power is transferred when the internal resistance of the source equals the resistance of the load. When extended to a circuit with a frequency-dependent signal, to obtain maximum power transfer the load impedance must be the complex conjugate of the source impedance.

2.3 Smith chart

The Smith chart (Figure 1) is used to determine the matching network.

2.3.1 Normalized impedance

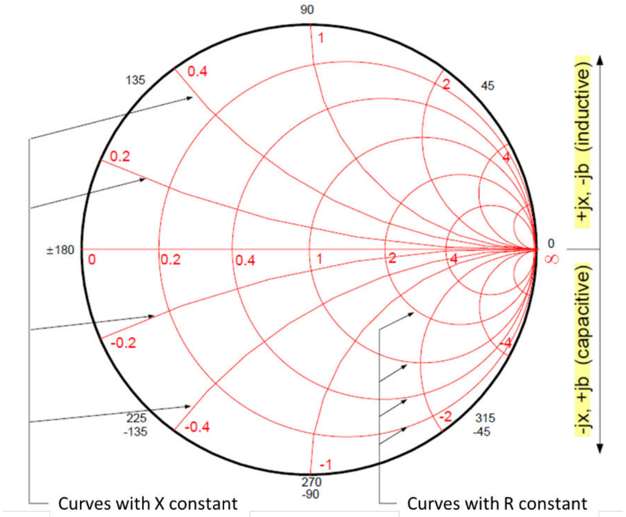
The normalized impedance z is a complex impedance (r is the real part and x the imaginary part): $\mathbf{z = r + jx = Z / Z_0}$ where Z_0 is the characteristic impedance and is often a constant (in our case $Z_0 = 50\ \Omega$).

For a capacitor $z = -j / (2\pi * f * C * Z_0)$, for an inductor $z = j (2\pi * f * L) / Z_0$.

2.3.2 Reading a Smith chart

The Smith chart is represented with the normalized impedance scale $z = Z / Z_0$.

Figure 1. Smith chart



If Z_0 = 50 Ω, when there is matching ($Z = Z_0$) the normalized impedance at 50 Ω is 1 and it is the center of the Smith chart. The goal in the search of a matching network is to converge towards this point.

The horizontal axis of the Smith chart represents pure resistors: at the left side, z = 0 (short circuit) and at the right side, z = ∞.

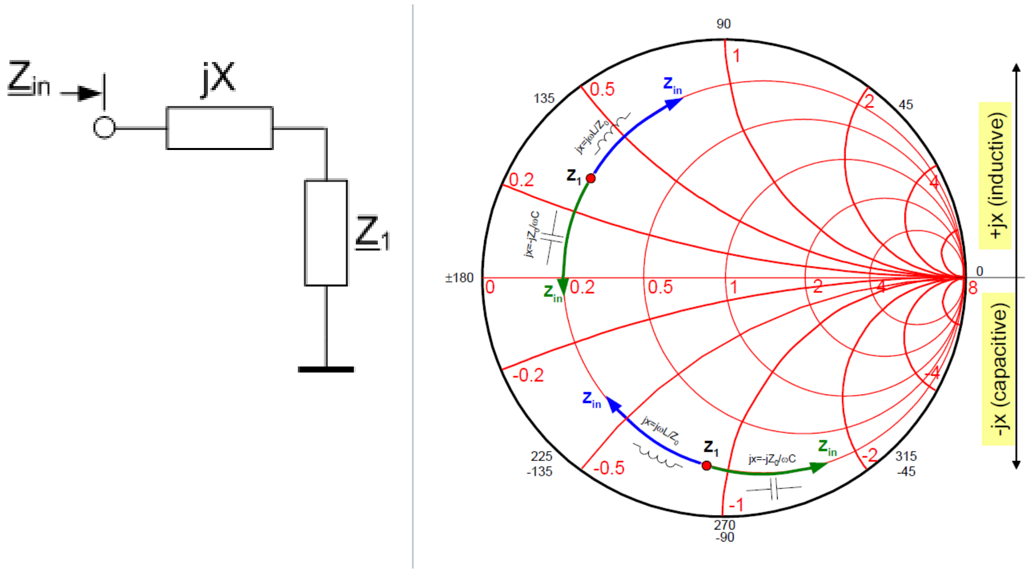
The region located above the X axis represents impedances with inductive reactance (positive imaginary part of the complex impedance) or capacitive susceptance (positive imaginary part of the complex admittance).

The region below the X axis represents impedances with capacitive reactance (negative imaginary part of the complex impedance) or inductive susceptance (negative imaginary part of the complex admittance).

Serial inductor or capacitor

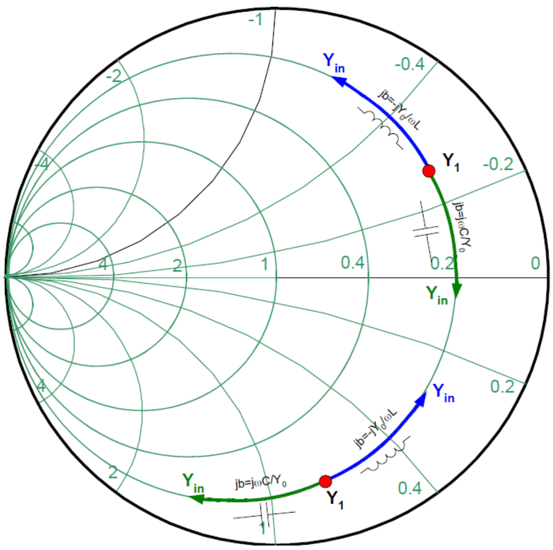
If an inductor or a capacitor is in series with the load impedance Z_1, the resulting impedance Z_{in} moves as shown in Figure 2.

Figure 2. Series connection



The Smith chart can be represented in normalized admittance (y = 1/z) scale, as shown in Figure 3.

Figure 3. Smith chart for admittance



Parallel inductor or capacitor

If an inductor or a capacitor is in parallel with the load admittance Y_1 the resulting impedance Y_{in} moves as shown in Figure 4.

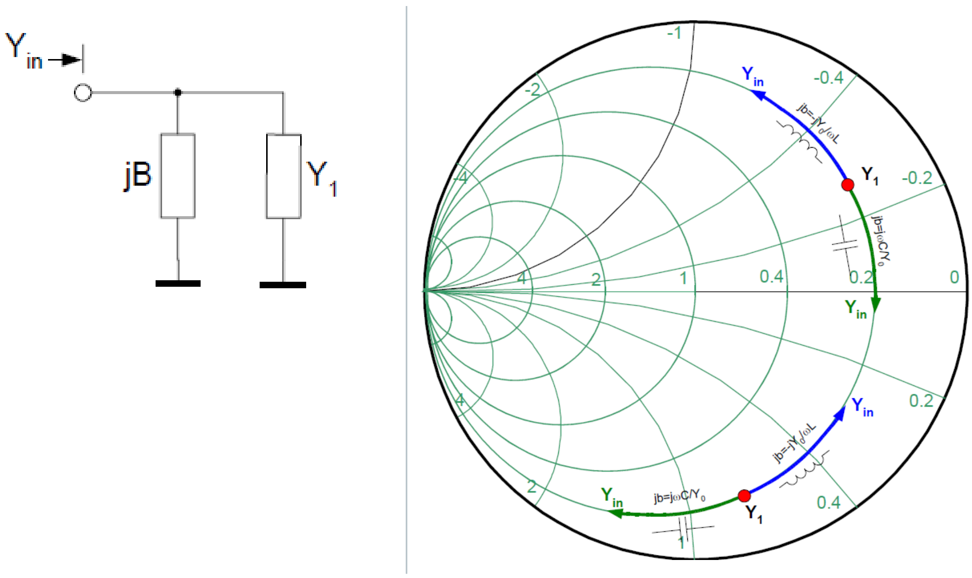


Figure 4. Parallel connection

In Figure 5, the Smith chart in impedance and admittance planes.

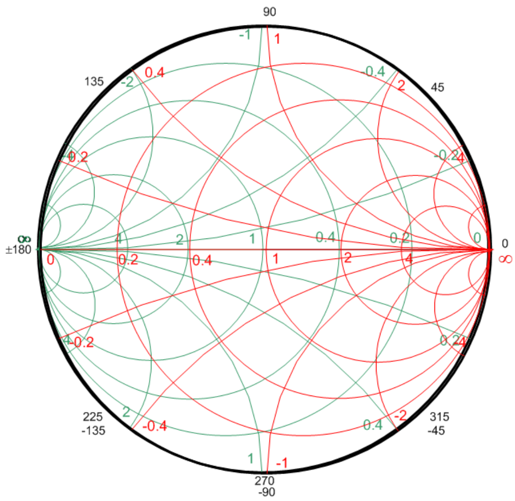


Figure 5. Smith chart with impedance and admittance

The circles with constant VSWR are additional information that can be retrieved from the Smith chart even when they are not represented. These circles have the same center, and as values the intersections between the circle and the right side of the horizontal axis from the center (see Figure 6).

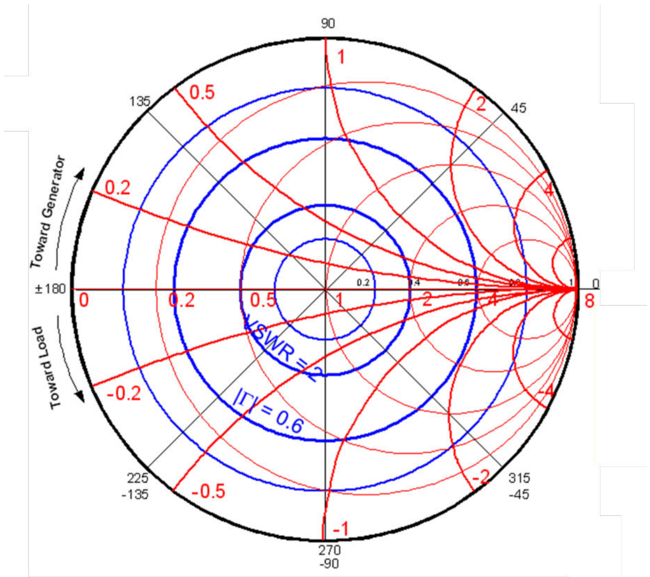


Figure 6. **Smith chart with VSWR circles**

Figure 7 is an example of the free software "Smith": starting from Z_L = (25.00 - j * 8.00) represented on the Smith chart by DP1, the goal is to obtain Z_{in} = 50 Ω. By adding (in series or in parallel) inductors or capacitors, the impedance converges towards the center of the graph.

