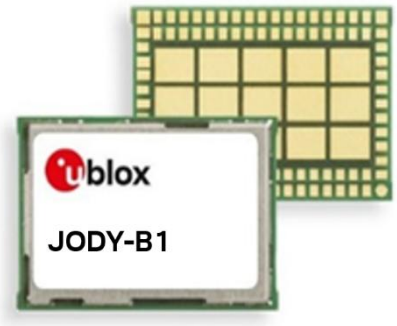
S

JODY-B1 series System Integration Manual

Host-based automotive modules with Dual-Mode Bluetooth®

System integration manual



Abstract

Targeted towards hardware and software application engineers, this document describes how to integrate JODY-B1 modules in application products and explains the hardware design-in, software, component handling, regulatory compliance, and production testing. It also lists the external antennas approved for use with the module.

UBXDOC-465451970-3820 - R01
C2-Restricted

Document information

Title	**JODY-B1 series System Integration Manual**	
Subtitle	Host-based automotive modules with Dual-Mode Bluetooth®	
Document type	System integration manual	
Document number	UBXDOC-465451970-3820	
Revision and date	R01	16-Jun-2025
Disclosure restriction	C2-Restricted	

Document status description	
Draft	For functional testing. Revised and supplementary data will be published later.
Objective Specification	Target values. Revised and supplementary data will be published later.
Advance Information	Data based on early testing. Revised and supplementary data will be published later.
Early Production Information	Data from product verification. Revised and supplementary data may be published later.
Production Information	Document contains the final product specification.

This document applies to the following products:

Product name	**Chipset**
JODY-B151-A	Qualcomm QCA8695AU

☞ For information about the related hardware, software, and status of listed product types, see also the data sheet [1].

Contents

1 System description

1.1 Overview

The JODY-B1 series System Integration Manual comprises compact modules based on the Qualcomm QCA8695AU chipset. The modules enable dual-mode Bluetooth® Classic (BR/EDR) and Bluetooth® Low Energy (LE) communication and are ideal for in-vehicle infotainment and telematics applications with common use cases that require Bluetooth.

JODY-B1 modules undergo qualification testing in accordance with u-blox Qualification Policy for automotive grade products based on AEC-Q104 and are manufactured in line with ISO/TS 16949 AEC-Q104. Host processor connections are made through high-speed UART or SPI for Bluetooth.

Radio type approvals for Europe (RED), the United States (FCC), and Canada (ISED) are planned.

1.2 Module architecture

JODY-B1 includes the Qualcomm QCA8695AU System-On-Chip (SoC). It is a Bluetooth baseband processor system with an integrated 2.4 GHz transceiver and fully integrated power management circuitry that provides power to the internal voltage domains of the SoC.

JODY-B1 supports an optional high selectivity LTE filter for the 2.4 GHz ISM band as shown in the block diagrams in the module data sheet [1].

☞ Coexistence filters are recommended for designs with co-located LTE devices operating in bands 7, 38, 40, or 41.

JODY-B1 supports an Universal Asynchronous Receiver Transmitter (UART) interface and HCI UART transport layer for host CPU connectivity.

A PCM/I2S interface is available to connect an external audio codec and external audio system.

2 Module integration

JODY-B1 shall be integrated into the application product together with a Host CPU. Figure 1 shows a typical integration.

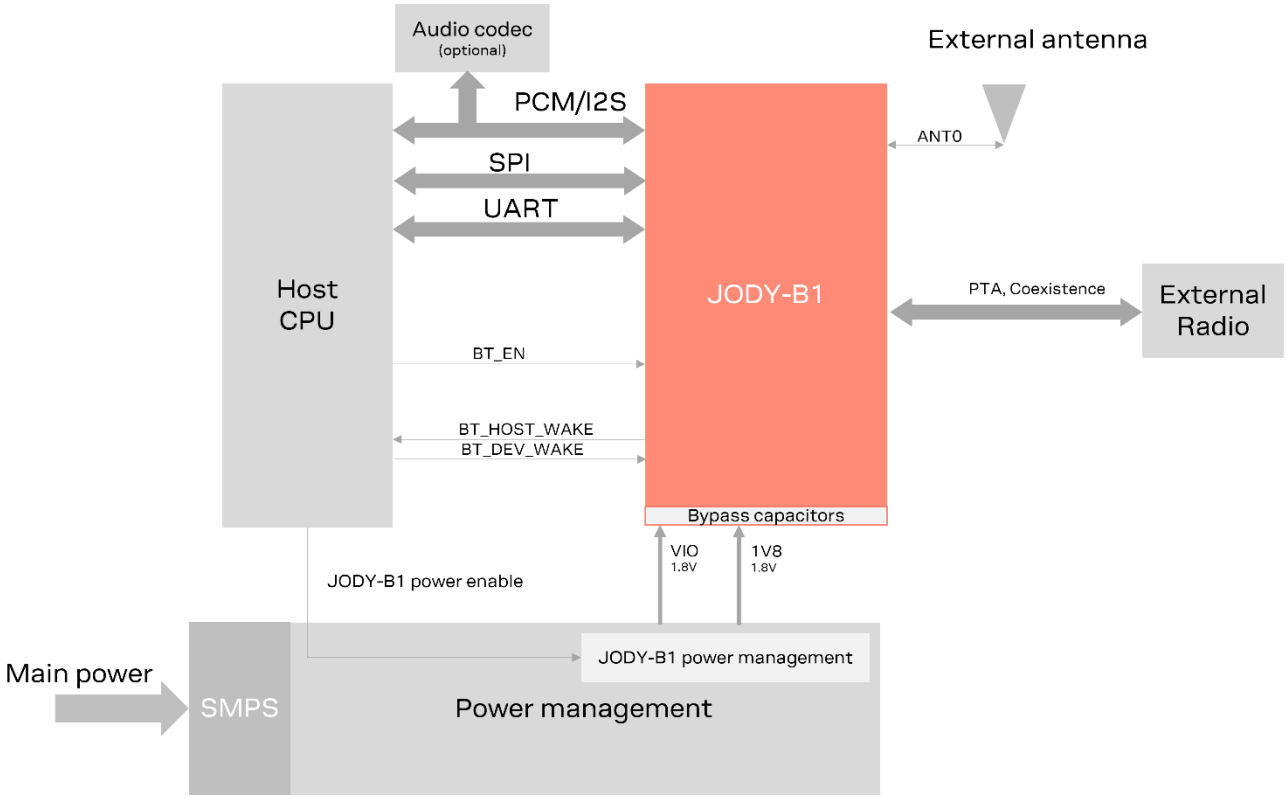


Figure 1: JODY-B1 integration in host system

- The UART interface is used for downloading the firmware and Bluetooth communication between the host and the controller.
- Bluetooth enable and host and module wake up signals are available to control JODY-B1 to and from host CPU.
- The module power is supplied through the **1V8** and **VIO** domain pins.
- For correct operation, it is important to correctly configure JODY-B1 with the settings and start-up sequences described in this document and in the JODY-B1 data sheet [1]. This configuration puts requirements for enabling the timing of power sources and the assertion of **BT_EN**.
- JODY-B1 includes a PCM/I2S interface that can be used to connect a codec for Bluetooth audio. If the interface is not used, it can be omitted.

2.1 Power supply interface

JODY-B1 series System Integration Manual power supply pins **1V8** and **VIO** pins must be sourced by a regulated DC power supply, such as an LDO or SMPS. The appropriate type for your design depends on the main power source of the application.

Pin	Name	I/O	Description	Remarks
3	VIO	PWR	VIO supply	1.8 V
4	1V8	PWR	1.8 V supply	1.8 V

Table 1: Power supply pins

The DC power supply can be taken from any of the following sources:

- Switched Mode Power Supply (SMPS)
- Low Drop Out (LDO) regulator

Module power up must strictly follow the defined power-up sequence. It is important to design the power management to comply with the recommended power-up sequence.

The current consumed through the supply pins by JODY-B1 series System Integration Manual modules can vary by several orders of magnitude depending on the operation mode and state. The current consumption can change from high consumption, experienced during Bluetooth transmission at maximum RF power level in connected-mode, to low current consumption during the low power idle-mode when power saving is enabled. Regardless of the chosen DC power supply, it is crucial that it can satisfy the high peak current consumed by the module. When designing the supply circuitry for the module, a contingency of at least 20% over the stated peak current is recommended.

2.2 Antenna interfaces

JODY-B1 series System Integration Manual modules include an antenna pin to connect an external antenna. Either integrated SMD or equivalent antennas mounted on application board or external antenna connected through antenna connector and coaxial cable can be used.

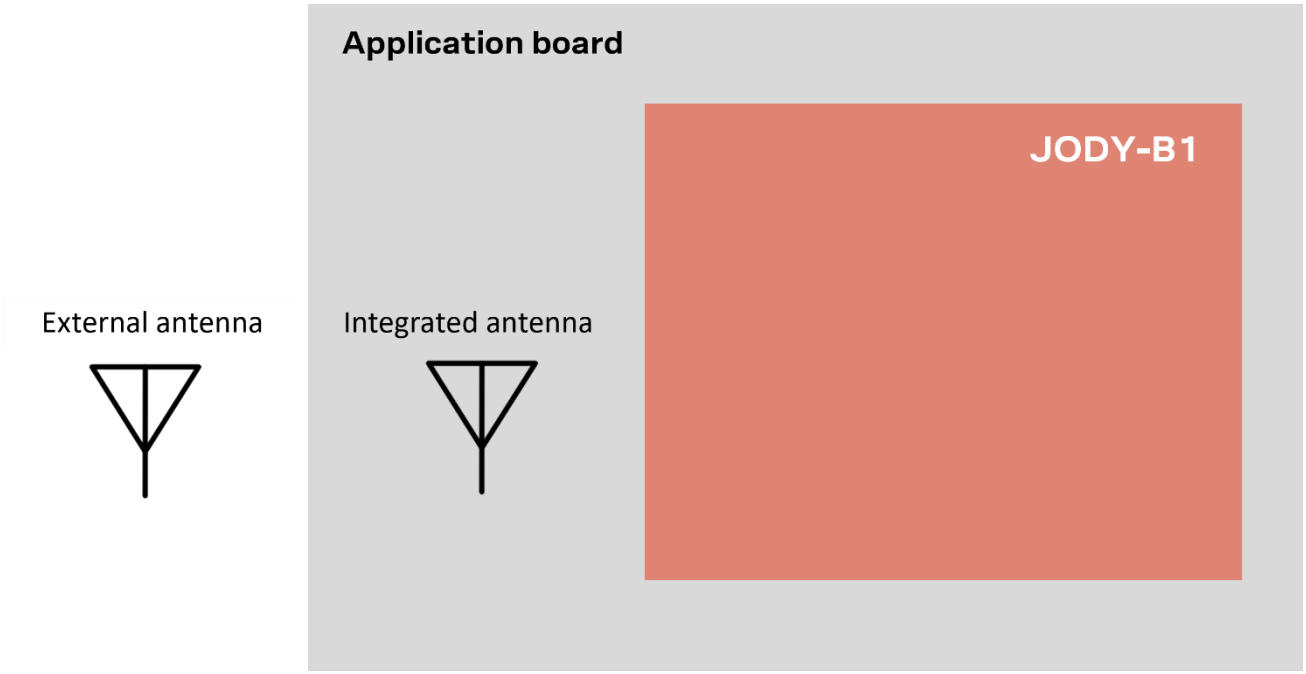


Figure 2: Antenna options

- External antenna: An external antenna of choice connected through a coaxial cable to an U.FL or Reverse Polarity SMA connector placed on the application PCB and connected to the module antenna pin.
- Integrated antenna: A permanent antenna included into the PCB application design. Ideally an SMD antenna mounted on the application PCB or a Flexible PCB antenna attached to the application product's housing.

