Application Note

CC23xx and CC27xx Hardware Configuration and PCB Design Considerations

Haiatullah Gholami

ABSTRACT

This application note provides design guidelines for the CC23xx and CC27xx SimpleLink™ ultra-low-power wireless microcontroller unit (MCU) platform. This document includes an overview of the different reference designs followed by RF front-end, schematic, PCB, and antenna design considerations. The application note also covers crystal oscillator tuning, optimum load impedance and a brief explanation of the different power supply configurations. The last section of this document provides a summary of steps to carry out at board bring-up.

Table of Contents

List of Figures

List of Tables

Trademarks

SimpleLink™, LaunchPads™, LaunchPad™, and SmartRF™ are trademarks of Texas Instruments.
Bluetooth® is a registered trademark of Bluetooth SIG.
All trademarks are the property of their respective owners.

1 Introduction

Designing high-performance wireless product with CC23xx and CC27xx device families careful hardware configuration and PCB layout. To optimize the performance of these systems, users must follow well-established design practices for RF performance, power efficiency, and regulatory compliance.

This application note provides comprehensive guidelines for hardware configuration and PCB design tailored to the CC23xx and CC27xx devices. This document addresses critical aspects such as RF front-end, crystal oscillator configuration, tuning, and layout. The recommendations are based on TI's extensive validation and reference designs; helping users accelerate time to market while minimizing design risks.

2 Reference Designs

LaunchPads™ are the main development platform for CC23xx and CC27xx devices. A LaunchPad™ includes optimized external RF components onboard and a PCB antenna providing an easy-to-use development environment with a single core software development kit (SDK) and rich tool set. Each CC23xx and CC27xx device family is featured on a dedicated LaunchPad with RF matching network and an antenna optimized for operation at one or more of the supported ISM bands. All TI LaunchPad design files, including Gerber-files and CAD source, are available for download on the products pages at TI.com and can be used as a reference design when integrating CC23xx or CC27xx into custom hardware.

2.1 LP-EM-CC2340R53

LP-EM-CC2340R53 is a single-ended a LaunchPad that operates in 2.4GHz ISM band and supports BLE 5.4 and earlier LE specifications. The RF front end enables up to +8dBm output power.

Featured device:	CC2340R53 RKP Package
ISM band:	2.4GHz
Antenna:	*2.4GHz Inverted F Antenna* application note
Design files:	*LP-EM-CC2340R53 Design Files*

2.2 LP-EM-CC2340R5

LP-EM-CC2340R5 is a single-ended a LaunchPad that operates in 2.4GHz ISM band and supports BLE 5.4 and earlier LE specifications. The RF front end enables up to +8dBm output power.

Featured device:	CC2340R5 RKP Package
ISM band:	2.4GHz
Antenna:	*2.4GHz Inverted F Antenna* application note
Design files:	*LP-EM-CC2340R5 Design Files*

2.3 LP-EM-CC2340R5-Q1

LP-EM-CC2340R5-Q1 is a single-ended LaunchPad that operates in 2.4GHz ISM band and supports BLE 5.4 and earlier LE specifications. The RF front end enables up to +8dBm output power.

Featured device:	CC2340R5-Q1
ISM band:	2.4GHz
Antenna:	*2.4GHz Inverted F Antenna* application note
Design files:	*LP-EM-CC2340R5-Q1 Design Files*

2.4 LP-EM-CC2340R5-RGE-4x4-IS24

LP-EM-CC2340R5-RGE-4x4-IS24 is a single-ended LaunchPad that operates in 2.4GHz ISM band and supports BLE 5.4 and earlier LE specifications. The RF front end enables up to +8dBm output power.

Featured device:	CC2340R5 RGE Package
ISM band:	2.4GHz
Antenna:	*2.4GHz Inverted F Antenna* application note
Design files:	*LP-EM-CC2340R5-RGE-4x4-IS24 Design Files*

2.5 LP-EM-CC2745R10-Q1

LP-EM-CC2745R10-Q1 is a single-ended LaunchPad that operates in 2.4GHz ISM band and supports BLE 5.4 and earlier LE specifications. The RF front end enables up to +10dBm output power.

Featured device:	CC2745R10-Q1
ISM band:	2.4GHz
Antenna:	*2.4GHz Inverted F Antenna* application note
Design files:	*LP-EM-CC2745R10-Q1 Design Files*

3 Schematic

3.1 Schematic Overview

Figure 3-1 shows the RF section and components described.

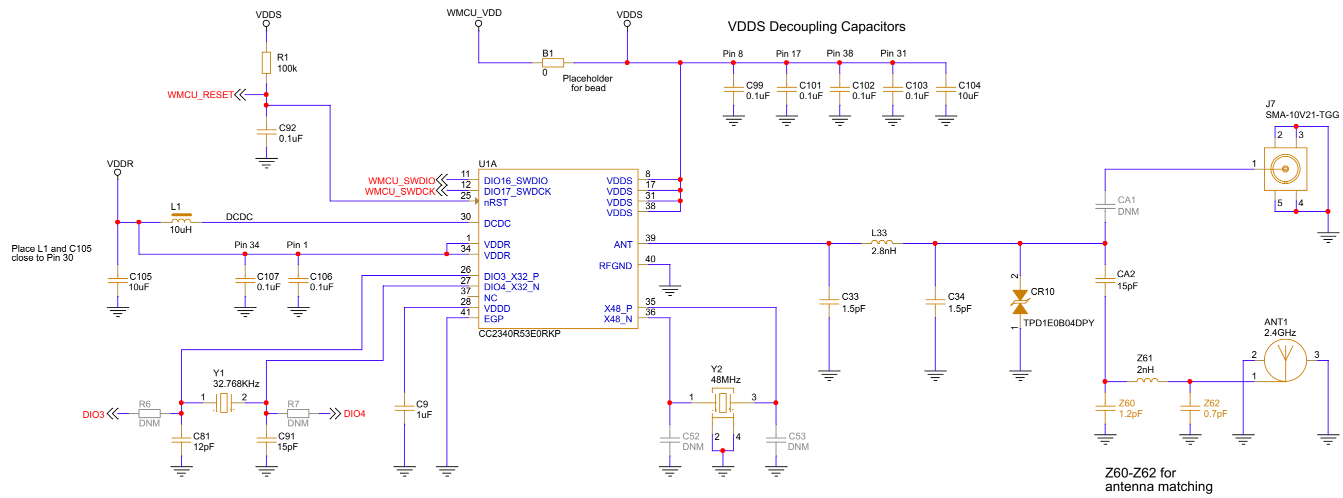

Figure 3-1. RF Section and Components of the CC2340R53 Schematic

3.1.1 48MHz Crystal

A 48MHz crystal is required as the frequency reference for the radio. For CC23xx and CC27xx, the internal Cap Array can be used instead of external load capacitors for the 48MHz crystal. For more details on how to set the internal Cap Array and how to tune the HFXT, refer to Section 6.4.

