S

MAYA-W2 antenna reference design

Antenna integration guidance

Application note

Abstract

This application note describes the integration of the antennas in the MAYA-W2 reference design, which was subsequently used to acquire the appropriate FCC and ISED grant. It highlights the module and antenna requirements, performance expectations, and explains the RF path implemented between the various components of the test setup used during the certification.

UBX-23009119 - R01
C1-Public

Document information

Title	MAYA-W2 antenna reference design	
Subtitle	Antenna integration guidance	
Document type	Application note	
Document number	UBX-23009119	
Revision and date	R01	29-Dec-2023
Disclosure restriction	C1-Public	

This document applies to the following products:

Product name
MAYA-W2 series

Contents

1 Introduction

This document describes the implementation of antennas in MAYA-W2 reference design, which was subsequently used to acquire the appropriate FCC and ISED grant. To leverage this existing u-blox grant, customers must copy this design exactly into their application product. Any deviation from this reference design must be filed with the FCC/ ISED to determine whether it can be considered as a "permissive change" to the original grant or is significantly different to warrant the application of a completely new equipment grant of certification (new FCC ID). See also the FCC Permissive Change Policy [1].

The given information should be sufficient to allow for a skilled person to implement the antenna design on an application product. It provides the designer with the necessary PCB layout details including microstrip type, dimensions and antenna interface requirements.

- MAYA-W261, and MAYA-W271 includes two RF antenna pins for use with external antennas.
- MAYA-W266 and MAYA-W276 includes one RF antenna pin that must be connected to an external antenna or looped back on the application PCB to the feed pin of the internal antenna.
- MAYA-W260 includes an U.FL connector on module and no RF trace is included in the certification grant.

2 Module description and requirements

The antenna ports **RF_ANT0**, and **RF_ANT1** have a nominal characteristic impedance of 50 Ω. To allow proper impedance matching along the RF path, each port must be connected to the related antenna through a 50 Ω transmission line. A bad termination of the pin can result in poor performance or even damage the RF section of the module. Antenna interface and antenna requirements are described in Table 1.

For optimal performance in multiradio mode, the isolation between the antennas must meet the requirements specified in Table 2.

Item	Requirements	Remarks
Impedance	50 Ω nominal characteristic impedance	The impedance of the antenna RF connection must match the 50 Ω impedance of the antenna pins.
Frequency range	2400 - 2500 MHz 5150 - 5850 MHz	For 802.11b/g/n/ax and Bluetooth. For 802.11a/n/ac/ax
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 (Voltage Standing Wave Ratio), refers to the amount of reflected power. It provides a measurement of how well the primary antenna RF connection matches the 50 Ω characteristic impedance of antenna pins. To maximize the amount of power transferred to the antenna, the impedance of the antenna termination must match the 50 Ω nominal impedance of antenna pins over the entire operating frequency range.
Efficiency	> -1.5 dB (> 70%) recommended > -3.0 dB (> 50%) acceptable	The radiation efficiency is the ratio of the radiated power to the power delivered to antenna input; the efficiency is a measure of how well an antenna receives or transmits.
Maximum gain		To comply with the radiation exposure limits of the various regulatory agencies, the peak antenna gain must not exceed that specified in the Approved antennas section in the datasheet [2].

Table 1: Summary of antenna interface requirements

For optimal performance in multiradio mode, the isolation and antenna correlation coefficient between the antennas must meet the requirements specified in Table 2.

Item	Requirements	Remarks
Isolation (in-band)	S_{21} > 25 dB recommended S_{21} > 20 dB acceptable	The S_{21} parameter represents the antenna-to-antenna isolation between the two antennas in their band of operation.
Isolation (out-of-band)	S_{21} > 35 dB recommended S_{21} > 30 dB acceptable	Out-of-band isolation is evaluated in the band of the aggressor. This ensures that the transmitting signal from the other radio is sufficiently attenuated by the receiving antenna to avoid any saturation or intermodulation effect at the receiver port.
Envelope correlation Coefficient (ECC)	ECC < 0.1 recommended ECC < 0.5 acceptable	The ECC parameter correlates the far-field parameters between antennas in the same system. Although MAYA-W1 does not currently support MIMO, a low ECC parameter is fundamental for improving the performance of any future implementation in MIMO-based systems.

Table 2: Summary of MIMO and Wi-Fi/Bluetooth coexistence requirements

3 Reference design of RF path

The FCC grant of certification for MAYA-W2 is based on a setup configuration that includes MAYA-W2 mounted on an evaluation board.

Figure 1 shows the reference design schematic and how to connect either the internal antenna or external antennas to connectors J10 and J11.