2.2.1 RF pins and connectors

An RF pin is used to connect the Bluetooth antenna. Table 2 describes the function of the RF pins on JODY-B1.

Pin	Name	I/O	Description	Remarks
24	ANT0	I/O, RF	Bluetooth	50 Ω

Table 2: RF pin allocation

⚠ For proper implementation of antennas in the application product, follow the RF interface options.

⚠ To implement a design compliant with the u-blox FCC certification Grant follow the instructions in the Antenna Integration application note [10].

2.2.2 Approved antenna designs

JODY-B1 modules come with a pre-certified antenna design that can save time and expense during the certification process. To leverage this benefit, customers are required to implement an antenna layout that is fully compliant with the u-blox reference design outlined in future versions of this document. Reference design source files are available from u-blox on request.[1]

For Bluetooth operation, JODY-B1 modules have been tested and approved for use with the antennas featured in the list of Pre-approved antennas .

To implement a design compliant with the u-blox FCC certification Grant follow the instructions in the Antenna Integration application note [10].

2.2.3 Integrated antennas

JODY-B1 modules allow an SMD antenna to be mounted on the application board, connected with a transmission line to the ANT0 pin.

For proper implementation of antennas in the application product, follow the RF interface options.

2.2.4 External antennas

External antennas can be used with JODY-B1 modules. The antennas are preferably connected to the module through coaxial cable and a U.FL or RPSMA connector.

External antennas are particularly suited for application products housed in metal casings that demand that the antennas are placed externally.

For proper implementation of antennas in the application product, follow the RF interface options.

⚠ To avoid invalidating the compliance and pre-certification of u-blox modules with the various regulatory bodies, use only external antennas included the list of Pre-approved antennas. u-blox modules may also be integrated with other antennas. In which case, OEM installers must certify their own designs with the respective regulatory agencies.

[1] Reference designs are only available after certification

2.3 System function interfaces

2.3.1 Power-up sequence

Figure 3 shows the recommended power-up sequence for JODY-B1. Startup is initiated by simultaneously applying the **1V8** and **VIO** power supplies. After both supplies are stable, assert **BT_EN** high after a minimum delay of **20 µs**.

Alternatively, if the power supplies are applied individually:

- Wait at least **20 µs** after **1V8** is applied.
- Wait at least **10 µs** after **VIO** is applied.
- Then, set **BT_EN** high to complete the startup sequence.

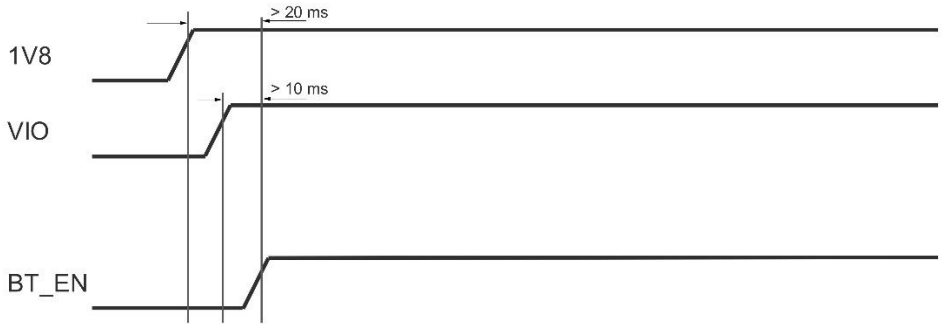


Figure 3: Power-up sequence of JODY-B1 module

BT_EN shall be held low during start up and set high when the power is stable, or later when the module must be turned on. **BT_EN** is powered by the **VIO** voltage domain.

Pin	Name	I/O	Description	Remarks
3	VIO	PWR	VIO supply (1.8 V)	
4	1V8	PWR	1.8 V analog power supply	
58	BT_EN	I	Bluetooth enable signal (active high) or reset, external weak pull down may be required	Assert low for power down or reset.

Table 3: Signals included in start-up sequence

☞ Power down mode can only be entered through **BT_EN** de-assertion by the host. For correct reset, **BT_EN** must be set low for a minimum of 100 ms.

☞ All supply voltages must be monotonic.

2.3.2 Reset

Although external reset is not a prerequisite for correct operation, it can be asserted by the host controller through **BT_EN** in the event of any abnormal module behavior.

JODY-B1 series modules are reset to a default operating state by any of the following events:

- Internal Power-On Reset is triggered when **1V8** and internal VCORE generated by an on-chip DC/DC converter supplied from **1V8** is good (power good 90%).
- **BT_EN** de-assert: The device is reset when the **BT_EN** input pin is low.

☞ A firmware download to the module is required after each reset.

2.3.3 Power-off sequence

JODY-B1 modules enter **Power Down** mode when **BT_EN** is set low. After de-assertion, **1V8** and **VIO** supplies can be disabled. The module then enters **Power Off** mode.

2.3.4 Wake-up signals

JODY-B1 series System Integration Manual modules provide wake-up input and output signals that handle the low-power modes. See also Power management.

The wake-up signals are used to exit JODY-B1 or host CPU from sleep modes. Wake-up signals are powered by the **VIO** voltage domain. These signals are optional, out-of-band, wake-up pins that shall be NC if not used.

Table 4 describes the various wake-up, input and output signals.

Pin	Name	I/O	Description
11	BT_DEV_WAKE	I	Host to Bluetooth Module wake-up signal (input)
12	BT_HOST _WAKE	O	Bluetooth Module to Host wake-up signal (output)

Table 4: Wake-up signal definitions

2.3.5 Power states

JODY-B1 series System Integration Manual modules have several operation states. The power states and general guidelines for Bluetooth operations are defined in Table 5.

General status	Power state	Description
Power-down	Not Powered	**1V8**, and **VIO** supplies not present or below the operating range. The module is switched off.
	Power Down	Asserting **BT_EN** while **1V8**, and **VIO** supplies are present powers down the module. This represents the lowest leakage mode of operation with active voltage rails. Register and memory states are not maintained in power-down mode. The module is automatically reset after exiting power-down mode, which means that the firmware must be downloaded again. If firmware is not downloaded, the device must be kept in its power-down state to reduce the leakage.
Normal operation	Active	Enables TX/RX data connection with the system running at the specified power consumption.
	Deep sleep	Low-power state used in the sleep state of many power-save modes. Memory is placed in low-power retention mode.

Table 5: Description of module power states

2.4 Host interfaces

2.4.1 High-speed UART interface

JODY-B1 series System Integration Manual modules use high-speed UART to connect to the host CPU for firmware download and Bluetooth communication. The UART interface supports the HCI UART transport layer as defined in the Bluetooth Core Specification, Version 5.0, Volume 4, Part A. In addition to communication with the host, the HCI UART interface also supports Bluetooth software (in-band) sleep control.

The main features of the UART interface include:

- Two pins for transmit and receive operations
- Two flow control pins (**RTS**/**CTS**)
- Supports standard baud rates and high throughput up to 3.2 Mbps. The default baud rate after reset is 115.2 kbps.

- Host baud rate tolerance is ±3%.

Table 6 describes the function of each of the UART signals.

Pin	Name	I/O	Description	Remarks
36	BT_UART_TX	O	UART serial output signal	Connect to Host RX
37	BT_UART_RX	I	UART serial input signal	Connect to Host TX
38	BT_UART_RTS	O	UART request-to-send output signal, active low	Connect to Host CTS
39	BT_UART_CTS	I	UART clear-to-send input signal, active low	Connect to Host RTS

Table 6: UART signal description

High-Speed UART signals are powered by the **VIO** voltage domain.

2.4.2 SPI interface

The JODY-B1 supports a Serial Peripheral Interface (SPI). Table 7 describes the module pins for the SPI interface.

Pin	Pin name	I/O	Description
61	BT_SPI_CLK	I	SPI clock input signal
64	BT_SPI_SS	I	SPI chip select signal
62	BT_SPI_MOSI	I	SPI data input signal
63	BT_SPI_MISO	O	SPI data output signal

Table 7: SPI interface description

2.4.3 PCM/I2S audio interface

JODY-B1 series System Integration Manual modules include a bi-directional, 4-wire, PCM digital audio interface to connect an external digital audio device like an audio codec.

The PCM interface is configured to operate in central or peripheral mode. In each case, the **PCM_IN** pin is the data receive terminal (input), and the **PCM_OUT** pin is the data transmit terminal (output). The **PCM_CLK** and **PCM_SYNC** pins function as inputs or outputs, depending on whether the PCM interface is configured as central or peripheral:

- When the JODY-B1 PCM interface is the central: **PCM_CLK** and **PCM_SYNC** are outputs from the device to the PCM bus peripheral(s).
- When the JODY-B1 PCM interface is the peripheral: **PCM_CLK** and **PCM_SYNC** are inputs to the device from the PCM bus central.

☞ PCM pins are shared with the I2S interface.

Table 8 describes the function of each of the PCM digital audio signals.

Pin	Name	I/O	Description	Remarks
16	PCM_CLK	I/O	PCM clock signal. Alternate function: I2S clock	Output if central, input if peripheral
15	PCM_SYNC	I/O	PCM frame sync signal. Alternate function: I2S word select	Output if central, input if peripheral
18	PCM_IN	I	PCM data input signal. Alternate function: I2S data in	
17	PCM_OUT	O	PCM data output signal. Alternate function: I2S data out	

Table 8: PCM digital audio signal descriptions

PCM/I2S signals are powered by the **VIO** voltage domain.

2.5 External coexistence interface

The included PTA interface enables coexistence with an external radio. Table 9 describes the function of each of the external coexistence signals.

Pin	Pin name	I/O	Description
90	PTA_PRI	I	PTA external radio priority signal
89	PTA_REQ	I	PTA request from the external radio
88	PTA_GNT	O	PTA external radio grant signal

Table 9: External coexistence interface description

2.6 JTAG

The module includes a JTAG test interface as shown in Table 10.

Pin	Pin name	I/O	Description
32	JTAG_TCK	I	JTAG test clock input signal
33	JTAG_TMS	I	JTAG controller select input signal
85	JTAG_TDO	O	JTAG test data output signal
86	JTAG_TDI	I	JTAG test data input signal

Table 10: JTAG interface description

2.7 Other remarks

2.7.1 Unused pins

JODY-B1 series modules have unconnected (NC) pins that are reserved for future use. These pins must be left unconnected on the application board.

3 Design-in

Follow the design guidelines in this chapter to optimize the integration of JODY-B1 series System Integration Manual modules in the final application board.