3.1.2 32.768kHz Crystal

The 32kHz crystal is used as the reference for the RTC clock. CC27xx family of devices do not require any external 32kHz crystal. The RC oscillator can be calibrated automatically to provide a sleep timer accurate enough for Bluetooth® Low Energy. Using an external crystal has an advantage that increases sleep clock accuracy and reduces the power consumption for Bluetooth Low Energy by having shorter receive windows around connection events.

For the CC23xx family of devices, the internal low-speed RC oscillator (32kHz) can be used as a reference instead of an external 32kHz crystal in some cases. See Section 12 for the device-specifc errata for which cases the LFOSC can be used.

The *Low Frequency Clock Source* can be set in Sysconfig *TI DEVICES* section under *Device Configuration* in *Device Configuration*.

1. LF XOSC: external crystal
2. LF RCOSC: internal RC oscillator, DIO23_X32P and DIO24_X32N can be left floating or used as GPIOs.
3. External LF clock: external clock provided by another IC. The clock has to be fed to the DIO23_X32P pin. Pin DIO24_X32N can be left floating or can be used a GPIO.

3.1.3 Filter

An LC filter is placed between the RF pin and the antenna. The filter has the function of attenuating the harmonics.

3.1.4 Decoupling Capacitors

In the reference designs, there are several decoupling capacitors. Take caution when placing the decoupling capacitors, as some pins with the same name are not connected internally. If the decoupling capacitor is placed at the incorrect pin, then the effect of decoupling capacitor gets reduced. The schematic in the reference design for every device shows where each decoupling capacitor needs to be placed at.

3.1.5 Antenna Components

A pi-match network is recommended between the LC filter and the antenna for antenna impedance matching. For more information, see Section 5.

3.1.6 RF Shield

An RF shield is used on some of the TI reference designs to reduce the radiation of spurious signals, in particular the third harmonic radiated power levels. TI recommends to have footprint for a shield in the design, especially for 20dBm devices. If the spurious emissions are within the expected range without the shield, then the shield can be set to Do Not Mount (DNM).

3.2 I/O Pins Drive Strength

The I/O pins have configurable drive strength and maximum current. All I/O pins support 2mA, while some pins support up to 10mA. Table 3-1 shows the I/O pins with high-drive capabilities.

Table 3-1. CC23xx/CC27xx Pins With up to 10mA Drive Strength

4 x 4 (RGE)	5 x 5 (RKP), 5 x 5 (RHB)	6 x 6 (RHA)
DIO12	DIO12	DIO2
DIO16_SWDIO	DIO16_SWDIO	DIO3
DIO17_SWDCK	DIO17_SWDCK	DIO9_SWDIO
DIO24_A7	DIO18	DIO10_SWDCK
	DIO19	DIO17_A8
	DIO24_A7	DIO18_A7

3.3 Bootloader Pins

The bootloader communicates with an external device over a 2-pin universal asynchronous receiver and transmitter (UART) or a 4-pin SPI. The SPI port has the advantage of supporting higher and more flexible data rates, but the port also requires more connections to the CC23xx and CC27xx devices. UART has the disadvantage of having slightly lower and possibly less flexible rates. However, the UART requires fewer pins and can be easily implemented with any standard UART connection.

There are three possible configurations for the serial interfaces for CC23xx devices and only one possible configuration for the serial interfaces for CC27xx devices. Table 3-2 and Table 3-3 specify which serial interface signals are configured to specific DIOs for CC23xx and CC27xx devices, respectively. Configuration of the ROM Bootloader is done between the FCFG and the CCFG. **There are defaults that are set in the FCFG that take effect if a valid CCFG is not present on start-up. The default configuration is dependent on the device package. Please see the device-specific technical reference manual for which configuration is the default configuration for a given packaging.** If users want to alter the defaults set by the FCFG, then update the CCFG to provide the behavior desired.

Table 3-2. CC23xx: Configuration of Serial Interfaces

Signal	Pin Configuration	serialIoCfgIndex == 0	serialIoCfgIndex == 1	serialIoCfgIndex == 2
UART_RX	Input with pull-up	DIO20	DIO12	DIO22
UART_TX	No pull (output when selected)	DIO6	DIO13	DIO20
SPI_CLK	Input with pull-up	DIO8	DIO24	DIO24
SPI_CS	Input with pull-up	DIO11	DIO11	DIO11
SPI_POCI	No pull (output when selected)	DIO12	DIO21	DIO12
SPI_PICO	Input with pull-up	DIO13	DIO13	DIO13

Table 3-3. CC27xx: Configuration of Serial Interfaces

Signal	Pin Configuration	serialIoCfgIndex == 0
UART_RX	Input with pull-up	DIO2
UART_TX	No pull (output when selected)	DIO1
SPI_CLK	Input with pull-up	DIO3
SPI_CS	Input with pull-up	DIO7
SPI_POCI	No pull (output when selected)	DIO5
SPI_PICO	Input with pull-up	DIO4

The bootloader can be configured to trigger unconditionally or can be configured to trigger with a pin. Pin trigger can be enabled in Sysconfig by going to the *TI DEVICES* section under *Device Configuration* in *Boot Configuration* and by setting the *Enable Pin Trigger* option. The Pin Trigger DIO can also be set at the same place in Sysconfig. The default Pin Trigger DIO for ROM bootloader can be found in Table 3-4.

Table 3-4. Default Bootloader Trigger Pin

CC23xx	CC27xx
DIO21	DIO21

3.4 Serial Wire Debug (SWD) Pins

Serial Wire Debug (SWD) is used for programming and doing debug on the device. The table below shows which two I/O pins are used for SWD.

Table 3-5. CC23xx and CC27xx SWD Pins

Signal	CC23xx	CC27xx
SWDIO	DIO16_SWDIO	DIO9_SWDIO
SWDCK	DIO17_SWDCK	DIO10_SWDCK

4 PCB Layout

4.1 Board Stack-Up

The distance from the top layer to the ground layer needs to match the reference design. Deviating from the recommended board stack-up can change the parasitics and, in some cases, can lead to a redesign of the filter. As an example, Figure 4-1 shows the board stack-up, which is used for all CC27xx LaunchPads. The board stack-up for all TI wireless devices can be found in the device-specific reference design.

