

Hardware Design Considerations for MKW41Z/31Z/21Z BLE and IEEE 802.15.4 Devices

1. Introduction

This application note describes Printed Circuit Board (PCB) design considerations for the MKW41Z, MKW31Z, and MKW21Z 48-pin Laminated QFN (LQFN) package. Included are layouts of the component copper layer, solder mask, and solder paste stencil. These recommendations are guidelines only and may need to be modified depending on the assembly house used and the other components on the board.

Contents

1.	Introduction	1
2.	48-pin LQFN component copper layer	2
2.1.	48-pin LQFN solder mask	2
2.2.	48-pin LQFN solder paste stencil	2
3.	48-pin LQFN package dimensions	4
3.1.	48-pin LQFN device marking details	4
4.	48-pin LQFN soldering profile.....	5
5.	Design and board layout considerations.....	6
5.1.	MKW41Z device footprint	7
5.2.	RF circuit topology and matching.....	9
5.3.	Antenna considerations	10
6.	Revision history	11



2. 48-pin LQFN component copper layer

Figure 1 shows a recommended component copper layer. This layer is also referred to as the top metal layer and is the layer to which the components are soldered. The footprint for the 48-pin LQFN package consists of 48 IC contact pads, and 16 centered ground pads. The copper pattern is shown in Figure 1. Use 0.25 mm via holes to connect to the ground plane layers. These are required for RF grounding and help to prevent solder float.

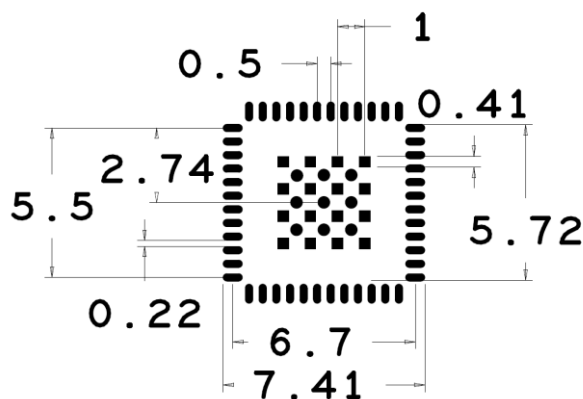


Figure 1. 48-pin LQFN component copper layer

2.1. 48-pin LQFN solder mask

The solder mask limits the flow of the solder paste during the reflow process. The following figure shows a recommended solder mask pattern. The pattern represents openings in the solder mask.

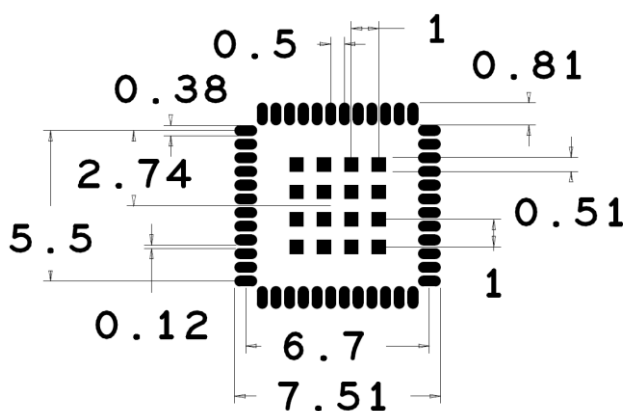


Figure 2. 48-pin LQFN solder mask pattern

2.2. 48-pin LQFN solder paste stencil

The solder paste stencil controls the pattern and thickness of the solder paste dispensed on the board. Figure 3 shows a recommended solder stencil pattern. Stencil thickness should be approximately 0.1 mm.

Other patterns and opening sizes can be used if too much solder is being applied. See the following section for more information.