3.1 Overview

Although all application circuits must be properly designed, and the following aspects of the application design require special attention:

- Module antenna integration. **ANT**.
  - Antennas and RF circuits affect the RF performance and certification compliance. It is important to follow the design instructions given here to reach specified performance. Further, to maintain compliance and subsequent certification of the application design, it is important to observe the applicable parts of antenna schematic and layout described in Antenna interfaces.
- Module power supply. **Power** and **GND**.
  - Power supply circuits might affect the RF performance. It is important to select a suitable device capable to source the adequate voltage and current. It is also important to implement adequate power and ground planes in PCB stack-up and to implement bypass capacitors for these supplies. See also Supply interfaces.
- High-speed interfaces, such as **PCIe, SDIO**, high-speed **UART**, and **PCM**.
  - High-speed interfaces are a potential source of noise that can affect the regulatory compliance of standards for radiated emissions. It is important to follow the schematic and layout design recommendations described in General high-speed layout guidelines.
- System functions: **Power Down, Reset** and Configuration.
  - Careful utilization of these pins in the application design is necessary to ensure that the module operates as it should. Specifically, observe the state and voltage level is correctly defined during module boot and operation. It is important to follow the pin design described in the General high-speed layout guidelines.
- Other pins: specific signals.
  - Careful utilization of these pins is necessary to ensure that the module operates as it should. It is important to follow the schematic and design layout recommendations.
- **NC** pins must not be connected.

3.2 RF interface options

Short range modules provide several RF-interface options for connecting external antennas. Module variants also provides internal or embedded antennas when using such module there are special considerations to account when designing the host PCB:

- The **ANT** ports have a nominal characteristic impedance of 50 Ω. For correct impedance matching, these ports must be connected to the respective antenna through a 50 Ω U.FL connector and coax or a transmission line – depending on the type of module connector. Poor termination of **ANT** pins will result in degraded performance of the module.

⚠ According to FCC regulations, the transmission line from the module antenna pin to the physical antenna (or antenna connector on the host PCB) is considered part of the approved antenna design. Therefore, module integrators must use exactly the antenna reference design used in the module FCC type approval or certify their own design.

For instructions on how to design circuits that comply with these requirements, see also Antenna interfaces.

3.2.1 Antenna design

To optimize the radiated performance of the final product, the selection and placement of both the module and antenna must be chosen with due regard to the mechanical structure and electrical design of the product. To avoid later redesigns, it is important to decide the positioning of these components at an early phase of the product design.

The compliance and subsequent certification of the RF design depends heavily on the radiating performance of the antennas.

To ensure that the u-blox RF certification of modules is extended through to the application design, carefully follow the guidelines outlined below.

- External antennas, including, linear monopole classes:
  - Place the module and antenna in any convenient area on the board. External antennas do not impose any restriction on where the module is placed on the PCB.
  - Select antennas with an optimal radiating performance in the operating bands. The radiation performance depends mainly on the antennas.
  - Choose RF cables that offer minimum insertion loss. Unnecessary insertion loss is introduced by low quality or long cables. Large insertion losses reduce radiation performance.
  - Use a high-quality 50 Ω coaxial connector for proper PCB-to-RF-cable transition.
- Integrated antennas, such as patch-like antennas:
  - Internal integrated antennas impose some physical restrictions on the PCB design:
    - Integrated antennas excite RF currents on its counterpoise, typically the PCB ground plane of the device that becomes part of the antenna; its dimension defines the minimum frequency that can be radiated. Therefore, the ground plane can be reduced to a minimum size that should be similar to the quarter of the wavelength of the minimum frequency that has to be radiated, given that the orientation of the ground plane related to the antenna element must be considered.
    - Find a numerical example to estimate the physical restrictions on a PCB, where: Frequency = 2.4 GHz → Wavelength = 12.5 cm → Quarter wavelength = 3.5 cm in free space or 1.5 cm on a FR4 substrate PCB.
- Choose antennas with optimal radiating performance in the operating bands. Radiation performance depends on the complete product and antenna system design, including the mechanical design and usage of the product. Table 11 summarizes the requirements for the antenna RF interface.
- Make the RF isolation between the system antennas as high as possible, and the correlation between the 3D radiation patterns of the two antennas as low as possible. In general, RF separation of at least a quarter wavelength between the two antennas is required to achieve a minimum isolation and low pattern correlation. If possible, increase the separation to maximize the performance and fulfill the requirements described in Table 11.

Item	Requirements	Remarks
Impedance	50 Ω nominal characteristic impedance	The impedance of the antenna RF connection must match the 50 Ω impedance of Antenna pins.
Frequency Range	2400 - 2500 MHz	Bluetooth.
Return Loss	S11 < -10 dB (VSWR < 2:1) recommended S11 < -6 dB (VSWR < 3:1) acceptable	The Return loss or the S11, as the VSWR, refers to the amount of reflected power, measuring how well the primary antenna RF connection matches the 50 Ω characteristic impedance of antenna pins. The impedance of the antenna termination must match as much as possible the 50 Ω nominal impedance of antenna pins over the operating frequency range, to maximize the amount of power transferred to the antenna.
Efficiency	> -1.5 dB (> 70%) recommended > -3.0 dB (> 50%) acceptable	Radiation efficiency is the ratio of the radiated power to the power fed to the antenna input: the efficiency is a measure of how well an antenna receives or transmits.
Maximum Gain		To comply with regulatory agencies radiation exposure limits the maximum antenna gain must not exceed the value specified in type approval documentation.

Table 11: Summary of antenna interface requirements

⚠ When operating dual antennas in the same 2.4 GHz band, sufficient isolation is critical for attaining an optimal throughput performance in Wi-Fi/Bluetooth coexistence mode.

Select antennas that provide:

- Optimal return loss (or VSWR) over all the operating frequencies.
- Optimal efficiency figure over all the operating frequencies.
- An appropriate gain that does not exceed the regulatory limits specified in some regulatory country authorities like the FCC in the United States.

A useful approach for the antenna micro-strip design is to place an U.FL connector close to the embedded PCB or chip antenna. The U.FL connector only needs to be mounted on units used for verification.

3.2.1.1 Integrated antenna design

If integrated antennas are used, the transmission line is terminated by the antennas themselves or by the antenna together with the connected coaxial cable and U.FL plug.

Consider the following the guidelines when designing the antenna:

- The antenna design process should commence at the same time as the mechanical design of the product. PCB mock-ups are useful in estimating overall efficiency and radiation path of the intended design during early development stages.
- Use antennas designed by an antenna manufacturer that provide the best possible return loss (or VSWR).
- Provide a ground plane large enough according to the related integrated antenna requirements. The ground plane of the application PCB may be reduced to a minimum size that must be similar to one quarter of wavelength of the minimum frequency that has to be radiated. The overall antenna efficiency may benefit from larger ground planes.
- Proper placement of the antenna and its surroundings is also critical for antenna performance. Avoid placing the antenna close to conductive or RF-absorbing parts, such as metal objects or ferrite sheets, as these may absorb part of the radiated power, shift the resonant antenna frequency of the antenna, or otherwise affect the antenna radiation pattern.
- Ensure that correct the installation and deployment of the antenna system, including PCB layout and matching circuitry, is done correctly. In this regard, it is recommended that you strictly follow the specific guidelines provided by the antenna manufacturer.

- Further to the custom PCB and product restrictions, antennas may also require tuning/matching to reach the target performance. It is recommended that you plan measurement and validation activities with the antenna manufacturer before releasing the end-product to manufacturing.
- The receiver section may be affected by noise sources like hi-speed digital busses. Avoid placing the antenna close to busses as DDR. Otherwise, consider taking specific countermeasures, like metal shields or ferrite sheets, to reduce the interference.
- Be aware of interaction between co-located RF systems, like LTE sidebands on 2.4 GHz band. Transmitted power may interact or disturb the performance of the module where specific LTE filter is not present.

3.2.1.2 RF transmission line design

RF transmission lines, such as those that connect from **ANT** pins to their related antenna connectors or antenna, must be designed with a characteristic impedance of 50 Ω.

Figure 4 shows the design options and the most important parameters for designing a transmission line on a PCB:

- Microstrip: track separated with dielectric material and coupled to a single ground plane.
- Coplanar microstrip: track separated with dielectric material and coupled to both the ground plane and side conductor.
- Stripline: track separated by dielectric material and sandwiched between two parallel ground planes.

The most common transmission line implementation is the coplanar microstrip, as shown in Figure 4.

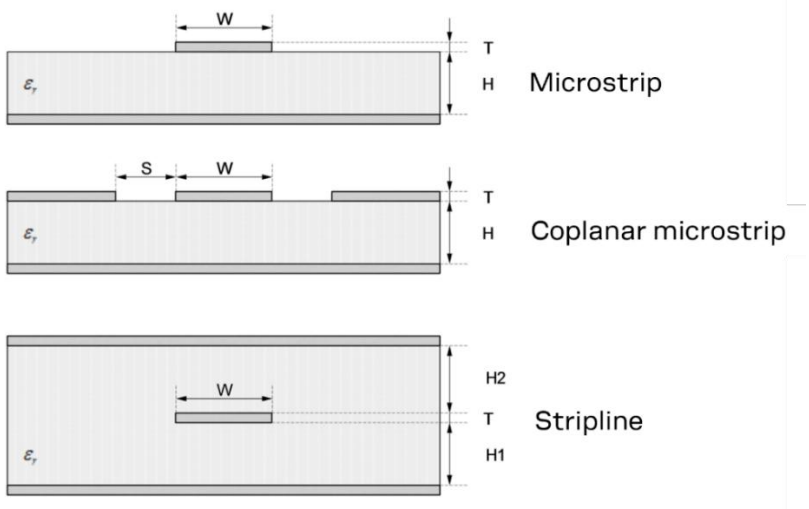


Figure 4: Transmission line trace design

Follow these recommendations to design a 50 Ω transmission line correctly:

- Designers must provide enough clearance from surrounding traces and ground in the same layer. In general, the trace to ground clearance should be at least twice that of the trace width. The transmission line should also be “guarded” by the ground plane area on each side.
- In the first iteration, calculate the characteristic impedance using tools provided by the layout software. Ask the PCB manufacturer to provide the final values usually calculated using dedicated software and production stack-ups. It is sometimes possible to request an impedance test coupon on side of the panel to measure the real impedance of the traces.
- Although FR-4 dielectric material can result in high losses at high frequencies, it can still be an appropriate choice for RF designs. In which case, aim to:
  - Minimize RF trace lengths to reduce dielectric losses.

    - If traces longer than few centimeters are needed, use a coaxial connector and cable to reduce losses.
    - For good impedance control over the PCB manufacturing process, design the stack-up with wide 50 Ω traces with width of at least 200 µm.
    - Contact the PCB manufacturer for specific tolerance of controlled impedance traces. As FR-4 material exhibits poor thickness stability it gives less control of impedance over the trace width.
- For PCBs with components larger than 0402 and dielectric thickness below 200 µm, add a keep-out, that is, some clearance (void area) on the ground reference layer below any pin on the RF transmission lines. This helps to reduce the parasitic capacitance to ground.
- Route RF lines in 45 ° angle and avoid acute angles. The transmission lines width and spacing to GND must be uniform and routed as smoothly as possible.
- Add GND stitching vias around transmission lines.
- Provide a sufficient number of vias on the adjacent metal layer. Include a solid metal connection between the adjacent metal layer on the PCB stack-up to the main ground layer.
- To avoid crosstalk between RF traces and Hi-impedance or analog signals, route RF transmission lines as far from noise sources (like switching supplies and digital lines) and any other sensitive circuit.
- Avoid stubs on the transmission lines. Any component on the transmission line should be placed with the connected pin located over the trace. Also avoid any unnecessary components on RF traces.