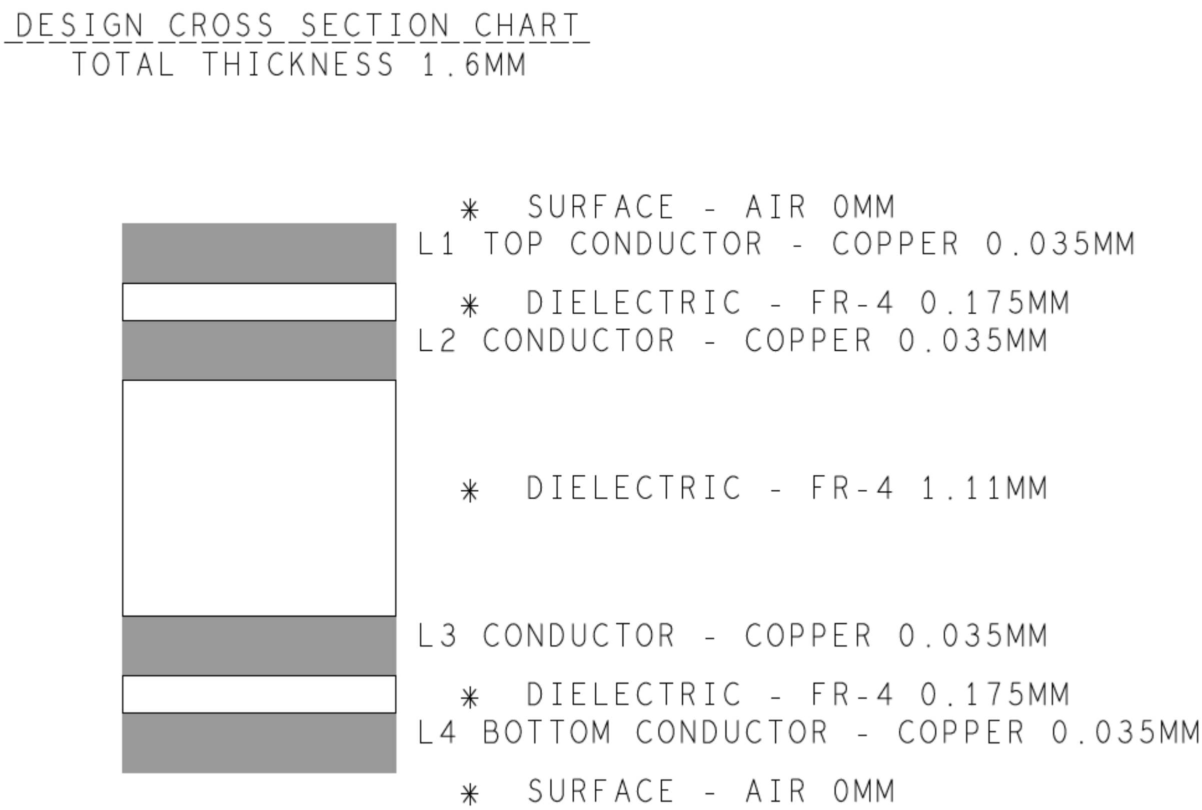

Figure 4-1. LP-EM-CC2745R10-Q1 Board Stack-Up

4.2 LC Filter

Lay out the LC filter so that crosstalk between the shunt components is minimized. Figure 4-2 shows three different layouts from worst to best. The advantage with the layout to the right is that the parasitic inductance in the PCB track (in black) between the shunt capacitor and the series inductor is in series with the inductor. In the middle figure, the parasitic inductance is in series with the shunt capacitor forming a series LC circuit.

If the design cannot use the reference design as-is (for example, use of a different component size), then the filter probably has to be re-tuned. Simulate both the TI reference design and the custom design using an electromagnetic simulator. The two designs have the same S21 and S22.

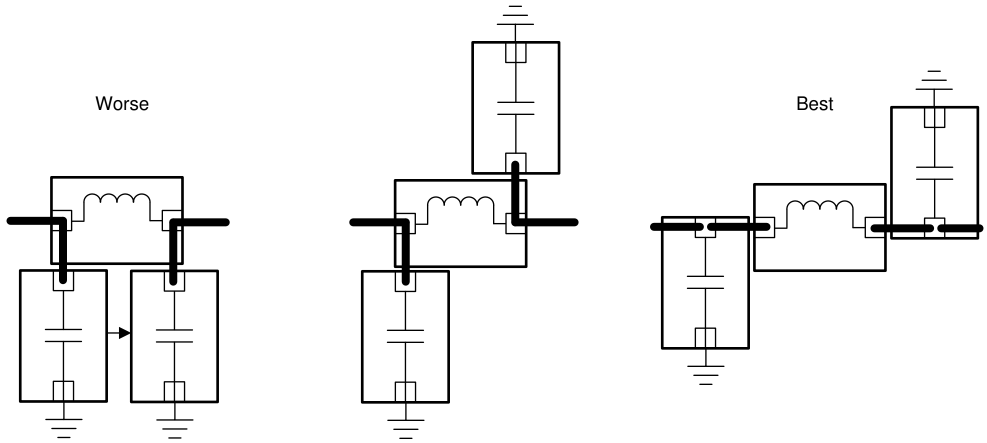

Figure 4-2. LC Filter PCB Layout Design Guideline

4.3 Decoupling Capacitors

General rules for decoupling capacitors:

- Make sure decoupling capacitors are on same layer as the active component for best results.
- Route power into the decoupling capacitor and then into the active component.
- Each decoupling capacitor needs to have a separate via to ground to minimize noise coupling (see Figure 4-3).
- Place the decoupling capacitor close to the pin for decoupling.
- The ground current return path between decoupling capacitor and chip needs to be short and direct (low impedance). For details, see Section 4.5.

The right side of the Figure 4-3 uses separate vias to ground has less noise coupling.

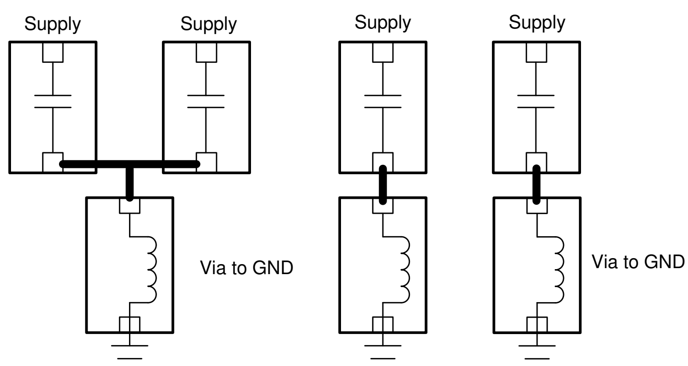

Figure 4-3. Decoupling Capacitors and VIA to Ground

4.4 Placement of Crystal Load Capacitors

The main oscillation loop current is flowing between the crystal and the load capacitors. Keep this signal path (crystal to C_{L1} to C_{L2} to crystal) as short as possible and use a symmetrical layout. Hence, both the ground connections of the capacitors need to always be as close as possible. Never route the ground connection between the capacitors or all around the crystal because this long ground trace is sensitive to crosstalk and EMI.

