User's Guide

CC33xx Hardware Integration

Dylan Hubbard, Jessica M. Torres, Jonathan Cohen, Andy Bui, and Josh Smith

ABSTRACT

This document provides necessary information on the WLAN and *Bluetooth*® Low Energy hardware operation to aid engineers in system design. The document reviews the integration process of TI's CC33xx devices into PCBs of final products. When designing a system around the TI chipset, TI recommends following the guidelines outlined in this user guide.

Table of Contents

List of Figures

List of Tables

Trademarks

SimpleLink™ is a trademark of Texas Instruments.
Wi-Fi™ is a trademark of Wi-Fi Alliance.
Bluetooth® is a registered trademark of Bluetooth SIG, Inc.
All trademarks are the property of their respective owners.

1 Introduction

1.1 Overview

The SimpleLink™ Wi-Fi™ CC33xx family of devices is where potential cost savings meet reliability, enabling engineers to connect more applications with confidence. CC33xx devices are available in a single-chip, or pre-certified module, integrating both Wi-Fi 6 and Bluetooth Low Energy (BLE) 5.4. The following devices are also pin-to-pin compatible within the relevant packaging family (for example, IC to IC or module to module):

Table 1-1. CC33xx Companion IC Device Comparison

Device	Feature		
	WLAN 2.4GHz	Bluetooth Low Energy 5.4	WLAN 5GHz
CC3300	✓		
CC3301	✓	✓	
CC3350	✓		✓
CC3351	✓	✓	✓

Table 1-2. CC33xx Companion Module Comparison

Device	Feature		
	WLAN 2.4GHz	Bluetooth Low Energy 5.4	WLAN 5GHz
CC3300MOD	✓		
CC3301MOD	✓	✓	
CC3350MOD	✓		✓
CC3351MOD	✓	✓	✓

This guide explains the hardware requirements and recommendations for integrating the CC33xx devices.

2 Schematic Considerations – CC33xx Devices

The CC33xx devices are designed to integrate easily with any system and require few external components. The digital interface to the host processor (MPU or MCU) is highly flexible based on the end application. Users can decide to use any combination of secure digital input output (SDIO), serial peripheral interface (SPI), or universal asynchronous receiver-transmitter (UART) for shared Wi-Fi and Bluetooth Low Energy communication.

This section explains the minimum requirements for optimizing an engine area schematic.

2.1 Schematic Reference Design

TI recommends following the reference design and guidelines for the CC33xx devices as closely as possible to achieve the full capabilities of the CC33xx devices (as listed in the device-specific data sheet) and to pass certification. The schematic designs recommended for the CC33xx and CC335x devices are accessed here:

CC330x Reference Design Files

CC335x Reference Design Files

Figure 2-1 and Figure 2-2 show the reference schematics for the CC330x and CC335x devices.

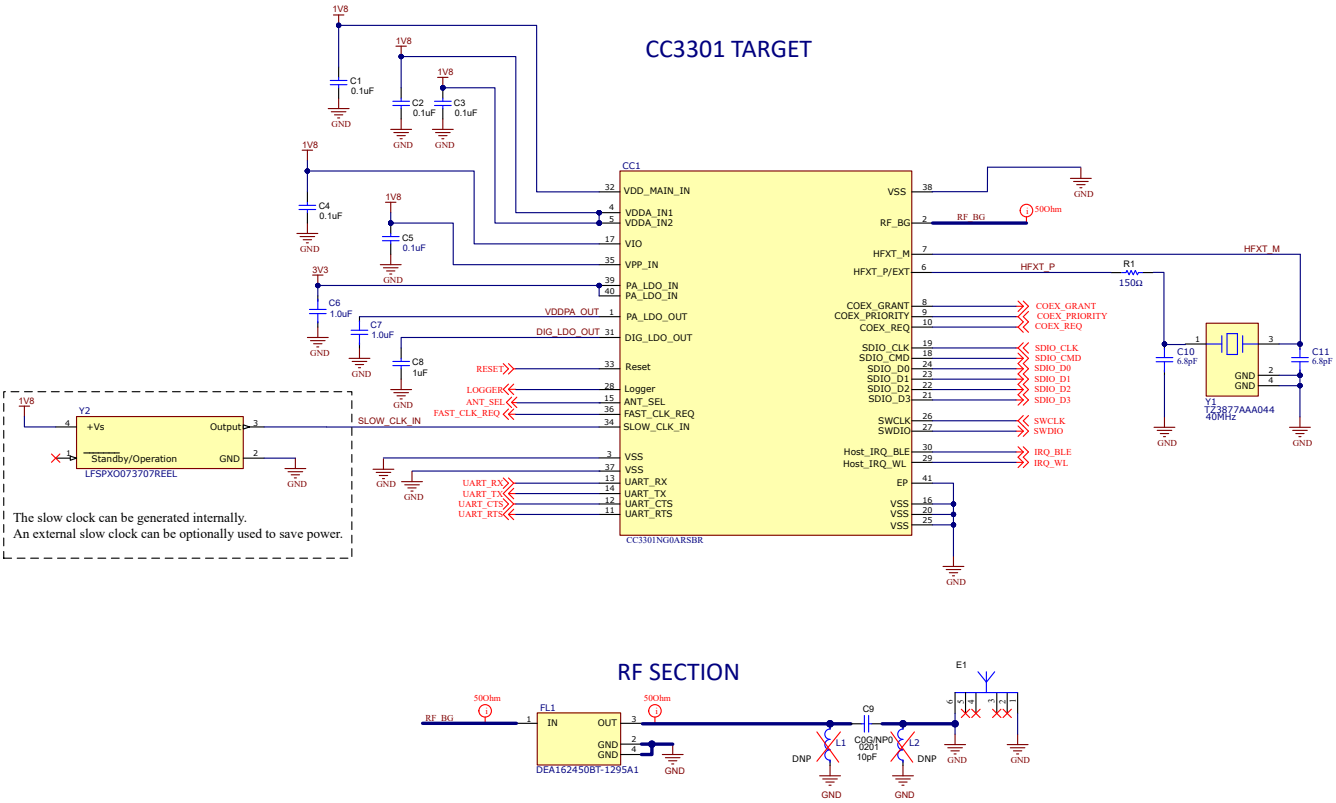


Figure 2-1. CC33xx Reference Schematic

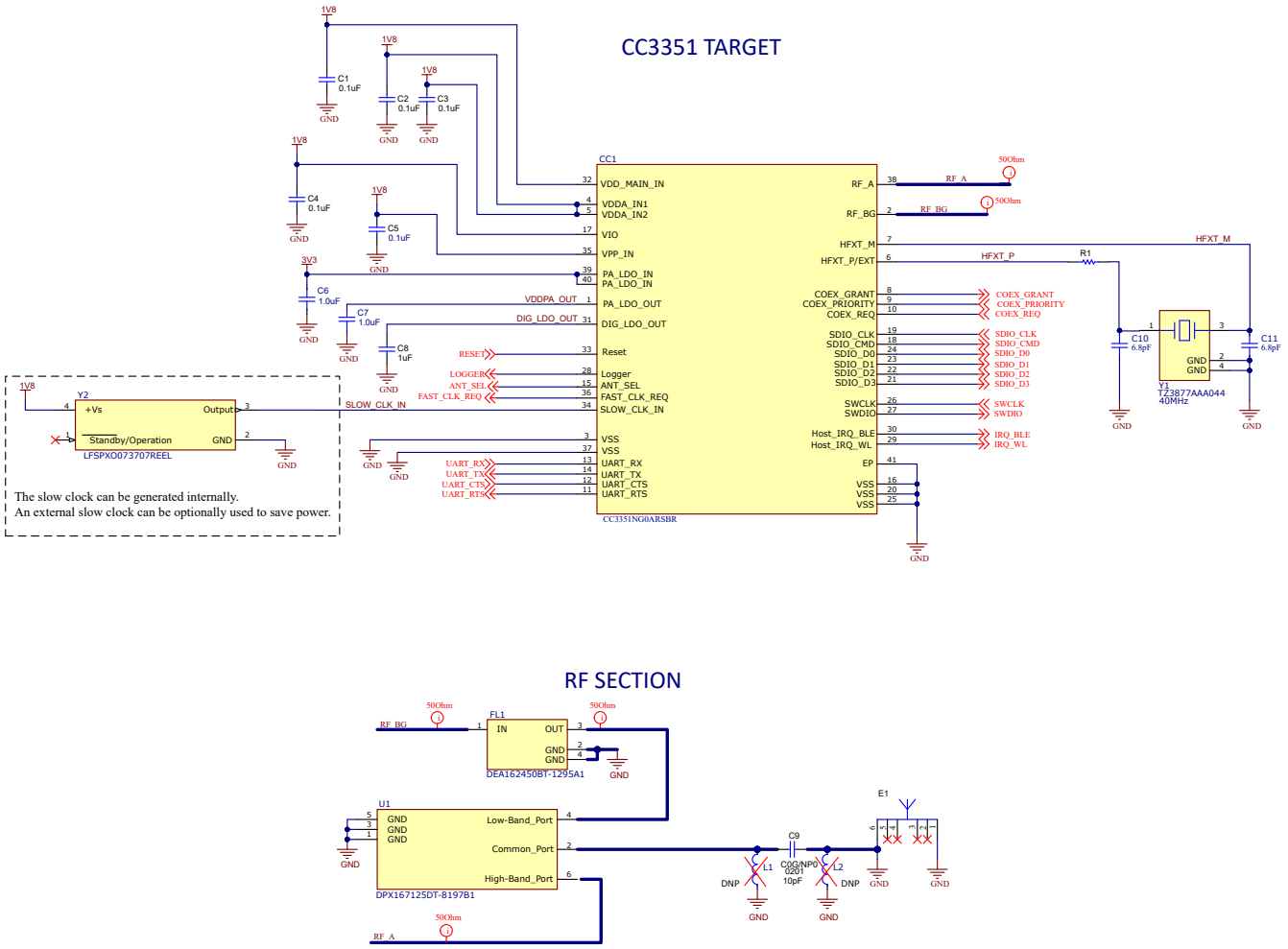


Figure 2-2. CC335x Reference Schematic

The only difference between the schematics for the CC33xx and CC335x devices is the added diplexer U1 for 5GHz support in the CC335x devices. For more information on the RF section of the schematic, see Section 2.4.

Table 2-1. Bill of Materials

Item	Designator of Reference	Quantity	Value	Part Number	Manufacturer	Description	Package Reference
1	C_1, C_2, C_3, C_4, C_5	5	0.1µF	GRM033C71A104KE14D	Murata	CAP, CERM, 0.1µF, 10V, ±10%, X7S, 0201 Matching component: CAP, CERM, 10pF, 50V, ±5%, C0G/NP0, 0201	0201
2	C_6, C_7	2	1µF	GRM033D70J105ME01D	Murata	Chip Multilayer Ceramic Capacitors for General Purpose, 0201, 1.0µF, X7T, +22%/-33%, 20%, 6.3V	0201
3	C_8	1	1µF	GRM155R70J105MA12D	Murata	CAP, CERM, 1µF, 6.3V, ±20%, X7R, 0402	0402
4	C_9	1	10pF	GJM0335C1E100JB01D	Murata	CAP, CERM, 10pF, 25V, ±5%, C0G/NP0, 0201	0201
5	C_{10}, C_{11}	2	6.8pF	GJM0335C1H6R8BB01	Murata	Chip Multilayer Ceramic Capacitors for General Purpose, 0201, 6.8pF, C0G, 30ppm/°C, 0.25pF, 50V	0201
6	R1	1	150Ω	RC0201FR-7D150RL	YAGEO	RES, 150, 1%, 0.05W, 0201	0201
7	CC1	1		CC33x1NG0ARSBR	Texas Instruments	CC33x1NG0ARSBR – Wi-Fi 6 and Bluetooth Low Energy 5.2 Combo Transceiver	WQFN40
8	Y2	1		TZ3877AAAO44	Tia-Saw Technology	Crystal Unit SMD 2.0 x 1.6 40.0MHz	SMT4_2MM05_1MM65
9	FL1	1		DEA162450BT-1295A1	TDK	2.45GHz Center Frequency Band Pass RF Filter, 100MHz Bandwidth, 1.8dB 0603, 3 PC Pad	SMT_FILTER_1MM60_0MM80
10	**Only for CC335x:** U1	1		DPX167125DT-8197B1	TDK	Multilayer Diplexer for 2.4-2.5GHz W-LAN and Bluetooth / 5-7GHz W-LAN	SMD6
11	**Optional:** Y1(1)	1		LFSPX0073707REEL	IQD Frequency Products	**Optional:** 32.768kHz XO (Standard) CMOS Oscillator 1.8V Enable/Disable 4-SMD, No Lead	SMT4_2MM0_1MM6

(1) The slow clock can be generated internally. An external slow clock can be optionally used to consume less power than sourcing the slow clock internally.