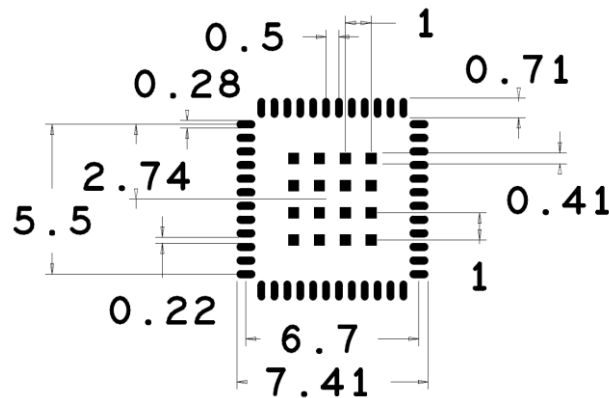


Figure 3. 48-pin LQFN solder stencil pattern

2.2.1. LGA problems with excess solder

Excess solder may cause the LGA to float or bridge between the package contacts. To use the correct amount of solder paste applied to the PCB, take into consideration the following:

- Stencil thickness
- Other components mounted on the PCB
- Manufacturing equipment
- Assembly house experience

3.1. 48-pin LQFN device marking details

Diagram illustrating the marking layout on a component, showing the following fields and their corresponding labels:

- Pin 1**: Points to the first pin location.
- Part Number**: Points to the marking **M41W**.
- Mask Number**: Points to the marking **XXX**.
- Assembly Site**: Points to the marking **ALYWZ** (circled).
- Date Code**: Points to the marking **MMMMM**.

NOTE

Hardware Design Considerations for MKW41Z/31Z/21Z BLE and IEEE 802.15.4 Devices, Application Note, Rev. 1,
03/2017

4. 48-pin LQFN soldering profile

The following figure shows the recommended soldering profile for the MKW41Z 48-pin LQFN package, in a board size approximately 3.20 inches × 2.10 inches.