Figure 5 shows a trace and ground design example. The top layer, to the left, shows the transmission line design connecting the RF pads to U.FL connectors. The purple rectangles are the top layer solder pads.

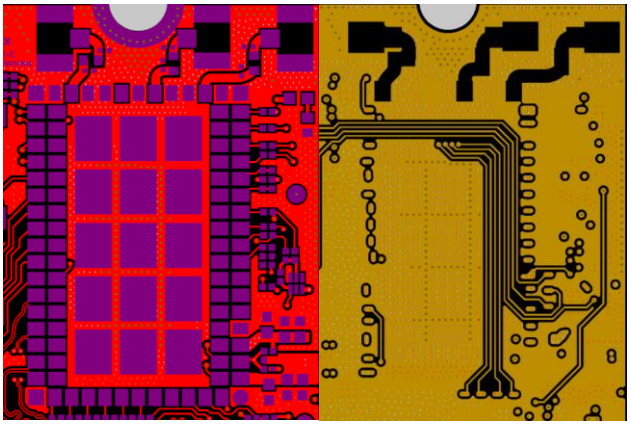


Figure 5: RF trace and ground design example showing top layer and inner layer 1

Figure 5 shows typical artwork implementing a coplanar microstrip on a PCB. The trace includes coplanar microstrip, impedance matching PI network, and U.FL. RF connectors. The ground clearance on inner layers allow for a wider microstrip, which is less lossy than a narrow one. A wider trace also has less impedance variation over PCB production batches due to the absolute tolerances in the PCB etching process.

Figure 6 shows layout of pins for U.FL connector. Especially consider the GND clearance under the signal pad.

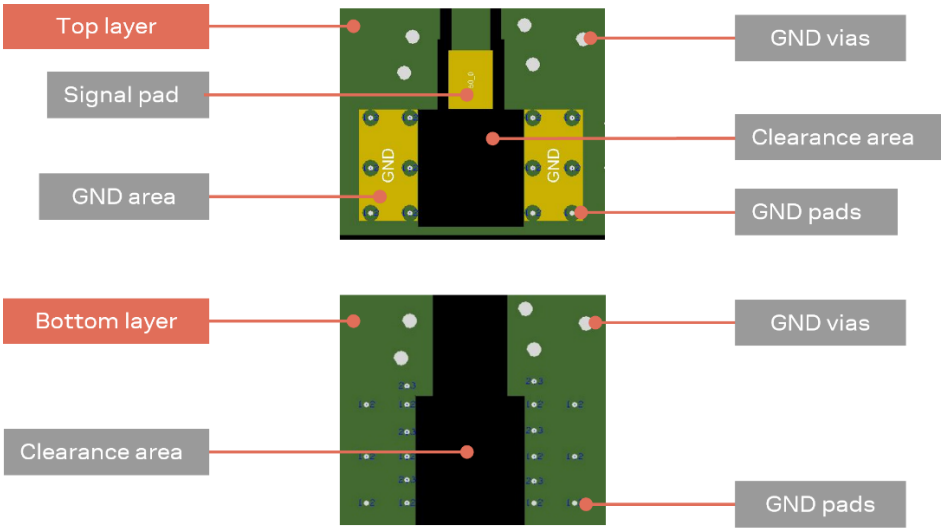


Figure 6: U.FL connector layout, top layer (top), and inner layer 1 (bottom)

3.3 Supply interfaces

3.3.1 Module supply design

Although the GND pins are internally connected, it is advisable to connect all available ground pins on the application board to solid ground with a good (low impedance) connection to external ground. This minimizes power loss, improves RF performance, and betters thermal performance.

Good connection of the module supply pins, supplied by a DC supply source, is required for accurate RF performance.

Consider the following guidelines when developing the schematic:

- All power supply pins must be connected to an appropriate DC source.
- Any series component with an Equivalent Series Resistance (ESR) greater than a few mΩ should be avoided. The only exception to this general rule is the use of ferrite beads for DC filtering. To avoid possible instability in the DC supply, only use ferrite beads if needed.
- For high-frequency filtering, additional bypass capacitors in the range of 100 nF to 1 µF are required on all supply pins. Offering low ESR/ESL resistance, a class II ceramic capacitor with an X7R or X5R dielectric is well suited for this purpose. For +105 °C applications X6S dielectric capacitors are recommended. Bypass capacitors of a smaller size can be chosen to minimize ESL (Equivalent Series Inductance) in the manufacturing process. The capacitor should be placed as close as possible to the module supply pin.
- In case of noise and immunity issues it is recommended to implement serial beads on especially RF supply traces.
- To help filter current spikes from the RF section and avoid ground bounce, a minimum bulk capacitance of 10 µF should be applied to the **1V8** rail (optionally on **VIO**) and placed close to the module supply pins. Offering low ESR/ESL resistance, a class II ceramic capacitor with an X7R or X5R dielectric is well suited for this purpose. Special care should be taken in the selection of X5R/X7R dielectrics due to capacitance derating versus DC bias voltage.

3.3.1.1 Guidelines for supply circuit design using an SMPS

When choosing between an SMPS or LDO to supply the modules, it is advisable to consider the acceptable power and thermal dissipation of the application product.

A Switched Mode Power Supply (SMPS) is generally recommended for converting the main supply to the module supply when the voltage difference is relatively high. In these circumstances, the use of an SMPS dissipate less power and subsequently generates less power dissipation and heat than an LDO. See also Regulated DC power supply.

By contrast, an LDO is generally simpler to use and does not generate the amount of noise an SMPS might. See also Guidelines for supply circuit design using a Low Drop-Out (LDO) linear regulator.

To avoid degrading the module stability, RF performance, or violating spurious emission standards, the characteristics of the SMPS should meet the following prerequisites:

- **Power capability:** The regulator, together with any additional filter in front of the module, must be capable of providing a voltage within the specified operating range. It must also be capable of delivering the specified peak current.
- **Low output ripple**: The peak-to-peak ripple voltage of the switching regulator must not exceed the specified limits. This requirement is appliable to both the voltage ripple generated by the SMPS at operating frequency and the high-frequency noise generated by power switching.
- **PWM/PFM mode operation**: It is advisable to select regulators that support a fixed Pulse Width Modulation (PWM) mode. Pulse Frequency Modulation (PFM) mode typically exhibits higher ripple and can affect RF performance. If power consumption is not a primary concern, PFM/PWM mode transitions should be avoided in favor of fixed PWM operation to reduce the peak-to-peak noise on voltage rails. Switching regulators with mixed PWM/PFM mode can be used provided that the PFM/PWM modes and transition between modes complies with the requirements.

3.3.1.2 Guidelines for supply circuit design using a LDO linear regulator

The use of a linear regulator is appropriate when the difference between the available supply rail and the module supply is relatively low. Linear regulators can also be considered for powering 1.8 V domains – particularly those having low current requirements and those cascaded from an SMPS-generated low voltage rail.

The characteristics of the Low Drop-Out (LDO) linear regulator used to power the voltage rails must meet the following prerequisites:

- **Power capabilities**: The LDO linear regulator must be able to provide a voltage within the specified operating range. It must also be capable of withstanding and delivering the maximum specified peak current while in “connected mode”.
- **Power dissipation**: The power handling capability of the LDO linear regulator must be checked to limit its junction temperature to the maximum rated operating range. The worst-case junction temperature can be estimated as shown below:

  $$T_{j,est} = (V_{in} - V_{out}) * I_{avg} * \theta_{ja} + T_a$$

  Where: θ_{ja} is the junction-to-ambient thermal resistance of the LDO package[2], I_{avg} is the current consumption of the given voltage rail in continuous TX/RX mode and T_a is the maximum operating temperature of the end product inside the housing.

[2] Thermal dissipation capability reported on datasheets is usually tested on a reference board with adequate copper area (see also JESD51 [9]). Junction temperature on a typical PCB can be higher than the estimated value due to the limited space to dissipate the heat. Thermal reliefs on pads also affect the capability of a device to dissipate heat.

3.4 Data communication interfaces

3.4.1 High-speed UART interface

The high-speed UART interface for the JODY-B1 complies with the HCI UART Transport layer. The module uses the settings shown in Table 12.

UART Settings	
Baud rate default after reset	115200 baud
Data bits	8
Parity bit	No parity
Stop bit	1 stop bit
Flow Control	RTS/CTS

Table 12: HCI UART transport layer settings

RTS/CTS flow control is used to prevent temporary UART buffer overrun.

- If RTS is 0 (output, active low), the module is ready to receive, and the host is allowed to send.
- If CTS is 0 (input, active low), the host is ready to receive, and the module is allowed to send.

☞ The use of hardware flow control with RTS/CTS is mandatory.

Baud rate			
9600	125000	720000	3000000
19200	230400	921600	3200000
38400	250000	1000000	
57600	460800	1600000	
115200	500000	2000000	

Table 13: Possible baud rates for the UART interface

The baud rate accuracy of the UART interface is +1.5/-2.5% and the host baud rate tolerance is ±3%.

3.5 Other interfaces and notes

Most pins have internal keeper resistors; then leave un-used pins open. Otherwise, it is good practice to follow below recommendation.

- Digital output signals might be left unconnected.
- Input signals without internal keeper resistors might be tied to GND or supply voltage through a pull resistor. The same applies for analog inputs.

For JODY-B1 inputs without keepers applies to the I2S and SPI interfaces.

3.6 General high-speed layout guidelines

These guidelines describe best practices for the layout of all high-speed interfaces. Designers should prioritize the layout of higher speed busses. Low frequency signals, other than those with high-impedance traces, are generally not critical to the layout.

⚠ Low frequency signals with high-impedance traces (such as signals driven by weak pull resistors) may be affected by crosstalk. For these high impedance traces, a supplementary isolation of 4*W from other busses is recommended.

3.6.1 General considerations for schematic design and PCB floor planning

- Verify which signal bus requires termination and add appropriate series resistor terminations to the schematics.
- Carefully consider the placement of the module with respect to the antenna position and host processor; minimize RF trace length first and then the communication interface length.
- SDIO bus routing must aim to keep layer-to-layer transition to a minimum.
- Verify the allowable stack-ups, and the controlled impedance dimensioning for antenna traces and busses, with the PCB manufacturer.
- Verify that the power supply design and power sequence are compliant with the specifications described in System function interfaces.

3.6.2 Component placement

- Accessory parts like bypass capacitors must be placed as close as possible to the module to improve filtering capability. Prioritize placing the smallest capacitors close to module pins.
- Do not place components close to the antenna area. Follow the recommendations of the antenna manufacturer to determine distance of the antenna in relation to other parts of the system. Designers should also maximize the distance of the antenna to High-frequency busses, like DDRs and related components. Alternatively, consider an optional metal shield to reduce interferences that might otherwise be picked up by the antenna and subsequently reduce module sensitivity.