2.2 Power Supply

There are two power rails that must be routed to the CC33xx devices:

- 1.8V: VDD_MAIN_IN, VDDA_IN1, VDDA_IN2, VIO, and VPP_IN
- 3.3V: PA_LDO_IN

The CC33xx devices have internal LDOs for regulating the digital core, memory, and power amplifier supplies. The output of the LDO can be measured from the DIG_LDO_OUT and PA_LDO_OUT signals.

Note

The output of the PA_LDO_OUT signal is not enabled until firmware is loaded onto the device.

For further information on the operating conditions for the supply pins, see Table 2-2.

Table 2-2. Required Device Power

Pin	Signal	Direction (I/O)	Required Voltage (Typical)
1	PA_LDO_OUT	O	N/A
31	DIG_LDO_OUT	O	N/A
17	VIO	I	1.8V
32	VDD_MAIN_IN	I	1.8V
4	VDDA_IN1	I	1.8V
5	VDDA_IN2	I	1.8V
35	VPP_IN	I	1.8V

Table 2-2. Required Device Power (continued)

Pin	Signal	Direction (I/O)	Required Voltage (Typical)
39	PA_LDO_IN	I	3.3V
40			

2.2.1 Power Input/Output Requirements

The following lists the supply connections in descending order of criticality:

- PA_LDO_OUT (pin 1): Provide decoupling capacitor (1.0µF)
- VDDA_IN1 (pin 4): Provide decoupling capacitor (0.1µF)
- VDDA_IN2 (pin 5): Provide decoupling capacitor (0.1µF)
- DIG_LDO_OUT (pin 31): Provide decoupling capacitor (1.0µF)
- VPP_IN (pin 35): Provide decoupling capacitor (0.1µF).
- VIO (pin 17): Provide decoupling capacitor (0.1µF)
- VDD_MAIN_IN (pin 32): Provide decoupling capacitor (0.1µF)

Prioritize the placements of the bypass capacitor in order of criticality to maximize RF performance.

2.2.2 Boot Sequence

For the CC33xx devices to function correctly, the proper boot-up sequence must be followed. Normally, this boot sequence is managed by the host driver, in regard to the interfaces between the CC33xx devices and the host.

Figure 2-3 shows the top-level boot-up sequence for the CC33xx devices when using an SDIO (secure digital input output) for the default setup of the host processor communication.

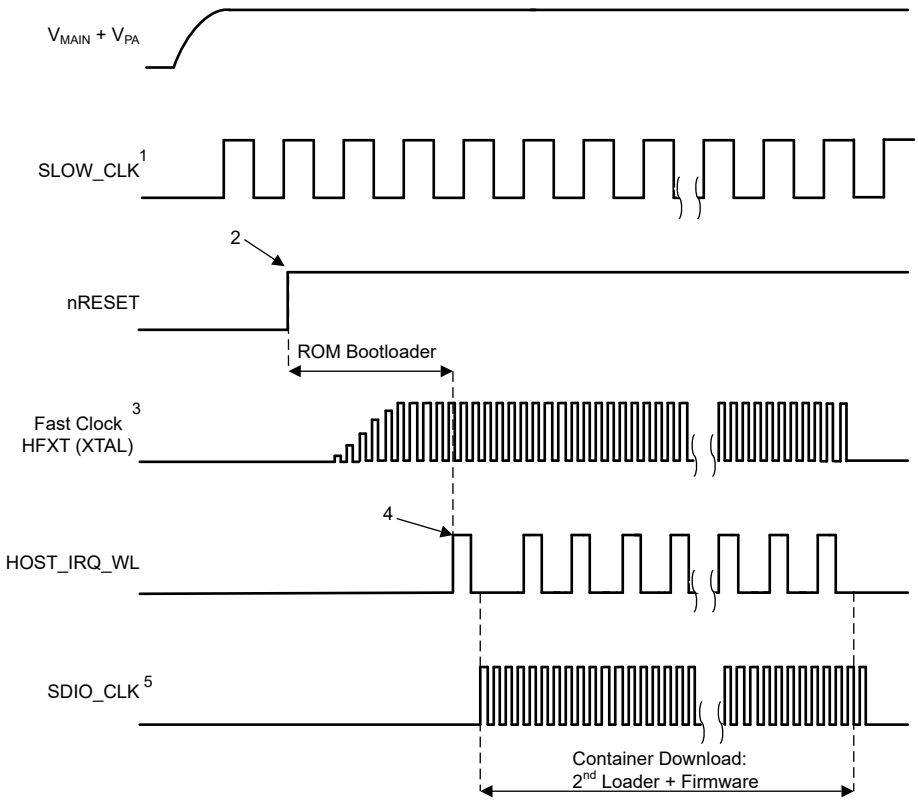


Figure 2-3. CC33xx Boot-Up Sequence

Note

1. For an external slow clock, verify that the clock is stable before the nReset pin is deasserted (high).
2. All power supplies (VDD_MAIN_IN, VDDA, V_{IO}, and V_{PA}) must be stable for at least 10µs before the nReset pin is deasserted.
   a. VDD_MAIN_IN and V_{IO} power supplies must be supplied from the same source to prevent glitches on the I/O during power up.
   b. VDDA_IN1/IN2 and PA_LDO_IN power supplies can be supplied independent of all other supplies.
3. The external fast clock (HFXT) stabilizes during the ROM bootloader. The clock is active until the CC33xx devices go into sleep mode after boot is finished.
4. The ROM bootloader is the first stage of boot for the CC33xx devices, boot begins when the nReset pin deasserts (device enable) and ends when the HOST_IRQ_WL line indicates *output high* from the device, signaling to the host that the boot stage is complete.
5. After the ROM bootloader is finished, the CC33xx devices are ready to receive two binary containers (secondary bootloader + firmware) from the host processor through the SDIO or SPI lines. These containers are downloaded in memory chunks. At the end of every chunk, the CC33xx devices send an acknowledge to the host by raising the HOST_IRQ_WL line *high*, signaling to the host that the memory chunk was successfully received and the device is now ready for the next chunk. The SDIO_CLK is driven by the host and is only active when the SDIO bus is active.
6. After the containers are downloaded to the CC33xx devices, the device completes initialization and goes to sleep.

2.2.2.1 SOP Modes

The logger (pin 28) and Host_IRQ_WL (pin 29) signals are considered *sense on power* (SOP) pins. When connecting these pins to a host, verify the Host_IRQ_WL (pin 29) pin stays at the *low* logic level during power-up, and the logger (pin 28) pin stays at the *high* logic level during power-up.

If these SOP pins are connected to a host that potentially affects the logic level of these lines, consider adding an optional pulldown and pullup resistor (or resistors).

2.2.3 Power Down Sequence

To improve reliability and proper boot up following the power-down sequence, follow the correct power-down sequence for the CC33xx devices.

To perform the correct shutdown sequence:

1. Deassert (low) the nRESET pin while the slow clock (if sourced externally) and all supplies to the device (VDD_MAIN_IN, VDDA, V_{IO}, and V_{PA}) are still stable and available.
2. Only deassert the power supplies to the chip *after* the nRESET signal is deasserted (low).

A successful shutdown sequence takes 1µs to complete after the nRESET pin is low. If the power supplies remain stable, the minimum time between two successive device enables is 1µs.

2.3 Clock Source

The CC33xx devices use two clocks for operation:

- A fast clock running at 40MHz for WLAN/BLE functions
- A slow clock running at 32.768kHz for low power modes

The fast clock must be generated externally. The slow clock can be generated internally by the devices or externally by an oscillator.

Note

A deviation in clock frequency is reflected as a deviation in radio frequency.

See *Simplelink Frequency Tuning* for more information on how to carefully select external loading capacitors (C_L) to complete frequency tuning according to specific board layouts.

2.3.1 Fast Clock

The CC33xx devices support a crystal-based fast clock (XTAL). The crystal is fed directly between the HFXT_P and HFXT_M pins with the appropriate loading capacitors and a 150Ω resistor. The following lists the design requirements:

1. Provide a 150Ω resistor on the HFXT_P (pin 6), close to the device and before the XTAL.
2. Connect the XTAL across the HFXT_P (pin 6) and HFXT_M (pin 7) pins.
3. Provide load capacitors (6.8pF) at both pins of the XTAL.

Note

The recommended load capacitor of 6.8pF is based on a TI board layout.

4. Tune the load capacitance, if required.

Note

The requirement of this step is based on the board layout implemented by the customer. This step is not always necessary for the board. See *Simplelink Frequency Tuning* for further guidance on this topic.

The fast clock component must meet the requirements shown in Table 2-3.

Table 2-3. External Fast Clock XTAL Specifications

Parameter	Test Conditions	MIN	TYP	MAX	Unit
Supported frequencies			40		MHz
Frequency accuracy	Initial + temperature + aging			±25 | ±20(2)	ppm
Load Capacitance, C_L (1)		5		13	pF
Equivalent series resistance, ESR				40	Ω
Drive level			100		µW

(1) Load capacitance, C_L= [C1 × C2] / [C1 + C2] + C_P, where C1 and C2 are the capacitors connected on the HFXT_P and HFXT_M pins, respectively, and C_P is the parasitic capacitance (typically 1pF to 2pF). For example, for C1 = C2 = 6.2pF and C_P = 2pF, then C_L = 5pF.
(2) When selecting an XTAL for a design implementing the CC330x devices, the accuracy requirement for frequency is ±25ppm. When selecting an XTAL for a design implementing the CC335x devices, the accuracy requirement for frequency is ±20ppm.

2.3.2 Slow Clock

The slow clock is generated by the internal oscillator of the device, but an external oscillator can also be used.

2.3.2.1 Slow Clock Generated Internally

To minimize external components, the slow clock can be generated by an internal oscillator. However, this clock is less accurate and consumes more power than sourcing the slow clock externally. For this scenario, the SLOW_CLK_IN pin must be left not connected.

2.3.2.2 Slow Clock Using an External Oscillator

For excellent power consumption, the slow clock can be generated externally by an oscillator or sourced elsewhere in the system. The external source must meet the requirements shown in Table 2-4. This clock can be fed into the SLOW_CLK_IN pin of the CC335xMOD and must be stable before the nReset pin is deasserted and the device is enabled.

Table 2-4. External Slow Clock Requirements

Parameter		Description	MIN	TYP	MAX	Unit
	Input slow clock frequency	Square wave		32768		Hz

Table 2-4. External Slow Clock Requirements (continued)

Parameter		Description	MIN	TYP	MAX	Unit
	Frequency accuracy	Initial + temperature + aging			±250	ppm
	Input duty cycle		30%	50%	70%	
T_r/T_f	Rise and fall time	10% to 90% (rise) and 90% to 10% (fall) of digital signal level			100	ns
V_{IL}	Input low level		0		0.35 × V_{IO}	V
V_{IH}	Input high level		0.65 × V_{IO}		1.95	V
	Input impedance		1			MΩ
	Input capacitance				5	pF

2.4 Radio Frequency (RF)

Routing out the RF_BG (pin 2) pin is required for radio frequency (RF) functionality when implementing the CC330x devices in a single band 2.4GHz configuration. Routing out the RF_A (pin 38) pin is an additional requirement for radio frequency (RF) functionality when implementing the CC335x devices in a dual band 5GHz configuration.

On the RF_BG trace, a band pass filter (BPF) is required along this path before reaching any radiative or conductive component. See Table 2-1 for the recommended BPF. TI also recommends implementing an impedance matching network (such as a PI or L network) before the antenna for better RF performance. Figure 2-4 is an example of a schematic design for the RF path when designing for a single band 2.4GHz configuration (CC330x). In this single band 2.4GHz use-case, ground the RF_A (pin 38) pin for better noise reduction.