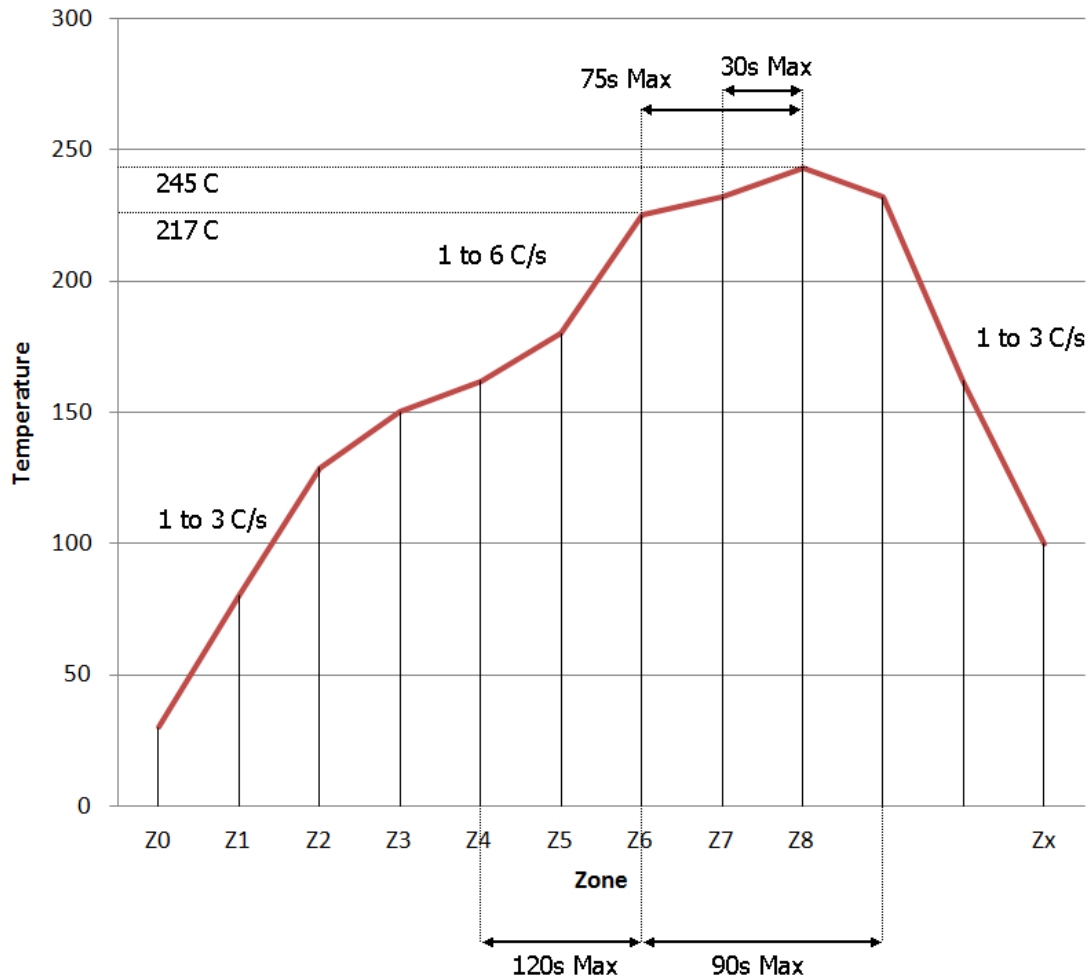


Figure 6. 48-pin LQFN soldering profile

5. Design and board layout considerations

To have successful wireless hardware development, the proper device footprint, RF layout, circuit matching, antenna design, and RF measurement capability are essential. RF circuit design, layout, and antenna design are specialties requiring investment in tools and experience. With available hardware reference designs from NXP, RF design considerations, and the guidelines contained in this application note, hardware engineers can successfully design BLE and IEEE 802.15.4 radio boards with good performance levels. The following figure shows the FRDM-KW41Z development board. It contains the MKW41Z device and all necessary I/O connections.

