AN12518

Hardware Design Considerations for MKW39A/38A/37A/38Z/37Z Bluetooth LE Devices

Rev. 0 — April 2020 Application Note

Contents

1 Introduction... 1
2 QFN component coppoer layer.............1
3 QFN package dimensions..................... 4
4 QFN soldering profile............................6
5 Design and board layout considerations....................................7

1 Introduction

This application note describes Printed Circuit Board (PCB) design considerations for the MKW39A/38A/37A/38Z/37Z 48-pin QFN (HVQFN-7 × 7 Pitch 0.5 mm) wettable flank 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.

2 QFN component coppoer layer

2.1 48-pin HVQFN

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 **Wettable** HVQFN package (7 × 7 × 0.85 mm) consists of 48 IC contact pads, and nine centered ground pads. The copper pattern is as 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.

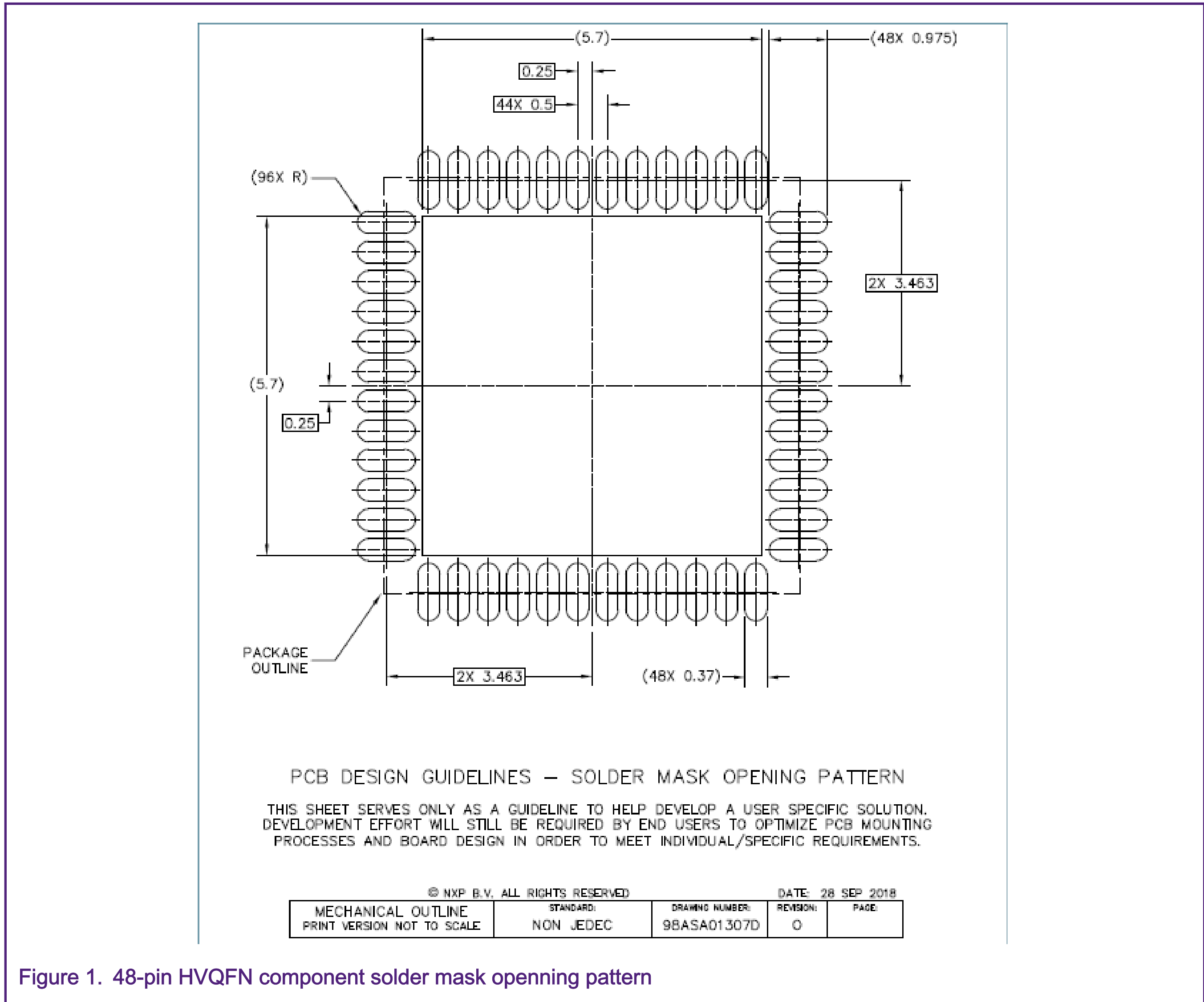


Figure 1. 48-pin HVQFN component solder mask openning pattern

2.1.1 48-pin HVQFN solder mask

The solder mask limits the flow of the solder paste during the reflow process. Figure 2 shows a recommended solder mask pattern. The pattern represents openings in the solder mask.