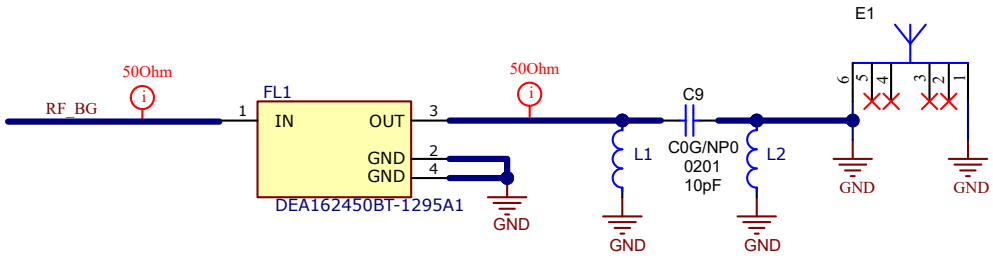


Figure 2-4. BG Band Reference Schematic

Additional routing is required for the RF_A (pin 38) pin when designing with the CC335x devices to enable a 5GHz RF with a 2.4GHz RF band. To use both RF bands, a diplexer is required. The RF_A pin must be connected directly to the high-band port of the diplexer, while the RF_BG pin must be routed through a BPF (as described previously) and then be connected to the Low_Band port of the diplexer. See Table 2-1 for the recommended diplexer. Figure 2-5 is an example of a schematic design for the RF path when designing for a dual band 2.4GHz + 5GHz configuration (CC335x).

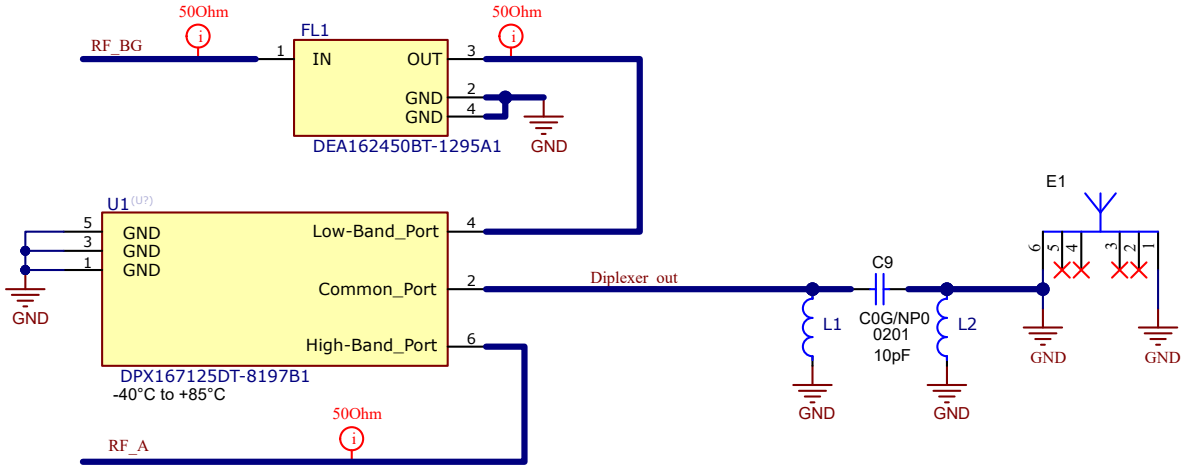


Figure 2-5. Dual Band RF Reference Schematic

Any deviation from these recommendations can cause performance to diverge from the specifications in the datasheet.

If implementing an RF switch (for utilizing antenna diversity), make the input of the RF switch the final filtered signal from the CC33xx devices. Specifically, when designing a dual band RF device that implements a CC335x device, route the input of the RF switch to the common port of the diplexer. When designing a single band RF device that implements a CC330x device, the input of the RF switch must be routed to the output of the BPF. The ANT_SEL (pin 15) pin can be routed and used as the switching signal. The RF_IN signal must be connected to the output of the BPF when designing a single band device (with CC330x), and the RF_IN signal must be connected to the common port of the diplexer when designing a dual band device (with CC335x), as seen in Figure 2-6.

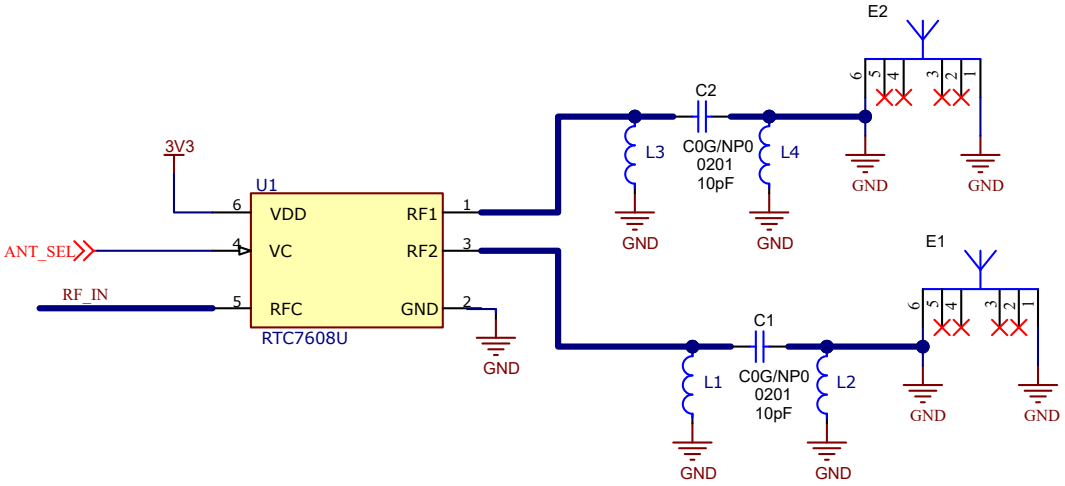


Figure 2-6. Antenna Diversity Reference Schematic

2.5 Digital Interfaces

All I/O signals on the CC33xx devices operate at 1.8V (typical). Designs must take in consideration the use of level shifters if utilized at a higher voltage in a system.

2.5.1 Reset

The nReset (pin 33) pin, an *active low* signal, must be connected and controlled by the host. If using a setup without a host, reset must be pulled high after power supplies are stable.

When reset is low, the device enters an active shutdown mode. After the device is re-enabled, firmware must be re-downloaded for proper operation.

2.5.2 Secure Digital Input Output (SDIO)

An SDIO is the main host interface for wireless communication. The CC33xx devices support a shared SDIO interface for both Wi-Fi and Bluetooth Low Energy protocols.

As per the SDIO specification, the host expects these data lines to be pulled up (SDIO_D0, SDIO_D1, SDIO_D2, SDIO_03, and SDIO_CMD). Take care that components (for example, level shifters) on the SDIO data lines do not change the state of these data lines to a low logic level.

2.5.2.1 SDIO Timing Diagram: Default Speed

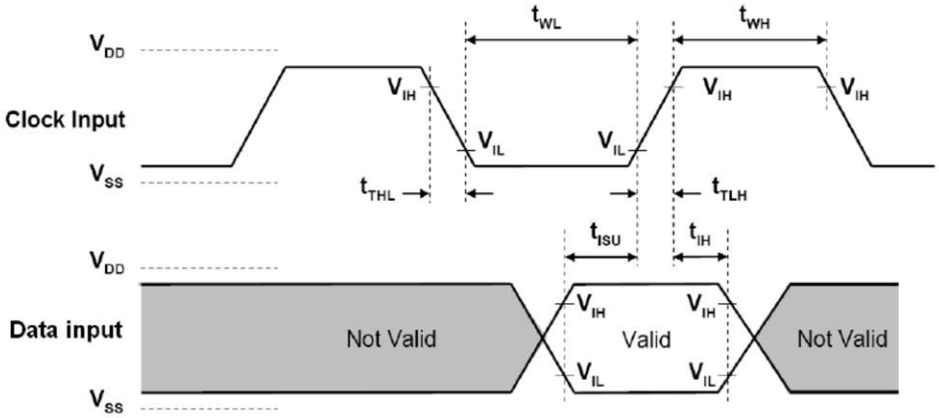


Figure 2-7. SDIO Default Input Timing

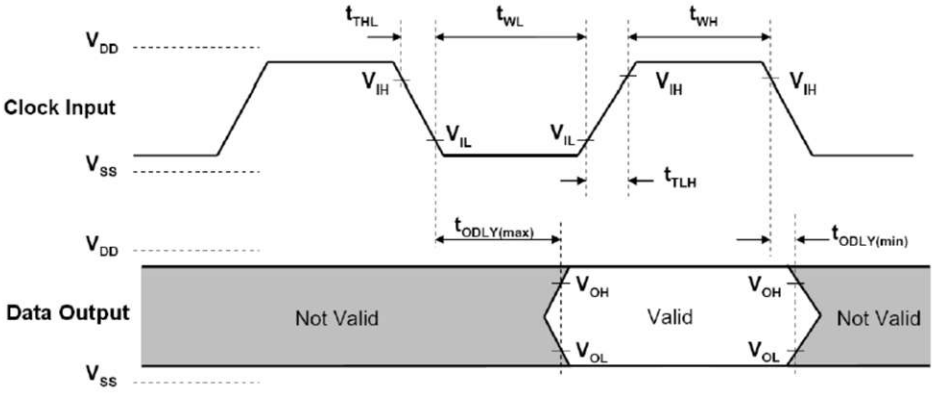


Figure 2-8. SDIO Default Output Timing

Table 2-5. SDIO Timing Parameters: Default Speed

Parameter	Description	MIN	MAX	Unit
f_{clock}	Clock frequency, CLK		26	MHz
t_{High}	High period	10		ns
t_{Low}	Low period	10		
t_{TLH}	Rise time, CLK		10	
t_{THL}	Fall time, CLK		10	
t_{ISU}	Setup time, input valid before CLK ↑	5		
t_{IH}	Hold time, input valid after CLK ↑	5		
t_{ODLY}	Delay time, CLK ↓ to output valid	2	14	
C_L	Capacitive load on outputs	15	40	pF

2.5.2.2 SDIO Timing Diagram: High Speed

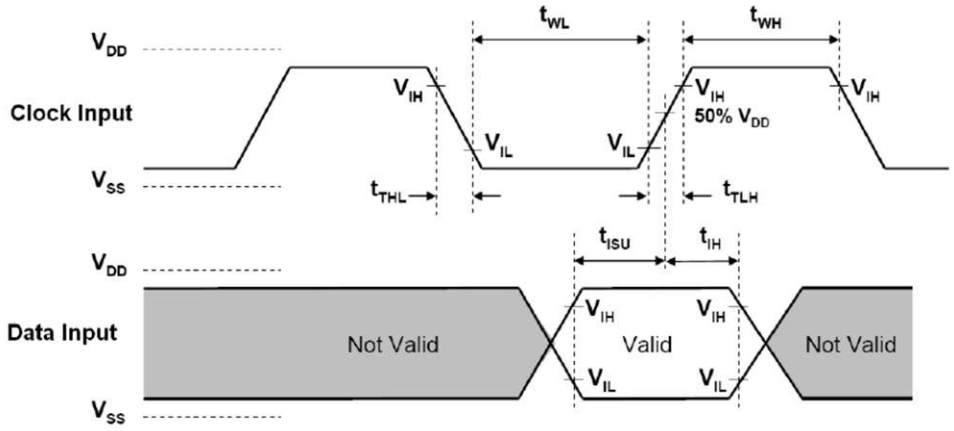


Figure 2-9. SDIO HS Input Timing

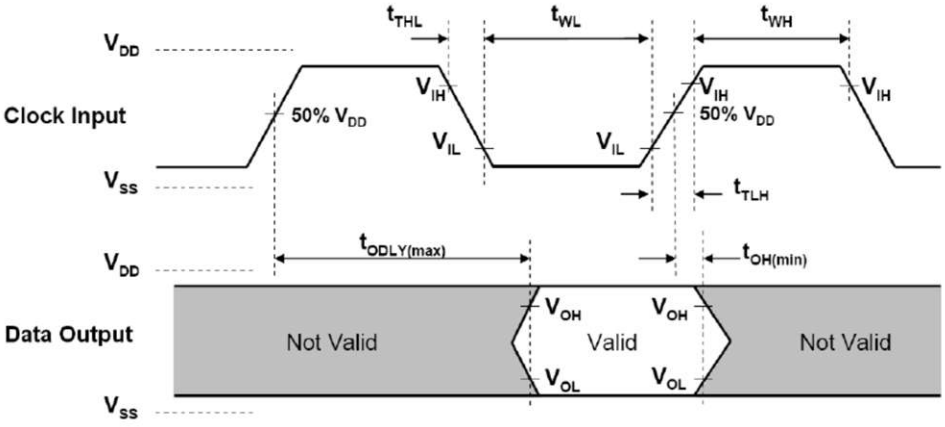


Figure 2-10. SDIO HS Output Timing

Table 2-6. SDIO Timing Parameters: High Speed

Parameter	Description	MIN	MAX	Unit
f_{clock}	Clock frequency, CLK		52	MHz
t_{High}	High period	7		ns
t_{Low}	Low period	7		
t_{TLH}	Rise time, CLK		3	
t_{THL}	Fall time, CLK		3	
t_{ISU}	Setup time, input valid before CLK ↑	6		
t_{IH}	Hold time, input valid after CLK ↑	2		
t_{ODLY}	Delay time, CLK ↑ to output valid	2	14	
C_L	Capacitive load on outputs	15	40	pF