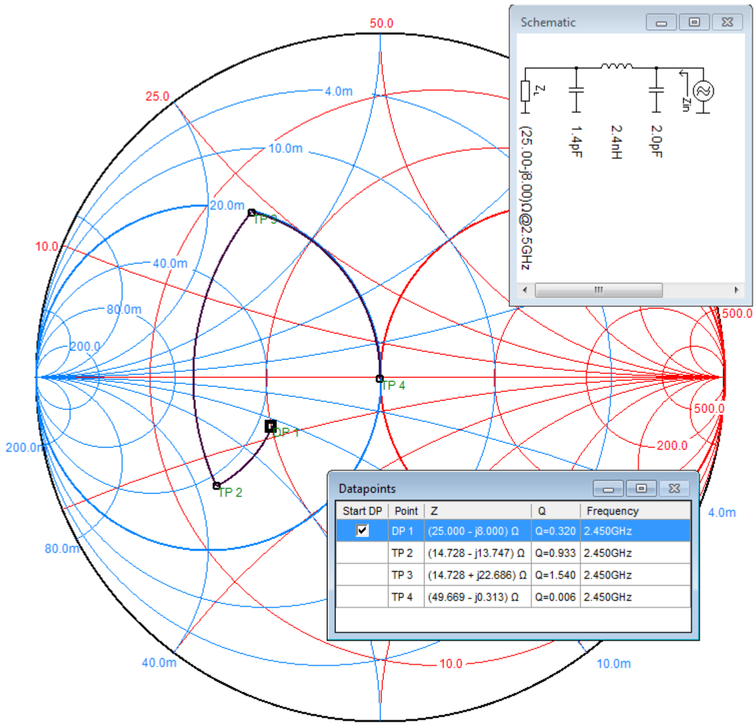


Figure 7. **Adapting a network with the Smith free software**

3 Reference board schematics

Refer to the following schematics packs available from www.st.com:

- NUCLEO-WBA52CG
- NUCLEO-WBA55CG
- NUCLEO-WBA65RI

4 Choosing the components

In the Bluetooth® LE bandwidth, and more generally at high frequencies, the choice of the external components is critical because they directly influence the performance of the application.

4.1 Capacitor

Capacitors are passive electrical components used to store energy in an electrical field. They are made with different construction techniques, materials (such as double-layer, polyester, and polypropylene), and sizes. For RF design, it is recommended to use ceramic capacitors on the surface mount version.

The equivalent circuit of a capacitor is represented in Figure 8. The resistor R_p represents its leakage current, while R_s is the equivalent serial resistor (ESR), representing all ohmic losses of the capacitor. The inductor L_s is the equivalent serial inductance (ESL), and its value is the function of the SRF (self-resonant frequency). From Figure 9, it can be appreciated that the impedance of the capacitor is capacitive at low frequencies, at the SRF is resistive, and inductive at higher frequencies.

Figure 8. Capacitor equivalent circuit

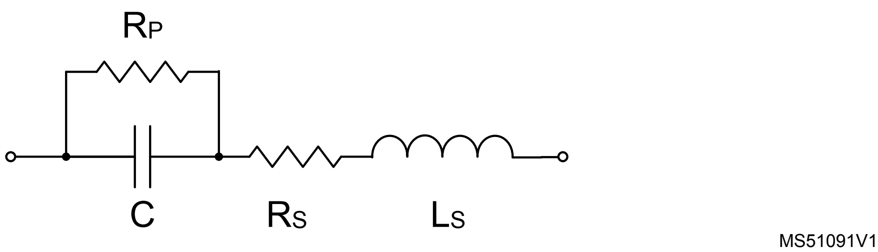
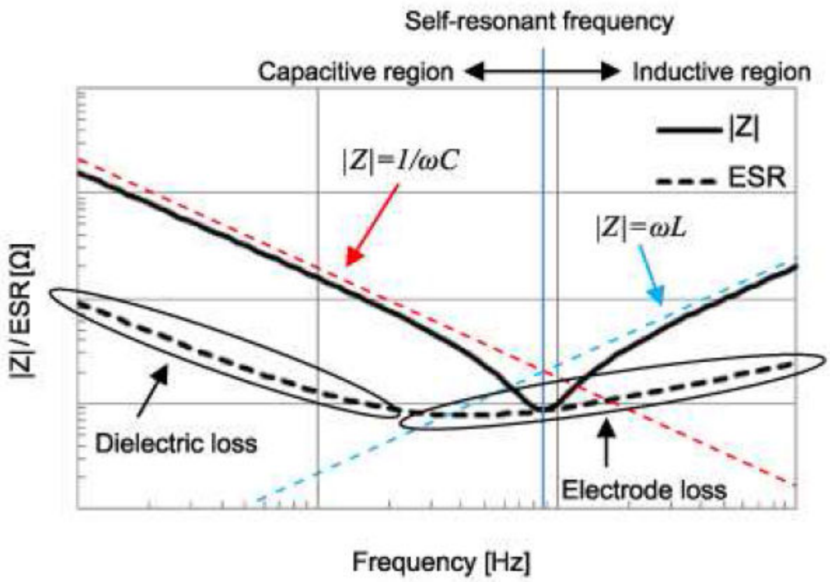


Figure 9. Capacitor impedance vs. frequency



For RF matching, multilayer ceramic capacitors offer linear temperature coefficients, low losses, and stable electrical properties over time (voltage and frequency). SMD (surface mount device) is used with a 0402 package, a good compromise between performance and handling.

For RF decoupling, the capacitance value must be chosen so that the frequency to be decoupled is close to or just above the self-resonant frequency of the capacitor.

For DC-DC converter, as the quality factor of a capacitor is inversely proportional to its ESR, a capacitor with low insertion loss and a good quality factor is recommended. The capacitor requires either an X7R or X5R dielectric.

Table 3. Capacitor temperature ranges

Minimum temperature		Maximum temperature		Variation over the temperature range	
Code	**Temperature**	**Code**	**Temperature**	**Code**	**Variation (%)**
X	-55 °C (-67 °F)	4	+65 °C (+149 °F)	P	± 10
		5	+85 °C (+185 °F)	R	± 15
Y	-30 °C (-22 °F)	6	+105 °C (+221 °F)	S	± 22
		7	+125 °C (+257 °F)	T	+22 / -33
Z	+10 °C (+50 °F)	8	+150 °C (+302 °F)	U	+22 / -56
		9	+200 °C (+392 °F)	V	+22 / -82

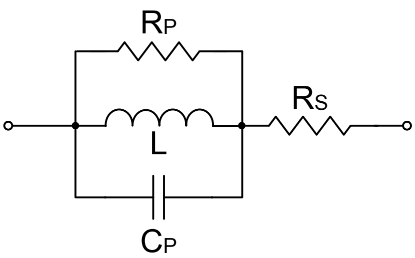
4.2 Inductor

An inductor is a passive electrical component used to store energy in its magnetic field. Inductors differ from each other for construction techniques and used materials.

For RF design, where a high Q (quality factor = Im[Z] / Re[Z]) is required to reduce insertion loss, it is generally recommended to use air core inductors. Those inductors do not use a magnetic core made of ferromagnetic material, but are wound on plastic, ceramic, or other nonmagnetic materials. SMD is also used with a 0402 package.

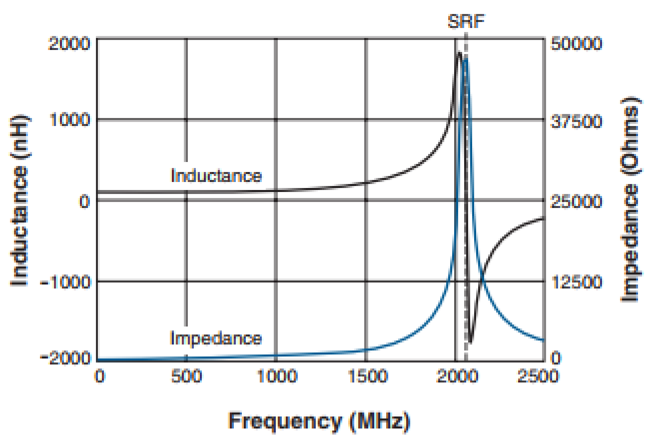
The equivalent circuit of an inductor is shown in Figure 10. The resistor R_s represents the losses due to the winding wire and terminations, its value increases with temperature. The resistor R_p represents the magnetic core losses, it varies with frequency, temperature and current. The capacitor C_p is associated with the windings.

Figure 10. Inductor equivalent circuit



As shown in Figure 11, at SRF the impedance and inductance are at their maximum. At lower/higher frequencies impedance and inductance increase/decrease with frequency.

Figure 11. Inductor impedance vs. frequency



For RF matching and decoupling, a good compromise between application cost and RF performance is to use an inductor with medium Q. For DC-DC converter, the nominal value is 10 µH. The inductor value affects the peak-to-peak ripple current, the output voltage ripple and the efficiency. The selected inductor has to be rated for its DC resistance and saturation current.

It is important to use the components shown in the schematics to obtain the best RF performance with the given PCB layout of the reference boards.

4.3 SMPS

The STM32WBA microcontrollers are available in packages for either LDO or SMPS. Refer to the document [1] available on www.st.com. The embedded SMPS (switched-mode power supply) runs at 3 MHz. It can be used to improve power efficiency for VDD above 1.71 V.

To operate properly the SMPS needs an input capacitor (C6), and inductor (L6) and an output capacitor (C23). The exact values depend upon the targeted performance, and on the available area on the PCB.

The additional inductor (L5 10 nH) is recommended to block noise from interfering with the supply.

Ferrite bead L4 and C21 can be used to filter out the switching noise of the SMPS towards the supply.