3.6.3 Layout and manufacturing

- Avoid stubs on high-speed signals. Test points or component pads should be placed over the PCB trace.
- Verify the recommended maximum signal skew for differential pairs and length matching of buses.
- Minimize the routing length; longer traces degrade signal performance. Ensure that maximum allowable length for high-speed busses is not exceeded.
- Ensure to track your impedance matched traces. Consult early with your PCB manufacturer for proper stack-up definition.
- RF, analog, and digital sections should have dedicated and clearly separated areas on the board.
- No digital routing is allowed in the GND reference plane area of RF traces (ANT pins and Antenna).
- Designers are strongly recommended to avoid digital routing beneath all layers of RF traces.
- Ground cuts or separation are not allowed below the module.
- As a first priority, minimize the length of the RF traces. Then, minimize bus length to reduce potential EMI issues related to the radiation of digital busses.
- All traces (Including low speed or DC traces) must couple with a reference plane (GND or power). High-speed busses should be referenced to the ground plane. If designers need to change the ground reference, an adequate number of GND vias must be added in the area of transition. This facilitates a low-impedance path between the two GND layers for the return current.
- Hi-speed busses are not allowed to change reference plane. If a change to the reference plane is unavoidable, some capacitors should be added in the area to provide a low impedance return path through the various reference planes.
- Trace routing should maintain a distance that is greater than 3*W from the edge of the ground plane routing.
- Power planes should maintain a safe distance from the edge of the PCB. The distance must be sufficient to route a ground ring around the PCB, and the ground ring must then be stitched to other layers through vias.
- Route the power supply in low impedance power planes. If you choose to route the power supply with traces, do not route loop structures.

⚠ The heat dissipation during continuous transmission at maximum power can significantly raise the temperature of application baseboards under modules. Avoid placing temperature sensitive devices close to the module and provide these devices with sufficient grounding to transfer generated heat to the PCB.

3.7 Module footprint and paste mask

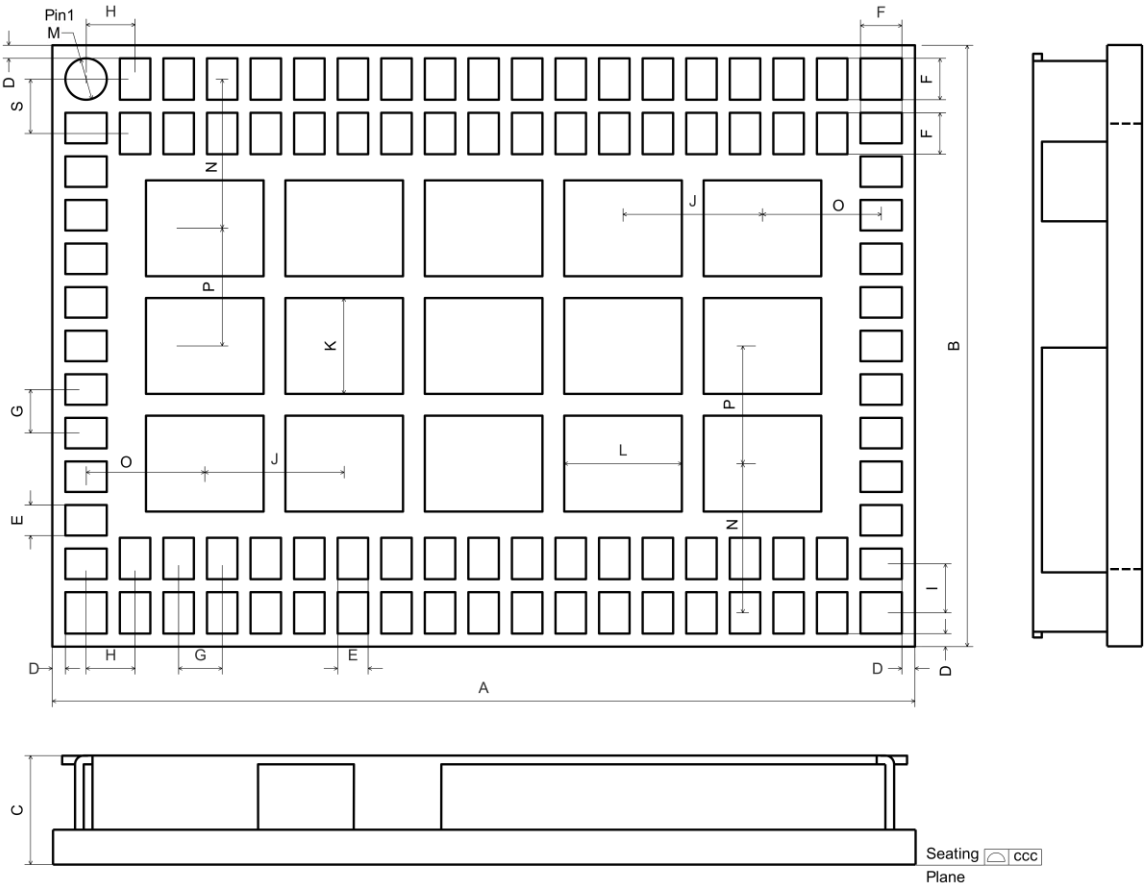


Figure 7: Recommended footprint for JODY-B1, bottom view

Figure 7 shows the pin layout of JODY-B1 series System Integration Manual modules. Dimensions are found in [1]. The proposed land pattern layout complements the pin layout of the module. Both Solder Mask Defined (SMD) and Non Solder Mask Defined (NSMD) pins can be used with adherence to the following considerations:

- All pins should be Non-Solder Mask Defined (NSMD)
- To help with the dissipation of the heat generated by the module, GND pads must have good thermal bonding to PCB ground planes.

The suggested stencil layout for the JODY-B1 module is to follow the copper pad layout exactly as described in Figure 7 for the outer pads, while the central pads should implement a special solder paste pattern with the following characteristics:

- Solder paste area should be split in several smaller parts, typically four to nine depending on copper pad area.
- Total solder paste area should cover about 50% to 60% of copper thermal pad area.
- Total solder paste area must not exceed 65% of copper thermal pad area.

Missing to consider solder paste optimization can lead to poor soldering quality in production.

A suggested stencil opening implementation is shown in Figure 8.

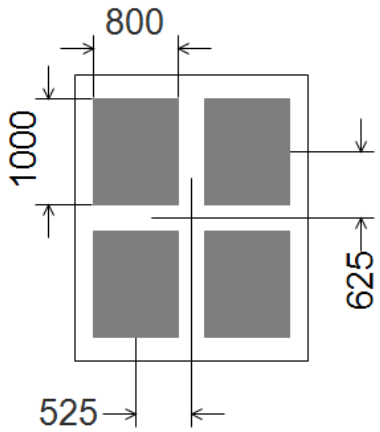


Figure 8: Stencil opening example for inner thermal pads (dimensions in µm)

⚠ The exact mask geometries, distances and stencil thicknesses must be adapted to the specific production process of the customer.

3.8 Thermal guidelines

JODY-B1 series System Integration Manual modules are designed to operate from -40 °C to +105 °C at an ambient temperature inside the enclosure box. The board generates heat during high loads that must be dissipated to sustain the lifetime of the components.

The improvement of thermal dissipation in the module decreases its internal temperature and consequently increases the long-term reliability of device applications operating at high ambient temperatures.

For best performance, layouts should adhere to the following guidelines:

- Vias specification for ground filling: $300/600\mu m$, with no thermal reliefs allowed on vias.
- Ground via densities under the module: $50\ vias/cm^2$; thermal vias can be placed in gaps between the thermal pads of the module.
- Minimum layer count and copper thickness: $4\ layers, 35\ \mu m$.
- Minimum board size: $55x70\ mm$.
- To optimize the heat flow from the module, power planes and signal traces should not cross the layers beneath the module.

These recommendations facilitate a design that is capable of achieving a thermal characterization parameter of $\psi_{JB} = TBD\ °C/W$ for -40 °C to +105 °C where, JB refers to the junction between the module and the bottom side of the main PCB characterization parameter.

Use the following hardware techniques to further improve thermal dissipation in the module and optimize its performance in customer applications:

- Maximize the return loss of the antenna to reduce reflected RF power to the module.
- Improve the efficiency of any component that generates heat, including power supplies and processor, by dissipating it evenly throughout the application device.
- Provide sufficient ventilation in the mechanical enclosure of the application.
- For continuous operation at high temperatures, particularly in high-power density applications or smaller PCB sizes, include a heat sink on the bottom side of the main PCB. The heat sink is best connected using electrically insulated / high thermal conductivity adhesive[3].

[3] Typically not required.

3.9 ESD guidelines

In compliance with the following European regulations, designers must implement proper protection measures against ESD events on any pin exposed to end users:

- ESD testing standard CENELEC EN 61000-4-2 [3]
- Radio equipment standard ETSI EN 301 489-1 [4]

The minimum requirements as per these European regulations are summarized in Table 14.

Application	Category	Immunity level
All exposed surfaces of the radio equipment and any ancillary equipment in the end product.	Contact discharge	4 kV
	Air discharge	8 kV

Table 14: Minimum ESD immunity requirements based on EN 61000-4-2

Compliance with the protection levels specified in EN 61000-4-2 [3] are fulfilled by including proper ESD protection in parallel to any susceptible trace that is close to areas accessible to end users.

⚠ Special care should be taken with the **ANT** pins that must be protected by choosing an ESD absorber or TVS diode with adequate parasitic capacitance. For 5 to 6 GHz operation, protection with maximum internal capacitance of 0.1 pF is advised.

3.10 Design-in checklists

3.10.1 Schematic checklist

☐ Check that the module pins have been properly numbered and designated in the schematic (including thermal pins). See Pin definition in the JODY-B1 data sheet [1].
☐ Power supply design complies with the voltage and power supply requirements described in the module data sheet [1].
☐ The Power-up sequence has been properly implemented.
☐ Adequate bypassing has been included in front of each power pin. See Power-up sequence.
☐ Each signal group is consistent with its own power rail supply or proper signal translation has been provided. See Pin definition in the JODY-B1 data sheet [1].
☐ Configuration pins are properly set at bootstrap. See Configuration pins.
☐ Unused pins are properly terminated. See Unused pins and 3.5.
☐ A pi-filter is provided in front of each antenna for final matching. High-speed UART interface.
☐ Additional RF co-location filters have been considered in the design. See block diagrams in the module data sheet [1].

3.10.2 Layout checklist

☐ PCB stack-up and controlled impedance traces follow the recommendations given by the PCB manufacturer. See RF transmission line design.
☐ All pins are properly connected, and the footprint follows u-blox pin design recommendations. See Module footprint and paste mask.
☐ Proper clearance has been provided between the RF and digital sections of the design. See Layout and manufacturing.
☐ Proper isolation has been provided between antennas (RF co-location, diversity, or multi-antenna design). See Layout and manufacturing.
☐ Bypass capacitors have been placed close to the module. See Component placement.
☐ Low impedance power path has been provided to the module. See Component placement.

☐ Controlled impedance traces have been properly implemented in the layout (both RF and digital) and the recommendations provided by the PCB manufacturer have been followed. See RF transmission line design and Component placement.
☐ 50 Ω RF traces and connectors follow the rules described in Antenna interfaces.
☐ Antenna integration has been reviewed by the antenna manufacturer.
☐ Proper grounding has been provided to the module for the low impedance return path and heat sink. See Layout and manufacturing.
☐ Reference plane skipping has been minimized for high frequency busses. See Layout and manufacturing.
☐ All traces and planes are routed inside the area defined by the main ground plane. See Layout and manufacturing
☐ u-blox has reviewed and approved the PCB[5].

[5] This is applicable only for end-products based on u-blox reference designs.

4 Software

The chapter describes the available software options for JODY-B1 series System Integration Manual modules, which are based on the Qualcomm QCA8695AU chipset. The drivers and firmware required to operate JODY-B1 series System Integration Manual modules are developed by Qualcomm.

☞ For the latest JODY-B1 series System Integration Manual software deliverables, contact your local support team.