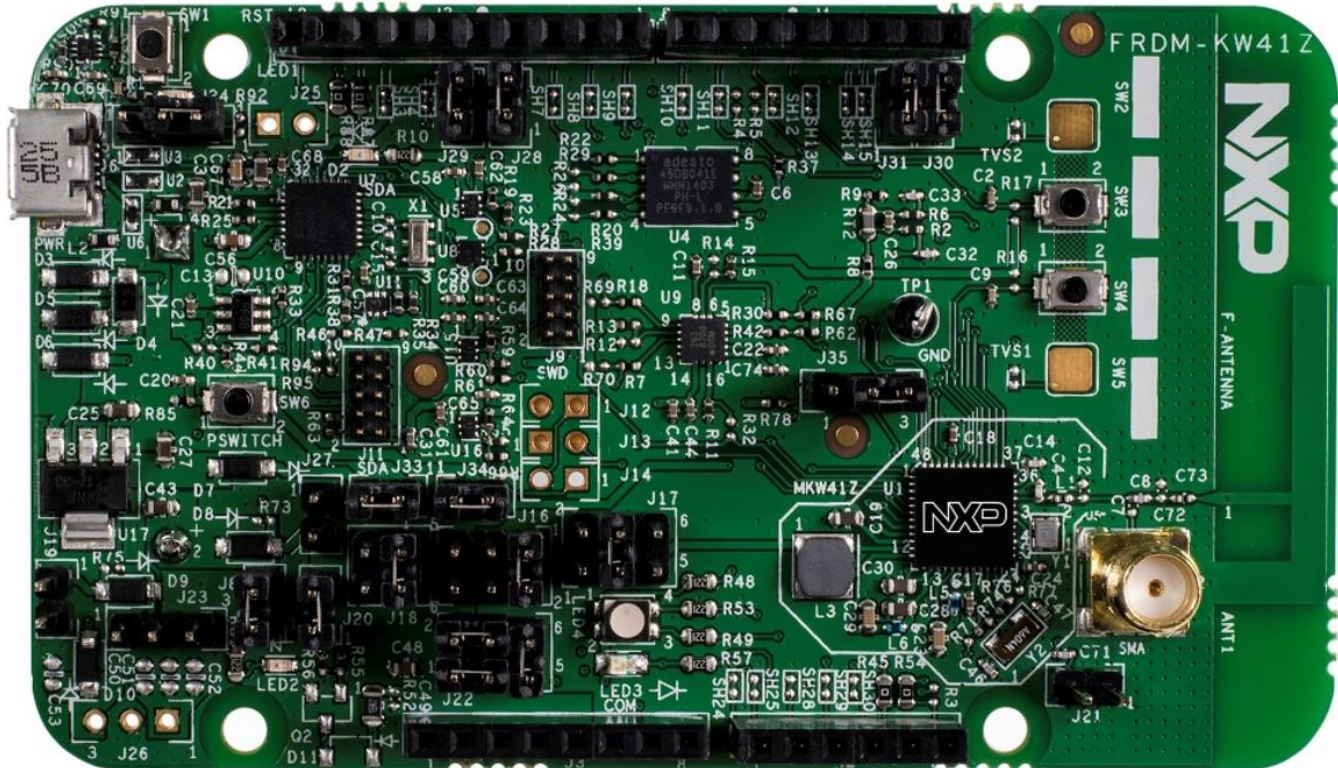


Figure 7. FRDM-KW41Z development board

The device footprint and layout are critical and the RF performance is affected by the design implementation. For these reasons, use of the NXP recommended RF hardware reference designs are important for successful board performance. Additionally, the reference platforms have been optimized for radio performance. If the recommended footprint and design are followed exactly in the RF region of the board; sensitivity, output power, harmonic and spurious radiation, and range will have a high likelihood of first time success.

The following subsections describe important considerations when implementing a wireless hardware design starting with the device footprint, RF circuit implementation, and antenna selection. The following figure shows an example of a typical layout with the critical RF section which must be copied exactly for optimal radio performance. The less critical layout area can be modified without reducing radio performance.

NOTE

Exact dimensions are not given in this document, but can be found in the design files for the FRDM-KW41Z and USB-KW41Z boards.