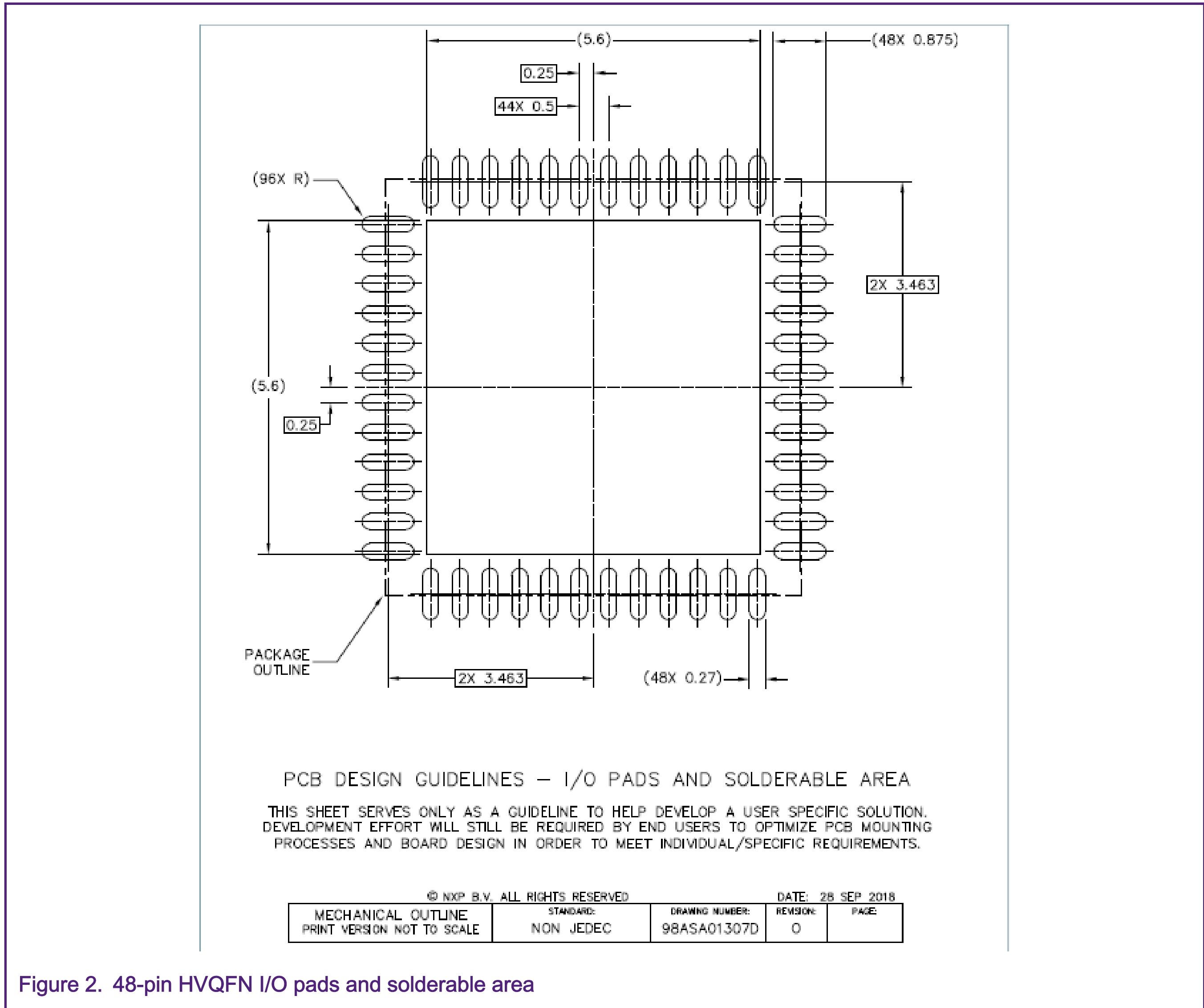


Figure 2. 48-pin HVQFN I/O pads and solderable area

2.1.2 48-pin HVQFN solder paster 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.

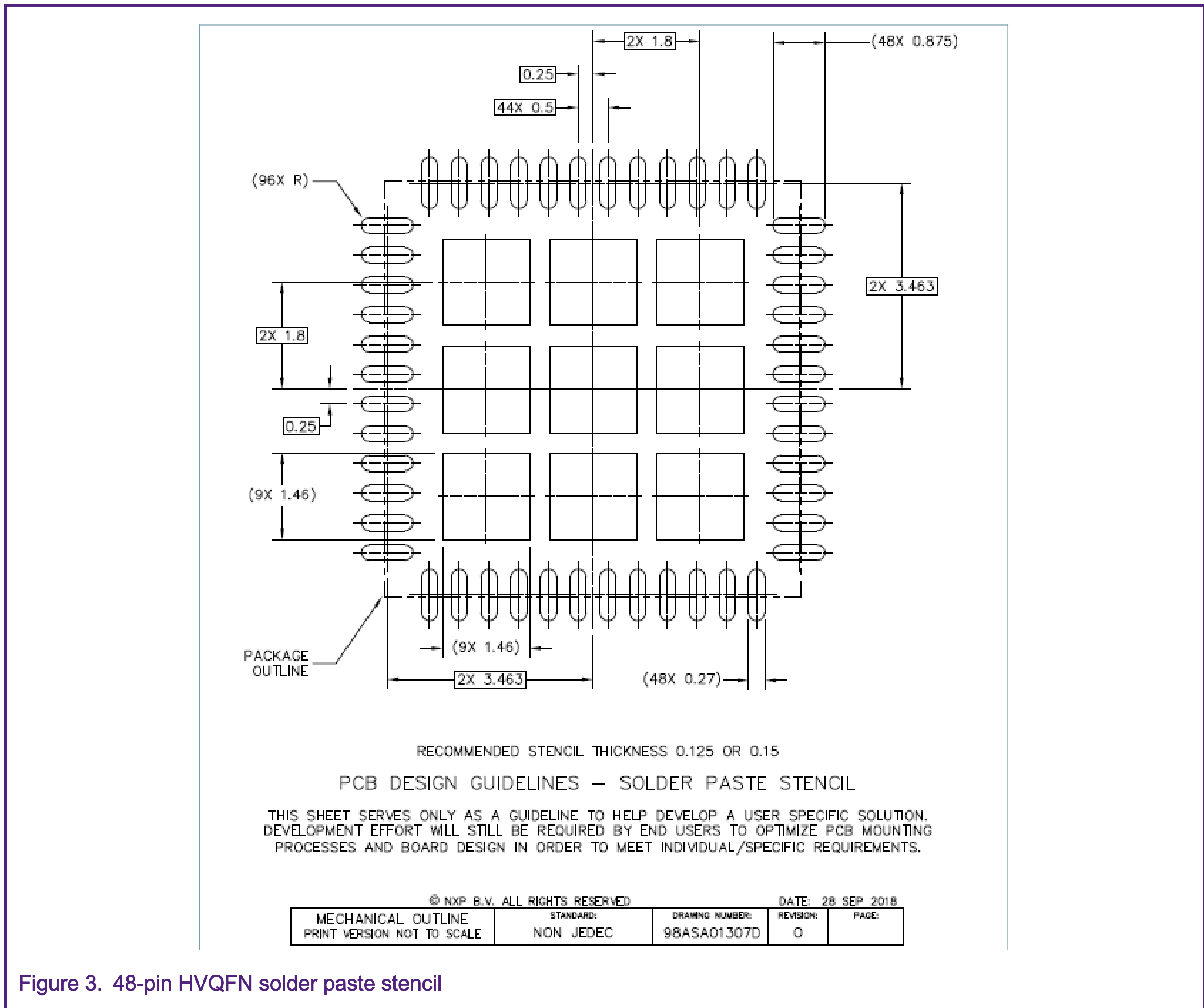


Figure 3. 48-pin HVQFN solder paste stencil

2.2 QFN problems with excess solder

Excess solder may cause the QFN 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 QFN package dimensions

Figure 4 shows the 48-pin HVQFN package dimensions.