Figure 12. Components for SMPS

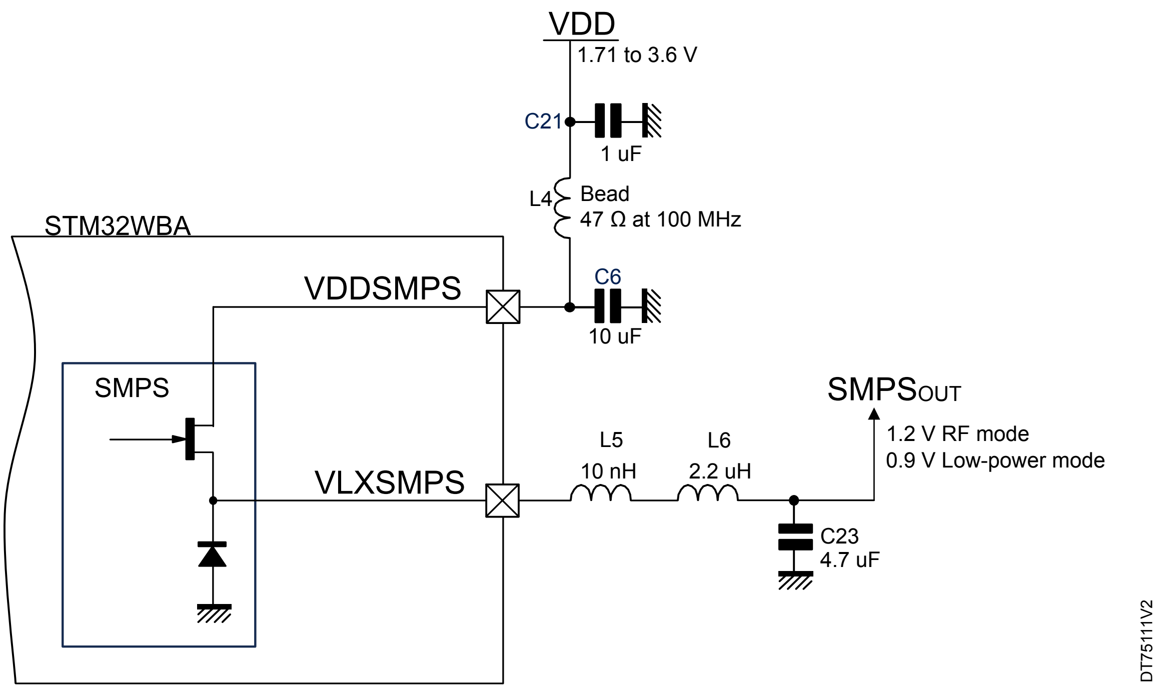


Table 4. Choice of components on STM32WBA55 Nucleo-64 board

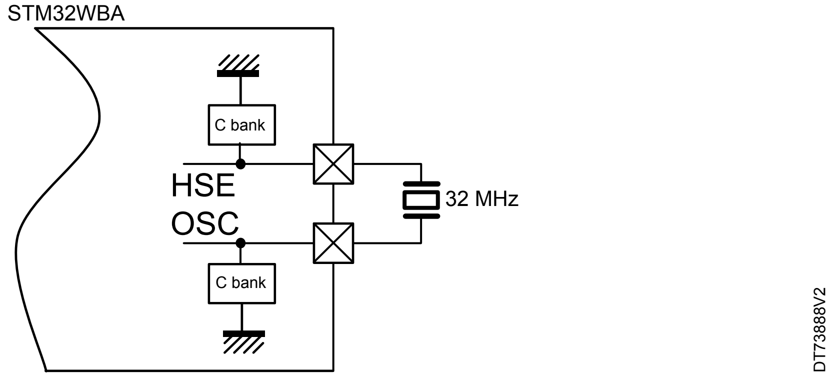
Designator	Value	Size (inch)	Manufacturer	Part number
C6	10 uF	0603	Murata	GRM188C81A106MA73
L4	47 Ω at 100 MHz	0603	TAIYO YUDEN	LSMGA160808T470RG
L5	10 nH	0402	Murata	LQG15HS10NJ02D
L6	2.2 uH	0805	Murata	LQM21PN2R2MGH
C21	1 uF	0402	Yageo	CC0402KRX5R6BB105
C23	4.7 uF	0402	Murata	GRM155R60J475ME47

4.4 External crystals

Two oscillators with external crystals are available on the STM32WBA microcontrollers.

For a more fundamental discussion of oscillators and the choice of crystals, see document [5]

Figure 13. **HSE oscillator**

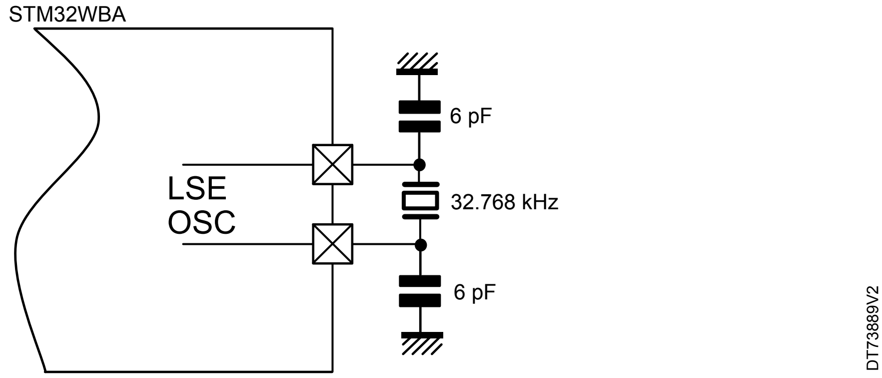


The HSE (high speed external) with 32 MHz frequency and 8 pF capacitance load is used by the RF subsystem. The crystal must be placed as close as possible to pins OSC_IN and OSC_OUT, to minimize output distortion and startup stabilization time. The HSE with the frequency tolerance must be reached during the start time of the HSE, which is 1 ms. The load capacitances are integrated on the chip and can be tuned according to the selected crystal via an internal register.

Table 5. **Crystals used for STM32WBA55 Nucleo-64**

Designator	Value	Package	Manufacturer	Part number
X1	32.768 kHz	1.6 x 1.0 mm	NDK	NX1610SE-32.768KHZ -EXS00A-MU01501
X2	32 MHz	1.6 x 1.2 mm	NDK	NX1612SA-32MHZ-EXS00A-CS09166

Figure 14. **LSE oscillator**

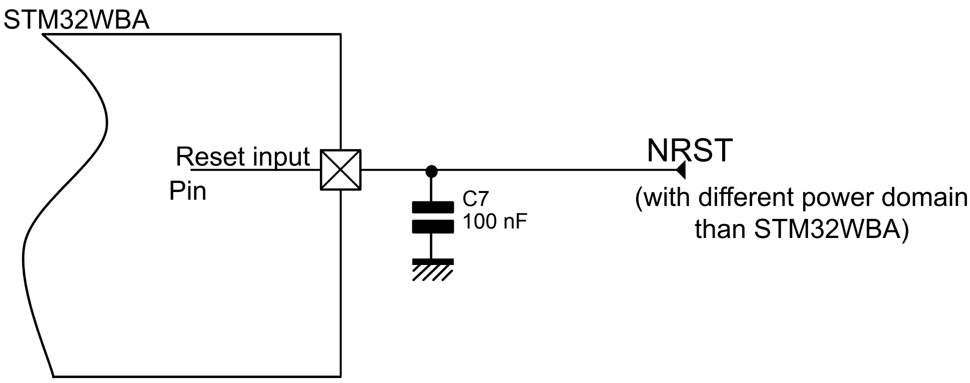


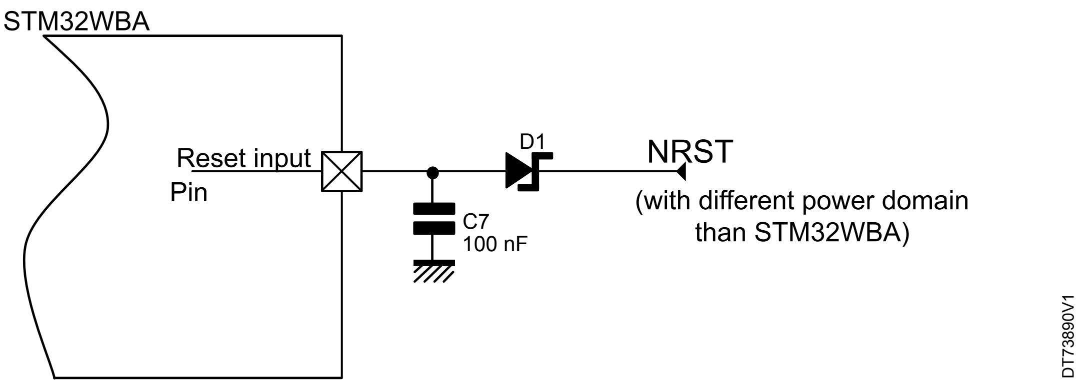
The LSE (low speed external) with 32.768 kHz frequency is used for the RTC subsystem. C_1 and C_2 must be tuned to meet to the recommended load capacitance C_0 of the selected crystal. Low power consumption and fast startup time are achieved with a low C_0 value. Inversely, a higher C_0 leads to a better frequency stability.

4.5 Reset

A 100 nF capacitor (C7) must be connected to the RESET pin. The additional diode is only needed if the external reset is coming from a device on another power domain than the STM32WBA circuit. If the power domains are the same, the diode is not needed.

Figure 15. **Connect RESET with 100 nF capacitor**

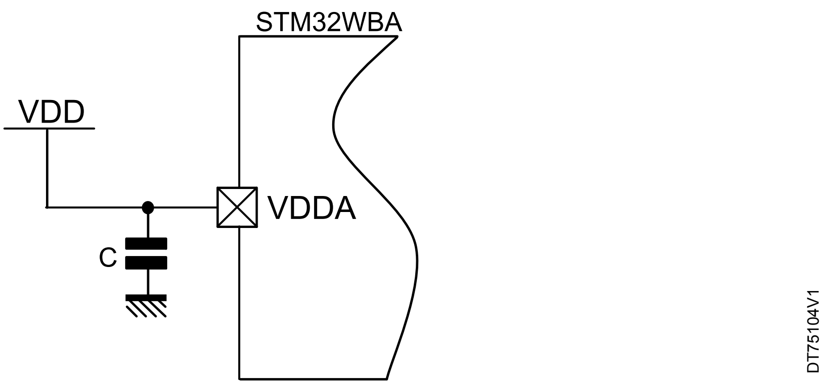




4.6 VDDA decoupling

The quality of the VDDA has a direct impact on the accuracy of the ADC and the comparator. The decoupling of this pin depends on the noise on the power supply and the desired performances of the ADC and the comparator. The minimum decoupling is a capacitor of 100 nF but we recommend a 1 uF capacitor in X7R technology. In the reference design an LC filter is used to improve the quality of the supply in case there is noise from the source or another part in the system. The cutoff frequency must be selected to match the noise in that case.

Figure 16. **Minimal decoupling of VDDA**



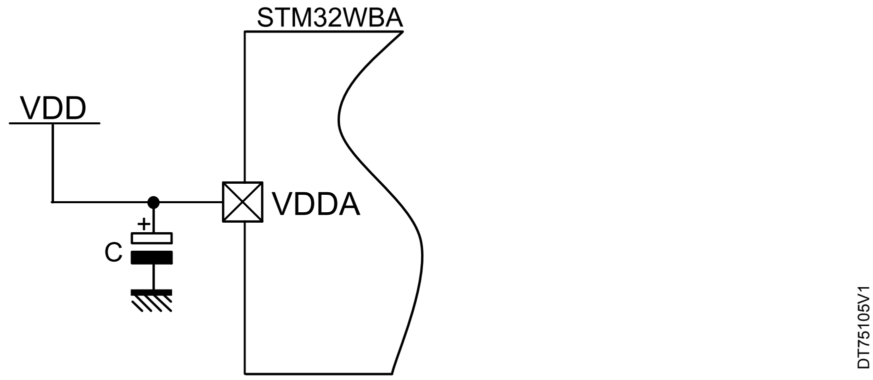


Figure 17. **Improved decoupling of VDDA**

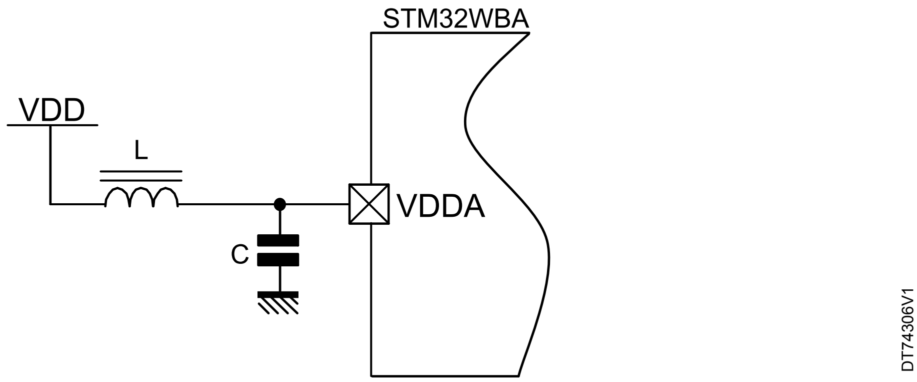


Figure 18. **Decoupling with LC filter for noisy environments**

4.7 RF output stage power supply

4.7.1 LDO package

The LDO configuration can drive RF output up to 10 dBm.