2.5.3 Serial Peripheral Interface (SPI)

The SPI signal lines can be used as a host interface for wireless communication. The CC33xx devices also support a shared SPI for both BLE and WLAN. The SPI lines on CC33xx include:

- SDIO_CMD (SPI PICO)
- SDIO_CLK (SPI clock)
- SDIO_D3 (SPI CS)
- SDIO_D0 (SPI POCI)

2.5.3.1 SPI Timing Diagram

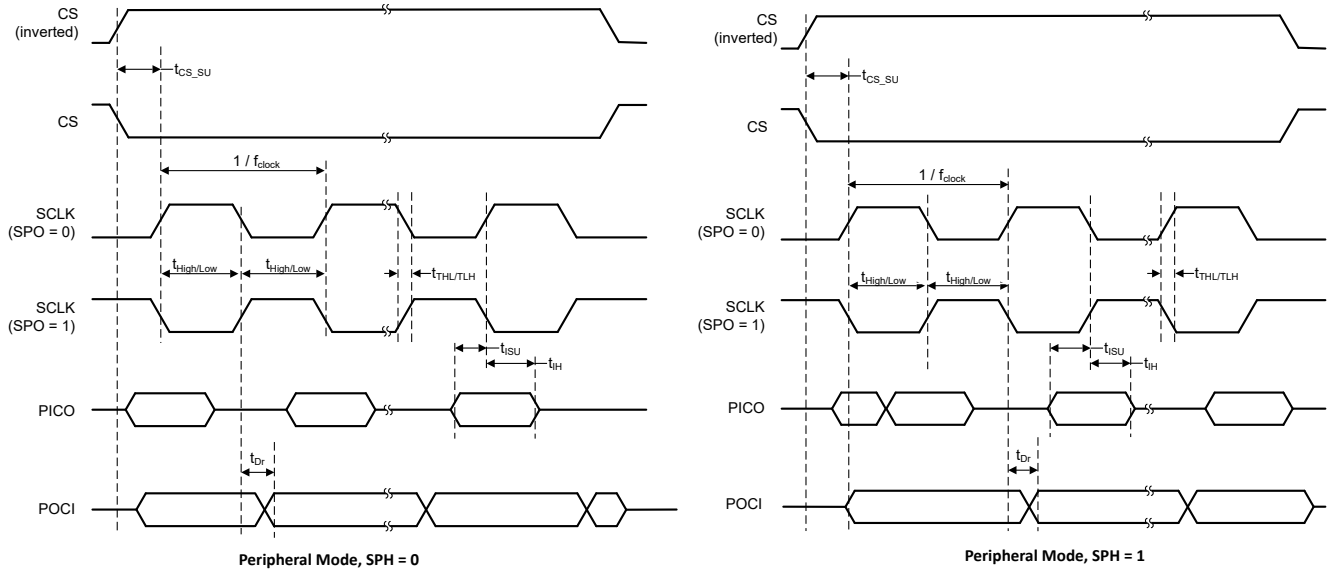


Figure 2-11. SPI Timing Diagram

Table 2-7. SPI Timing Parameters

Parameter	Description	MIN	MAX	Unit
f_{clock}	Clock frequency, CLK		26	MHz
t_{High}	High period	10		ns
t_{Low}	Low period	10		
t_{TLH}	Rise time, CLK		3	
t_{THL}	Fall time, CLK		3	
t_{CSsu}	CS setup time, CS valid before CLK ↑	3		
t_{ISU}	PICO, input valid before CLK ↑	3		
t_{IH}	PICO hold time, input valid after CLK ↑	3		
t_{Dr}, t_{Df} – Active	Delay time, CLK ↑/↓ to output valid	2	10	
t_{Dr}, t_{Df} – Sleep	Delay time, CLK ↑/↓ to output valid		12	
C_L	Capacitive load on outputs	15	40	pF

2.5.4 Universal Asynchronous Receiver-Transmitter (UART)

UART is the main host interface for BLE, which supports a host controller interface (HCI) transport layer. When a UART is used with a host, take care to connect the UART correctly:

- Pin 14 is the UART_TX of the device and must be connected to the host side of the RX.
- Pin 13 is the UART_RX of the device and must be connected to the host side of the TX.
- Pin 12 is the UART_CTS of the device and must be connected to the host side of the RTS.
- Pin 13 is the UART_RTS of the device and must be connected to the host side of the CTS.

2.5.4.1 UART Timing Diagram

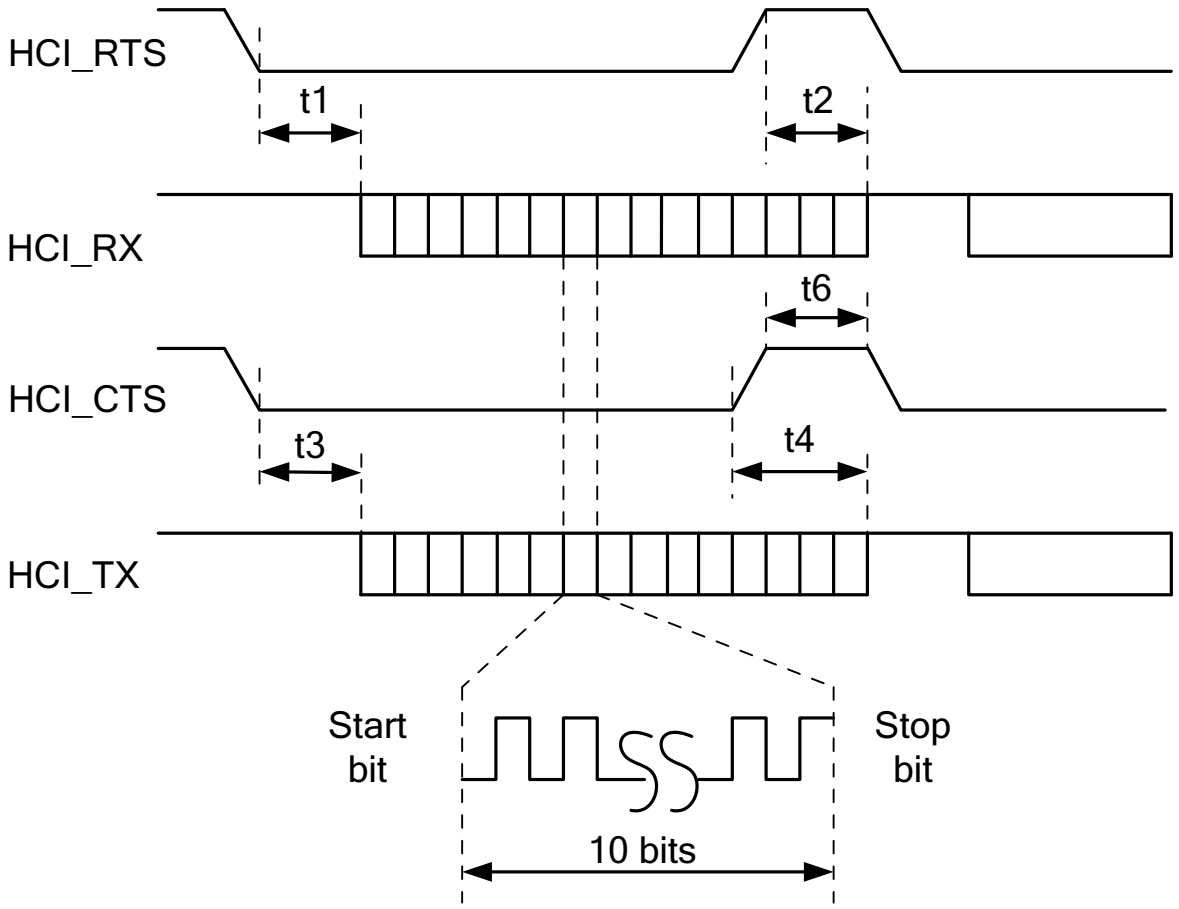


Figure 2-12. UART Timing Diagram

Table 2-8. UART Timing Parameters

Parameter	Condition	MIN	TYP	MAX	Unit
Baud rate		37.5		4364	kbps
Baud rate accuracy per byte	Receive | Transmit	–2.5		+1.5	%
Baud rate accuracy per bit	Receive | Transmit	–12.5		+12.5	%
CTS low to TX_DATA on		0	2		ms
CTS high to TX_DATA off	Hardware flow control			1	Byte
CTS high pulse width		1			bit
RTS low to RX_DATA on		0	2		ms
RTS high to RX_DATA off	Interrupt set to 1/4 FIFO			16	Byte

2.5.5 Serial Wire Debug (SWD)

The two serial wire debug pins on the CC33xx devices include the SWCLK (pin 26) and SWDIO (pin 27) pins. These signals are used for various types of debug (such as RF testing) and must be routed to test points for troubleshooting.

2.5.6 WLAN Interrupt Request (HOST_IRQ_WL)

The CC33xx devices require an interrupt request (IRQ) line between the individual device and the host, in addition to the serial host interface (SDIO, SPI, or UART). This IRQ line is important for the proper function of the host driver; where the host receives the signal from the CC33xx device any time the device tries to send something on the host interface, and the CC33xx signals an acknowledgment to the host when a message is received from the host.

By default, this IRQ line is a separate dedicated GPIO between the host and the CC33xx device. The option to use an SDIO in-band interrupt also exists; where the interrupt is enabled over the SDIO bus, reducing the total number of pins necessary to interface with the host.

2.5.7 Logger

The logger pin (pin 28) of the CC33xx devices is an output tracer for firmware logs. This pin is useful for an in-depth debug of the firmware running on the CC33xx devices. TI provides parsers and documentation for how to read these logs in the CC33xx driver release. The firmware logs are normally not necessary unless undergoing deep debugging with TI, as other debugging tools exists as part of the driver. In the event that these logs are necessary, connecting the logger pin routed to a test point is a good option for access.

2.5.8 Coexistence

The coexistence feature is a means to organize wireless packet traffic for communication protocols operating in the same frequency band. The CC33xx devices behave as the coexistence primary device and communicates to the coexistence secondary device. The CC33xx devices support a three-wire packet traffic arbitration (PTA) interface for coexistence. The coexistence signals on the CC33xx devices include:

- COEX_GRANT (pin 8) – an input signal, controlled by the coexistence primary device. Indicates the response of the PTA decision.
- COEX_PRIORITY (pin 9) – an output signal, controlled by the coexistence secondary device. Indicates the priority of a request signal.
- COEX_REQ (pin 10) – an output signal, controlled by the coexistence secondary device. Indicates a request to use the shared frequency band.

These three signals on the CC33xx devices must be routed to the matching coexistence pins on the coexistence secondary device. Coexistence is enabled by default and can be configured between internal and external in the INI file. Coexistence can be implemented with either one antenna (for both the primary and secondary device) or two antennas for each device. Figure 2-13 and Figure 2-14 provide visual examples of utilizing the coexistence feature with a Bluetooth Low Energy device.