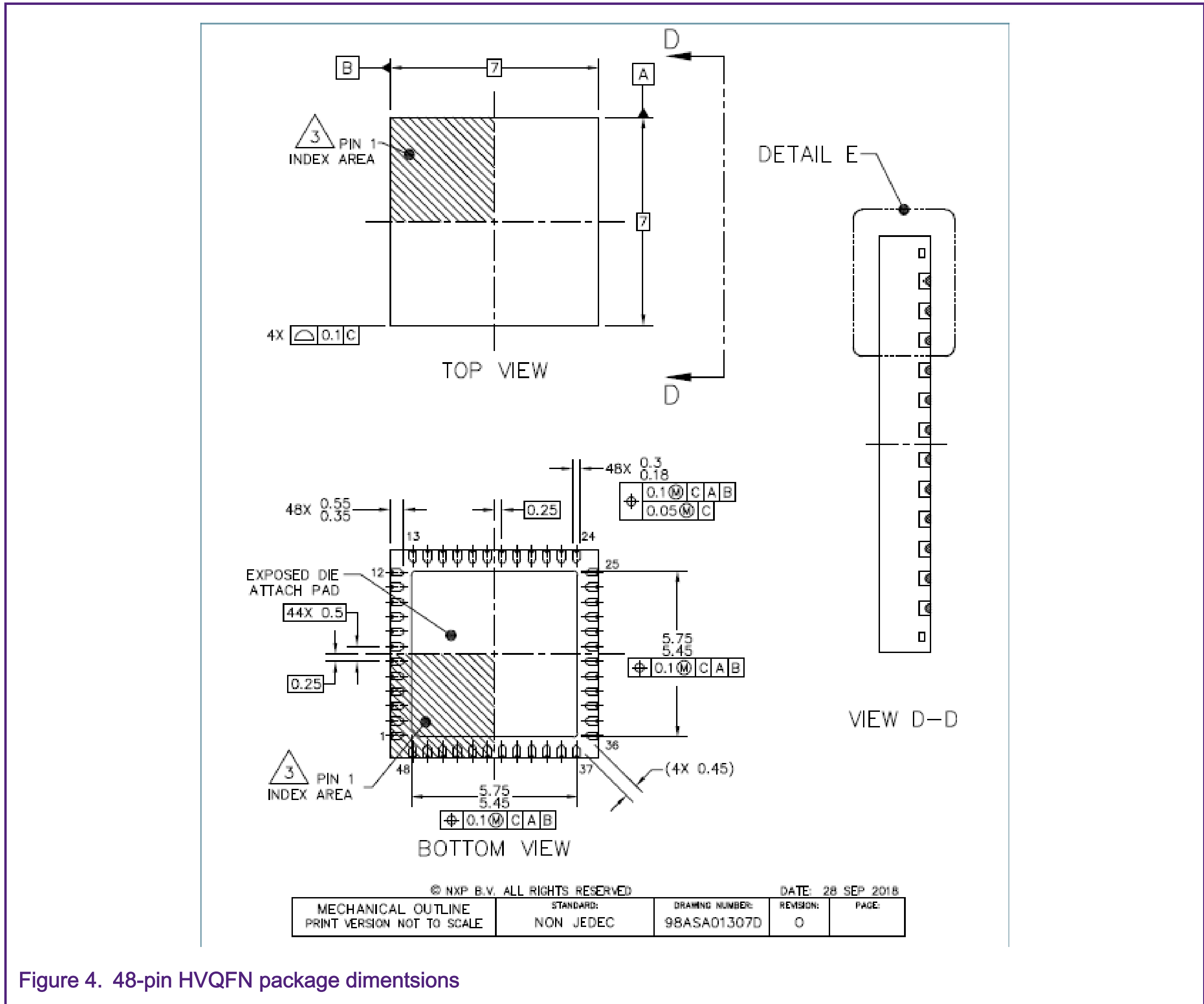


Figure 4. 48-pin HVQFN package dimentsions

3.1 48-pin HVQFN device marking details

The MKW39A, MKW38A, MKW37A, MKW38Z and MKW37Z devices are in the 48-pin HVQFN (7 × 7 × 0.85 mm). Figure 5 shows device marking examples for the HVQFN device.