4.5 Current Return Path

There needs to be a solid ground plane from the capacitor ground pad back to the chip. If there is break in the ground plane, then this can result in a longer return path for the current. A longer return path can lead to reduced RF performance and higher spurious emissions.

4.6 DC/DC Regulator

The DCDC components must be placed close to the DCDC pin. The capacitor at the DCDC regulator output (DCDC pin) must have a short and direct ground connection to the chip (low impedance). Keep a solid ground plane from the capacitor ground pad back to the chip.

4.7 Antenna Matching Components

A pi-network is recommended for antenna impedance matching. Place the antenna matching components as close to the antenna as possible.

4.8 Transmission Lines

Traces in the LC filter are too short to be considered transmission lines, but longer traces, such as from the LC filter, towards the antenna needs to have a 50Ω impedance. TXLine is a free tool for PCB trace impedance calculations and is available at: TXLine Transmission Line Calculator.

4.9 Electromagnetic Simulation

If the design does not follow the reference design (for example, different filter component placement or component size), then TI recommends to use Advanced Design System (ADS) or similar to simulate and compare the impedances and S-parameters of the custom design with the reference design. Changes to the filter component values can be required if the custom design deviates too much from the reference design.

5 Antenna

The existing antenna documentation available is mainly oriented towards antennas that operate at a single frequency. There are two antenna selection guides are available: Antenna Selection Quick Guide application note and Antenna Selection Guide application note. There is a CC-Antenna-DK2 and Antenna Measurements Summary application note available with complete documentation. All antenna documentation that is available from TI can be accessed from the Antenna Selection Quick Guide application note because there are hyperlinks to all antenna documentation, antenna measurement reports, and all antenna reference designs.

TI recommends to include an antenna matching network to tune and to reduce the mismatch losses of the antenna. For a single-band antenna, the recommendation is to always include a pi-match network prior to the antenna (see Figure 5-1). Only two of the three footprints or components are required. The impedance of the antenna determines if footprint or component ANT1 or ANT3 is used. ANT2 is always used. Even if the antenna is matched exactly, ANT2 can be set as a 0Ω resistor. For more details on antenna matching, refer to Antenna Impedance Measurement and Matching application note.

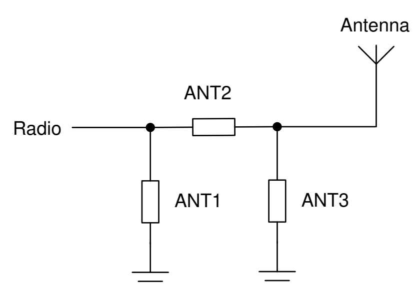

Figure 5-1. Recommended Antenna PI-Match Network for Single-Band Antennas

6 Crystal Tuning

6.1 CC23xx and CC27xx Crystal Oscillators

The CC23xx and CC27xx devices have two crystal oscillators. The high frequency crystal oscillator (HFXT) runs at 48MHz and is mandatory to operate the radio. The low frequency crystal oscillator (LFXT) is used for RTC timing and only required when accurate RTC timing is necessary. For example, synchronous protocols such as Bluetooth Low Energy (see Section 3.1.2).

Both crystal oscillators are pierce type oscillators are shown in Figure 6-1. In this type of oscillator, the crystal and the load capacitors form a pi-filter providing a 180° phase shift to the internal amplifier keeping the oscillator locked at the specified frequency. For this frequency to be correct, the load capacitance must be dimensioned properly based on the capacitive load (CL) parameter of the crystal.

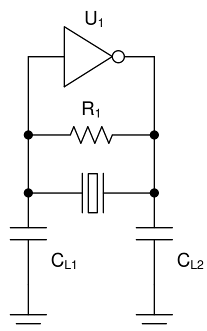

Figure 6-1. Pierce-Type Oscillator

A key difference between the oscillators is that the high frequency oscillator has internal variable load capacitance inside the IC and does not require external load capacitors. The low frequency oscillator needs to have external capacitors to operate properly.

6.2 Crystal Selection

When selecting a crystal part, refer to the device-specific CC23xx and CC27xx data sheets in Section 12 that lists requirements for the crystal parameters. All of these requirements must be fulfilled to make sure of proper operation of the oscillators and proper operation of the device. Refer to the Crystal Oscillator and Crystal Selection for the CC13xx, CC26xx, and CC23xx Family of Wireless MCUs application note for recommended crystals.

6.3 Tuning the LF Crystal Oscillator

The frequency of the 32kHz crystal oscillator is set by properly dimensioning the load capacitors relative to the wanted load capacitance, CL, of the crystal. From the crystal, the two capacitors are placed in series. The PCB traces and the pads add some parasitic capacitance. Equation 1 shows how to calculate the total effective capacitance value.

$$CL = \frac{C1xC2}{C1 + C2} + C_{parasitic} \approx \frac{load \quad capacitor \quad value}{2} + C_{parasitic} \tag{1}$$

The last simplification requires that C1 and C2 are equal.