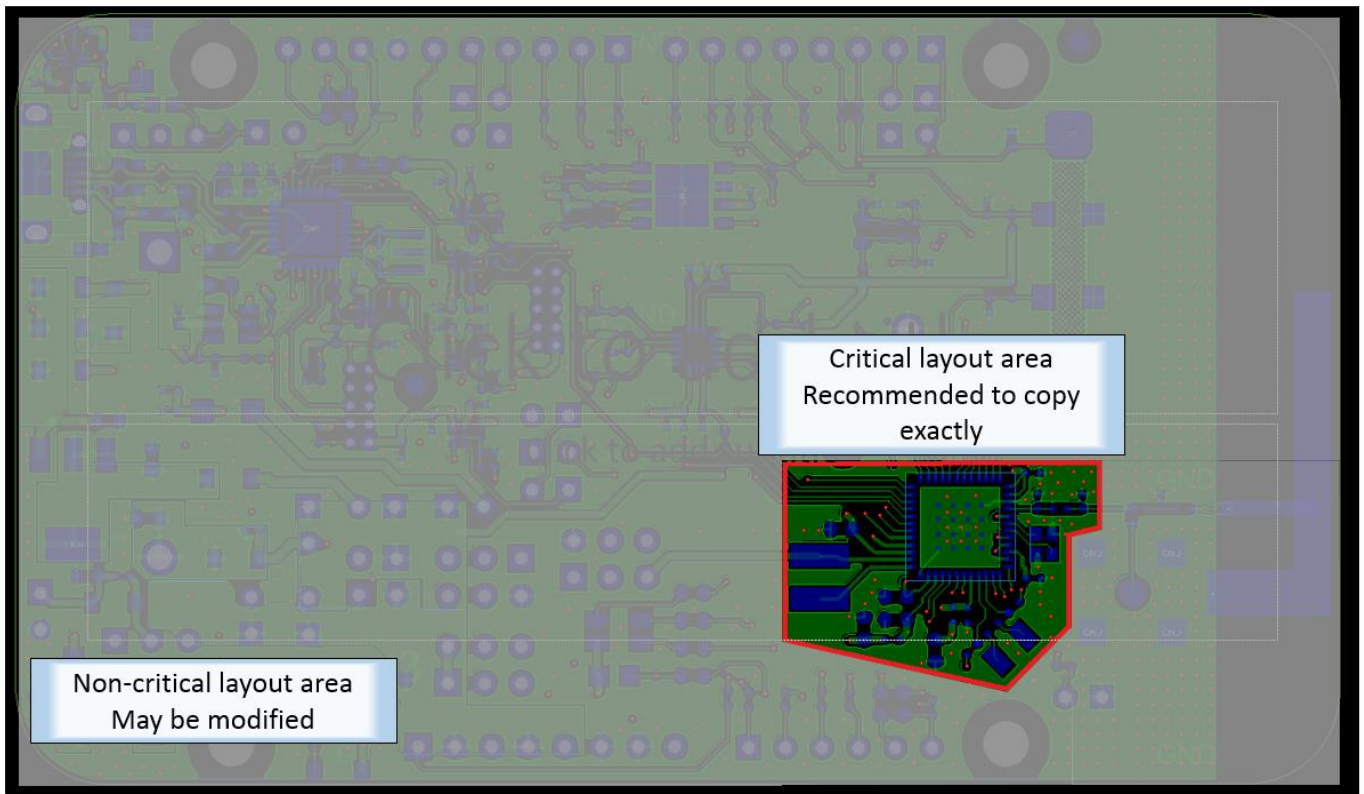


Figure 8. Critical layout areas

5.1. MKW41Z device footprint

The performance of the wireless link is largely influenced by the device's footprint. As a result, a great deal of care has been put into creating a footprint so that receiver sensitivity and output power are optimized to enable board matching and minimal component count. NXP highly recommends copying the die flag exactly as it is shown in the following figure; this includes via locations as well. Deviation from these parameters can cause performance degradation.