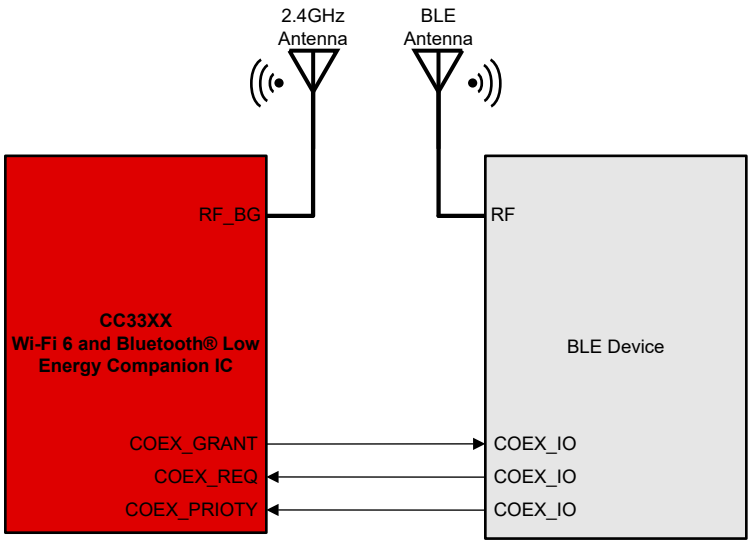


Figure 2-13. CC33xx Coexistence With Dual Antennas

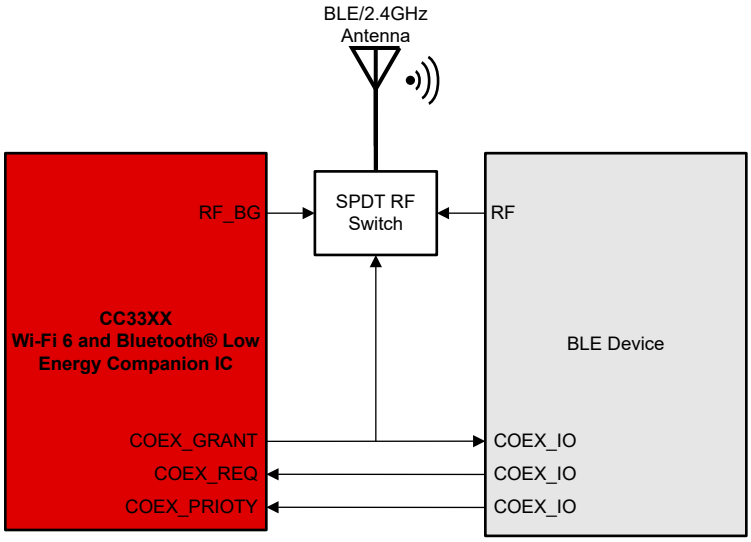


Figure 2-14. CC33xx Coexistence With Single Antenna

3 Layout Considerations – CC33xx Devices

The CC33xx devices are designed to integrate easily with any system and require few external components. The digital interface to the host processor (MPU or MCU) is highly flexible based on the end application. Users can decide to use any combination of SDIO, SPI, or UART for shared Wi-Fi and Bluetooth Low Energy communication.

This section explains the minimum requirements for optimizing an engine area layout.

3.1 Layout Reference Design

TI recommends following the reference design and guidelines for the CC33xx devices as closely as possible to achieve the full capabilities of the CC33xx devices (as listed in the device-specific data sheet) and to pass certification. These layout guidelines are especially important in the engine area, which includes the sensitive RF components and traces.

The following list of designs incorporate the layout guidelines given throughout this document. Refer to these as reference sources:

- CC330x Reference Design Files
- CC335x Reference Design Files
- BP-CC3351
- M2-CC3351

The figures below include the engine area of the CC33xx devices on the top layer (layer 1) and ground layer (layer 2) for all three designs.

Before proceeding with any hardware build involving the CC33xx devices, TI recommends submitting the design for review.

Note

The engine area (as shown in the reference design) can be rotated according to the design requirements.

3.1.1 Reference Design – CC330x Single Band Layout

Figure 3-1 is sampled from the CC330x reference design files.

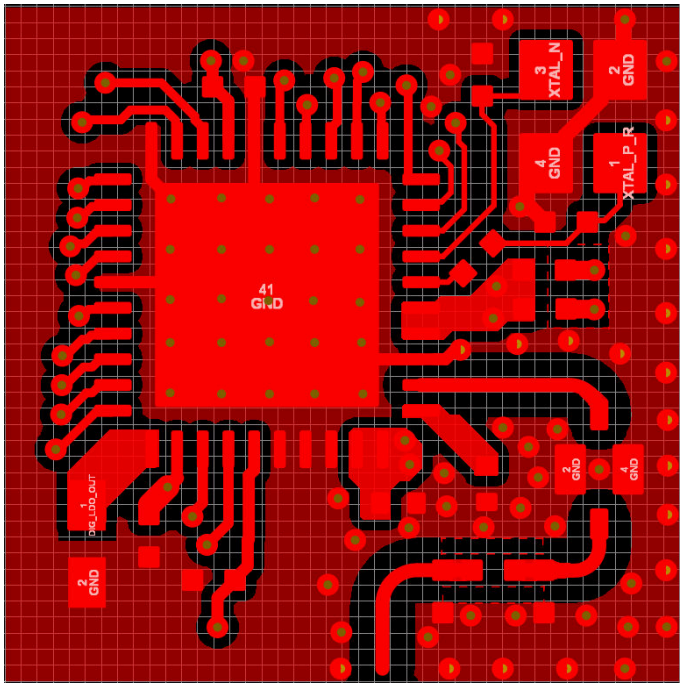


Figure 3-1. CC330x Reference Design, Top (Layer 1)

Figure 3-2 is sampled from the CC330x reference design files.

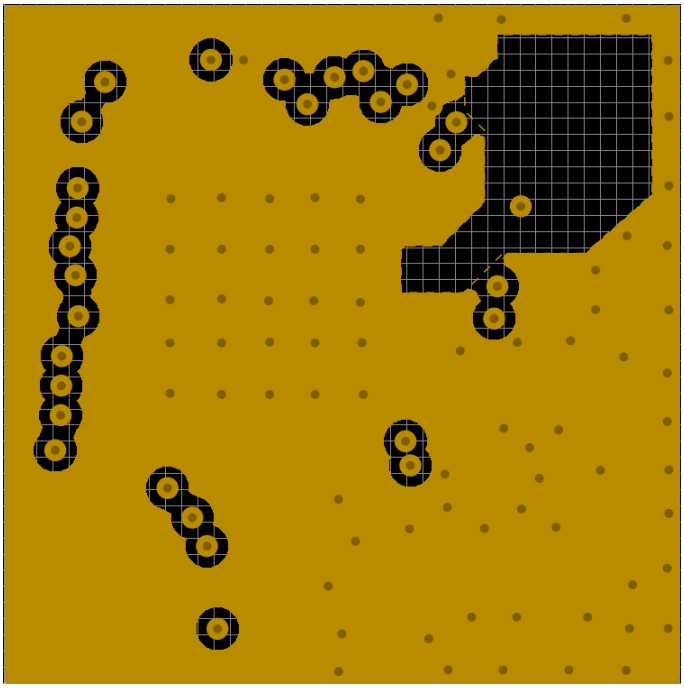


Figure 3-2. CC330x Reference Design, Ground (Layer 2)

3.1.2 Reference Design – CC335x Dual Band Layout

Figure 3-3 is sampled from the CC335x reference design files.

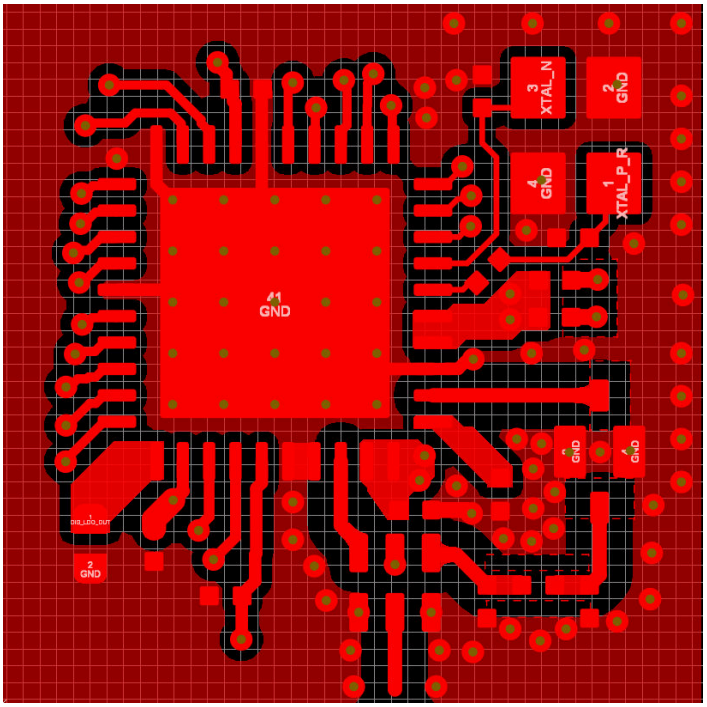


Figure 3-3. CC335x Reference Design, Top (Layer 1)

Figure 3-4 is sampled from the CC335x reference design files.

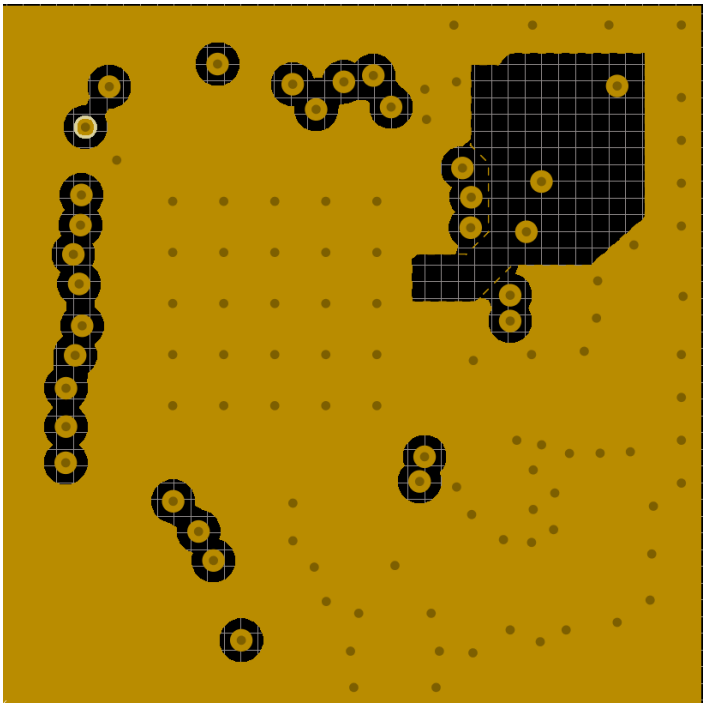


Figure 3-4. CC335x Reference Design, Ground (Layer 2)

3.1.3 BP-CC3351 Design Layout

Figure 3-5 is sampled from the BP-CC3351 design files.

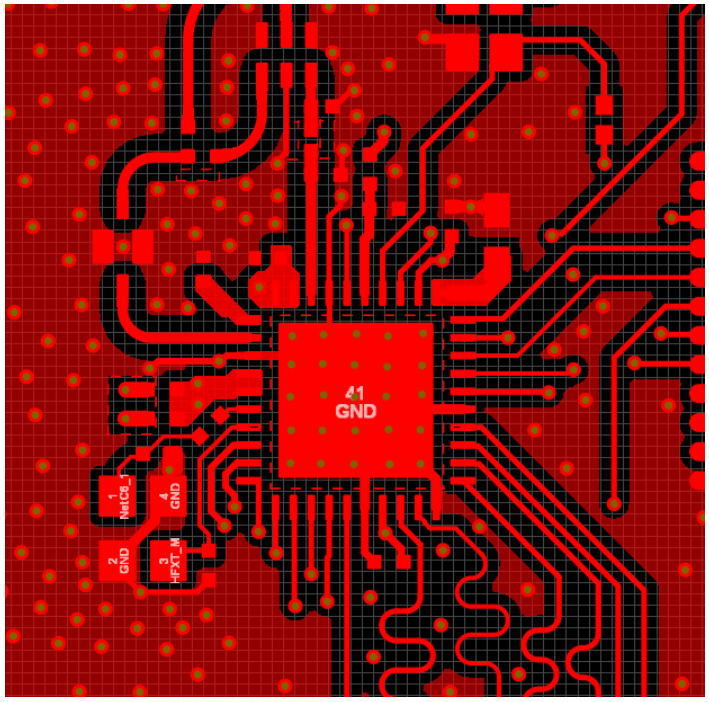


Figure 3-5. BP-CC3351 Layout, Top (Layer 1)

Figure 3-6 is sampled from the BP-CC3351 design files.