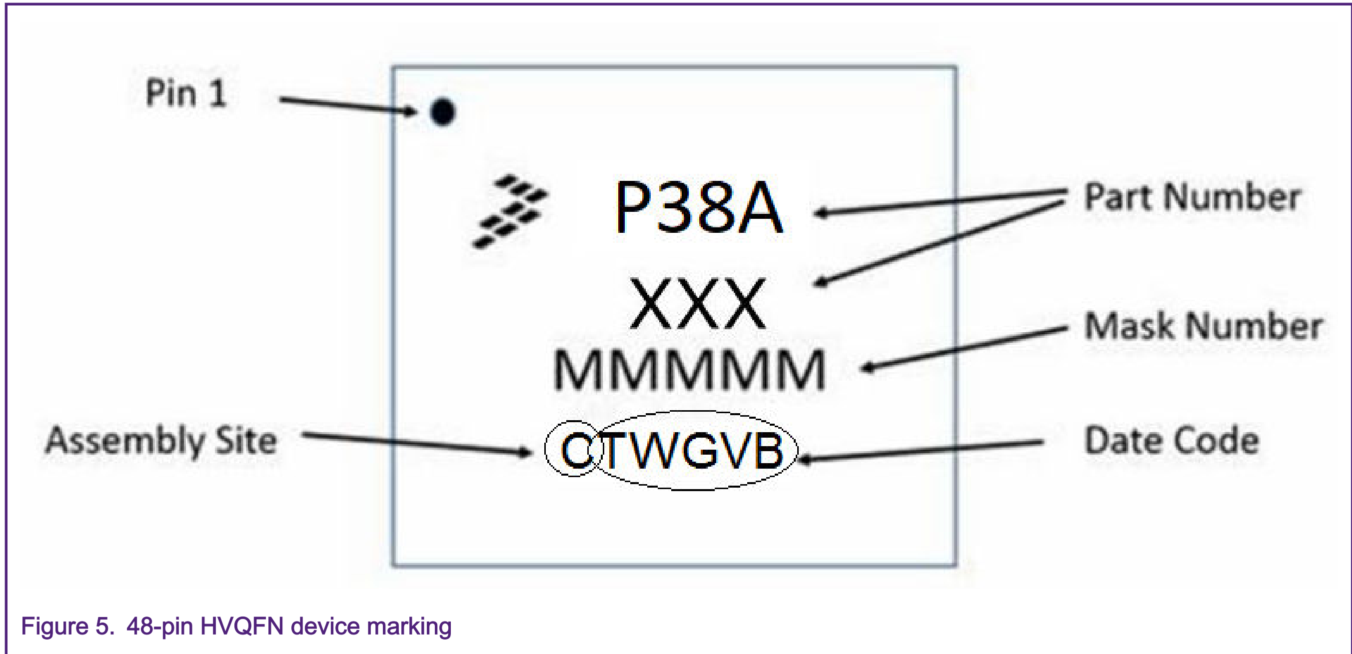


Figure 5. 48-pin HVQFN device marking

NOTE

Your device part number may differ from the part number, as shown in Figure 5.

4 QFN soldering profile

Figure 6 shows the recommended soldering profile for the 48 pins - HVQFN package, in a board size approximately 3.20 × 2.10 inches.

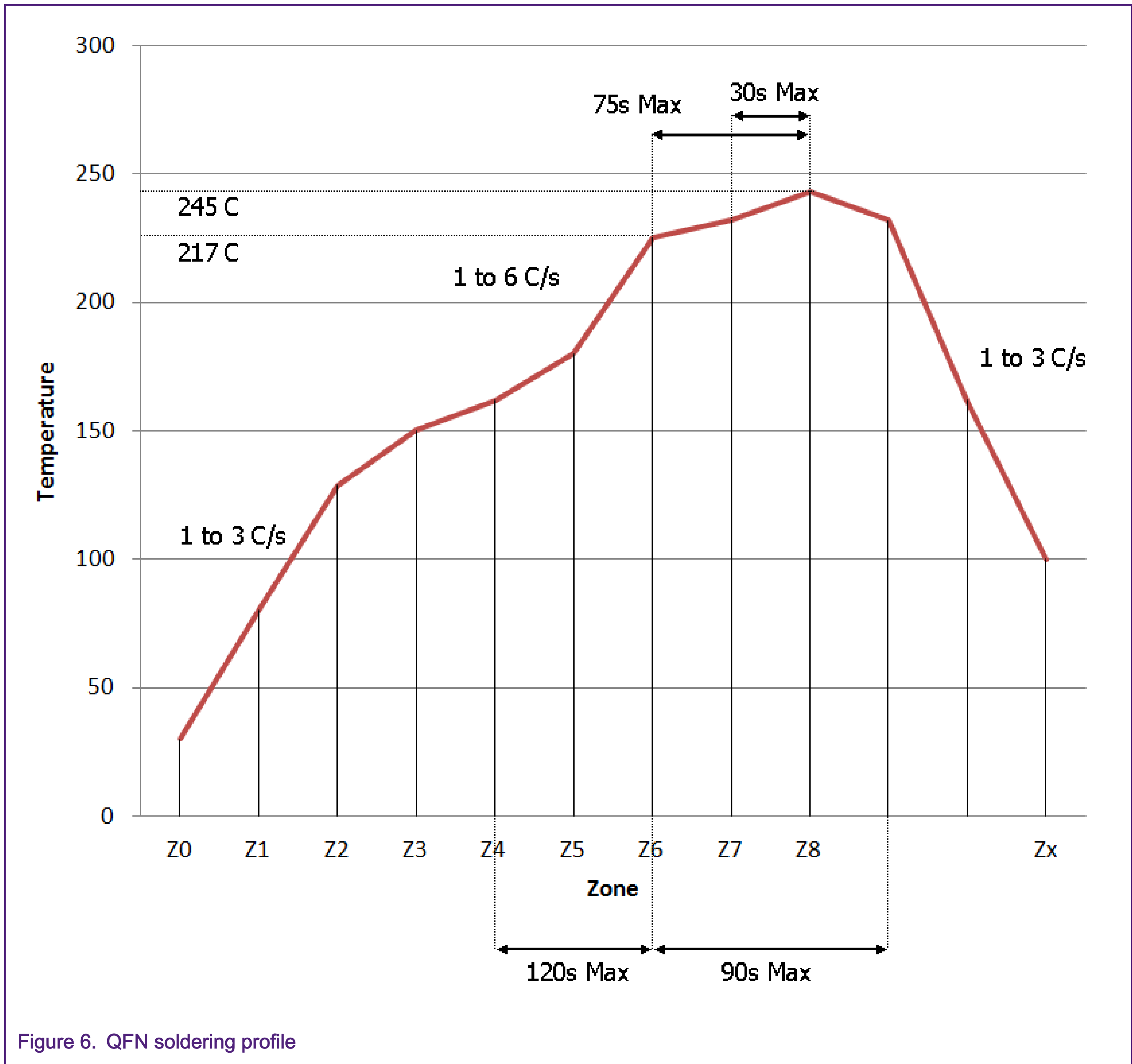


Figure 6. QFN soldering profile

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 radio boards with good performance levels. Figure 7 shows the FRDM-KW38 development board. It contains the MKW38A device and all necessary I/O connections.