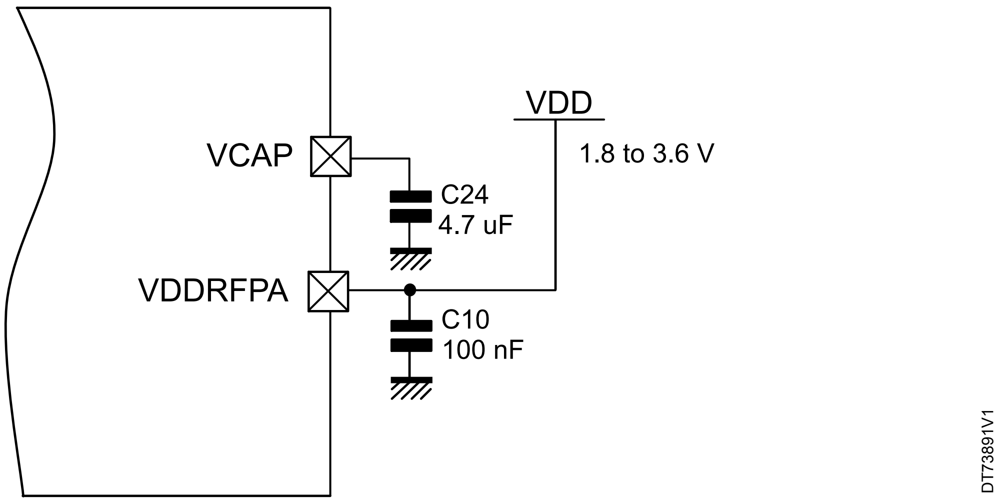
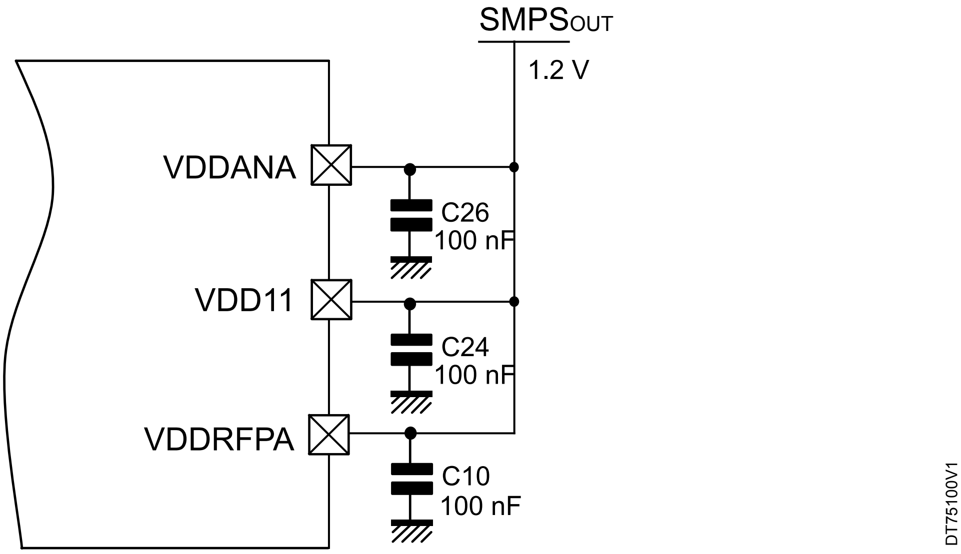


Figure 19. **LDO configuration**

4.7.2 SMPS packages

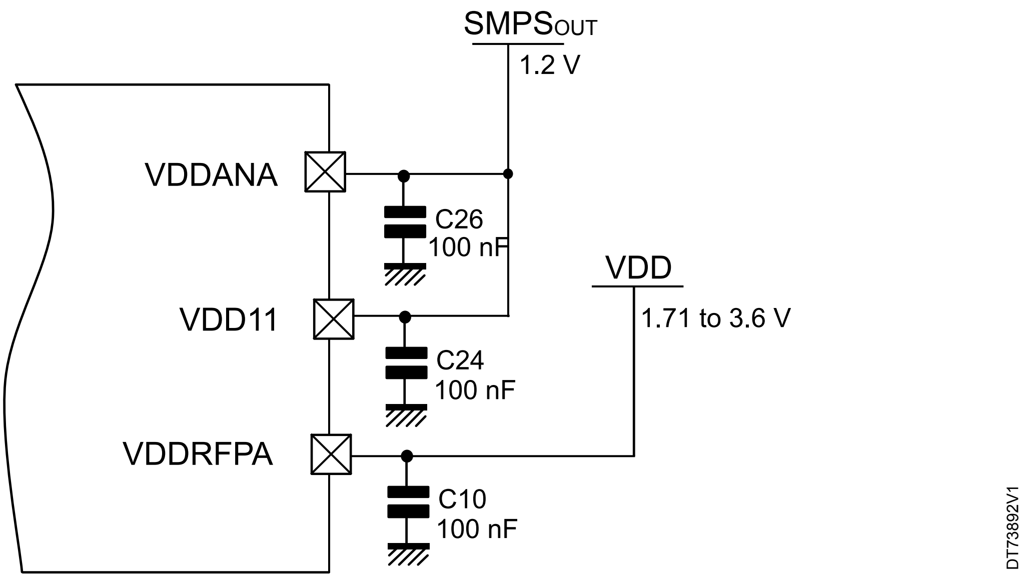
The SMPS packages enable two methods to supply the internal RF power amplifier. The first method takes the benefit of SMPS to reduce power consumption. This can drive the RF amplifier up to 6 dBm.

Figure 20. **SMPS configuration for low-power**



For high RF power output, up to 10 dBm, the supply of the power amplifier, VDDRFPA, is connected to the main VDD supply.

Figure 21. **SMPS configuration for high RF power**



5 STM32WBA and antenna RF matching

Maximum power is transferred from the chip (source) to the antenna (load) if the source impedance matches the load impedance. As a first approximation, both impedances could be considered to be 50 Ω but in reality the impedances are expressed as complex numbers and the imaginary components should have opposite signs. Thus, for correct matching, the source impedance X+jY should be matched by load X-jY. The matching circuit also rejects harmonics outside the 2.4 GHz band and incorrect matching can compromise the compliance of the radio with RED and FCC standards.

The design of the matching circuit is always a compromise. The impedance varies from part to part, by frequency and by power. The impedance of the circuit when transmitting is different from the impedance when receiving. There is also the cost of the circuit to consider, and the time it takes to design it. For this reason, there is no single best solution for the matching circuit. For a high quality design with the best rejection, it is recommended to design a matching circuit with 5 SMD components as described below. If the time to market is important, then the design phase can be shortened by using IPD *MLPF-WB-04D3*. If low product cost is imperative, it is possible to use a design with 3 components for a narrowed usage, for example, a reduced output power.

5.1 Five element solutions

Five passive components are needed to filter the harmonic signals on the RF output of the STM32WBA processors. The choice of components must consider the drift of component values over time, over temperature and variations from one PCB to another.

The optimum impedance in RX and TX for the STM32WBA is 50 ohms. Thus, on the SoC side, the filter must present 50 ohms. On the antenna side, the filter must present the conjugated value on the antenna (on the band). Note that the required filter is not necessarily one of 50 ohms to 50 ohms but one of 50 ohms to (Z_ant). This allows the limitation of the number of passive components and the insertion loss.

Figure 22 shows the implementation on the STM32WBA55 Nucleo-64 board with a PCB antenna.

Figure 22. Implementation on STM32WBA55 Nucleo-64

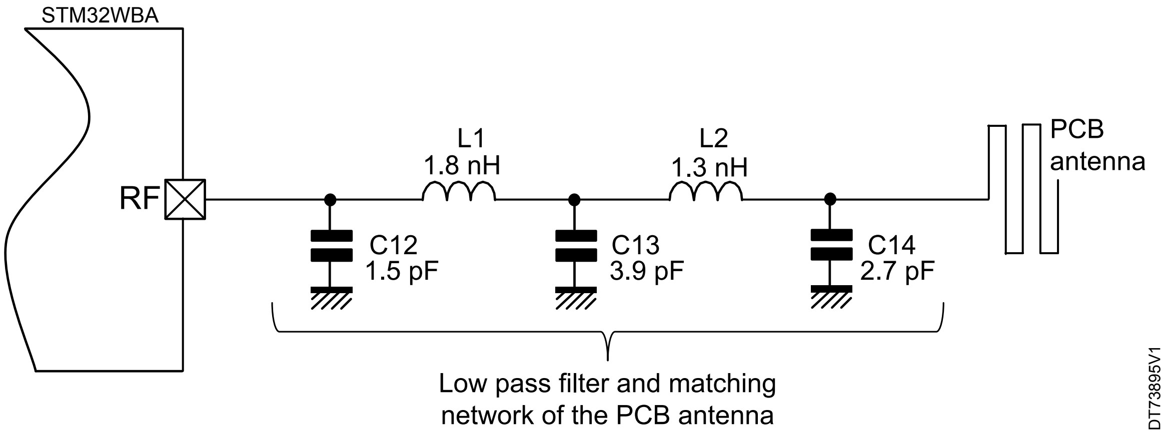


Table 6. Components for STM32WBA55 Nucleo-64

Designator	Value	Size (inch)	Manufacturer	Part number
L1	1.8 nH	0402	Murata	LQG15HS1N8S02
L2	1.3 nH	0402	Murata	LQG15HS1N3S02
C12	1.5 pF	0402	Murata	GRM1555C1H1R5CA01
C13	3.9 pF	0402	Murata	GRM1555C1H3R9CA01
C14	2.7 pF	0402	Murata	GJM1555C1H2R7BB01

Figure 23 shows the same topology dimensioned for 50 ohms input/output.