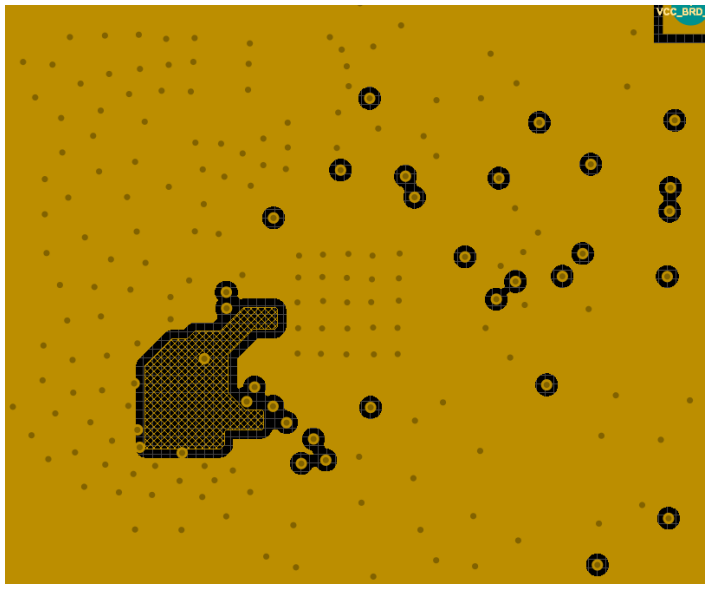


Figure 3-6. BP-CC3351 Layout, Ground (Layer 2)

3.1.4 M2-CC3351 Design Layout

Figure 3-7 is sampled from the M2-CC3351 design files.

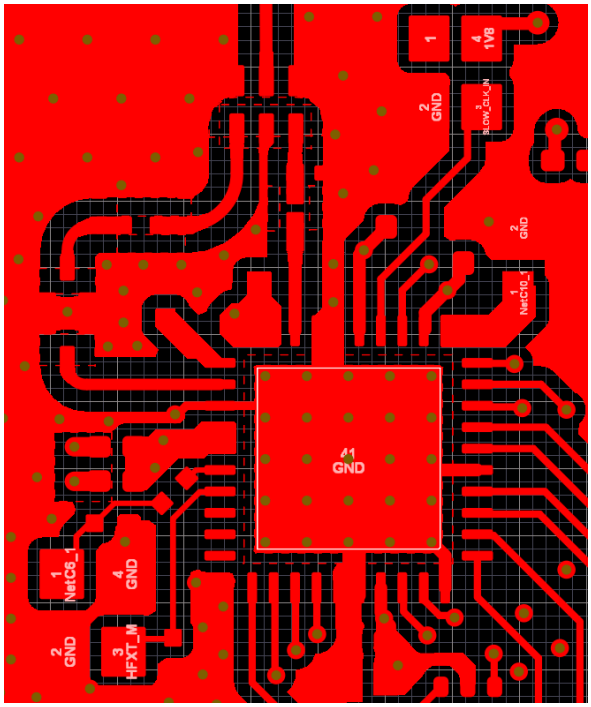


Figure 3-7. M2-CC3351 Layout, Top (Layer 1)

Figure 3-8 is sampled from the M2-CC3351 design files.

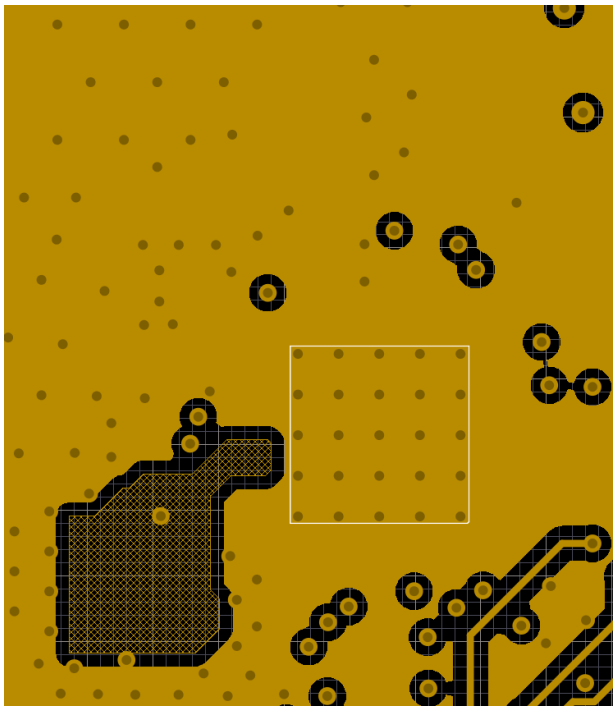


Figure 3-8. M2-CC3351 Layout, Ground (Layer 2)

3.2 IC Thermal Pad

Underneath the IC is one continuous ground plane on the top layer with 25 vias evenly distributed, as shown in Figure 3-9. This is important for thermal dissipation and RF performance.

Figure 3-9 is sampled from the CC330x reference design files.

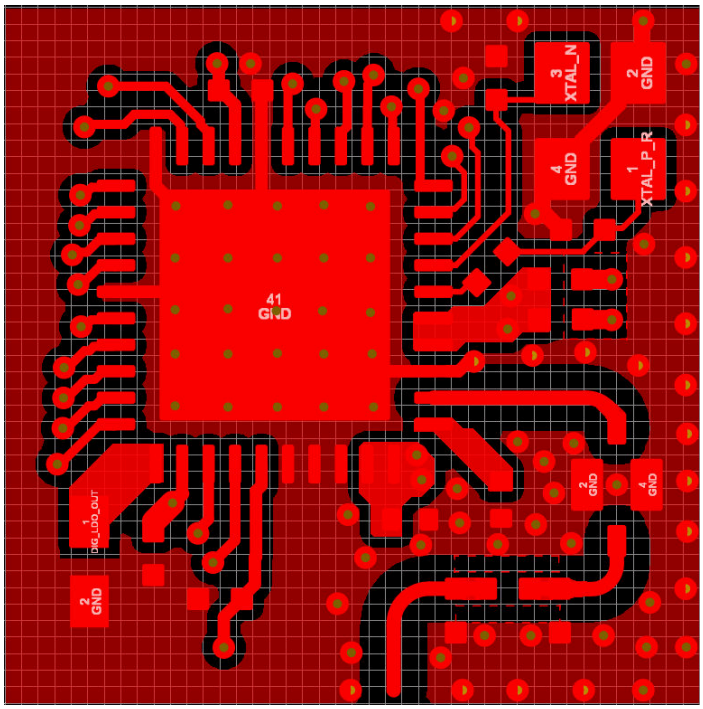


Figure 3-9. Reference Design Thermal Pad

3.3 Radio Frequency (RF)

Figure 3-10 and Figure 3-11 show the RF section, as depicted in the CC330x and CC335x reference designs. Special care must be taken when designing this area of the layout to achieve peak performance. A poor layout can cause performance degradation for the output power, EVM, sensitivity, and spectral mask.

Figure 3-10 is sampled from the CC330x reference design files.

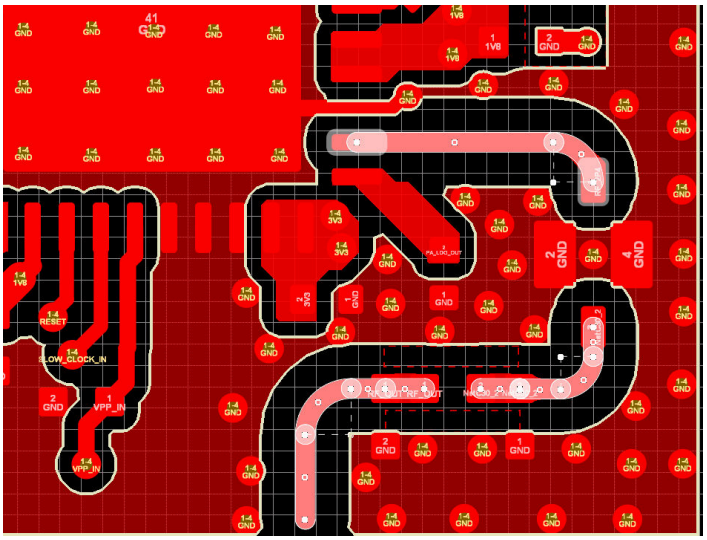


Figure 3-10. CC330x Reference Design RF Path

Figure 3-11 is sampled from the CC335x reference design files.

In Figure 3-11, the blue traces are the RF_BG trace (2.4GHz), the beige trace is the RF_A trace (5GHz), and the green trace is the combined RF trace connected to the common port of the diplexer, which is routed to an antenna.

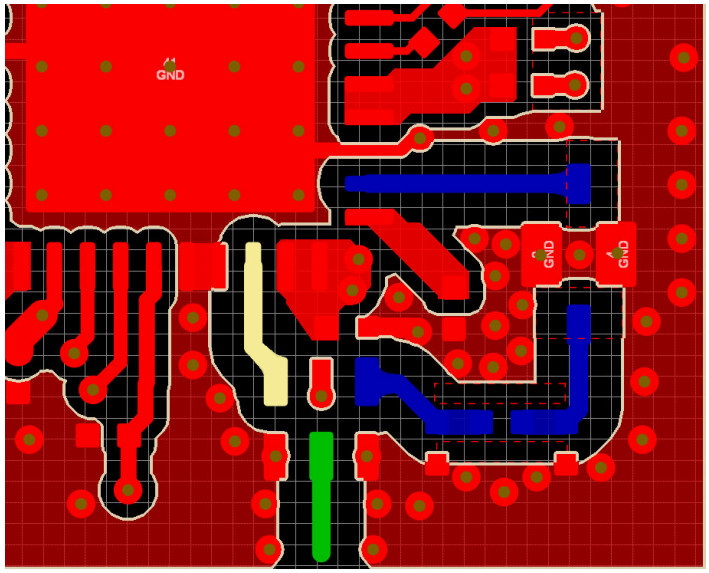


Figure 3-11. CC335x Reference Design RF Path

Follow the reference design as closely as possible and comply with the following list of guidelines:

Note

The guidelines for RF layout listed below are relevant for designs with the CC330x and CC335x devices, unless otherwise noted.

- The RF trace has a constant 50Ω characteristic impedance. This value is achieved by matching the CPWG based on the dielectric, layer stackup, ground plane spacing, and trace thickness. These parameters must remain consistent throughout the length of the trace.
- The entire RF trace is only on the top layer of the PCB, and the layer immediately underneath is one constant ground plane (with the exception of the XTAL cutout) for the trace reference.
- The RF trace is as clean and straight as possible with no components before the antenna, beside the band pass filter and matching filter. This action avoids unwanted component-to-component coupling. If a straight RF trace is not possible, rounded curves are acceptable.
- The RF trace is as isolated as possible from other components to decrease noise. Ground planes surround the RF trace, and the distance between ground through stitching is less than 1/8th of the minimum wavelength.
- The band pass filter is as far as possible (within design space limits) from the RF_BG (pin 2) pin and the VDDA decoupling capacitors on pins 4 and 5.
- A ground via is placed between the two ground pads for the band pass filter (BPF). The ground plane on both sides of the BPF is connected to enable one common ground plane for the entire area. There is increased ground through stitching in the ground plane between the BPF and the PA_LDO_OUT decoupling capacitor (pin 1).
- There is no high frequency signal traces or test points close to the RF trace.
- **CC335x only:** The diplexer is placed in such a way that the RF_A trace is as short as possible (the trace connecting the RF_A (pin 38) pin to the high port of the diplexer). The RF_A trace takes layout priority over the RF_BG trace when routing.

Another factor of impact in RF performance is the stackup. As an example, Table 3-1 contains the stackup (from top to bottom) used in the BP-CC3351 design.

Table 3-1. Stackup (From Top to Bottom) Used in All CC33xx EVMs

Layer	Name	Thickness	ε_r
	Top Soldermask		
	Top Solder	1.00mil	3.5
1	Top Layer	1.85mil	
	Dielectric 1	5.48mil	4.2
2	L02_GND	1.26mil	
	Dielectric 2	42.82mil	4.2
3	L03_PWR	1.26mil	
	Dielectric 3	5.48mil	4.2
4	Bottom Layer	1.85mil	
	Bottom Solder	1.00mil	3.5
	Bottom Soldermask		

The RF signal from the device is routed to the antenna using a coplanar waveguide (CPW-G) structure. This structure offers maximum isolation across the filter gap and shielding to the RF lines. Taking into account the stackup and trace measurements is imperative for achieving a 50Ω impedance. Figure 3-12 and Figure 3-13 illustrate examples of calculating trace impedance using the BP-CC3351 EVM as an example.

This image is sampled from the BP-CC3351 design files.