The best way to measure the frequency accuracy of the oscillator is to output the 32kHz clock signal on an I/O pin. The frequency can be measured using a frequency counter without affecting the oscillator. The following code snippet outputs the selected 32kHz clock source to DIO19. Only one specific I/O can be used to output the 32kHz clock for CC23xx and CC27xx (see Table 6-1).

```
/* INCLUDES
*/
#include <ti/drivers/GPIO.h>
#include DeviceFamily_constructPath(inc/hw_types.h)
#include DeviceFamily_constructPath(inc/hw_memmap.h)
#include DeviceFamily_constructPath(inc/hw_ckmd.h)
#include DeviceFamily_constructPath(inc/hw_ioc.h)
#include DeviceFamily_constructPath(inc/hw_pmctl.h)
// ...
int main()
{
// ...

/** Add the following after Board_init();
* Be sure IOID used below is not used by any entries in PIN or
* GPIO tables from the board files.
* The clock source can be switched with constant clockSrc.
*/

uint8_t clockSrc = 0xF; // for LF crystal clock

// drive output low first
GPIO_setConfig(19, GPIO_CFG_OUTPUT | GPIO_CFG_OUT_LOW);

// Configure the IOC.IOC19.PORTCFG MMR to select DTB
HWREG(IOC_BASE + IOC_O_IOC19) &= ~IOC_IOC19_PORTCFG_M;
HWREG(IOC_BASE + IOC_O_IOC19) |= IOC_IOC19_PORTCFG_DTB;

// Make sure the DTB mux selects in IOC (and if required in
// source clock IP) are reset that zero is driven on DTB0.
// ULLSEL mux select (select CKMD)
HWREG(IOC_BASE + IOC_O_DTBCFG) &= ~IOC_DTBCFG_ULLSEL_M;
HWREG(IOC_BASE + IOC_O_DTBCFG) |= 0x1 << IOC_DTBCFG_ULLSEL_S; // 0x1 to route CKMD to DTB0

// Enable IOC.DTBOE.EN0
HWREG(IOC_BASE + IOC_O_DTBOE) &= ~IOC_DTBOE_EN0_M;
HWREG(IOC_BASE + IOC_O_DTBOE) |= IOC_DTBOE_EN0_EN;

// select which clock (CKMD) to output on DTB0 (DTB[0])
HWREG(CKMD_BASE + CKMD_O_DTBCTL) &= ~CKMD_DTBCTL_CLKSEL_M;
HWREG(CKMD_BASE + CKMD_O_DTBCTL) |= (clockSrc) << CKMD_DTBCTL_CLKSEL_S;

// enable DTB output
HWREG(CKMD_BASE + CKMD_O_DTBCTL) &= ~CKMD_DTBCTL_EN_M;
HWREG(CKMD_BASE + CKMD_O_DTBCTL) |= CKMD_DTBCTL_EN;

// ...
}
```

Table 6-1. I/O for Outputting the 32kHz Clock

CC23xx	CC27xx
DIO19	DIO27

6.4 Tuning the HF Crystal Oscillator

The HFXT has internal variable load capacitors (Cap Array Q1 and Cap Array Q2) in the IC and does not require external capacitors to be mounted. The load capacitance is set in the Sysconfig *TI DEVICES* section under *Device Configuration* by enabling *Override HFXT Cap Array Trims* and setting the appropriate values for Cap Array Q1 and Cap Array Q2. To avoid having a load capacitor imbalance on the two sides of the crystal, the maximum difference between Cap Array Q1 and Cap Array Q2 cannot be more than one. If Override HFXT Cap Array Trims is not enabled, then the fault values for Cap Array Q1 and Cap Array Q2 are used.

Table 6-2 shows the resulting total capacitance measured on an evaluation board versus Cap Array values. Note, that the resulting capacitance value includes parasitic capacitances, which is why the lowest setting is not 0pF. There are two regions where an increase in the Cap Array values do not change the effective capacitance. The two aforementioned regions are when the Cap Array value goes from 15 to 16 and from 47 to 48. If an automated search algorithm is used for finding the optimum Cap Array value, then take caution to remove either 15 or 16 and 47 or 48 from the search algorithm.

The best way to measure the accuracy of the HF crystal oscillator is to output an demodulated carrier wave from the radio and measuring the frequency offset from the wanted frequency using a spectrum analyzer. The relative offset of crystal frequency, typically stated in parts per million (ppm), is the same as the relative offset of the RF carrier.

For testing purposes, Cap Array values can be adjusted in SmartRF™ Studio. This simplifies tuning greatly by allowing on-the-fly updates of the load capacitance. The optimum value found in SmartRF Studio can then be entered into Sysconfig in the applicable software project.

Table 6-2. Cap Array

Cap Array Value (Q1 = Q2)	Measured Capacitance on Reference Board for CC23xx (pF)	Measured Capacitance on Reference Board for CC27xx (pF)
1	3.7	3.9
2	3.7	4.0
3	3.8	4.1
4	3.9	4.1
5	4.0	4.2
6	4.1	4.3
7	4.1	4.4
8	4.2	4.4
9	4.3	4.5
10	4.4	4.6
11	4.4	4.7
12	4.5	4.7
13	4.6	4.8
14	4.6	4.9
15	4.7	5.0
16	4.7	5.0
17	4.8	5.0
18	4.9	5.1
19	5.0	5.2
20	5.1	5.3
21	5.2	5.4
22	5.3	5.5
23	5.3	5.6
24	5.4	5.7
25	5.5	5.8
26	5.6	5.9
27	5.7	6.0
28	5.8	6.1
29	5.9	6.2
30	6.0	6.3
31	6.1	6.4
32	6.3	6.6
33	6.4	6.7

Table 6-2. Cap Array (continued)

Cap Array Value (Q1 = Q2)	Measured Capacitance on Reference Board for CC23xx (pF)	Measured Capacitance on Reference Board for CC27xx (pF)
34	6.5	6.8
35	6.6	6.9
36	6.7	7.0
37	6.8	7.2
38	6.9	7.3
39	7.1	7.4
40	7.2	7.5
41	7.3	7.6
42	7.4	7.8
43	7.5	7.9
44	7.6	8.0
45	7.7	8.1
46	7.8	8.2
47	7.9	8.3
48	7.9	8.3
49	8.1	8.5
50	8.3	8.7
51	8.4	8.8
52	8.6	9.0
53	8.7	9.2
54	8.9	9.3
55	9.0	9.5
56	9.2	9.7
57	9.3	9.8
58	9.5	10.0
59	9.6	10.1
60	9.8	10.3
61	9.9	10.5

7 Optimum Load Impedance

The matching environment for optimum performance is determined through a combination of load- and source-pull measurements, given as a terminating load or source impedance. This requires comprehensive measurements to characterize the nonlinear response of the RF front-end.

The parameters considered include:

- Transmit output power
- Transmit efficiency
- Transmit harmonic power levels
- Transmit output spectrum
- Receiver sensitivity

The operating conditions considered include:

- Frequency
- Voltage range
- Transmit power settings
- Package parasitics

Additionally, the effect of temperature variation on transmit/receive performance must also be considered.

These impedance locations are typically located in different regions of the Smith chart and a design space giving the best tradeoff between transmit and receive performance is identified for a given set of operating conditions.

The identified target impedances can be highly dependent on the power and ground planes of the application circuit and accurate measurement system calibration, along with the effects of differential and common current components due to the PCB layout. While detailed simulations of the PCB using EDA tools can add confidence to a design, simulation inaccuracies (such as component models) add additional errors that can be difficult to account for.

Due to the number of parameters that must be considered and amount of testing required for a robust design, TI recommends to follow the reference design.

8 PA Table

The PA table for the various devices is provided in SmartRF Studio. The txpower values used in the table are selected to provide the lowest possible device-to-device variation. In addition, the txpower setting has a built-in temperature compensation that gives a very low output variation as a function of temperature.