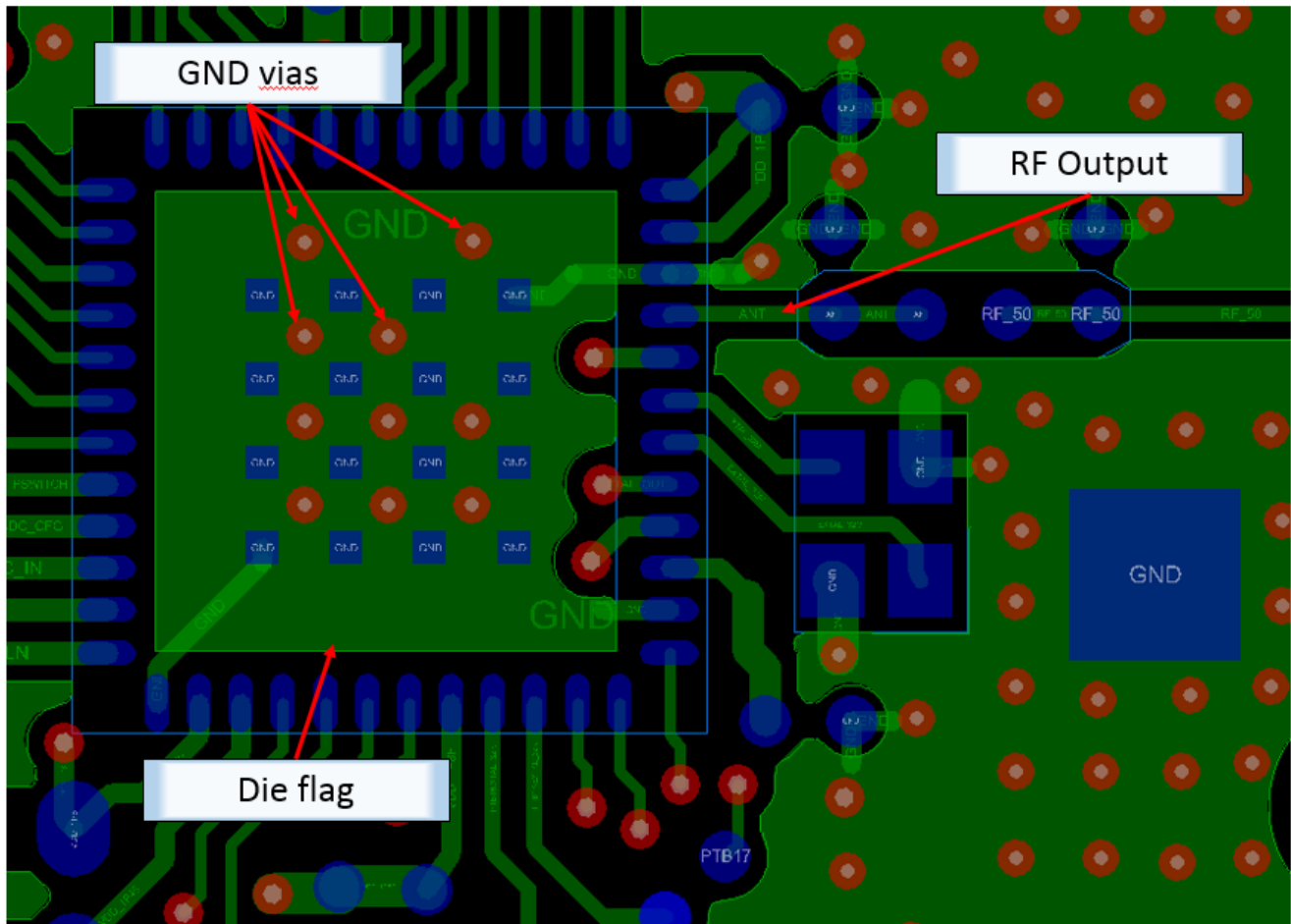


Figure 9. Critical layout of die flag area

Figure 9 shows the critical areas of the device die flag. These are the following:

- Ground vias and locations
- RF output and ground traces
- Die flag shape
- Test pins

As shown in Figure 9, regarding transmission lines, it is important to copy not just the physical layout of the circuit, but also the PCB stackup. Any small change in the thickness of the dielectric substrate under the transmission line will have a significant change in impedance, all this information can be found on the fabrication notes for each board design. As an illustration, consider a 50 ohm trace that is 18 mils wide over 10 mils of FR4. If that thickness of FR4 is changed from 10 to 6 mils, the impedance will only be about 36 ohms.

When the top layer dielectric becomes too thin, the layers will not act as a true transmission line; even though all the dimensions are correct. There is not universal industry agreement on which thickness at which this occurs, but NXP prefers to use a top layer thickness of no less than 8-10 mils. The use of a correct substrate like the FR4 with a dielectric constant of 4.3 will assist you in achieving a good RF design.

Hardware Design Considerations for MKW41Z/31Z/21Z BLE and IEEE 802.15.4 Devices, Application Note, Rev. 1, 03/2017

5.2. RF circuit topology and matching

Transmission lines have several shapes such as microstrip, coplanar waveguide, and stripline. For BLE and 802.15.4 applications built on FR4 substrates, the types of transmission lines typically take the form of microstrip or coplanar waveguide (CPW). These two structures are defined by the dielectric constant of the board material, trace width, and the board thickness between the trace and the ground.