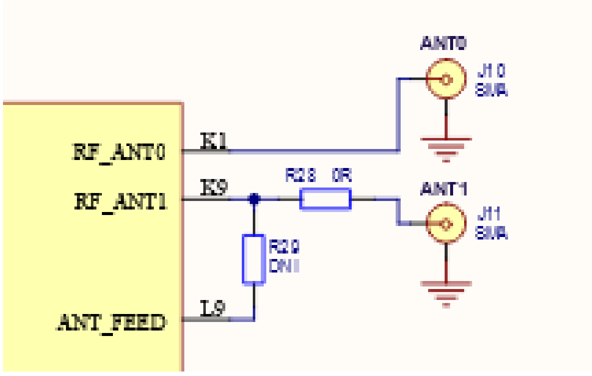


Figure 1: Reference design schematic – showing RF_ANT0 and RF_ANT1 antenna paths

- For MAYA-W266 and MAYA-W276 with internal antenna
  - Mount a 0 Ω resistor, R29, to connect to the internal antenna **ANT_FEED** pin.
  - R28 must be no mount.
  - Leave pin **RF_ANT0** unconnected.
- For MAYA-W266 and MAYA-W276 with external antenna
  - Mount a 0 Ω resistor, R28, to connect to an external antenna.
  - R29 must be no mount.
  - Leave pin **RF_ANT0** unconnected.
- For MAYA-W261, and MAYA-W271 for connecting two external antennas
  - Mount a 0 Ω resistor, R28, to connect to connector J11.
  - R29 must be no mount.

3.1 PCB RF trace routing

The antenna traces are implemented with coplanar microstrips routed on the top layer with GND reference plane on layer 3, as shown in Figure 4. GND on Layer 2 is masked underneath the RF trace. The two coplanar ground planes beside the signal trace must be connected to the lower ground plane using vias. It is advisable to place the vias with a maximum distance of 0.5 mm to the coplanar ground edge with a maximum pitch of 2 mm. The top side should be coated with generic LPI solder stop mask.

The copper clearance on right-hand side of the module and shown in Figure 2, Figure 3, and Figure 4 is only needed when the internal PCB trace antenna is used. If external antennas are used, the module can be placed in an arbitrary position of the application PCB.

For further information about the keep-free area, see the MAYA-W2 system integration manual [1].

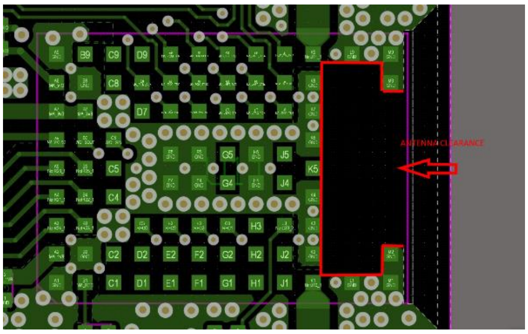


Figure 2. Definition of keep-free area, antenna clearance

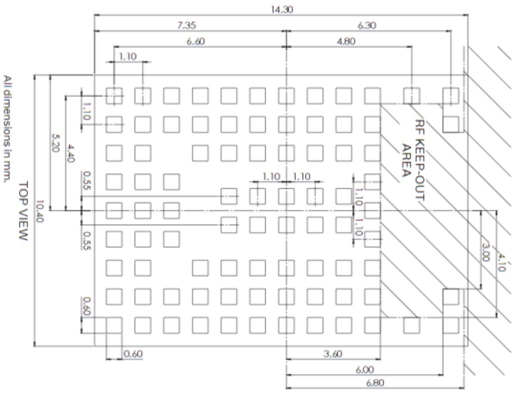


Figure 3. MAYA-W2 mechanical outline showing measures of the keep free-area, RF KEEP-OUT AREA

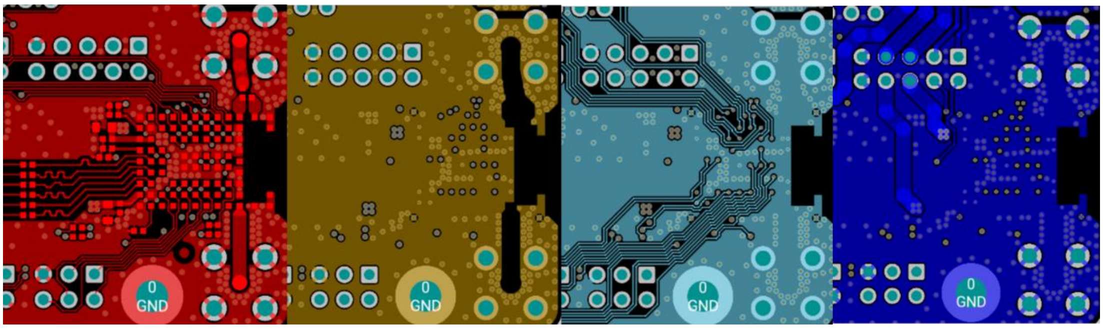


Figure 4: MAYA-W2 EVK layers

3.2 PCB trace dimensions

3.2.1 MAYA-W266 and MAYA-W276 internal antenna

Figure 5 shows the traces connecting **RF_ANT1** to **ANT_FEED** through the 0 Ω R29 resistor. The dimensions of the traces are defined in Table 3. Do not mount R28.