The PA used is designed to be highly effective on maximum power. With maximum power, the PA is in saturation and because of this, the device-to-device variation is low. For lower power settings, the PA is in the linear region and the output power is dependent on the transistor gain, which has a higher device-to-device variation.

9 Power Supply Configuration

9.1 Introduction to Power Supply

The CC23xx devices have three power rails that are exposed on external pins: VDDS, VDDR and VDDD. The CC27xx devices has one more power rail: VDDIO. VDDS is the main power source for the wireless microcontroller and must be supplied externally with 1.71V to 3.8V. VDDR is an internal power rail that is supplied from the internal DC/DC converter, or the internal Global LDO. VDDR is regulated to approximately 1.5V. VDDD is supplied internally by either digital LDO or micro LDO depending on the power state. This power rail is trimmed to approximately 1.28V and requires an external decoupling capacitor of 1µF. VDDIO supply powers the split rail IO supply for some GPIOs and must be supplied externally with 1.71V to 3.8V.

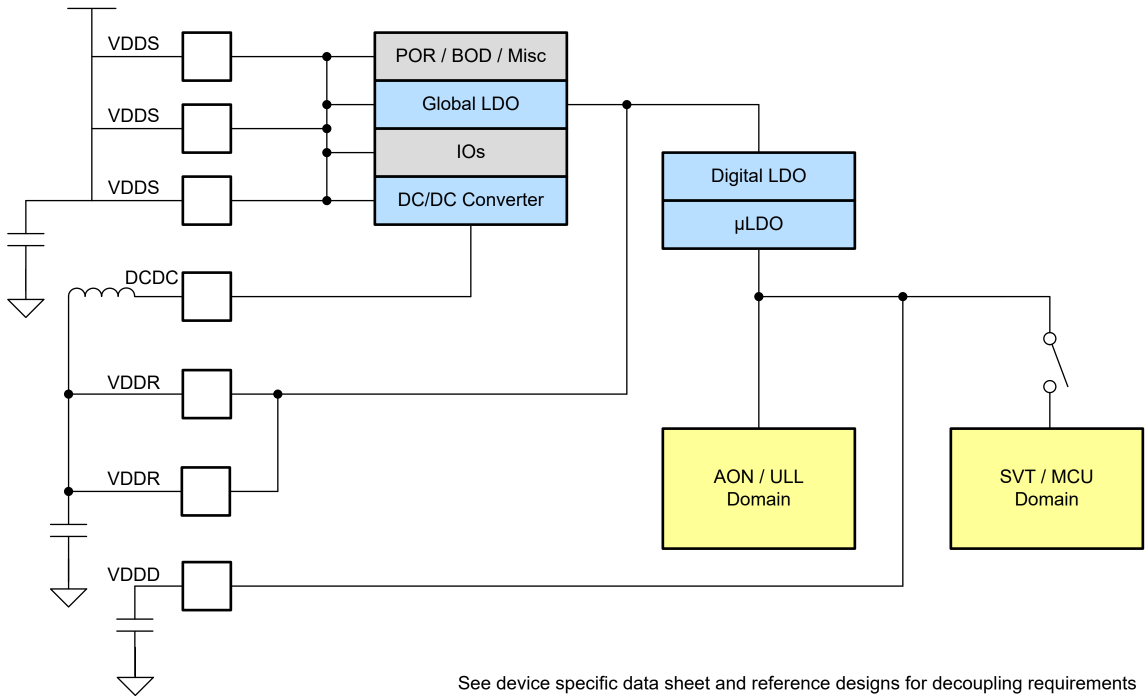

Figure 9-1. Power Supply System

9.2 DC/DC Converter Mode

Maximum efficiency is obtained by using the internal DC/DC converter, and requires an external inductor and capacitor. Place the components as close as possible to the CC23xx or CC27xx device and have a short current return path from the DC/DC capacitor ground to the pad on the chip. The VDDS pins are not connected together internally. Check the reference design for where each decoupling capacitor needs to be placed. The actual value of decoupling capacitors and the DC/DC inductor can vary from device to device. For the actual values, see the device-specific reference design.

When operating in DC/DC mode, the power system dynamically switches between the Global LDO and DC/DC converter depending on the required load to achieve maximum efficiency. If VDDS drops below 2.2V, then the DC/DC converter is less efficient than the LDO and the device runs in global LDO mode. For systems operating with VDDS less than 2.2V, consider global LDO to save component cost and board area.

To utilize the DC/DC converter, the customer has to select *DCDC* in the Sysconfig in *TI DEVICES -> Device Configuration -> Voltage Regulator*. See Figure 9-2 for more details.

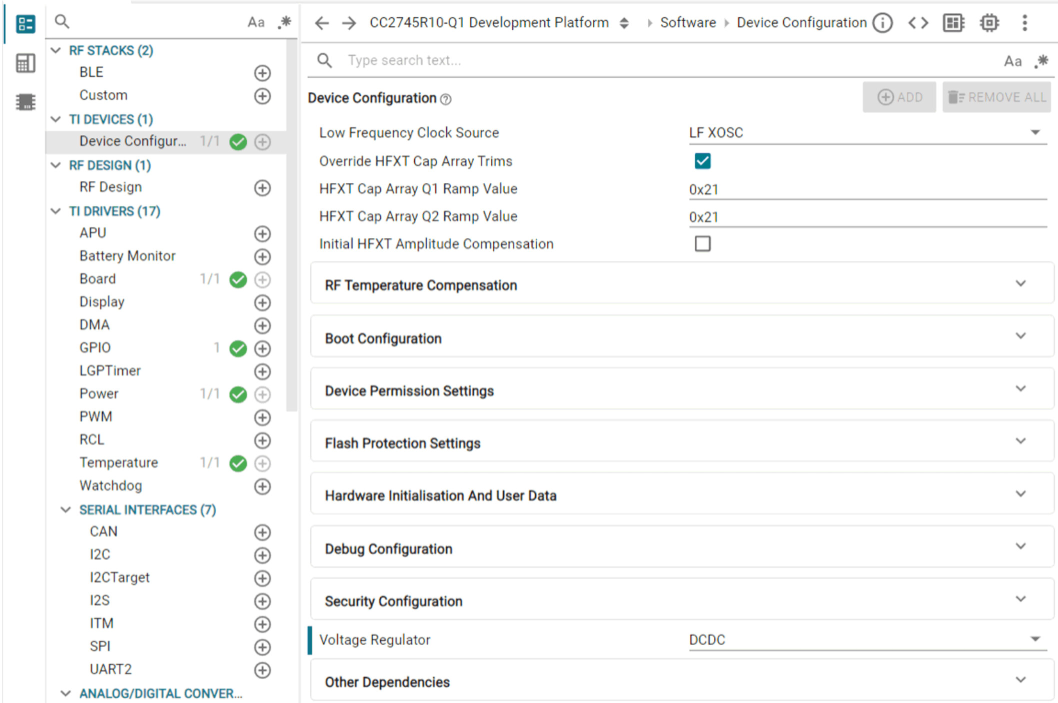

Figure 9-2. DC/DC Mode

9.3 Global LDO Mode

To save cost and PCB area, the DC/DC inductor can be removed and VDDR can be supplied from the Global LDO at the cost of higher power consumption. See the device-specific data sheet for how much increase in current consumption is expected. In this mode, a bulk capacitor on VDDR is still required and placed close to the VDDR pin.