Additionally, for CPW, the transmission line is defined by the gap between the trace and the top edge ground plane. These parameters are used to define the characteristic impedance of the transmission line (trace) that is used to convey the RF energy between the radio and the antenna.

KW41 has a single ended RF output with a 2 component matching network composed of a shunt capacitor and a series inductor. These two elements transform the device impedance to 50 ohms. The value of these components may vary depending on your specific board layout. The recommended RF-matching network is shown in [Figure 10](#).

Avoid routing traces near or parallel to RF transmission lines or crystal signals. Maintaining a continuous ground under an RF trace is critical to maintaining the characteristic impedance of that trace. Avoid any routing on the ground layer that will result in disrupting the ground under the RF traces.

Complexity is the main factor that will determine whether the design of an application board can be two-layer, four-layer, or more. The recommended board stackup for either a two-layer or four-layer board design is as follows:

- Two-layer stackup:
 - Top: RF routing of transmission lines, signals, and ground
 - Bottom: RF reference ground, signal routing, and general ground
- 4-layer stackup:
 - Top: RF routing of transmission lines
 - L2: RF reference ground
 - L3: DC power
 - Bottom: signal routing

For more information, see NXP application note *NXP IEEE 802.15.4 / ZigBee Package and Hardware Layout Considerations* (document [ZHDCRM](#))