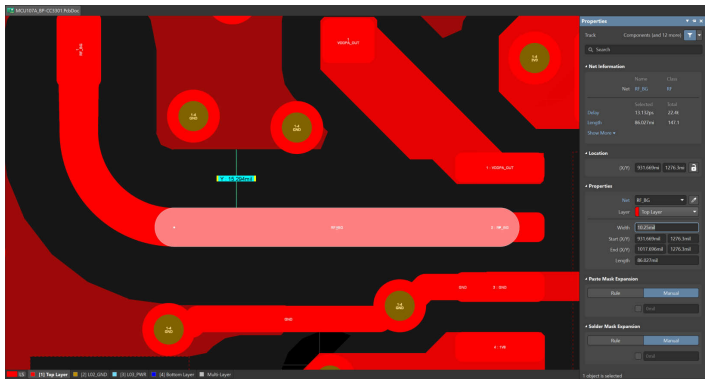


Figure 3-12. Example of Taking RF Trace Measurements

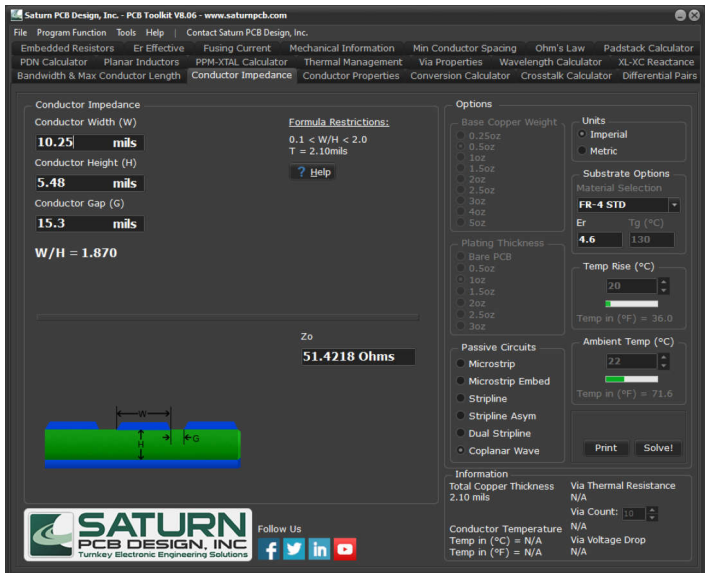


Figure 3-13. Example of Calculating Trace Impedance

3.4 XTAL

Figure 3-14 shows the placement and layout around the 40MHz XTAL and the connections to the CC33xx devices.

Figure 3-14 is sampled from the BP-CC3351 design files.

Figure 3-14. 40MHz XTAL From BP-CC3351

Follow the list of guidelines when integrating the XTAL:

- The traces connecting the XTAL to the CC3351 devices (XTAL_P and XTAL_M) is as short as possible with matching trace length.
- Place a 150Ω resistor on the XTAL_P pin as close as possible to the CC33xx devices.
- The two loading capacitors must be parallel to the edge of the XTAL.
- On the layer below the crystal (layer 2), place a cutout underneath the area of the XTAL and the loading capacitors. Verify that the layer below layer 2 (layer 3) has good ground underneath the same area. See Figure 3-15 for a visual representation.
- Wherever possible, increase ground through stitching around the XTAL for better isolation.

Figure 3-15 is sampled from layer 3 of the M2-CC3351 design files.

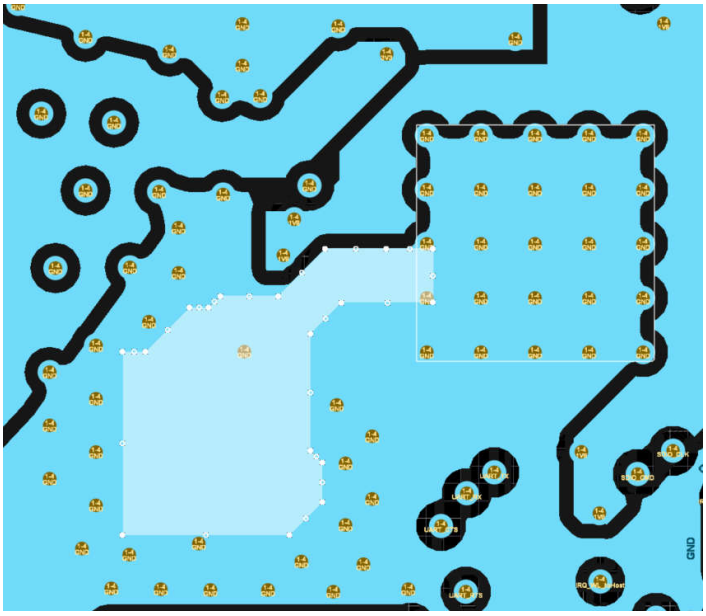


Figure 3-15. Reference Layout for Layer Under XTAL Cutout

3.5 Power Supplies

The power supplies, ground traces, and decoupling capacitors are important for improving layout. Since the decoupling capacitors can be close to the RF pins and traces of the device and power supplies, traces must be thick enough to support the necessary current to the device.

- PA_LDO_OUT (pin 1): TI suggests placing the decoupling capacitor close to the device pin and having a thick enough trace to have a low impedance path to the capacitor. See Figure 3-16 for a visual representation.
- VDDA_IN1 and VDDA_IN2 (pins 4 and 5): The supply side of the decoupling capacitors must be shorted together with a polygon region with two power vias (one for each decoupling capacitor). The ground side of each capacitor must go directly to ground by separate vias (not shorted together) and be isolated from the rest of the ground plane on the top layer.
- For the 1.8V power delivery, a thick trace or power plane must be used to carry the required amount of current consumption in the VDD_MAIN_IN, VIO, VDDA_IN1, VDDA_IN2, and VPP_IN pins combined. See Table 2-2 for maximum current consumption.
- The 1.8V path must be located around the device on a layer that is not the top layer or ground layer (place the path on layer 3 or 4). This way, the power path cannot interrupt the RF trace on the top layer (layer 1) or the continuous ground layer (layer 2). Only one via is used for each 1.8V power supply. The 1.8V supply currents must not flow under the device.
- For the 3.3V power delivery, a thick trace or a power plane must be used to carry the required amount of current consumption of the PA_LDO_IN pins. See Table 2-2 for more information. The power delivery must also be placed on a layer that is not the top layer or ground layer (layer 3 or 4).
- PA_LDO_IN (pins 39 and 40): These two pins must be shorted together with a solid region. The decoupling capacitor must be placed close to the device. Use two vias (if possible) to deliver the 3.3V rail.
- The ground for pins 37 and 38 must be shorted together with a solid region. This solid region must be connected to the thermal ground pad of the device.
- The ground for pin 3 must be shorted to the thermal pad under the device and to the ground plane that is adjacent to the RF trace.

Figure 3-16 is sampled from the BP-CC3351 design files.

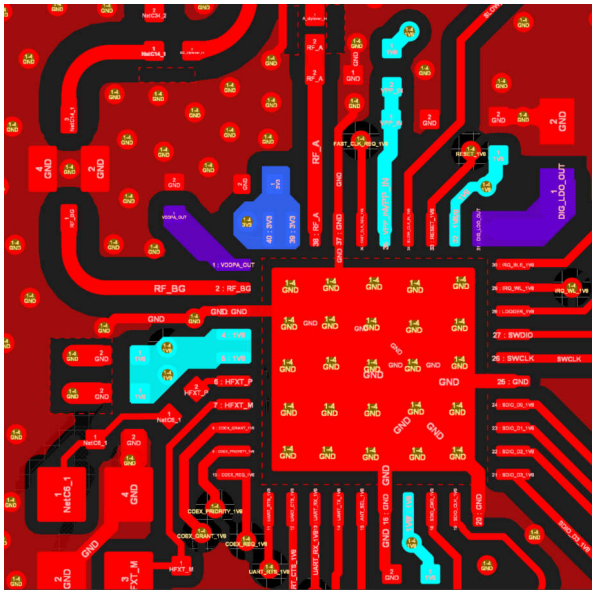


Figure 3-16. Reference Layout of CC33xx Power Supplies

Figure 3-17 is sampled from the M2-CC3351 design files.

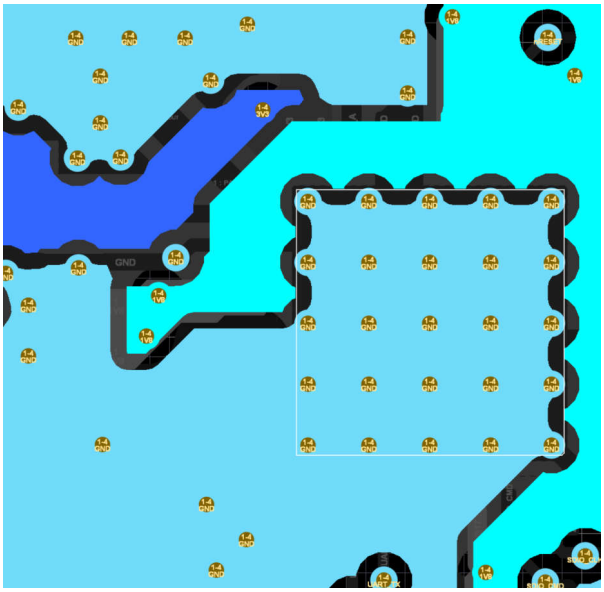


Figure 3-17. Reference Layout of the Power Layer

3.6 SDIO

SDIO is the main communication interface with the host processor for WLAN functions and can also be used for BLE functions in shared SDIO protocols. These signals are especially sensitive because of the clock, and must be designed as such.

The SDIO lines include SDIO_CLK, SDIO_CMD, SDIO_D0, SDIO_D1, SDIO_D2, and SDIO_D3. The SDIO_CLK signal, in particular, is very sensitive and must be regarded as such. To maintain reliable SDIO communication, take the following layout considerations into account:

- TI recommends that the SDIO lines have at least 5mils of width.
- The SDIO traces are as far away as possible from the other digital or analog signal traces.
- Ti recommends adding ground shielding around the SDIO bus.

- The SDIO_CLK must be isolated from all other signals with ground vias (stitching vias) and adjacent ground planes. The signal trace has a clearance of at least twice the trace width of the other SDIO signals.
- Route the SDIO lines in parallel to each other with lengths as short as possible (to decrease propagation delay), and have a clearance of 1.5x the trace width.
- The lengths of the SDIO traces must be length matched within 100mil tolerance to provide the sampled data at the same time on all of the traces. See Figure 3-18 for a visual example of length tuning.

Figure 3-18 is sampled from the BP-CC3351 design files.

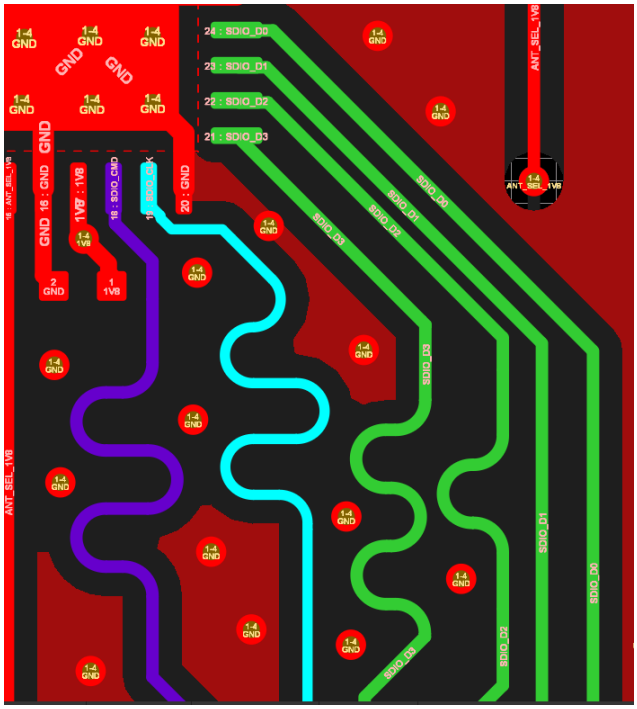


Figure 3-18. Reference Layout for SDIO Signals

4 Schematic Considerations – CC33xxMOD

The CC33xxMOD devices are designed to integrate easily with any system and require few external components. The digital interface to the host processor (MPU or MCU) is highly flexible based on the end application. Users can decide to use any combination of SDIO, SPI, or UART for shared Wi-Fi and Bluetooth Low Energy communication.