Figure 23. **Five elements output for 50 to 50 ohms**

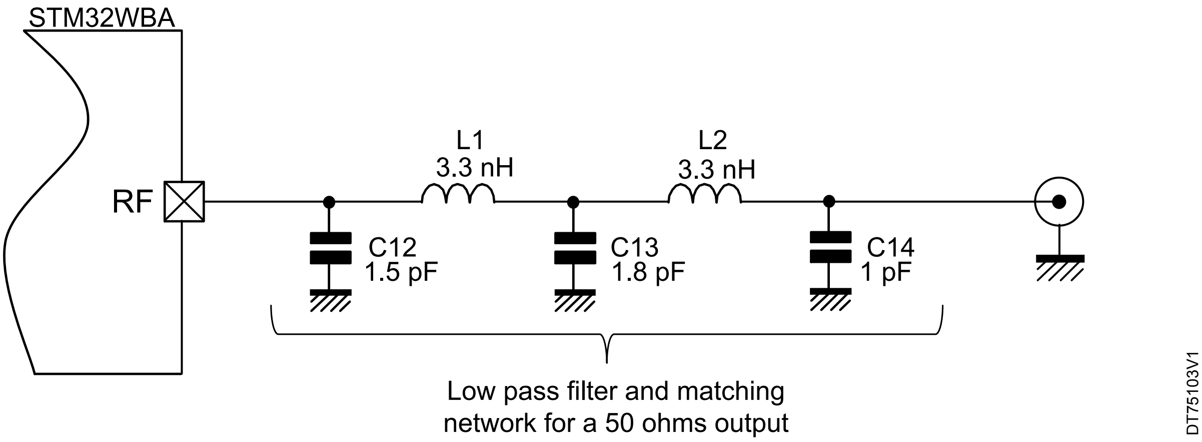


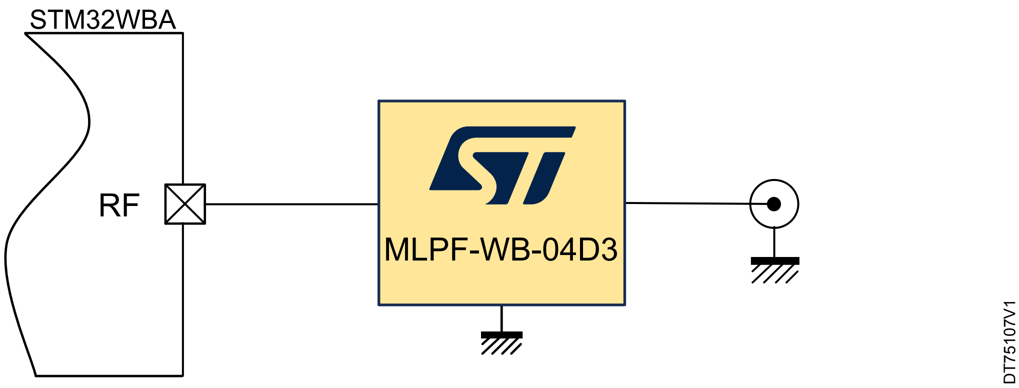
Table 7. **Components for a 50 ohms matching network**

Designator	Value	Size (inch)	Manufacturer	Part number
L1 & L2	3.3 nH	0402	Murata	LQG15HS3N3S02
C12	1.5 pF	0402	Murata	GRM1555C1H1R5CA01
C13	1.8 pF	0402	Murata	GRM1555C1H1R8CA01
C14	1 pF	0402	Murata	GRM1555C1H1R0CA01

5.2 IPD solution

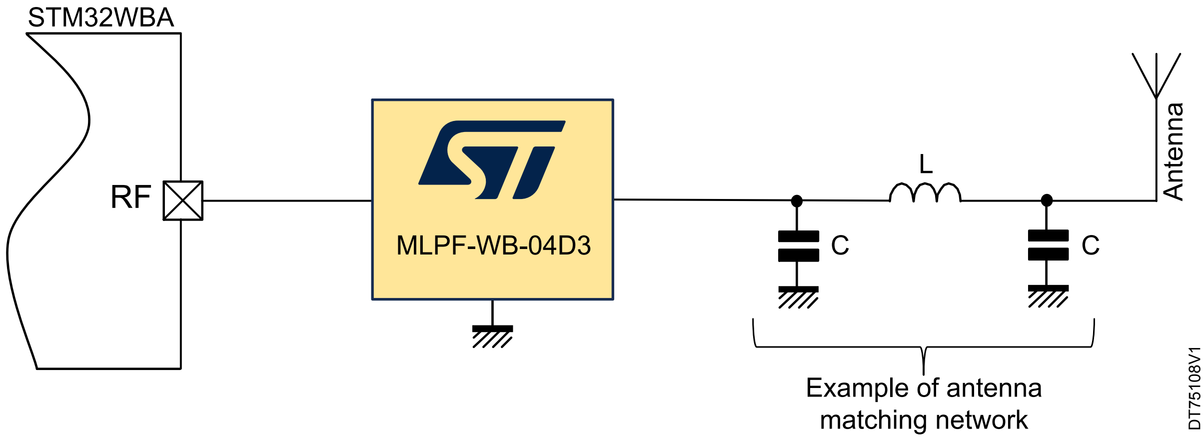
To reduce the number of discrete components in the design, ST proposes offers an IPD (Integrated Passive Device) for the filtering. This IPD with part number MLPF-WB-04D3 is a low-pass filter for a 50 to 50 ohms configuration. It meets the requirements for RF certification (RED, FCC) of designs based on STM32WBA processors with antennas that are matched to 50 ohms.

Figure 24. **STM32WBA with MLPF-WB-04D3 IPD**



If the antenna is not perfectly matched to 50 ohms, it is mandatory to add a matching network as shown in Figure 25. The number of components and the type of components, and their values depend on the antenna impedance.

Figure 25. STM32WBA with the IPD and an additional matching network



5.3 Three element solutions

Expert designers may choose to implement the matching network with three elements instead of five. This implementation needs a strong optimization with the type of components used and the layout of the PCB.

The schematics below show two solutions. Figure 26 concerns a 50 ohms output and Figure 27 provides values for the STM32WBA55 Nucleo-64 PCB up to +10 dBm with a meander antenna. These are only examples and they may not give satisfying results for other designs.

Figure 26. Low-pass filter with three elements for 50 ohms output

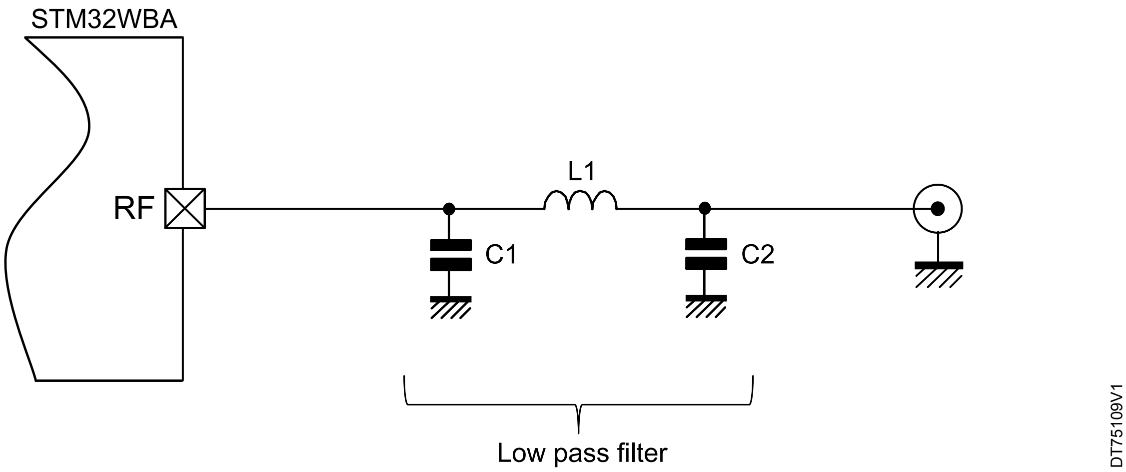


Table 8. Possible components for low-pass filter with three elements for 50 ohms output

Designator	Value	Size (inch)	Manufacturer	Part umber
L1	2.4 nH	0402	Murata	LQG15HS2N4S02
C1	1.6 pF	0402	Murata	GJM1555C1H1R6BB01
C2	1.6 pF	0402	Murata	GJM1555C1H1R6BB01

If the antenna is not perfectly matched to 50 ohms, it is necessary to modify the network. The input impedance is always 50 ohms, but the output must be the conjugated value of the antenna impedance.

Below is an example with the meander antenna.

Figure 27. Low-pass filter with three elements for meander antenna

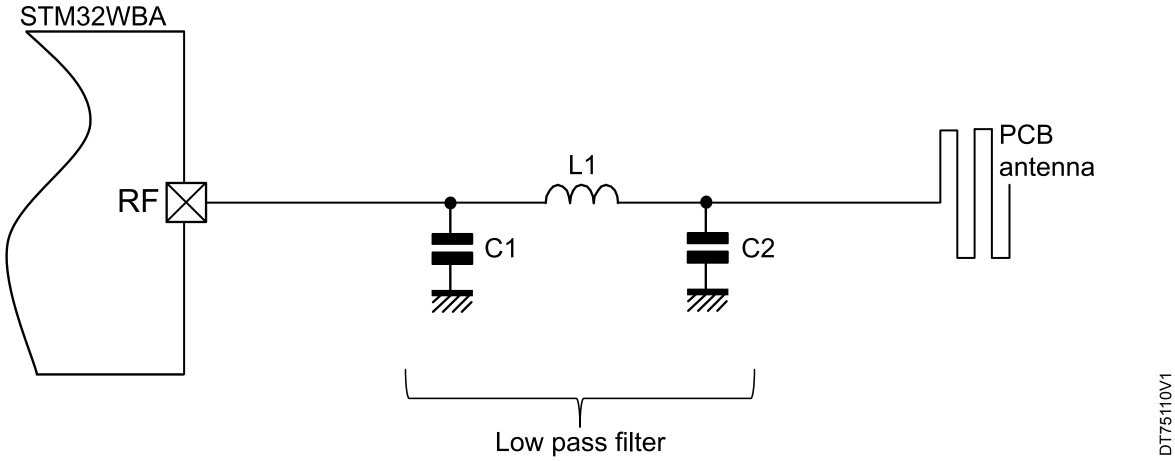


Table 9. Possible components for low-pass filter with three elements for meander antenna

Designator	Value	Size (inch)	Manufacturer	Part number
L1	5.1 nH	0402	Murata	LQG15HS5N1S02
C1	1.6 pF	0402	Murata	GJM1555C1H1R6BB01
C2	1.5 pF	0402	Murata	GJM1555C1H1R5CB01

5.4 Impedance by package

For the design of a five (or three) element matching circuit, it is important to consider the variant of the microcontroller as the impedance varies from package to package. The following table gives the most suitable impedances obtained from load pull measurements at different output power levels for transmitting and source pull measurements for receiving.