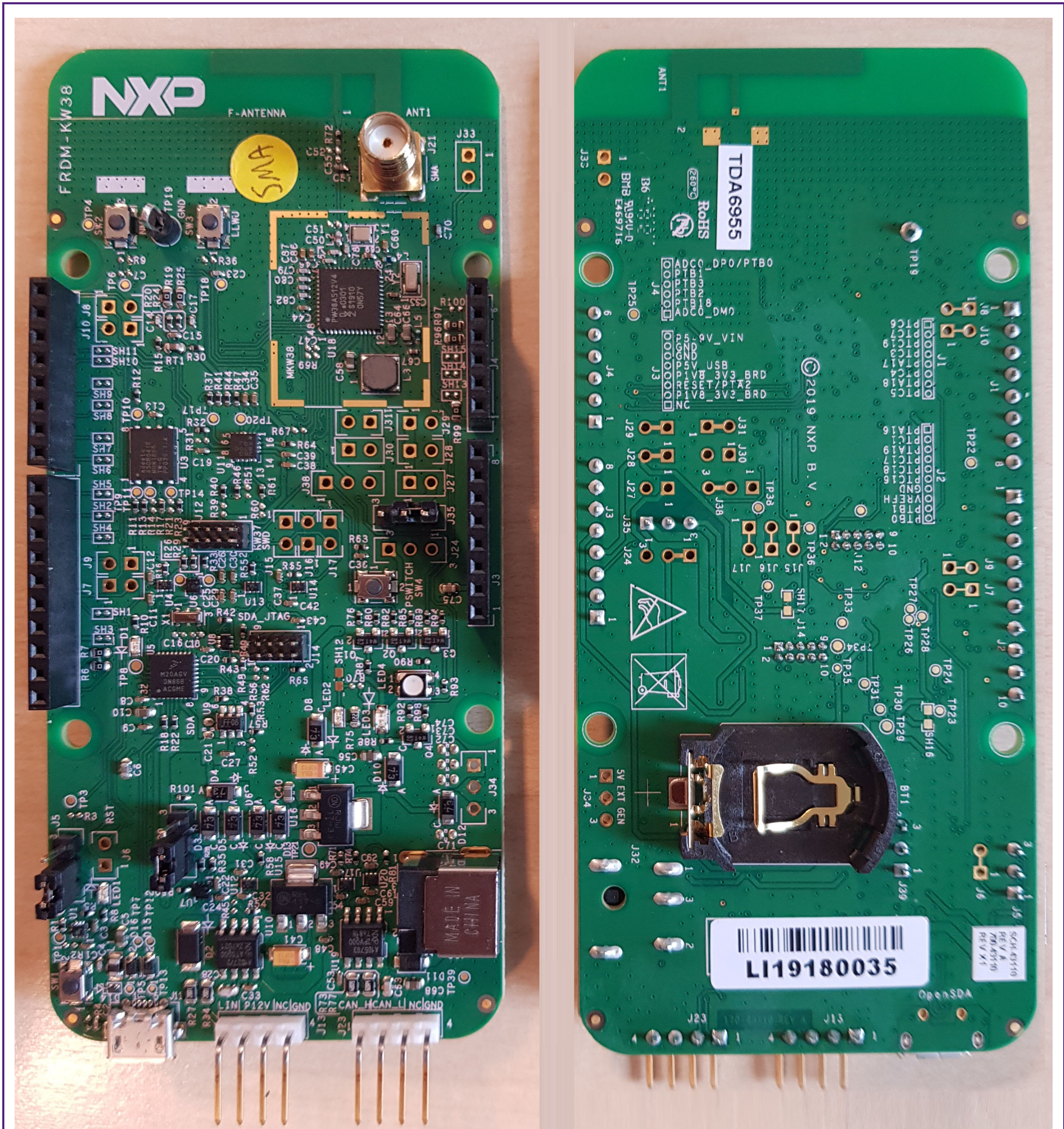


Figure 7. FRDM-KW38 development board

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. Figure 8 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-KW38 board.

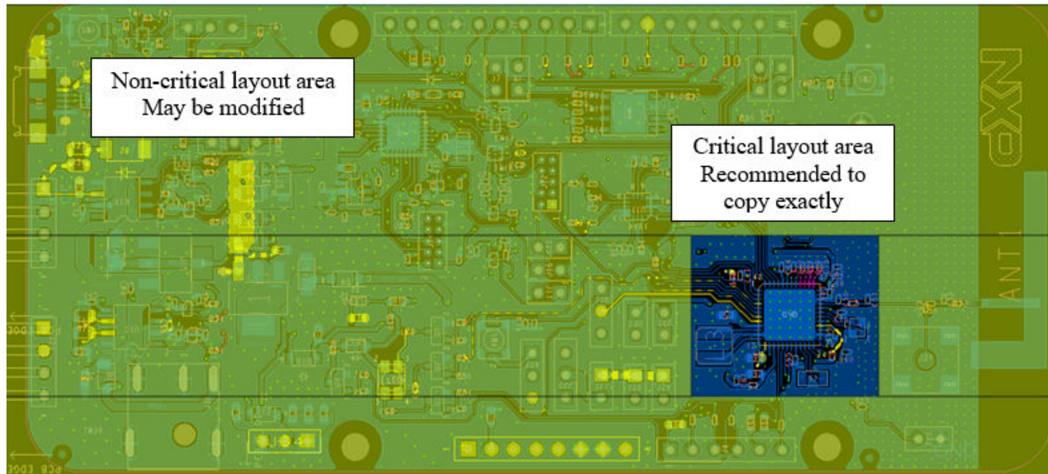


Figure 8. Critical layout areas

WARNING

The HVQFN 48 pins wettable package is pin to pin compatible from the KW36 to the KW38 product but the LPUART0 pins used for the bootloader are not:

- KW36: PTC2 (Rx) & PTC18 (Tx)
- KW38: TPC6 (Rx) & PTC7 (Tx)

al/MKW36A512RM.pdf

Chapter 11
Kinetis Flashloader

11.1 Chip-Specific Information

This device has various peripherals (LPUART, I2C, SPI, CAN) supported by the Kinetis Flashloader. The following table shows the pads used by the Kinetis Flashloader.

Table 11-1. Kinetis Flashloader Peripheral Pinmux

Peripheral	Instance	Alt mode	Port
LPUART	0	4	PTC2, LPUART0_RX PTC18, LPUART0_TX
I2C	0	3	PTB0, I2C0_SCL PTB1, I2C0_SDA
SPI	0	2	PTC16, SPI0_SCK PTC17, SPI0_SOUT PTC18, SPI0_SIN PTC19, SPI0_PCS0
CAN	0	9	PTC3, CAN0_TX PTC4, CAN0_RX

MKW39A512RM.pdf

Chapter 11
Kinetis Flashloader

11.1 Chip-Specific Information

This device has various peripherals (LPUART, I2C, SPI, CAN, LIN) supported by the Kinetis Flashloader. The LIN peripheral is supported in KW38 family only. The following table shows the pads used by the Kinetis Flashloader.