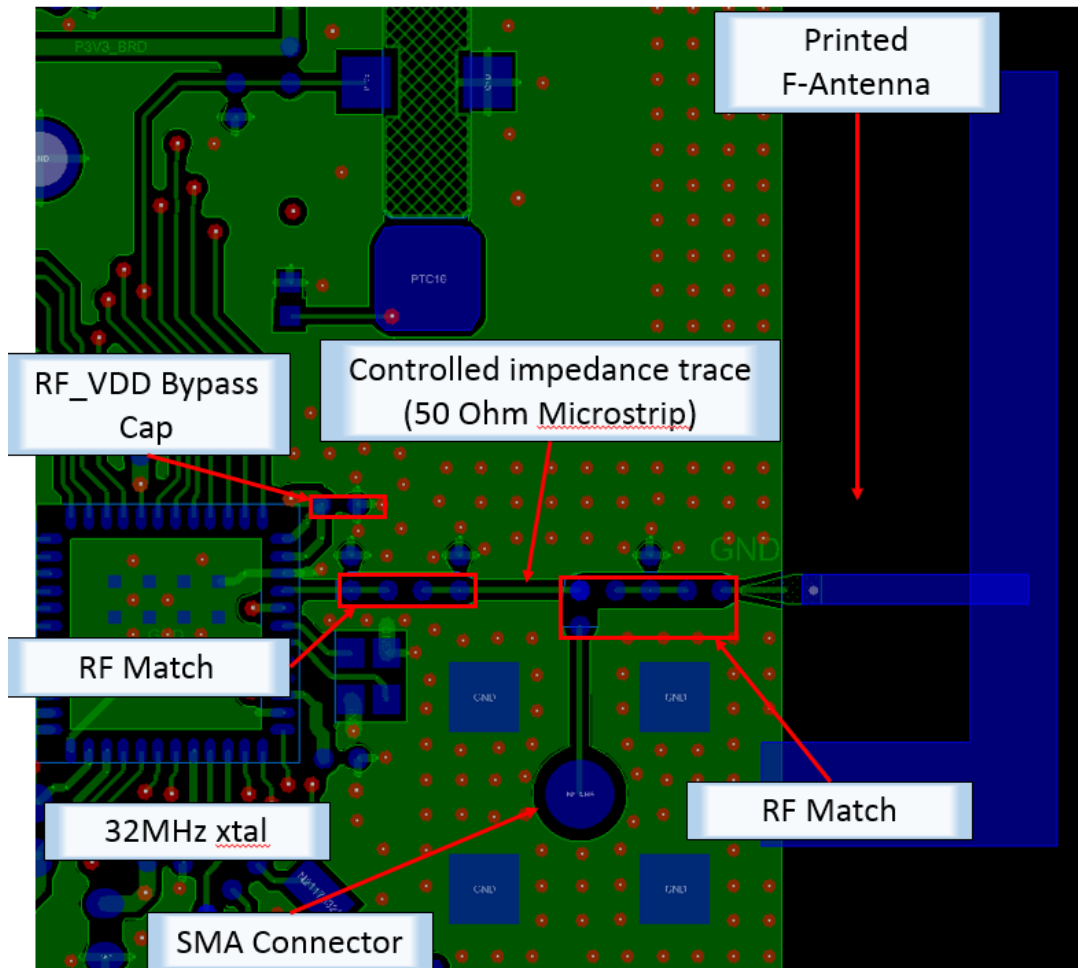


Figure 10. RF matching network

5.3. Antenna considerations

There are a large variety of antenna types available to choose from when designing for a wireless system. These include small footprint chip antennas, trace antennas, loop monopole, and dipole, each with their own set of pros and cons depending on the goal of the application. NXP recommends using one of the proven antenna implementations used in many of our hardware reference designs. For more information on compact antenna designs, see the NXP application note *Compact Integrated Antennas* (document [AN2731](#)).

Steps for good antenna performance:

- Be mindful of critical dimensions:
 - Critical dimensions should be copied exactly.
 - Customer final board sizes may differ from the NXP reference designs. As a result, the last leg of the trace antenna should be made longer to allow for final board tuning.
 - Antenna tuning may be required to operate at the proper frequency. Ideally, the minimum return loss needs to be centered at 2445 MHz. 10 dB return loss looking into the antenna

- at the band edges is sufficient to achieve good range and receive sensitivity.
- Antenna impedance is 50 ohm.
 - This is maintained from balun to antenna feed.
 - The example uses microstrip topology but co-planer waveguide with ground can also be used if desired. In this case the dimensions will change so care should be taken when changing from one topology to another.
- The antenna should be reasonably clear of metallic objects and oriented properly with the ground plane.
- Always check the antenna in its final environment, including the PCB, components, case enclosure, hand effects (if appropriate), and battery. Plastic and other materials in the near-field may cause detuning.
- Actual antenna performance can be evaluated in a variety of ways, such as range testing, measuring radiated signal level under controlled conditions, and characteristic testing in an anechoic chamber.

6. Revision history

Table 1. Revision history

Revision number	Date	Substantive changes
0	11/2016	Initial release
1	03/2017	Figure 5 part number updated

How to Reach Us:

Home Page:

nxp.com

Web Support:

nxp.com/support

Information in this document is provided solely to enable system and software implementers to use NXP products. There are no express or implied copyright licenses granted hereunder to design or fabricate any integrated circuits based on the information in this document. NXP reserves the right to make changes without further notice to any products herein.

NXP makes no warranty, representation, or guarantee regarding the suitability of its products for any particular purpose, nor does NXP assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation consequential or incidental damages. "Typical" parameters that may be provided in NXP data sheets and/or specifications can and do vary in different applications, and actual performance may vary over time. All operating parameters, including "typicals," must be validated for each customer application by customer's technical experts. NXP does not convey any license under its patent rights nor the rights of others. NXP sells products pursuant to standard terms and conditions of sale, which can be found at the following address:

nxp.com/SalesTermsandConditions.

NXP, the NXP logo, NXP SECURE CONNECTIONS FOR A SMARTER WORLD, and Tower, are trademarks of NXP B.V. All other product or service names are the property of their respective owners.

ARM, the ARM Powered logo, and Cortex are registered trademarks of ARM Limited (or its subsidiaries) in the EU and/or elsewhere. All rights reserved.

© 2016-2017 NXP B.V.

Document Number: AN5377

Rev. 1

03/2017