Table 10. Impedance by package

Microcontroller	Package	Load pull 0 dBm	Load pull 5 dBm	Load pull 10 dBm	Source pull -95 dBm
STM32WBA55	UFQFPN48	37.6 - j14.0	22.7 + j9.3	31.6 - j3.1	45.1 + j11.3
STM32WBA54	UFQFPN32	25.3 - j11.9	23.9 + j16.4	30.5 + j2.5	-
STM32WBA55	WLCSP41	22.7 + j9.3	20.5 + j31.9	26.3 + 23.7	37.1 + j39.5
STM32WBA55	UFBGA59	31.6 - j3.1	26.3 + j23.7	31.8 + j13.7	50.8 + j10.6
STM32WBA65	WLCSP88	31.8 + j13.7	20.5 + j13.9	26.3 + j23.7	38.2 + j24.6
STM32WBA65	UFBGA121	30.6 + j8.1	18.9 + j27.2	23.9 + j16.4	30.5 + j31.5
STM32WBA65	UFQFPN48	33.8 - j8.6	22.7 + j9.3	30.5 + j2.5	46.9 + j2.6

6 PCB stack

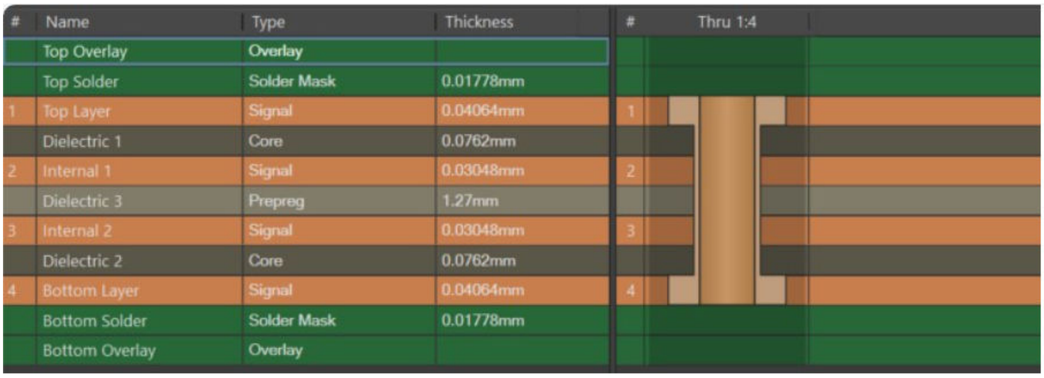
The Nucleo designs based on UFQFPN48 and UFBGA59 use a PCB stack of 4 layers. For the 121 balls BGA (STM32WBA65PI), 8 layers are used.

Figure 28. 4-layer PCB stack for design with BGA package

#	Name	Type	Thickness
	Top Overlay	Overlay	
	Top Solder	Solder Mask	0.01778mm
1	Top Layer	Signal	0.04064mm
	Dielectric 1	Core	0.0762mm
2	Internal 1	Signal	0.03048mm
	Dielectric 3	Prepreg	1.27mm
3	Internal 2	Signal	0.03048mm
	Dielectric 2	Core	0.0762mm
4	Bottom Layer	Signal	0.04064mm
	Bottom Solder	Solder Mask	0.01778mm
	Bottom Overlay	Overlay	

DT73893V1

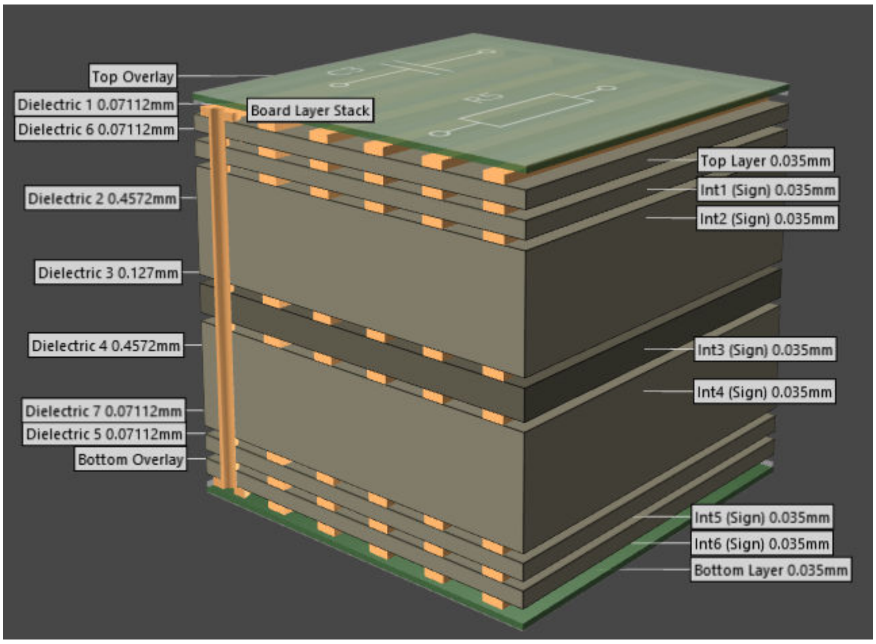
Figure 29. 4-layer PCB stack for design with QFN package



#	Name	Type	Thickness
	Top Overlay	Overlay	
	Top Solder	Solder Mask	0.01778mm
1	Top Layer	Signal	0.04064mm
	Dielectric 1	Core	0.0762mm
2	Internal 1	Signal	0.03048mm
	Dielectric 3	Prepreg	1.27mm
3	Internal 2	Signal	0.03048mm
	Dielectric 2	Core	0.0762mm
4	Bottom Layer	Signal	0.04064mm
	Bottom Solder	Solder Mask	0.01778mm
	Bottom Overlay	Overlay	

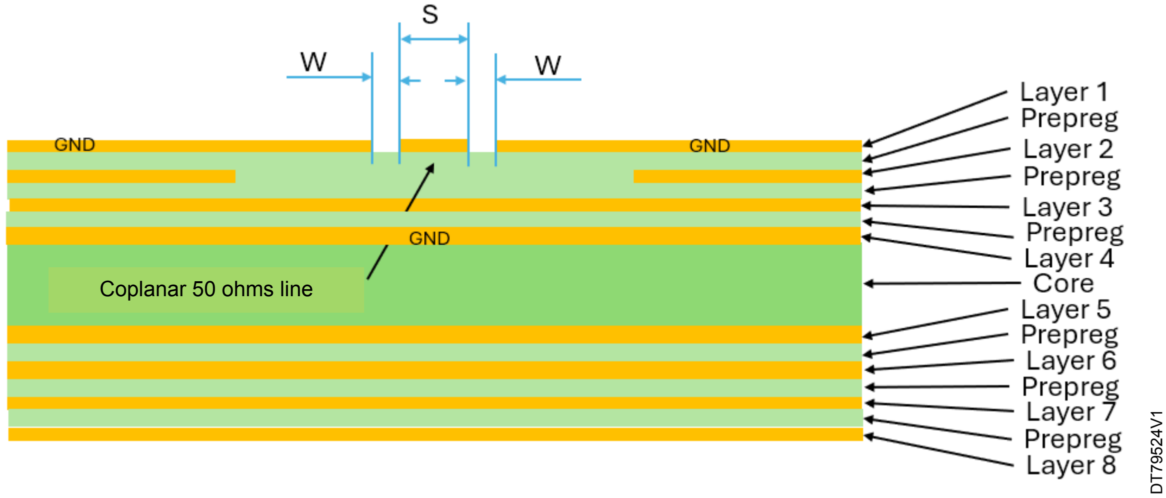
DT73894V1

Figure 30. 8-layer PCB stack for desgin with BGA 121 ball package



DT77088V1

Figure 31. Structure of 8-layer PCB stack



7 Layout recommendations for reference boards

7.1 Power supplies and decoupling

Power supply routing is important in RF design and, if not carefully made, can affect the system performance undesirably. Proper routing, bypassing, and decoupling avoid noise coupling effects and affecting the performance of the system. A source of high frequency noise on a PCB can be the transient demand of current by active devices, casing high frequency harmonics to be generated. The generated noise can travel to the power supply pins of the devices, degrading performance, to prevent this it must be bypassed to the ground plane using a capacitor providing a low impedance path.

Also, the digital section switches rail to rail rapidly, generating high frequency harmonics, which can couple with the power supply lines if not routed and decoupled properly. Also, there may be undesired coupling between the power supply lines. The power supply and digital lines must be routed away from the RF section, and decoupling must be done to isolate the corresponding power supply pin from the high frequency noise on the other sections. Additionally, the bypass/decoupling capacitor must be carefully selected considering the SRF (above this frequency the capacitor behaves as an inductor, hence a capacitor is effective only up to the SRF).

A common practice is to use a "star" configuration for the power supplies nets. A larger decoupling capacitor (4.7 to 10 uF) is mounted at the "root" of the star, and smaller capacitors (100 nF) at the end of each of the star branches near the power supply pins of the chip.

The star configuration avoids long ground return paths that would result if all the pins connected to the same power supply net are connected in series. A long ground return path causes a parasitic inductance that can lead to unintended feedback loops.

7.2 Grounding shunt-connected components

For shunt-connected (grounded) components (such as power supply decoupling capacitors), the recommended practice is to use at least two grounding vias for each component. This reduces the effect of via parasitic inductance. Via ground 'islands' can be used for groups of shunt-connected components.

7.3 IC ground plane (exposed pad)

Most devices require a solid ground plane on the component layer (TOP or BOTTOM of the PCB) directly underneath the component. This ground plane carries DC and RF return currents through the PCB to the assigned ground plane. The secondary function of the exposed pad is to provide a thermal heat-sink, so that the exposed pad must include the maximum number of vias allowed by the PCB design rules. These vias are ideally thru-vias (penetrating through all the PCB layers).

7.4 Isolation

Care must be taken to prevent unintended coupling between signal lines. Some examples of potential coupling and preventive measures:

- RF transmission lines: keep them as far apart as possible. The grounded coplanar waveguide provides excellent isolation between lines. It is impractical to achieve better than approximately -45 dB between RF lines on small PCBs.
- High-speed digital signal lines must be routed separately on a layer different from that of the RF signal line to prevent coupling. Digital noise (for example, clocks) can couple onto RF signal lines, and these can be modulated onto RF carriers.
- V_{DD} and power lines must be routed on a dedicated layer. Provide adequate decoupling/bypass capacitors at the main V_{DD} distribution nodes and at V_{DD} branches. The value of the bypass capacitances must be set based on the overall frequency response of the RF device, and the expected frequency distribution nature of any digital noise from clocks. These lines must also be separated from the RF lines.

7.5 Component orientation

Inductors on the PCB generate a magnetic field that can couple with other components and RF line. To obtain a good isolation between those components, place them orthogonally and, if not possible, space them as much as possible.

7.6 RF

To ensure sufficient spurious and harmonic rejection, an RF low-pass filter is required at the device output as explained in Section 5.

It is recommended to avoid long tracks between each component on the RF path to minimize possible phase rotation with respect to harmonics, avoid unwanted radiation (radiation is proportional to the length) and obviously minimize losses due to the track length. It is also recommended not to cover the RF tracks with solder resist when possible.

7.7 Crystals

It is recommended to place the crystals on the top layer and to keep them as close as possible to the oscillator pins of the device. Long lines must be avoided.

Do not route signals close to the crystal wires and pads. Also, leave a gap between the ground wires and the crystal pads to avoid a capacitance that would shift the frequency. The crystal must be kept far from SMPS and RF components (matching, filter, antenna).