☞ When using the internal antenna GND clearance must be implemented in accordance with the guidelines in the MAYA-W2 antenna reference designsystem integration manual [1].

☞ X represents horizontal axis. Y represent vertical axis.

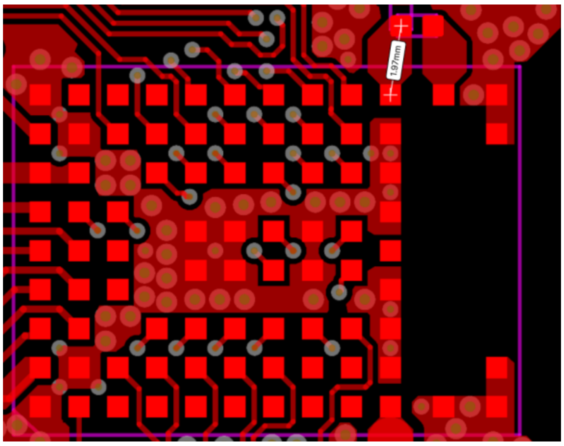


Figure 5: PCB artwork of connection from RF_ANT1 to ANT_FEED

Axis	Dimension	Comment
Trace length	1.97 mm	Point to point length between center of pads.
ΔX	0.30 mm	
ΔY	1.95 mm	
Trace width	1.25 mm	Tapered from 0.6mm with 45° degrees angle to widest trace width of 1.25mm

Table 3. Dimensions of the traces connecting RF_ANT1 to ANT_FEED through R29

3.2.2 MAYA-W261 and MAYA-W271, and MAYA-W266 and MAYA-W276 with single external antenna

For MAYA-W261 and MAYA-W271 and MAYA-W266 and MAYA-W276 that are used together with external antennas, implement the microstrips – in accordance with Figure 6 and the dimensions described in Table 4, Table 5, and Table 6 – together with the relevant part of the trace described in the PCB trace dimensions. This constitutes the full connection from MAYA to connector J11.

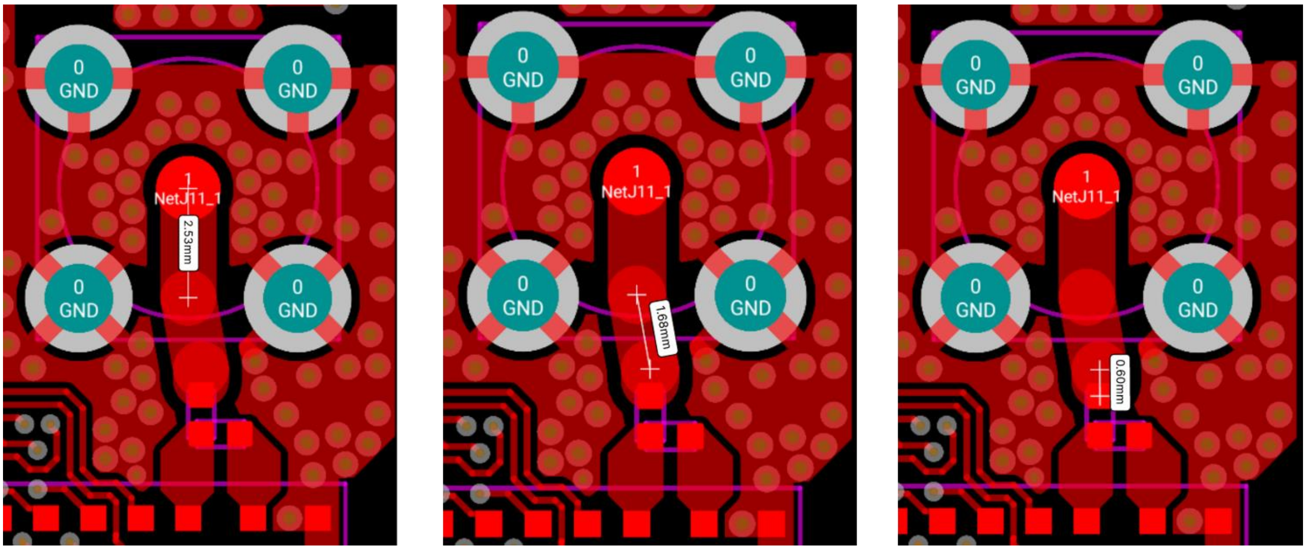


Figure 6: RF_ANT1 connection to connector J11

Axis	Dimension	Comment
Trace length	2.35 mm	Point to point length
ΔX	0 mm	
ΔY	2.35 mm	
Trace width	1.30 mm	