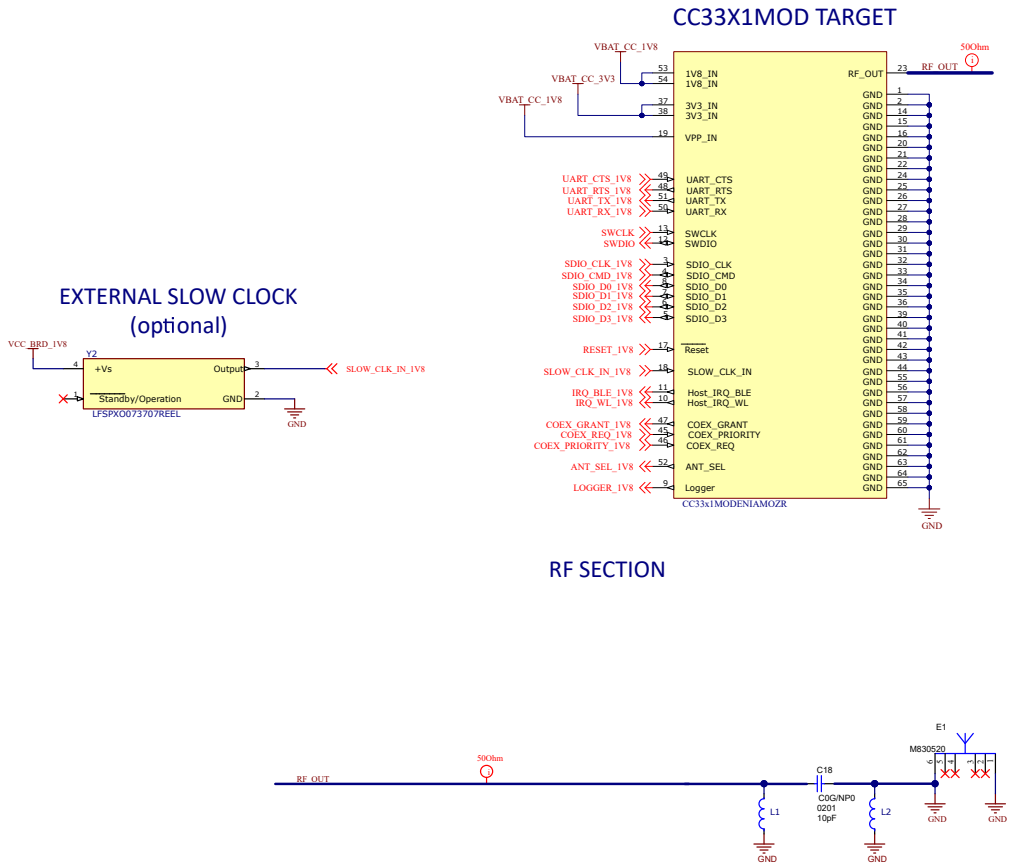
This section explains the differences between the schematic requirements for the CC33xx single chip and the CC33xxMOD pre-certified modules. All other guidance for the CC33xxMOD schematic is similar to the CC33xx devices and can be followed accordingly.

4.1 Schematic Reference Design – CC33xxMOD

TI recommends following the reference design and guidelines for the CC33xxMOD devices as closely as possible to achieve the full capabilities of the CC33xxMOD devices (as listed in the device-specific datasheet) and to pass certification. The schematic designs recommended for the CC33xxMOD devices are accessed here:

CC33xxMOD Reference Design Files

Figure 4-1 shows the reference schematic for the CC33xxMOD devices.



A. The slow clock can be generated internally. An external slow clock can be optionally used to consume less power than sourcing the slow clock internally.

Figure 4-1. CC33xxMOD Reference Schematic

4.2 Power Supply

There are two power rails that must be routed to the CC33xxMOD devices:

- 1.8V: 1V8_IN, VPP_IN
- 3.3V: 3V3_IN

See Section 4.2 for further information on the operating conditions for the supply pins.

Table 4-1. Required Device Power

Pin	Signal	Direction (I/O)	Required Voltage (Typical)
19	VPP_IN	I	1.8V
53	1V8_IN	I	1.8V
54	1V8_IN	I	1.8V
37	3V3_IN	I	3.3V
38	3V3_IN	I	3.3V

5 Layout Considerations – CC33xxMOD

The CC33xxMOD devices are designed to integrate easily with any system and require few external components. The digital interface to the host processor (MPU or MCU) is highly flexible based on the end application. Users can decide to use any combination of SDIO, SPI, or UART for shared Wi-Fi and Bluetooth Low Energy communication.

This section explains the differences between the schematic requirements for the CC33xx single chip and the CC33xxMOD pre-certified modules. All other guidance for the CC33xxMOD layout is similar to the CC33xx devices and can be followed accordingly.

5.1 CC33xxMOD RF Layout Recommendations

The RF section of this wireless module gets top priority in terms of layout. Importantly, the RF section must be laid out correctly to maintain optimum performance from the module. A poor layout can cause low-output power, EVM degradation, sensitivity degradation, and mask violations.

Figure 5-1 shows the RF placement and the routing of the CC33xxMOD module with an external antenna.

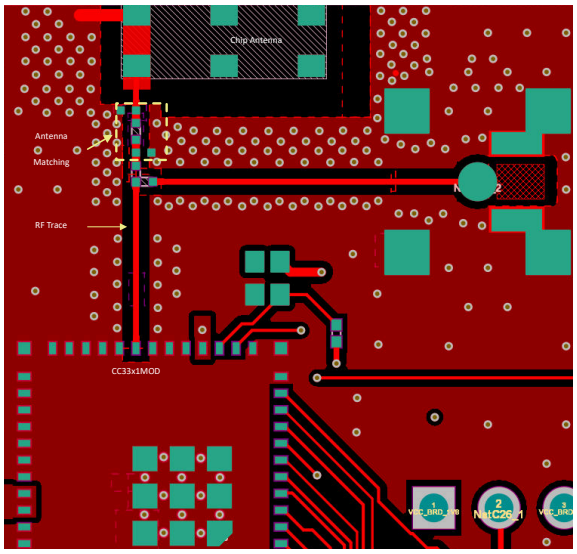


Figure 5-1. RF Section Layout

Follow these RF layout recommendations for the CC33xxMOD devices:

- RF traces must have 50Ω impedance and must be no shorter than 9.24mm.
- RF trace bends must be made with gradual curves, and 90° bends must be avoided.
- RF traces must not have sharp corners.
- There must be no traces or ground under the antenna section.
- RF traces must have via stitching on the ground plane beside the RF trace on both sides.
- RF traces must be as short as possible. The antenna, RF traces, and the module must be on the edge of the PCB product in consideration of the product enclosure material and proximity.

Follow the list of general layout recommendations:

- Have a solid ground plane and ground vias under the module for a stable system and thermal dissipation.
- Do not place signal traces underneath the module on a layer where the module is mounted.

6 CC33xxMOD Regulatory Compliance

The CC33xxMOD family of devices are designed to be easy to use and enhance the time to market for customers.

Follow these high-level steps to complete integration:

1. Design your carrier PCB following:
   - The design guidelines listed in the Important Notice to OEM integrators .
   - Using an antenna from our Antenna Installation Guidelines.
2. Evaluate the regions where you plan to ship your product and determine the appropriate regulatory bodies:
   - For regions with a pre-existing TI certification:
     a. TI modules are designed to enable easy reuse of certification.
     b. Full integration steps depend on the region, but in most use-cases, integration is limited to paperwork or spot checking for compliance.

     a. For other regions:
        – Work with the appropriate regulatory body to determine the steps for compliance.

Reports for the CC33xxMOD can be found on the SIMPLELINK-CC33XX-REPORTS page.

IDs are listed in Table 6-1 and Table 6-2:

Table 6-1. CC33xxMOD Certification

Model Name	Regulatory Body	ID
CC33MOD	FCC	Z64-CC33SBMOD
	ISED	461I-CC33SBMOD
	CE / ETSI	N/A
	Japan / TELEC	201-250389 (Test Grade: 01, CC3301MOD)
	Japan / TELEC	201-250390 (Test Grade: 00, CC3300MOD)

Table 6-2. CC335xMOD Certification

Model Name	Regulatory Body	ID
CC33MOD-DB	FCC	Z64-CC33DBMOD
	ISED	461I-CC33DBMOD
	CE / ETSI	N/A
	Japan / TELEC	201-250530 (Test Grade: 51, CC3351MOD)
	Japan / TELEC	201-250531 (Test Grade: 50, CC3350MOD)

To leverage the IDs in Table 6-1 and Table 6-2 at a regulatory test facility, TI recommends submitting a Module Certification Request form and obtaining a letter of authorization from TI for FCC and ISED.

6.1 CC33xxMOD OEM Integration Manual

Important Notice to OEM integrators

1. This module is limited to OEM installation ONLY.
2. This module is limited to installation in mobile or fixed applications, according to Part 2.1091(b).
3. Refer to the datasheet for the device for trace design instruction. As a module using trace design, this module relies on the manufacturers of the host product to prepare the antenna connector. A unique antenna connector must be used on the Part 15 authorized transmitters used in the host product. The module manufacturer provides a list of acceptable unique connectors (see Table 6-3).

4. Separate approval is required for all other operating configurations, including portable configurations with respect to Part 2.1093 and different antenna configurations.
5. For FCC Part 15.31 (h) and (k): The manufacturer of the host product is responsible for additional testing to verify compliance as a composite system. When testing the host device for compliance with Part 15 Subpart B, the manufacturer of the host product is required to show compliance with Part 15 Subpart B while the transmitter module (or modules) are installed and operating. The modules must be transmitting, and the evaluation must confirm that the intentional emissions of the module are compliant (for example, fundamental and out of band emissions). The manufacturer of the host product must verify that there are no additional unintentional emissions other than what is permitted in Part 15 Subpart B or emissions are complaint with the rules of the transmitter (or transmitters).

Antenna Installation

(1) The antenna must be installed such that 20cm is maintained between the antenna and users.
(2) The transmitter module cannot be co-located with any other transmitter or antenna.
(3) Only antennas of the same type and with equal or less gains (see Table 6-3) can be used with this module. Other types of antennas and higher gain antennas possibly require additional authorization for operation.

Table 6-3. Certified Antenna List

Brand	Part Number	Antenna Type
Kyocera AVX	M830520	Chip antenna
Kyocera AVX	W4P42X8W04-U100D3B0A	External PCB antenna
TDK	ANT162442DT-2001A2	Chip antenna
Yageo/Pulse	ANT3216A063R2455A	Chip antenna
Yageo/Pulse	W5028T	Dipole(stick) antenna
Yageo/Pulse	W3078T	Monopole(ceramic) antenna
Yageo/Pulse	W3917B0100T	Dipole(flexible PC) antenna
Molex	204281-1100	External PCB antenna
Molex	146153-1100	External PCB antenna
Molex	204281-0100	External PCB antenna
Molex	146153-0100	FPC
Unictron	H2B1PD1A1C385L	External PCB antenna
Unictron	H2B1PC1A1C095L	External PCB antenna
Laird	WTS2450	Whip antenna
TEConnectivity	001-0012	Whip antenna
Chang Hong	DA-2458-02	Whip antenna
Ezurio	001-0016	Flex PIFA

Note

CC33xxMOD is only certified with the M830520 antenna. CC335xMOD does support all of the antennas in Table 6-3.

EMI Considerations

Refer to KDB 996369 D04 and D02 for guidance regarding compliance of the host device.

Information to Make Changes

Only Grantees are permitted to make permissive changes. Please contact TI if the host integrator expects the module to be used differently than as granted: connectivity-wifi-cert@list.ti.com

7 Revision History

NOTE: Page numbers for previous revisions may differ from page numbers in the current version.

Changes from December 14, 2023 to January 8, 2026 (from Revision * (December 2023) to Revision A (January 2026)) **Page**

- Updated with module information....3
- Removed links and added tables....3
- Added additional schematic information....4
- Removed the *Power-Up Sequence* section....7
- Added the *Boot Sequence* section....7
- Added additional content for a dual band configuration....10
- Added the *WLAN Interrupt Request (HOST_IRQ_WL)* section....16
- Added the *Logger* section....16
- Added the *Schematic Considerations – CC33xxMOD* section and subsections....30
- Added the *Layout Considerations – CC33xxMOD* section and subsections....31
- Added the *CC33xxMOD Regulatory Compliance* section and subsections....32

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.

Last updated 10/2025