7.8 Check list

1. Verify that the bypass capacitors are as close as possible to the power supply pins that they are meant to bypass.
2. Ensure that each decoupling capacitor only decouples the specific pins recommended on the reference design and that the capacitor is of the correct value and type.
3. Verify that the stack-up matches the reference design. If the design is 4-layer or higher, verify that the ground plane is INNER1 layer just below the TOP/components layer.
4. Changing the spacing/stack-up affects the matching in the RF signal path and must be carefully accounted.
5. A solid ground plane must be placed below the device and the RF path. No ground plane must be placed below the antenna unless recommended by the manufacturer.
6. Verify that the RF signal path matches the reference design as close as possible (components must be arranged in a very similar way and oriented the same way).
7. The crystals must be as close as possible to the oscillator pins of the device. Long lines to the oscillator must be avoided.
8. No GPIOs net nearby the crystal area.
9. Verify that the ground pours on the TOP layer are stitched to the INNER ground plane and to the BOTTOM layer plane with many vias, especially around the RF path. Vias on the rest of the board must be no more than a tenth of a wavelength apart.
10. If the device uses a battery, do not place it under the antenna, because it acts as a ground plane.
11. The board must specify impedance-controlled traces, meaning that the layer spacing and FR4 permittivity must be controlled and known.

Important considerations for the antennas

1. The reference design must be copied exactly with the same stack-up.
2. Changes to feed line length of the antenna also change input impedance matching.
3. Metals close, plastic enclosures, and the human body change the antenna input impedance and resonance frequency.
4. For multiple antennas on the same board, use antenna polarization and directivity to isolate them.
5. For chip antennas, verify that the spacing from and the orientation respect to the ground plane is correct (as specified in the antenna datasheet).

7.9 Undesired effects

- Sensitivity/spurs (receive mode; RF input):
  - GPIO coupling through RF lines, ground, power supply
  - XTAL coupling through RF lines, ground, power supply
  - SMPS coupling through RF lines, ground, power supply
  - Digital activities through ground, power supply

- Pulling:
  - VCO pulled by the PA, the XTAL or/and the GPIO
  - PA pulled by the VCO, the XTAL or/and the GPIO
  - XTAL pulled by the VCO, the PA or/and the GPIO

8 Example layouts

Figure 32. STM32WBA55 with SMPS on QFN48

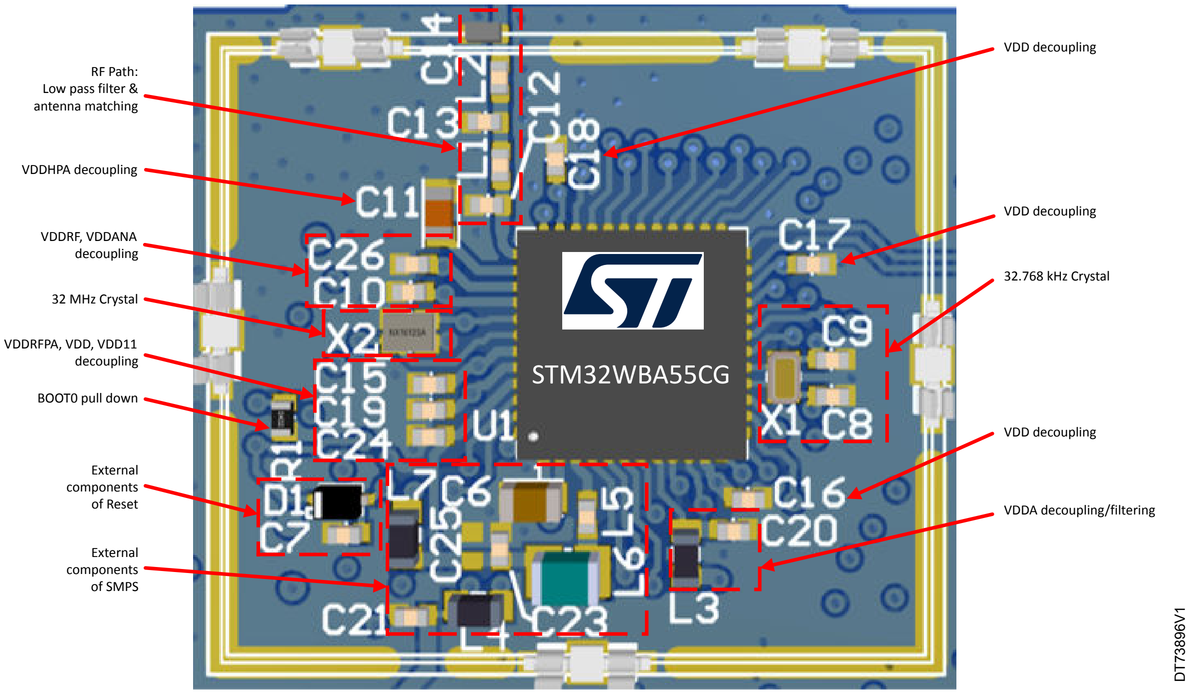
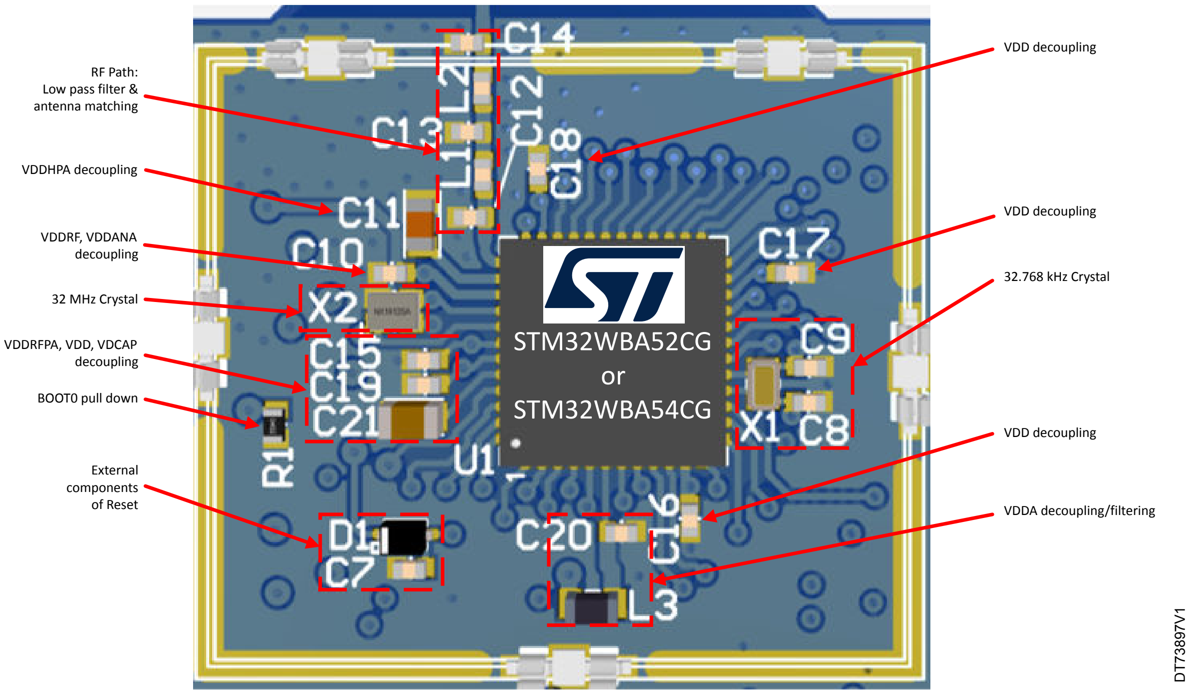


Figure 33. STM32WBA52 or STM32WBA54 with LDO on QFN48



Note: The layout of a board with WBA64 for LDO with package QFPFN48 is similar to the layout in Figure 33.