5 Handling and soldering

⚠ JODY-B1 series modules are Electrostatic Sensitive Devices that demand the observance of special handling precautions against static damage. Failure to observe these precautions can result in severe damage to the product.

5.1 ESD handling precautions

As the risk of electrostatic discharge in the RF transceivers and patch antennas of the module is of particular concern, standard ESD safety practices are prerequisite. See also Figure 9.

Consider also:

- When connecting test equipment or any other electronics to the module (as a standalone or PCB-mounted device), the first point of contact must always be to local GND.
- Before mounting an antenna patch, connect the device to ground.
- When handling the RF pin, do not touch any charged capacitors. Be especially careful when handling materials like patch antennas (~10 pF), coaxial cables (~50-80 pF/m), soldering irons, or any other materials that can develop charges.
- To prevent electrostatic discharge through the RF input, do not touch any exposed antenna area. If there is any risk of the exposed antenna being touched in an unprotected ESD work area, be sure to implement proper ESD protection measures in the design.
- When soldering RF connectors and patch antennas to the RF pin on the receiver, be sure to use an ESD-safe soldering iron (tip).

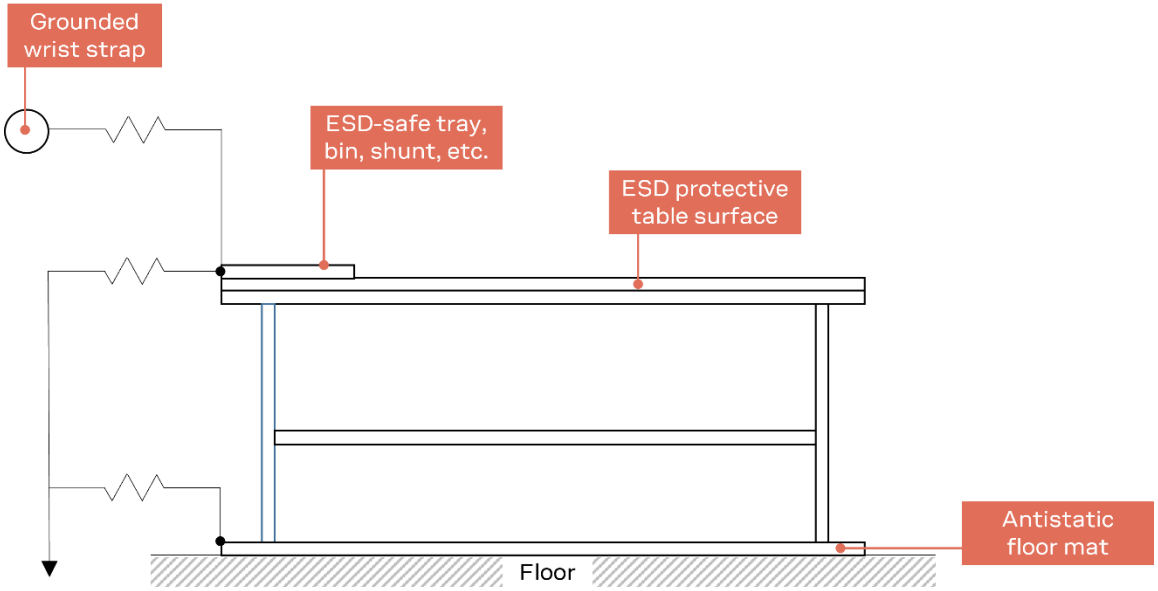


Figure 9: Standard workstation setup for safe handling of ESD-sensitive devices

5.2 Packaging, shipping, storage, and moisture preconditioning

For information pertaining to reels, tapes, or trays, moisture sensitivity levels (MSL), storage, shipment, and drying preconditioning, see the JODY-B1 series data sheet [1] and Packaging information reference guide [2].

5.3 Reflow soldering process

☞ JODY-B1 is approved for two-time reflow processes.

JODY-B1 modules are surface mounted devices supplied on a multi-layer FR4-type PCB with gold-plated connection pads. The modules are produced in a lead-free process using lead-free soldering paste. The thickness of solder resist between the host PCB top side and the bottom side of the JODY-B1 module must be considered for the soldering process.

JODY-B1 modules are compatible with industrial reflow profile for RoHS solders, and “no-clean” soldering paste is strongly recommended.

The reflow profile used is dependent on the thermal mass of the entire populated PCB, the heat transfer efficiency of the oven, and the type of solder paste that is used. The optimal soldering profile must be trimmed for the specific process and PCB layout

⚠ The target values shown in Table 15 and Figure 10 are given as general guidelines for a Pb-free process only. For further information, see also the JEDEC J-STD-020E [6] standard.

Process parameter		Unit	Target
Pre-heat	Ramp up rate to T_{SMIN}	K/s	3
	T_{SMIN}	°C	150
	T_{SMAX}	°C	200
	t_S (from 25°C)	s	150
	t_S (Pre-heat)	s	110
Peak	T_L	°C	217
	t_L (time above T_L)	s	90
	T_P	°C	245-250
	t_P (time above T_P -5°C)	s	30
Cooling	Ramp-down from T_L (max)	K/s	6
General	$T_{to\ peak}$	s	300
	Allowed reflow soldering cycles	-	2

Table 15: Recommended reflow profile

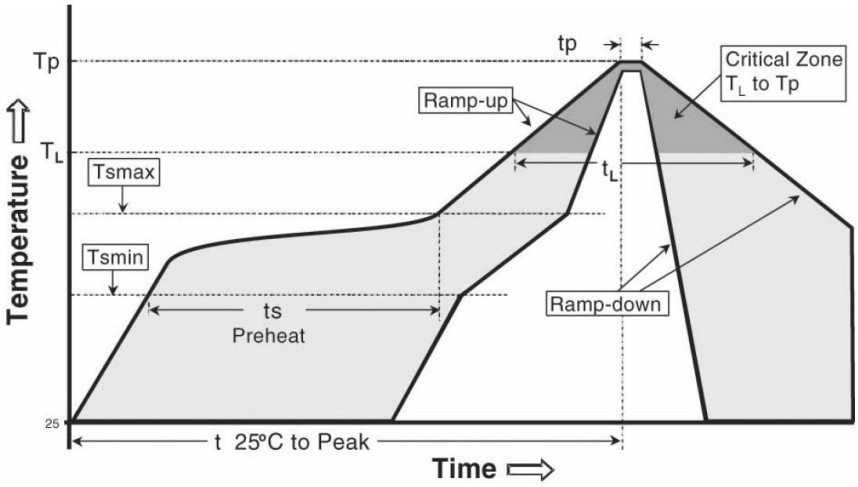


Figure 10: Reflow profile

☞ The lower value of T_P and slower ramp down rate is preferred.

5.3.1 Cleaning

Cleaning the modules is not recommended. Residues underneath the modules cannot be easily removed with a washing process.

- Cleaning with water will lead to capillary effects where water is absorbed in the gap between the baseboard and the module. The combination of residues of soldering flux and encapsulated water leads to short circuits or resistor-like interconnections between neighboring pins. Water will also damage the sticker and the ink-jet printed text.
- Cleaning with alcohol or other organic solvents can result in soldering flux residues flooding into the housing, areas that are not accessible for post-wash inspections. The solvent will also damage the label and the ink-jet printed text.
- Ultrasonic cleaning will permanently damage the module and the crystal oscillators in particular.

For best results use a “no clean” soldering paste and circumvent the need for a cleaning stage after the soldering process.

5.3.2 Other notes

- Boards with combined through-hole technology (THT) components and surface-mount technology (SMT) devices may require wave soldering to solder the THT components. Only a single wave-soldering process is allowed for boards populated with the modules. Miniature Wave Selective Solder processes are preferred over traditional wave soldering processes.
- Hand-soldering is not recommended.
- Rework is not recommended.
- Conformal coating can affect the performance of the module, which means that it is important to prevent the liquid from flowing into the module. The RF shields do not provide protection for the module from coating liquids with low viscosity; therefore, care is required while applying the coating. Conformal coating of the module will void the warranty.
- Grounding metal covers: Attempts to improve grounding by soldering ground cables, wick, or other forms of metal strips directly onto the EMI covers is done so at the customer’s own risk and will void the module warranty. The numerous ground pins on the module are adequate to provide optimal immunity to interferences.
- The modules contain components which are sensitive to Ultrasonic Waves. Use of any Ultrasonic Processes (cleaning, welding, etc.) may damage the module. The use of ultrasonic processes together with the module will void the warranty.

6 Regulatory compliance

6.1 General requirements

JODY-B1 series System Integration Manual modules are designed to comply with the regulatory demands of Federal Communications Commission (FCC), Innovation, Science and Economic Development Canada (ISED)[6] and the CE mark. This chapter contains instructions on the process needed for an integrator when including the JODY-B1 module into an end-product.

- Any deviation from the process described may cause the JODY-B1 series System Integration Manual module not to comply with the regulatory authorizations of the module and thus void the user's authority to operate the equipment.
- Any changes to hardware, hosts or co-location configuration may require new radiated emission and SAR evaluation and/or testing.
- The regulatory compliance of JODY-B1 does not exempt the end-product from being evaluated against applicable regulatory demands; for example, FCC Part 15B criteria for unintentional radiators [8].
- The end-product manufacturer must follow all the engineering and operating guidelines as specified by the grantee (u-blox).
- The JODY-B1 is for OEM integrators only.
- Only authorized antenna(s) may be used. Refer to JODY-B1 data sheet [1] for the list of authorized antennas. In the end-product, the JODY-B1 module must be installed in such a way that only authorized antennas can be used.
- The end-product must use the specified antenna trace reference design, as described in the JODY-B1 antenna reference design application note [14].
- Any notification to the end user about how to install or remove the integrated radio module is NOT allowed.

⚠ If these conditions cannot be met or any of the operating instructions are violated, the u-blox regulatory authorization will be considered invalid. Under these circumstances, the integrator is responsible to re-evaluate the end-product including the JODY-B1 series System Integration Manual module and obtain their own regulatory authorization, or u-blox may be able to support updates of the u-blox regulatory authorization. See also Antenna requirements.

6.1 European Union regulatory compliance (pending)

JODY-B1 series modules comply with the essential requirements and other relevant provisions of Radio Equipment Directive (RED) 2014/53/EU.

For information about the regulatory compliance of JODY-B1 series modules against requirements and provisions in the European Union, see the JODY-B1 Declaration of Conformity [15].

6.1.1 CE End-product regulatory compliance

6.1.1.1 Safety standard

In order to fulfill the safety standard EN 60950-1 [7], the JODY-B1 module must be supplied with a Class-2 Limited Power Source.

[6] Formerly known as IC (Industry Canada).

6.1.2 CE Equipment classes

In accordance with Article 1 of Commission Decision 2000/299/EC[7], JODY-B1 is defined as either Class-1 or Class-2 radio equipment, the end-product integrating JODY-B1 inherits the equipment class of the module.

☞ For guidance on end product marking in according with RED, see http://ec.europa.eu/

⚠ The EIRP of the JODY-B1 module must not exceed the limits of the regulatory domain that the module operates in. Depending on the host platform implementation and antenna gain, integrators have to limit the maximum output power of the module through the host software. For information about the corresponding maximum transmit power levels of Pre-approved antennas.