Table 11-1. Kinetis Flashloader Peripheral Pinmux

Peripheral	Instance	Alt mode	Port
LPUART	0	4	PTC6, LPUART0_RX PTC7, LPUART0_TX
I2C	0	3	PTB0, I2C0_SCL PTB1, I2C0_SDA
SPI	0	2	PTC16, SPI0_SCK PTC17, SPI0_SOUT PTC18, SPI0_SIN PTC19, SPI0_PCS0
CAN	0	9	PTC3, CAN0_TX PTC4, CAN0_RX
LIN (LPUART1)	1	3 3 GPIO	PTA17, LIN_TX_LS PTA18, LIN_TX_LS PTC5, LIN_SLIP

5.1 MKW38A 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 shown in Figure 9. This includes via locations as well. Deviation from these parameters can cause performance degradation.

SOT917 is a recommendation.

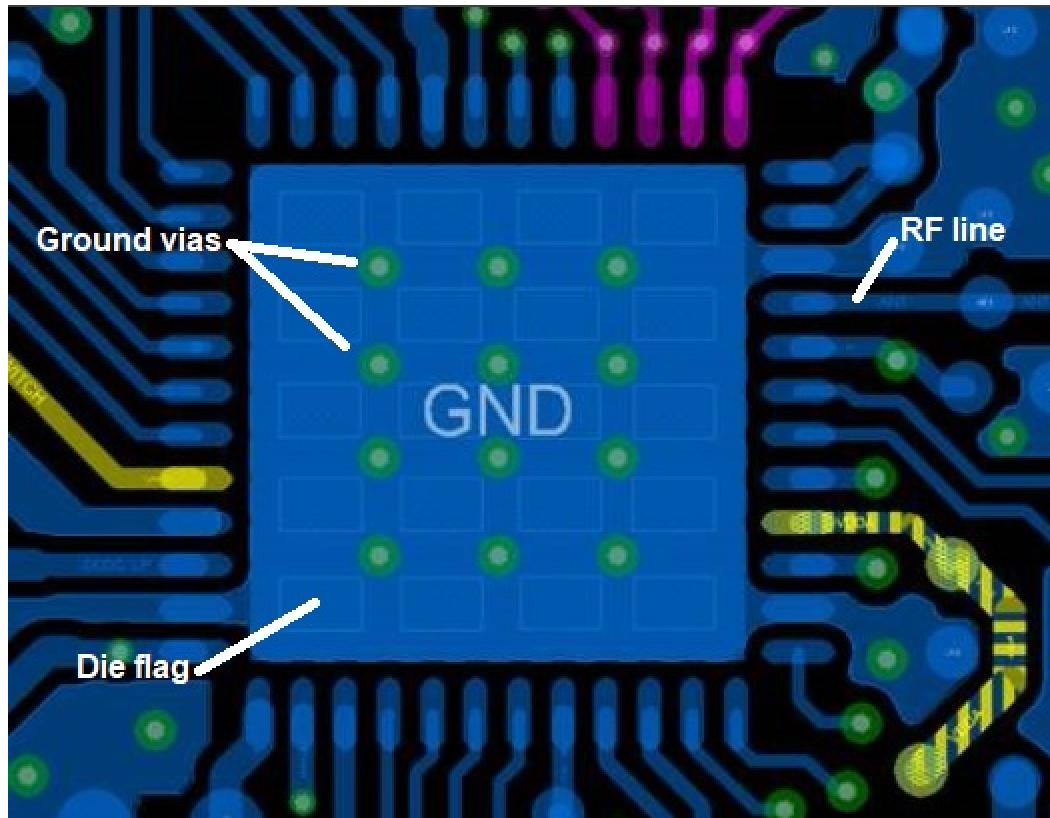


Figure 9. Critical layout of die flag area

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

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

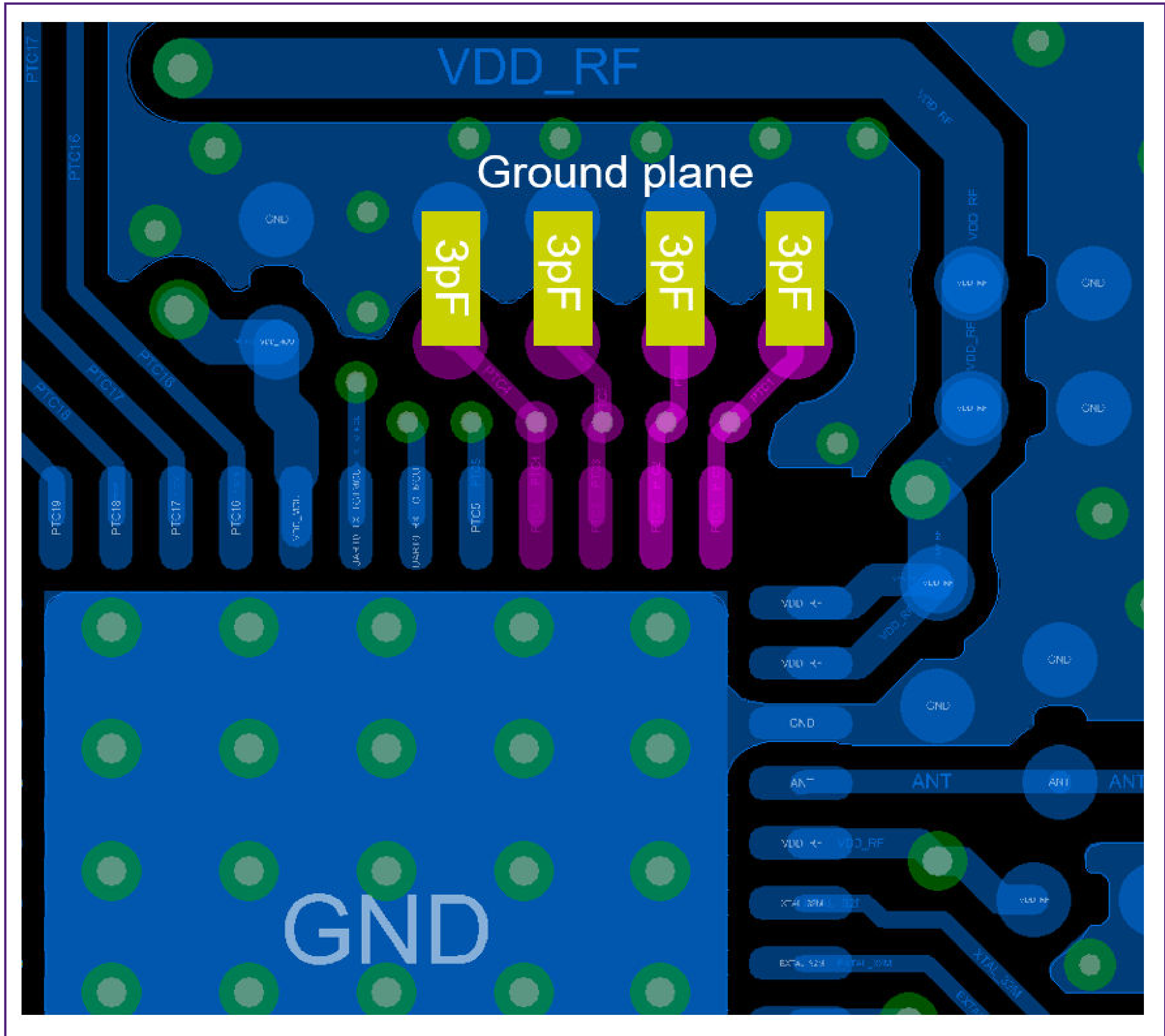
As shown in Figure 9 regarding transmission lines, it is important to copy not just the physical layout of the circuit, but also the PCB stackup. Any small change in the thickness of the dielectric substrate under the transmission line will have a significant change in impedance, all this information can be found on the fabrication notes for each board design. As an illustration, consider a 50 Ω 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.