Figure 34. STM32WBA54 with LDO on QFN32

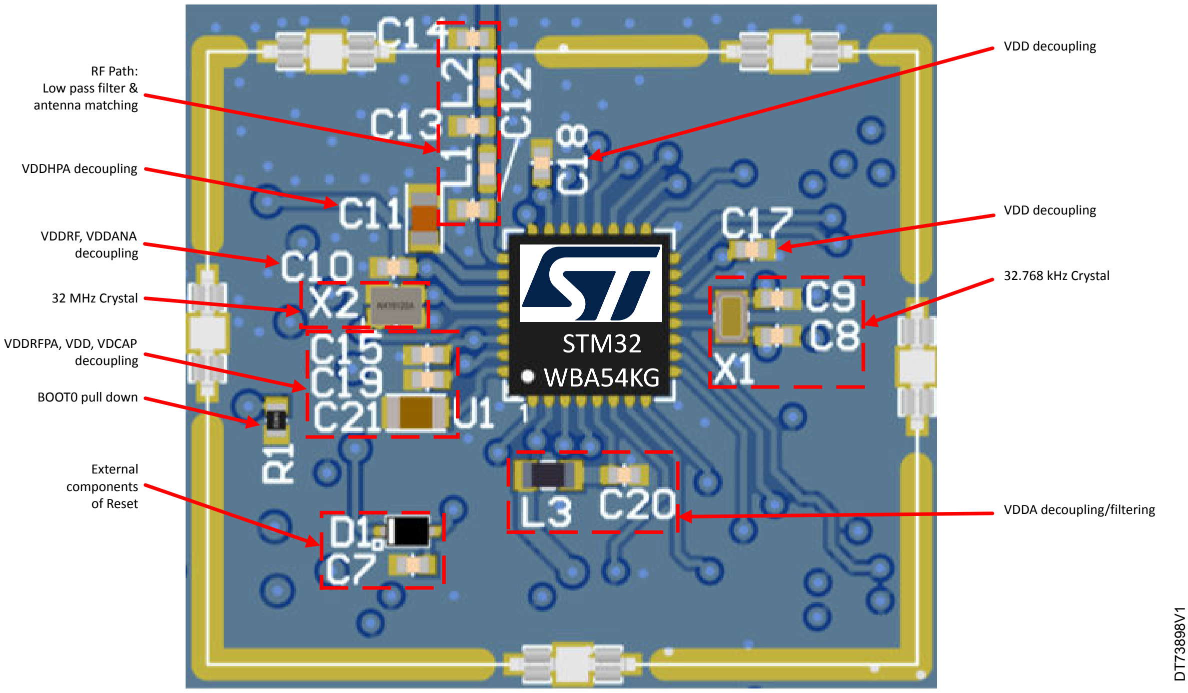


Figure 35. STM32WBA55 with SMPS on BGA

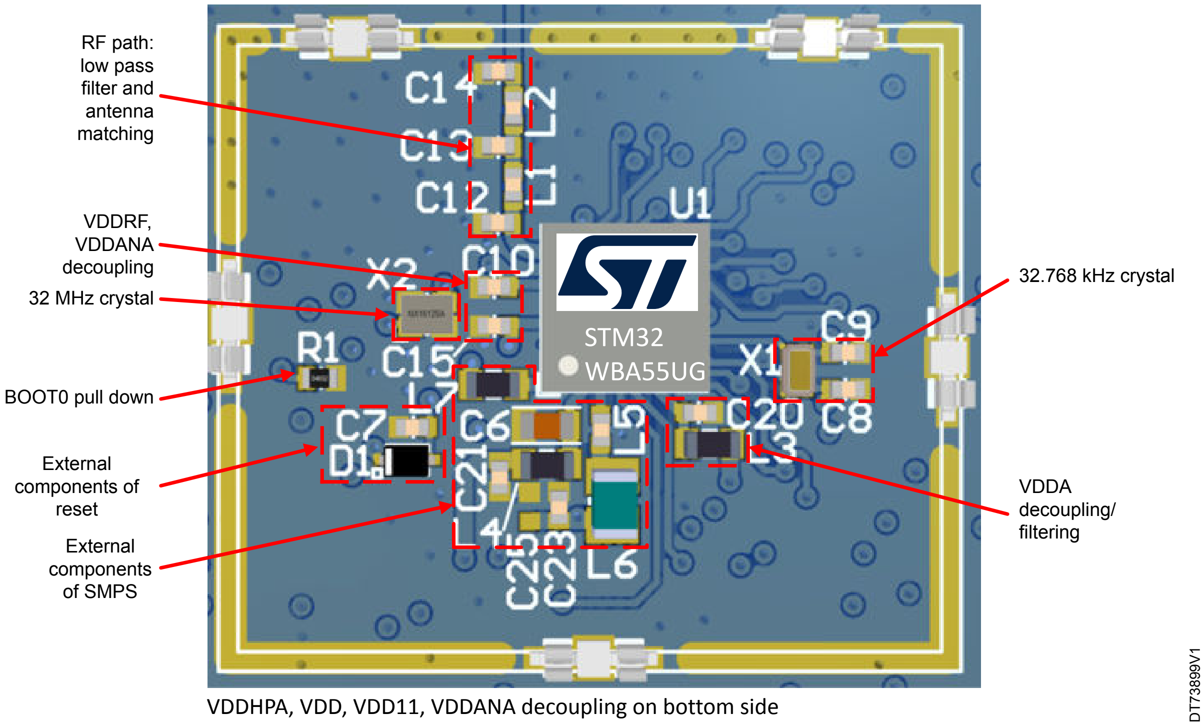


Figure 36. STM32WBA65 with SMPS on BGA121

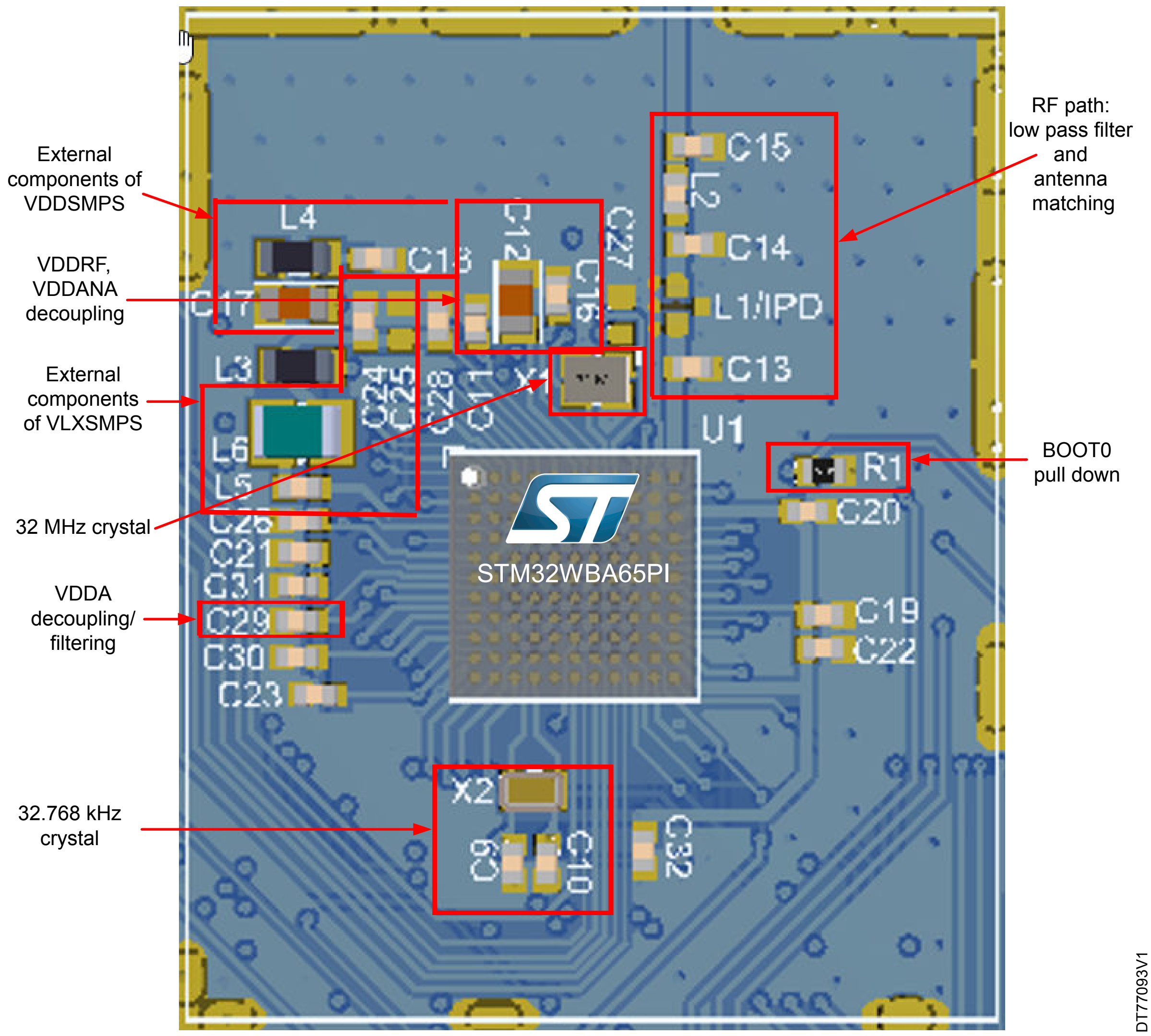
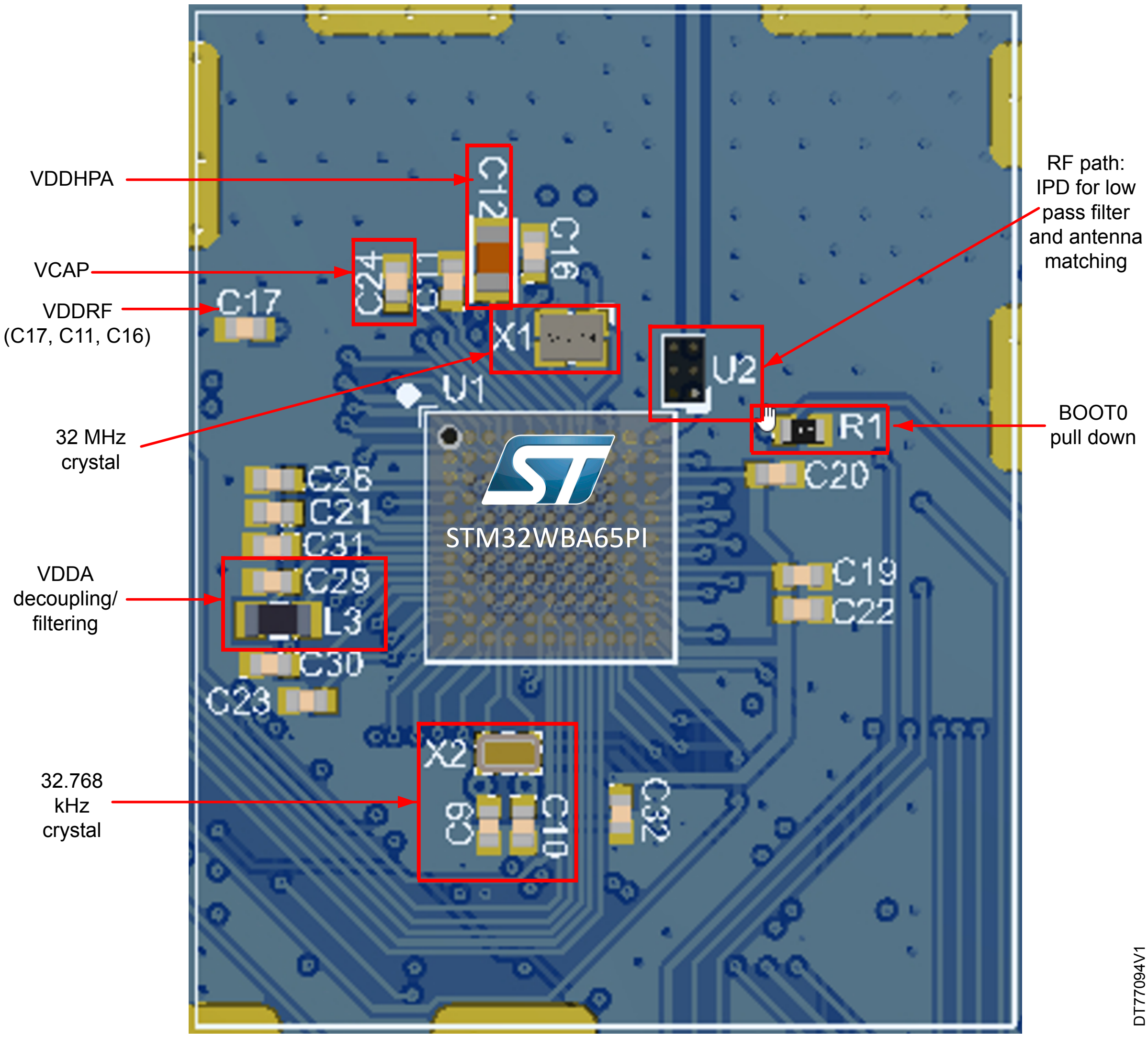


Figure 37. STM32WBA65 with LDO on BGA121



9 Conclusion

The STM32WBA microcontrollers integrate a high performance RF front-end.

To achieve the best TX and RX performance, several aspects have to be addressed during the PCB design:

- Choice of the PCB technology (number of layers, substrate technology)
- Computation of the antenna matching and filtering network
- Floor plan and critical RF component placement and routing
- Placement of the SMPS and LSE components (if used)

This application note provides useful guidelines to help the user to reach the performance specified in the datasheets.

Revision history

Table 11. **Document revision history**

Date	Version	Changes
02-May-2024	1	Initial release.
08-Jul-2024	2	Updated: • Figure 12. Components for SMPS • Figure 13. HSE oscillator • Figure 14. LSE oscillator • Figure 15. Connect RESET with 100 nF capacitor • Table 4. Choice of components on STM32WBA55 Nucleo-64 board • Table 5. Crystals used for STM32WBA55 Nucleo-64
30-Jul-2025	3	Replaced STM32WBA5xx by STM32WBA throughout the document Updated: • Section Introduction • Table 2. Applicable products • Section 1: General information • Section 3: Reference board schematics • Section 8: Example layouts • Section 6: PCB stack Added: • Table 1 • Section 5: STM32WBA and antenna RF matching

Contents

List of tables

List of figures