To utilize the Global LDO, users have to select *GLDO* in Sysconfig in *TI Devices* -> "*Device Configuration* -> *Voltage Regulator*. See Figure 9-3 for more details.

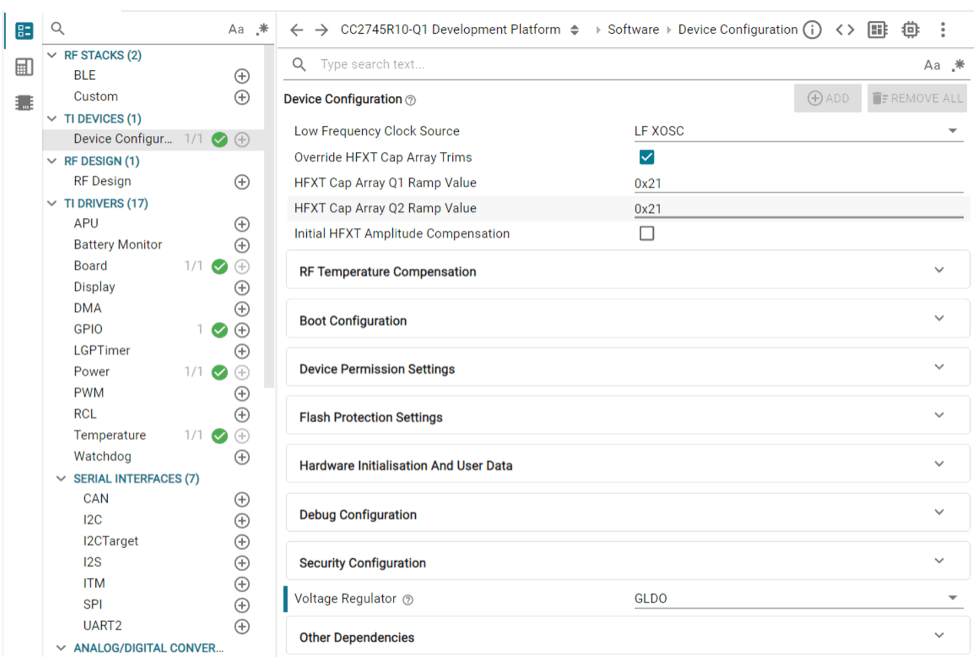

Figure 9-3. Global LDO Mode

10 Board Bring-Up

Before starting to develop software or doing range testing, TI recommends to do conducted measurements to verify that the board has the expected performance. Typically, the sensitivity, output power, harmonics, and current consumption needs to be measured to verify the hardware design.

The required measurements depend on the type of board and application. If this is a design with 10m range requirement, then the checkout does not need to be as detailed as for a design with a range extender. For an application and other designs that require high performance, having access to a spectrum analyzer and a signal generator with the option to send RF packets is highly recommended.

Different measurement methods are discussed in the following sections. Users need to select the methods applicable for the board. For more detailed tests, refer to the Basic RF Testing of CCxxxx Devices application report.

10.1 Power On

When powering on the board for the first time, check that the voltages on the following pins are as expected.

CC23xx and CC27xx

- VDDR = 1.5V
- VDDD = 1.28V

Do not measure directly on the X48P and X48N pins since this can damage the device.

10.2 RF Test: SmartRF Studio

To use SmartRF Studio for testing, the board needs a connector that enables a debugger to be connected directly to the RF chip. Use a LP-XDS110/LP-XDS110ET for the CC23xx and CC27xx. The required pins to be connected are VDDS (not required when the device is powered externally), GND, RESET, SWDIO and SWDCK.

1. Connect a debugger to the board. Open SmartRF Studio and verify that the device is visible in the list of connected devices.
2. Place two known, good boards with 2m distance. In this context, *known, good boards* are LaunchPads from TI. Use a predefined PHY setting in SmartRF Studio that is a closest match to the PHY that is used in the final product.
3. Set one board to Packet RX and the other to Packet TX and transmit 100 packets. Confirm that the packets are received and note the RSSI for the received packets.
4. Replace the board used in TX with the device under test (DUT). Repeat the test described in #3.
5. Replace the board used in RX with the DUT. Replace the board used in TX with a good known board. Repeat the test described in #3.
6. If possible, then the measurements need to be done with a *known, good* antenna first and then repeated with the antenna that is going to be used in the final design later. A poorly tuned antenna can cause significant loss in sensitivity or output power.
7. If the results are satisfactory, then change the settings from the predefined setting to the RF settings planned to be used in the final product. Repeat the tests described in #3 to #5 with the wanted RF settings.

If the RSSI deviates from the reference, then review the schematic and layout.

10.3 RF Test: Conducted Measurements

For high performance designs, TI recommends to perform conducted measurements to verify the performance before setting up an RF link.

10.3.1 Sensitivity

1. Disconnect the antenna and perform conducted measurements at the SMA connector or solder a semi rigid coax cable at the 50Ω point.
2. Configure the board under test and use the Packet RX option in SmartRF Studio 8 similar to the test described in Section 10.2. In Packet RX mode, users can set an expected packet count.

3. Use a LaunchPad as a transmitter. Use coax cables and attenuation between the LaunchPad SMA connector and the 50Ω point on the custom board. To get an accurate number using this method is difficult since the exact values of output power and attenuation are normally not known. Some energy also travels over the air from the EM to the DUT. In addition, background noise can impact the results. To get more accurate results, place the receiver in a shielded box.
4. SmartRF Studio calculates the packet error rate (PER) and bit error rate (BER).

If the conducted sensitivity is poor, then check for the following.

- Are the settings the same as the recommended values from SmartRF Studio? If the sensitivity is good when using SmartRF Studio and not with the settings used for the project, then the settings have to be reviewed.
- What is the frequency difference between the DUT and the signal source? Frequency offset can be measured by transmitting an demodulated continuous wave.
- Is the schematic, including all component values, in accordance with the reference design?
- Is the layout in accordance with the reference design?