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.

Recommendation to reach acceptable EMC during the radio certification:

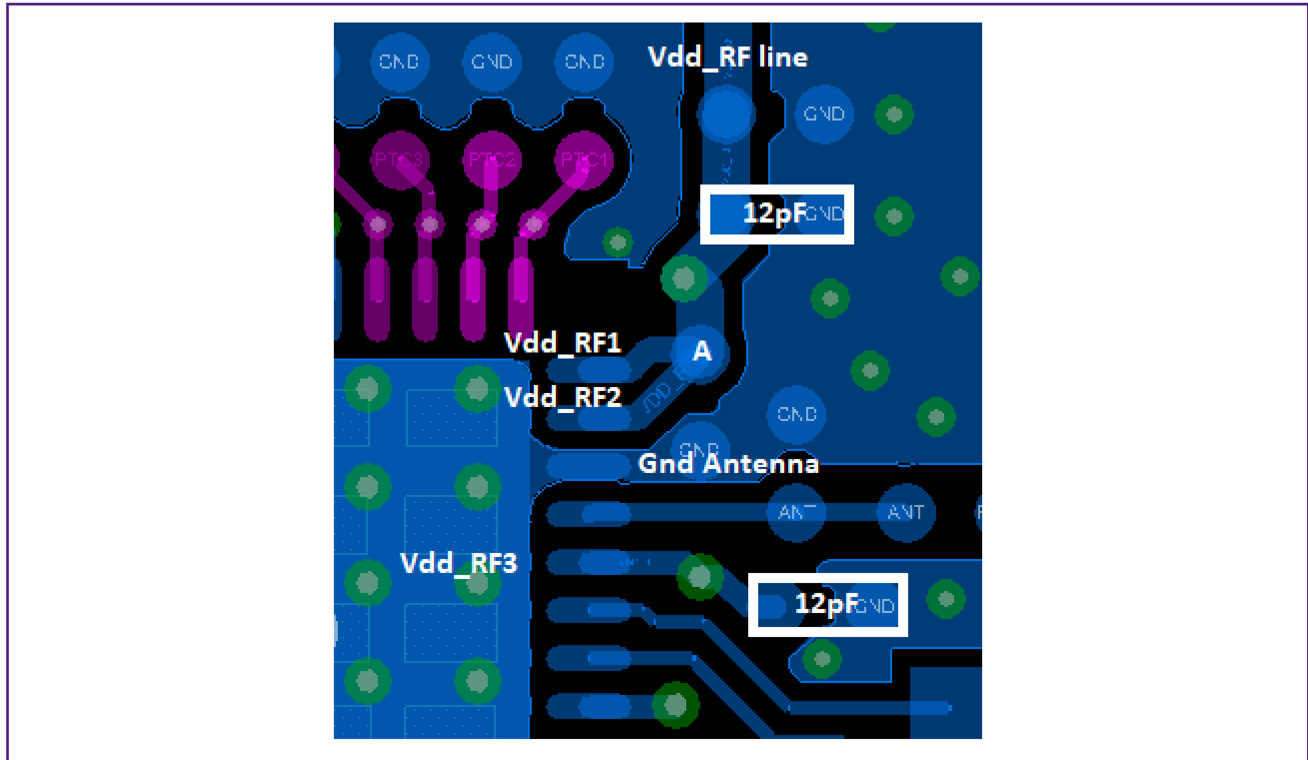
- A specific attention must be taken on 4 pins PTC1, 2, 3 & 4 if they are used on application.
- Four decoupling capacitors of 3pF are mandatory on those pins and be positioned closely to the KW38 pins.
- Wires from those four pins must be underlayer
- NXP recommend to put the vias under the KW38 package in case of the customer HW design rules allows it.

Find an example of the recommended layout:



Recommendation to perform a good `Vdd_RF` supply layout:

- `Vdd_RF1` and `Vdd_RF2` lines must have the same length as possible linked to pointA (Y connection).
- 12 pF decoupling capacitor from `Vdd_RF` wire must be connected to the Ground Antenna. Purpose is to get the path as short as possible from `Vdd_RF1`/`Vdd_RF2` to the ground antenna.
- 12 pF decoupling capacitor from `Vdd_RF3` pin must be as close as possible. Return to ground must be as short as possible. So vias (2 in image below) must be placed near to the decoupling capacitor to get close connection to the ground layer.



5.2 RF circuit topology and matching

Transmission lines have several shapes such as microstrip, coplanar waveguide, and stripeline. For Bluetooth LE applicationsbuilt 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.

KW38 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 Ω. The value of these components may vary depending on your specific board layout. Figure 10 shows the recommended RF-matching network.