6.2 Great Britain regulatory compliance (pending)

For information about the regulatory compliance of JODY-B1 series modules against requirements and provisions in Great Britain, see also the JODY-B1 UKCA Declaration of Conformity [16].

6.2.1 UK Conformity Assessed (UKCA)

☞ The United Kingdom is made up of the Great Britain (including England, Scotland, and Wales) and the Northern Ireland. Northern Ireland continues to accept the CE marking. The following notice is applicable to Great Britain only.

JODY-B1 series modules have been evaluated against the essential requirements of the Radio Equipment Regulations 2017 (SI 2017 No. 1206, as amended by SI 2019 No. 696).

For guidance on end product marking in accordance with UKCA, see https://www.gov.uk/guidance/using-the-ukca-marking.

6.3 United states/Canada End-product regulatory compliance

u-blox represents that the modular transmitter fulfills the FCC/ISED regulations when operating in authorized modes on any host product given that the integrator follows the instructions as described in this document. Accordingly, the host product manufacturer acknowledges that all host products referring to the FCC ID or ISED certification number of the modular transmitter and placed on the market by the host product manufacturer need to fulfil all of the requirements mentioned below. Non-compliance with these requirements may result in revocation of the FCC approval and removal of the host products from the market. These requirements correspond to questions featured in the FCC guidance for software security requirements for U-NII devices, FCC OET KDB 594280 D02 [12].

⚠ The modular transmitter approval of JODY-B1, or any other radio module, does not exempt the end product from being evaluated against applicable regulatory demands.

The evaluation of the end product shall be performed with the JODY-B1 module installed and operating in a way that reflects the intended end product use case. The upper frequency measurement range of the end product evaluation is the 5th harmonic of 2.4 GHz as described in KDB 996369 D04.

[7] 2000/299/EC: Commission Decision of 6 April 2000 establishing the initial classification of radio equipment and telecommunications terminal equipment and associated identifiers.

The following requirements apply to all products that integrate a radio module:

- Subpart B – UNINTENTIONAL RADIATORS
  To verify that the composite device of host and module comply with the requirements of FCC part 15B, the integrator shall perform sufficient measurements using ANSI 63.4-2014.
- Subpart C – INTENTIONAL RADIATORS
  It is required that the integrator carries out sufficient verification measurements using ANSI 63.10-2013 to validate that the fundamental and out of band emissions of the transmitter part of the composite device complies with the requirements of FCC part 15C.

When the items listed above are fulfilled, the end product manufacturer can use the authorization procedures as mentioned in Table 1 of 47 CFR Part 15.101, before marketing the end product. This means the customer has to either market the end product under a Suppliers Declaration of Conformity (SdoC) or to certify the product using an accredited test lab.

The description is a subset of the information found in applicable publications of FCC Office of Engineering and Technology (OET) Knowledge Database (KDB). We recommend the integrator to read the complete document of the referenced OET KDB's.

- KDB 178919 D01 Permissive Change Policy
- KDB 447498 D01 General RF Exposure Guidance
- KDB 594280 D01 Configuration Control
- KDB 594280 D02 U-NII Device Security
- KDB 784748 D01 Labelling Part 15 18 Guidelines
- KDB 996369 D01 Module certification Guide
- KDB 996369 D02 Module Q&A
- KDB 996369 D04 Module Integration Guide

6.3.1 United States compliance statement (FCC)

JODY-B1 series modules have modular approval and comply with FCC 47 CFR Part 15C §15.247. Operation is subject to the following two conditions:

- This device may not cause harmful interference, and
- This device must accept any interference received, including interference that may cause undesired operation.

⚠ Any changes or modifications NOT explicitly APPROVED by u-blox could cause the JODY-B1 series System Integration Manualmodule to cease to comply with FCC rules part 15 thus void the user's authority to operate the equipment.

⚠ JODY-B1 series modular transmitter is only FCC authorized for the specific rule parts listed on the FCC grant. The host product manufacturer is responsible for compliance to any other FCC rules that apply to the host not covered by the modular transmitter grant of certification.

The internal / external antenna(s) used for this module must provide a separation distance of at least 20 cm from all persons and must not be co-located or operating in conjunction with any other antenna or transmitter.

Table 16 shows the FCC IDs allocated to JODY-B1 series modules.

Model	FCC ID
JODY-B151-01A	XPYJODYB1

Table 16: FCC IDs for different variants of JODY-B1 series modules

For FCC end-product labeling requirements, see End product labeling requirements.

6.3.2 Canada compliance statement (ISED)

JODY-B1 series modules are certified for use in accordance with the Canada Innovation, Science and Economic Development Canada (ISED) Radio Standards Specification (RSS) RSS-247 Issue 4 and RSS-Gen. Table 17 shows the ISED certification IDs allocated to the JODY-B1 series modules.

Model	ISED certification ID
JODY-B151-01A	8595A-JODYB1

Table 17: ISED IDs for different variants of JODY-B1 modules

Operation is subject to the following two conditions:

1. This device may not cause interference, and
2. This device must accept any interference, including interference that may cause undesired operation of the device.

☞ Any notification to the end user of installation or removal instructions about the integrated radio module is NOT allowed. Unauthorized modification could void authority to use this equipment.

This equipment complies with ISED RSS-102 radiation exposure limits set forth for an uncontrolled environment. This equipment should be installed and operated with minimum distance 20 cm between the radiator and your body.

This radio transmitter IC: 8595A-JODYB1 has been approved by ISED to operate with the antenna types listed in Approved antennas with the maximum permissible gain indicated. Antenna types not included in this list, having a gain greater than the maximum gain indicated for that type, are strictly prohibited for use with this device.

Le présent appareil est conforme aux CNR d'ISED applicables aux appareils radio exempts de licence. L'exploitation est autorisée aux deux conditions suivantes :

(1) l'appareil ne doit pas produire de brouillage, et

(2) l'utilisateur de l'appareil doit accepter tout brouillage radioélectrique subi, même si le brouillage est susceptible d'en compromettre le fonctionnement.

Cet équipement est conforme aux limites d'exposition de rayonnement d'ISED RSS-102 déterminées pour un environnement non contrôlé. Cet équipement devrait être installé et actionné avec la distance minimum 20 cm entre le radiateur et votre corps.

Cet émetteur radio, IC: 8595A-JODYB1 été approuvé par ISED pour fonctionner avec les types d'antenne énumérés dans la section Approved antennas avec le gain maximum autorisé et l'impédance nécessaire pour chaque type d'antenne indiqué. Les types d'antenne ne figurant pas dans cette liste et ayant un gain supérieur au gain maximum indiqué pour ce type-là sont strictement interdits d'utilisation avec cet appareil.

The internal / external antenna(s) used for this module must provide a separation distance of at least 20 cm from all persons and must not be co-located or operating in conjunction with any other antenna or transmitter.

For ISED end-product labeling requirements, see End product labeling requirements.

☞ The approval type for all JODY-B1 variants is a single modular approval. Due to ISED Modular Approval Requirements (Source: RSP-100 Issue 10), any application which includes the module must be approved by the module manufacturer (u-blox). The application manufacturer must provide design data for the review procedure.

6.3.3 Referring to the u-blox FCC/ISED certification ID

If the General requirements, United states/Canada End-product regulatory compliance and all Antenna requirements are met, the u-blox modular FCC/ISED regulatory authorization is valid and the end-product may refer to the u-blox FCC ID and ISED certification number. U-blox may be able to support updates to the u-blox regulatory authorization by adding new antennas to the u-blox authorization for example. See also Antenna requirements.

⚠ To use the u-blox FCC / ISED grant and refer to the u-blox FCC ID / ISED certification ID, the integrator must confirm with u-blox that all requirements associated with the Configuration control and software security of end-products are fulfilled.

6.3.4 Obtaining own FCC/ISED certification ID

Integrators who do not want to refer to the u-blox FCC/ISED certification ID, or who do not fulfil all requirements to do so may instead obtain their own certification. With their own certification, the integrator has full control of the grant to make changes.

Integrators who want to base their own certification on the u-blox certification can do so via a process called “Change in ID” (FCC) / “Multiple listing” (ISED). With this, the integrator becomes the grantee of a copy of the u-blox FCC/ISED certification. U-blox will support with an approval letter that shall be filed as a Cover Letter exhibit with the application.

☞ For modules where the FCC ID / ISED certification ID is printed on the label, the integrator must replace the module label with a new label containing the new FCC/ISED ID. For a description of the labeling requirements, see also the JODY-B1 series System Integration Manual data sheet [1].

⚠ It is the responsibility of the integrator to comply with any upcoming regulatory requirements.

6.3.5 Antenna requirements

In addition to the general requirement to use only authorized antennas, the u-blox grant also requires a separation distance of at least 20 cm from the antenna(s) to all persons. The antenna(s) must not be co-located with any other antenna or transmitter (simultaneous transmission) as well. If this cannot be met, a Permissive Change as described below must be made to the grant.

☞ To support verification activities that may be required by certification laboratories, customers applying for Class-II Permissive changes must implement the setup described in Software.

6.3.5.1 Separation distance

If the required separation distance of 20 cm cannot be fulfilled, a SAR evaluation must be performed. This consists of additional calculations and/or measurements. The result must be added to the grant file as a Class II Permissive Change.

6.3.5.2 Co-location (simultaneous transmission)

If the module is to be co-located with another transmitter, additional measurements for simultaneous transmission are required. The results must be added to the grant file as a Class II Permissive Change.

6.3.5.3 Adding a new antenna for authorization

If the authorized antennas and/or antenna trace design cannot be used, the new antenna and/or antenna trace designs must be added to the grant file. This is done by a Class I Permissive Change or a Class II Permissive Change, depending on the specific antenna and antenna trace design.

- Antennas of the same type and with less or same gain as those included in the list of Pre-approved antennas can be added under a Class I Permissive Change.
- Antenna trace designs deviating from the u-blox reference design and new antenna types are added under a Class II Permissive Change.

⚠ Integrators intending to refer to the u-blox FCC ID / ISED certification ID must contact their local support team to discuss the Permissive Change Process. Class II Permissive Changes are subject to NRE costs.

6.3.6 Configuration control and software security of end-products

☞ "Modular transmitter" hereafter refers to the JODY-B1 module with FCC ID: XPYJODYB1.

As the end-product must comply with the requirements addressed by the OET KDB 594280 [11], the host product integrating the JODY-B1 must comply with the following requirements:

- Upon request from u-blox, the host product manufacturer will provide all of the necessary information and documentation to demonstrate how the requirements listed below are met.
- The host product manufacturer will not modify the modular transmitter hardware.
- The configuration of the modular transmitter when installed into the host product must be within the authorization of the modular transmitter at all times and cannot be changed to include unauthorized modes of operation through accessible interfaces of the host product.
- The host product uses only authorized firmware images provided by u-blox and/or by the manufacturer of the RF chipset used inside the modular transmitter.
- The modular transmitter must when installed into the host product have a regional setting that is compliant with authorized US modes and the host product is protected from being modified by third parties to configure unauthorized modes of operation for the modular transmitter, including the country code.
- The host product into which the modular transmitter is installed does not provide any interface for the installer to enter configuration parameters into the end product that exceeds those authorized.
- The host product into which the modular transmitter is installed does not provide any interface to third parties to upload any unauthorized firmware images into the modular transmitter and prevents third parties from making unauthorized changes to all or parts of the modular transmitter device driver software and configuration.