10.3.2 Output Power

1. Disconnect the antenna and perform conducted measurements at the SMA connector or solder a semi-rigid coax cable at the 50Ω point.
2. Preferred method: use a spectrum analyzer (SA). Use 2MHz RBW for measuring output power.
3. If an SA is not available, then use a LaunchPad with a SMA connection point. Use coaxial cables and attenuation between the LaunchPad SMA connector and the 50Ω point on the custom board. Use SmartRF Studio and set the Launchpad in Continuous RX and read the RSSI. Note, that the RSSI has a given tolerance so the measurement is not as accurate as the preferred method.

For more guidance on RF testing, refer to Basic RF Testing of CCxxxx Devices.

10.4 Hardware Troubleshooting

This section covers some of the common causes for poor performance.

10.4.1 No Link: RF Settings

To get a link between two RF chips the two RF chips have to operate on the same frequency and with the same RF settings.

10.4.2 No Link: Frequency Offset

For narrow band systems, a large frequency offset between the TX and RX devices can result in no link or a very poor link. The minimum required RX bandwidth to make sure reception is calculated by using Equation 2.

$$RX \quad BW = Signal \quad Bandwidth + 4 \cdot ppm \quad Crystal \quad \cdot RF \quad Frequency \quad of \quad Operation \tag{2}$$

The FSK the signal bandwidth can be approximated as data rate + 2×frequency deviation (Carson’s rule).

10.4.3 Poor Link: Antenna

An antenna needs a matching network to tune and reduce the mismatch losses of the antenna. If the antenna is not tuned, then energy is lost both in TX and RX and the link budget is lower. For more details, refer to Section 5.

10.4.4 Bluetooth Low Energy: Device Does Advertising But Cannot Connect

If using the 32kHz crystal oscillator as RTC source, then:

- Incorrect load capacitors for the 32.768kHz crystal can cause frequency offset
- 32kHz crystal does not start up (incorrect load capacitors, crystal missing, soldering issues)

If using the 32kHz RC oscillator as RTC source, then:

- Calibration is not configured correctly. For more information, see the Bluetooth Low Energy Stack User's Guide that is provided with the SDK.

Incorrect RTC frequency leads to the device missing the connection events and, breaking the link with the central device.

To debug this problem, the 32kHz clock can be output on an I/O pin and measured with a frequency counter. For more information, refer to Section 6.3. By outputting the clock on a pin, users need to always measure the _selected_ RTC clock source, and measure without affecting the clock source (which probing the crystal can do).

If using a 32.768kHz crystal, then make sure the crystal part is within the requirements outlined in the device-specific CC234x and CC27xx data sheets. Make sure that the load capacitors are dimensioned properly as shown in Section 6.3.

Verify that the BLE-Stack has been configured with the correct Sleep Clock Accuracy. The default value for a Central and Peripheral device is 50ppm and 40ppm, respectively. The Sleep Clock Accuracy can be adjusted with the HCI_EXT_SetSCACmd API. See hci.h or the TI Vendor Specific API Guide included in the SDK.

10.4.5 Poor Sensitivity: Background Noise

A RF channel receives all radio traffic in the selected frequency span. In addition to the wanted signal, the channel also receives background noise. Part of the background noise is other RF traffic on the selected band. To receive a RF packet, the received signal has to have a given SNR. If the background noise increases, then the practical sensitivity is poorer.

For example, if the conducted sensitivity is -100dBm, the required SNR is 7dB and the background noise is -90dBm, then the practical radiated sensitivity is -83dBm.

Before doing a range test, measure the background noise. One method is to turn off all known TX sources, attach a LaunchPad or a *known, good* board to SmartRF Studio, select the Continuous RX tab and press play. The average of the resulting graph can be used as an input to find the practical sensitivity.

10.4.6 High Sleep Power Consumption

- The chip is not going into the lowest power modes when a debugger is connected.
- Software: Use the gpioshutdown examples in the relevant SDK.
- When measuring current draw on a Launchpad, remove all jumpers.
- Make sure that every IC on the board is powered down.
- If the application is configured to use the 32kHz crystal, then check that this is connected and that the oscillator is running.
- Make sure no input pins are in tri-state.

11 Summary

This application note provides guidelines for designing a custom hardware using the CC23xx and CC27xx device families. The document begins with an overview of available reference designs for these devices, followed by key considerations for schematic and PCB layout. Finally, there are guidances on tuning the crystal oscillators and instructions of how to carry out the board bring-up.

12 References

- Cadence, TXLine Transmission Line Calculator
- Texas Instruments, *2.4GHz Inverted F Antenna*, application note
- Texas Instruments, *LP-EM-CC2340R53 Design Files*
- Texas Instruments, *LP-EM-CC2340R5 Design Files*
- Texas Instruments, *LP-EM-CC2340R5-Q1 Design Files*
- Texas Instruments, *LP-EM-CC2340R5-RGE-4x4-IS24 Design Files*
- Texas Instruments, *LP-EM-CC2745R10-Q1 Design Files*
- Texas Instruments, Antenna Selection Quick Guide, application note
- Texas Instruments, Antenna Selection Guide, application note
- Texas Instruments, CC-Antenna-DK2
- Texas Instruments, CC-Antenna-DK2 and Antenna Measurements Summary, application note
- Texas Instruments, Basic RF Testing of CCxxxx Devices, application note
- Texas Instruments, Crystal Oscillator and Crystal Selection for the CC13xx, CC26xx, and CC23xx Family of Wireless MCUs, application note
- Texas Instruments, Antenna Impedance Measurement and Matching, application note
- Texas Instruments, CC2340R SimpleLink™ Family of 2.4GHz Wireless MCUs, errata
- Texas Instruments, CC2340R5-Q1 SimpleLink™ Wireless MCU Device, errata
- Texas Instruments, CC2340R SimpleLink™ Family of 2.4GHz Wireless MCUs, data sheet
- Texas Instruments, CC2340R5-Q1 SimpleLink™ Bluetooth® 5.3 Low Energy Wireless MCU, data sheet
- Texas Instruments, CC274xR-Q1, CC274xP-Q1 Automotive SimpleLink™ Bluetooth® 5.4 Low Energy Wireless MCU, data sheet

13 Revision History

Changes from Revision * (May 2025) to Revision A (February 2026)	Page
• Updated Table 3-2 and Table 3-3	6

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, regulatory or other requirements.

These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you fully indemnify TI and its representatives against any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to TI's Terms of Sale, TI's General Quality Guidelines, or other applicable terms available either on ti.com or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products. Unless TI explicitly designates a product as custom or customer-specified, TI products are standard, catalog, general purpose devices.

TI objects to and rejects any additional or different terms you may propose.