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:

- 2-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 IEEE 802.15.4/ ZigBee Packageand Hardware Layout Considerations* (document ZHDCRM)

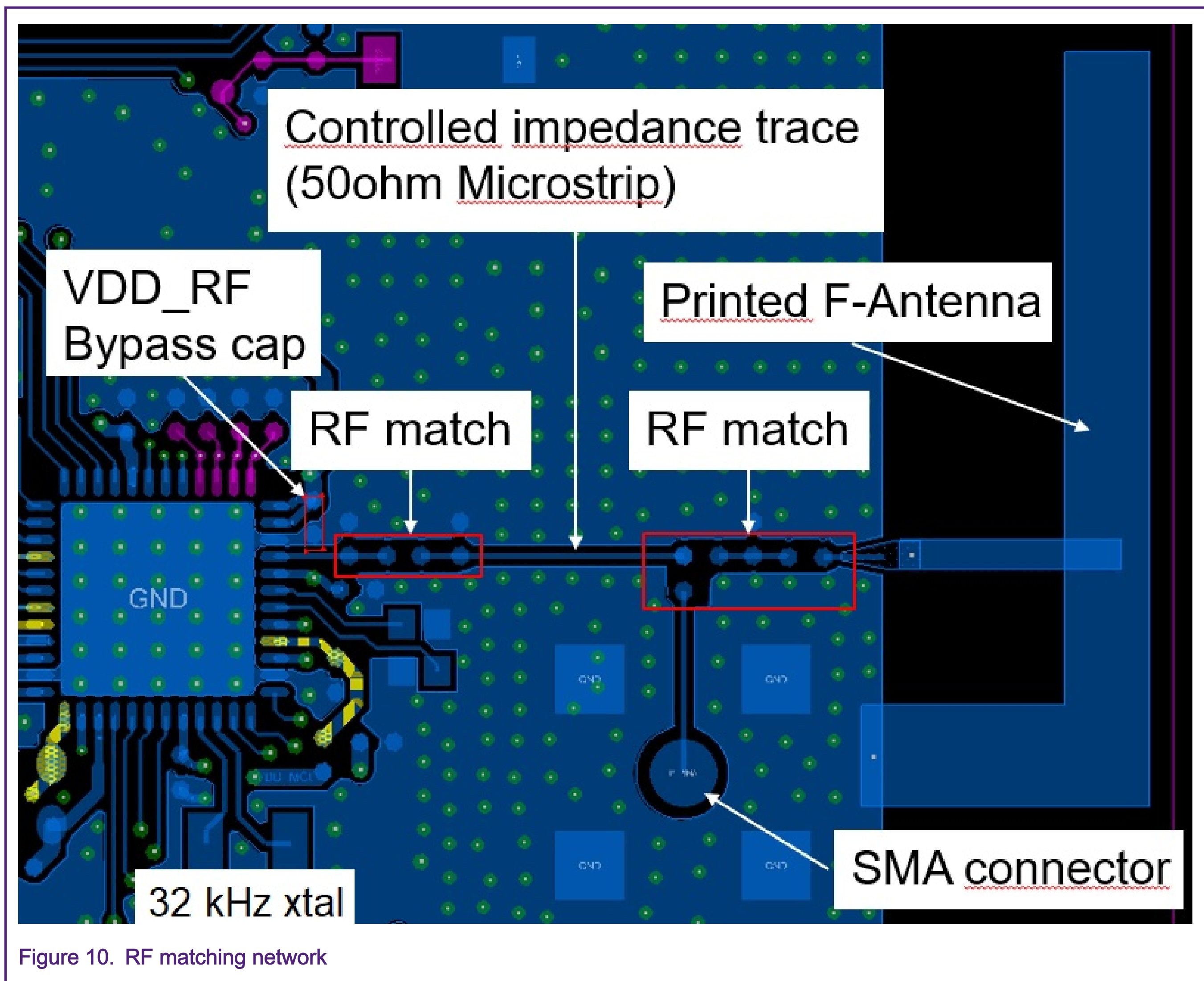


Figure 10. RF matching network

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 *Compact Integrated Antennas* (document AN2731).

Steps for good antenna performance:

- Be mindful of critical dimensions:
  - Critical dimensionsshould 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 Ω.

  - This is maintained from RF matched port/pin 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.

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.

While NXP has implemented advanced security features, all products may be subject to unidentified vulnerabilities. Customers are responsible for the design and operation of their applications and products to reduce the effect of these vulnerabilities on customer’s applications and products, and NXP accepts no liability for any vulnerability that is discovered. Customers should implement appropriate design and operating safeguards to minimize the risks associated with their applications and products.

NXP, the NXP logo, NXP SECURE CONNECTIONS FOR A SMARTER WORLD, COOLFLUX, EMBRACE, GREENCHIP, HITAG, I2C BUS, ICODE, JCOP, LIFE VIBES, MIFARE, MIFARE CLASSIC, MIFARE DESFire, MIFARE PLUS, MIFARE FLEX, MANTIS, MIFARE ULTRALIGHT, MIFARE4MOBILE, MIGLO, NTAG, ROADLINK, SMARTLX, SMARTMX, STARPLUG, TOPFET, TRENCHMOS, UCODE, Freescale, the Freescale logo, AltiVec, C-5, CodeTEST, CodeWarrior, ColdFire, ColdFire+, C-Ware, the Energy Efficient Solutions logo, Kinetis, Layerscape, MagniV, mobileGT, PEG, PowerQUICC, Processor Expert, QorIQ, QorIQ Qonverge, Ready Play, SafeAssure, the SafeAssure logo, StarCore, Symphony, VortiQa, Vybrid, Airfast, BeeKit, BeeStack, CoreNet, Flexis, MXC, Platform in a Package, QUICC Engine, SMARTMOS, Tower, TurboLink, UMEMS, EdgeScale, EdgeLock, eIQ, and Immersive3D are trademarks of NXP B.V. All other product or service names are the property of their respective owners. AMBA, Arm, Arm7, Arm7TDMI, Arm9, Arm11, Artisan, big.LITTLE, Cordio, CoreLink, CoreSight, Cortex, DesignStart, DynamIQ, Jazelle, Keil, Mali, Mbed, Mbed Enabled, NEON, POP, RealView, SecurCore, Socrates, Thumb, TrustZone, ULINK, ULINK2, ULINK-ME, ULINK-PLUS, ULINKpro, μVision, Versatile are trademarks or registered trademarks of Arm Limited (or its subsidiaries) in the US and/or elsewhere. The related technology may be protected by any or all of patents, copyrights, designs and trade secrets. All rights reserved. Oracle and Java are registered trademarks of Oracle and/or its affiliates. The Power Architecture and Power.org word marks and the Power and Power.org logos and related marks are trademarks and service marks licensed by Power.org.

© NXP B.V. 2020. All rights reserved.

For more information, please visit: http://www.nxp.com
For sales office addresses, please send an email to: salesaddresses@nxp.com

Date of release: April 2020
Document identifier: AN12518