☞ OET KDB 594280 D01 [11] lists the topics that must be addressed to ensure that the end-product specific host meets the Configuration Control requirements.

☞ OET KDB 594280 D02 [12] lists the topics that must be addressed to ensure that the end-product specific host meets the Software Security Requirements for U-NII Devices.

6.3.7 End product labeling requirements

For an end-product using the JODY-B1 series modules, there must be a label containing, at least, the following information:

This device contains FCC ID: XPYJODYB1 IC: 8595A-JODYB1

(XYZ) represents the FCC "Grantee Code", this code may consist of Arabic numerals, capital letters, or other characters, the format for this code will be specified by the Commission's Office of

Engineering and Technology[10]. (CN) is the Company Number registered at ISED. (UPN) is the Unique Product Number decided by the grant owner.

The label must be affixed on an exterior surface of the end product such that it will be visible upon inspection in compliance with the modular labeling requirements of OET KDB 784748. The host user manual must also contain clear instructions on how end users can find and/or access the FCC ID of the end product.

The label on the JODY-B1 module containing the original FCC ID acquired by u-blox can be replaced with a new label stating the end-product's FCC/ISED ID in compliance with the modular labeling requirements of OET KDB 784748.

FCC end product labeling

The outside of final products containing the JODY-B1 module must display in a user accessible area a label referring to the enclosed module. This exterior label can use wording such as the following: "Contains Transmitter Module FCC ID: XPYJODYB1" or "Contains FCC ID: XPYJODYB1".

In accordance with 47 CFR § 15.19, the end product shall bear the following statement in a conspicuous location on the device:

> This device complies with Part 15 of the FCC Rules. Operation is subject to the following two conditions:
> This device may not cause harmful interference, and
> This device must accept any interference received, including interference that may cause undesired operation.

The following statement must be included in the end-user manual or guide:

> Changes or modifications to this unit not expressly approved by the party responsible for compliance could void the user's authority to operate the equipment.
> This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates uses and can radiate radio frequency energy and, if not installed and used in accordance with the instructions, may cause harmful interference to radio communications. However, there is no guarantee that interference will not occur in a particular installation. If this equipment does cause harmful interference to radio or television reception, which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:
> Reorient or relocate the receiving antenna.
> Increase the separation between the equipment and receiver.
> Connect the equipment into an outlet on a circuit different from that to which the receiver is connected.
> Consult the dealer or an experienced radio/TV technician for help.

ISED end product labeling

The ISED certification label of a module shall be clearly visible at all times when installed in the host device; otherwise, the host device must be labeled to display the ISED certification number for the module, preceded by the words "Contains transmitter module", or the word "Contains", or similar wording expressing the same meaning, as follows: "Contains transmitter module IC: 8595A-JODYB1".

[10] 47 CFR 2.926

L'étiquette d'homologation d'ISED d'un module donné doit être posée sur l'appareil hôte à un endroit bien en vue en tout temps. En l'absence d'étiquette, l'appareil hôte doit porter une étiquette sur laquelle figure le numéro d'homologation du module d'ISED, précédé des mots « Contient un module d'émission », ou du mot « Contient », ou d'une formulation similaire allant dans le même sens et qui va comme suit : « Contient le module d'émission IC: 8595A-JODY-B1.

The end product shall bear the following statement in both English and French in a conspicuous location on the device:

Operation is subject to the following two conditions: This device may not cause interference, and This device must accept any interference, including interference that may cause undesired operation of the device.
Son utilisation est soumise aux deux conditions suivantes : Cet appareil ne doit pas causer d'interférences et il doit accepter toutes interférences reçues, y compris celles susceptibles d'avoir des effets indésirables sur son fonctionnement.

When the device is so small or for such use that it is not practicable to place the statements above on it, the information shall be placed in a prominent location in the instruction manual or pamphlet supplied to the user or, alternatively, shall be placed on the container in which the device is marketed. However, the FCC/ISED ID label must be displayed on the device as described above.

In case, where the final product will be installed in locations where the end-consumer is unable to see the FCC/ISED ID and/or this statement, the FCC/ISED ID and the statement shall also be included in the end-product manual.

6.4 Pre-approved antennas

For the specifications that must be fulfilled in the end product that uses radio type approval of the JODY-B1 module, see the JODY-B1 antenna reference design application note [14].

The JODY-B1 antenna reference design application note provides PCB layout details and electrical specifications.

The JODY-B1 modules has been tested and approved for use with the antennas described in Table 18.

Manufacturer	Part number	Antenna type	Peak gain [dBi]	Validated regulatory domain
Linx Technologies	ANT-DB1-RAF-RPS	Dipole antenna	**4.1**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)
Chang Hong	DA-2458-02-SMR	Dipole antenna	**2.85**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)
TE Connectivity	001-0012	Dipole antenna	**2.0**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)
Taoglas	GW.59.3153	Dipole antenna	**3.8**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)
Laird	MAF94051	Dipole antenna	**2.1**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)
Molex	1461530050	PCB patch	**3.2**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)
Molex	2042810100	PCB patch	**2.0**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)
Unictron	H2B1PD1A1C385L	PCB patch	**2.7**	US/Canada (FCC/ISED) EU/Great Britain (RED/UKCA)

Table 18: List of approved antennas

7 Product testing

7.1 u-blox in-line production testing

As part of our focus on high quality products, u-blox maintain stringent quality controls throughout the production process. This means that all units in our manufacturing facilities are fully tested and that any identified defects are carefully analyzed to improve future production quality.

The Automatic test equipment (ATE) deployed in u-blox production lines logs all production and measurement data – from which a detailed test report for each unit can be generated. Figure 11 shows the ATE typically used during u-blox production.

u-blox in-line production testing includes:

- Digital self-tests (firmware download, MAC address programming)
- Measurement of voltages and currents
- Functional tests (host interface communication)
- Digital I/O tests
- Measurement and calibration of RF characteristics in all supported bands, including RSSI calibration, frequency tuning of reference clock, calibration of transmitter power levels, etc.
- Verification of Wi-Fi and Bluetooth RF characteristics after calibration, like modulation accuracy, power levels, and spectrum, are checked to ensure that all characteristics are within tolerance when the calibration parameters are applied.

Figure 11: Automatic test equipment for module test

7.2 OEM manufacturer production test

As all u-blox products undergo thorough in-series production testing prior to delivery, OEM manufacturers do not need to repeat any firmware tests or measurements that might otherwise be necessary to confirm RF performance. Testing over analog and digital interfaces is also unnecessary during an OEM production test.

OEM manufacturer testing should ideally focus on:

- Module assembly on the device; it should be verified that:
  - Soldering and handling process did not damage the module components
  - All module pins are well soldered on the application board
  - There are no short circuits between pins
- Component assembly on the device; it should be verified that:
  - Communication with host controller can be established
  - The interfaces between module and device are working
  - Overall RF performance test of the device including antenna

In addition to this testing, OEMs can also perform other dedicated tests to check the device. For example, the measurement of module current consumption in a specified operating state can identify a short circuit if the test result deviates that from that taken against a “Golden Device”.

The standard operational module firmware and test software on the host can be used to perform functional tests (communication with the host controller, check interfaces) and perform basic RF performance testing. Special manufacturing firmware can also be used to perform more advanced RF performance tests.

Appendix

A Glossary

Abbreviation	Definition
AEC	Automotive Electronics Council
AP	Access Point
API	Application Programming Interface
ATE	Automatic Test Equipment
BT	Bluetooth
CDM	Charged Device Model
CE	European Conformity
CTS	Clear to Send
DC	Direct Current
DDR	Double Data Rate
DFS	Dynamic Frequency Selection
DHCP	Dynamic Host Configuration Interface
EDR	Enhanced Data Rate
EEPROM	Electrically Erasable Programmable Read-Only Memory
EIRP	Equivalent Isotropic Radiated Power
EMI	Electromagnetic Interference
ESD	Electro Static Discharge
ESL	Equivalent Series Inductance
ESR	Equivalent Series Resistance
FCC	Federal Communications Commission
GND	Ground
GPIO	General Purpose Input/Output
HBM	Human Body Model
HS	High-Speed
HCI	Host Controller Interface
ISED	Innovation, Science and Economic Development Canada
I2C	Inter-Integrated Circuit
KDB	Knowledge Database
LAN	Local Area Network
LDO	Low Drop Out
LED	Light-Emitting Diode
LPO	Low Power Oscillator
LTE	Long Term Evolution
MAC	Medium Access Control
MMC	Multi Media Card
MWS	Mobile Wireless Standards
NRE	Non-recurring engineering
NSMD	Non Solder Mask Defined
OEM	Original equipment manufacturer
OET	Office of Engineering and Technology
OS	Operating System

Abbreviation	Definition
PCB	Printed Circuit Board
PCI	Peripheral Component Interconnect
PCIe	PCI Express
PCM	Pulse-code modulation
PHY	Physical layer (of the OSI model)
PMU	Power Management Unit
RF	Radio Frequency
RSDB	Real Simultaneous Dual Band
RST	Request to Send
SDIO	Secure Digital Input Output
SMD	Solder Mask Defined
SMPS	Switching Mode Power Supply
SMT	Surface-Mount Technology
SSID	Service Set Identifier
STA	Station
TBD	To be Decided
THT	Through-Hole Technology
UART	Universal Asynchronous Receiver-Transmitter
VCC	IC power-supply pin
VIO	Input offset voltage
VSDB	Virtual Simultaneous Dual Band
VSWR	Voltage Standing Wave Ratio
WFD	Wi-Fi Direct
WLAN	Wireless local area network
WPA	Wi-Fi Protected Access

Table 19: Explanation of the abbreviations and terms used

Related documents

[1] JODY-B1 series data sheet, UBXDOC-1953704497-141
[2] Packaging information reference, UBX-14001652
[3] IEC EN 61000-4-2 – Electromagnetic compatibility (EMC) – Part 4-2: Testing and measurement techniques – Electrostatic discharge immunity test
[4] ETSI EN 301 489-1 – Electromagnetic compatibility and Radio spectrum Matters (ERM); ElectroMagnetic Compatibility (EMC) standard for radio equipment and services; Part 1: Common technical requirements
[5] IEC61340-5-1 – Protection of electronic devices from electrostatic phenomena – General requirements
[6] JEDEC J-STD-020E – Moisture/Reflow Sensitivity Classification for Nonhermetic Surface Mount Devices
[7] ETSI EN 60950-1:2006 – Information technology equipment – Safety – Part 1: General requirements
[8] FCC Regulatory Information, Title 47 – Telecommunication
[9] JESD51 – Overview of methodology for thermal testing of single semiconductor devices
[10] Antenna Integration application note, UBX-TBD
[11] FCC guidance 594280 D01 Configuration Control v02 r01,
[12] FCC guidance 594280 D02 U-NII Device Security v01r03
[13] JODY-B1 product summary, UBX-TBD
[14] JODY-B1 antenna reference design application note, UBX-TBD
[15] JODY-B1 RED Declaration of Conformity, UBX-TBD
[16] JODY-B1 UKCA Declaration of Conformity, UBX-TBD

☞ For product change notifications and regular updates of u-blox documentation, register on our website, www.u-blox.com.

Revision history

Revision	Date	Name	Comments
R01	16-Jun-2025	lber	Initial release

Contact

u-blox AG

Address: Zürcherstrasse 68
8800 Thalwil
Switzerland

For further support and contact information, visit us at www.u-blox.com/support.