Table 4: Dimensions of microstrip shown in Figure 6, upper segment (left hand image)

Axis	Dimension	Comment
Trace length	1.68 mm	Point to point length
ΔX	0.30 mm	
ΔY	1.65 mm	
Trace width	1.30 mm	

Table 5: Dimensions of microstrip shown in Figure 6, middle segment (middle image)

Axis	Dimension	Comment
Trace length	0.60 mm	Point to point length
ΔX	0 mm	
ΔY	0.60 mm	
Trace width	1.30 mm	

Table 6: Dimensions of microstrip shown in Figure 6, lower segment (right hand image)

For MAYA-W261 and MAYA-W271 Figure 7 and Table 7 show the implementation of the connection from **RF_ANT0** to connector J10.

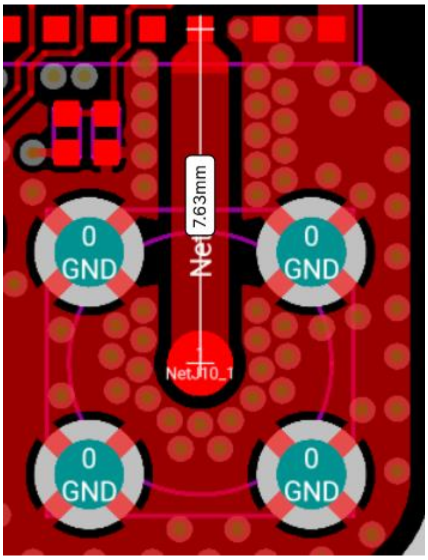


Figure 7: RF_ANT0 connection to connector J10

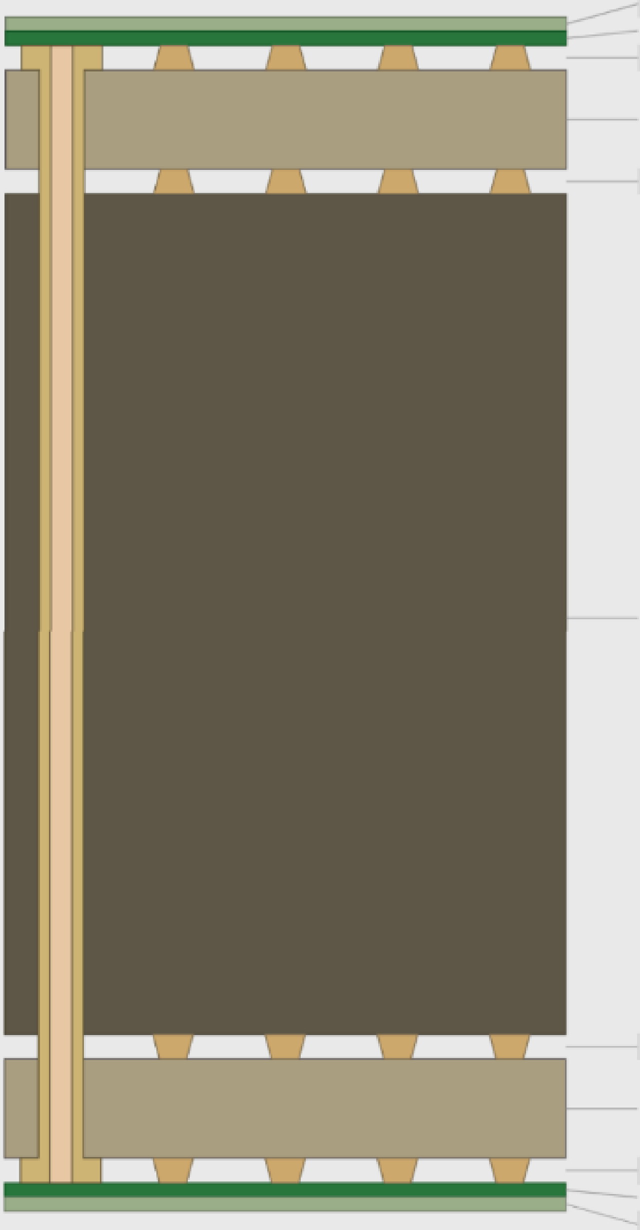
Axis	Dimension	Comment
Trace length	7.63 mm	Point to point length
ΔX	0 mm	
ΔY	7.63 mm	
Trace width	1.30 mm	

Table 7: Strip connecting RF_ANT0 to J10

4 MAYA-W2 EVB PCB

4.1 Stack-up

MAYA-W2 EVK reference design is designed on a four-layer PCB, as shown in Figure 8.



	Layer	Type	Thk (mm)
-	Top Overlay	Overlay	
-	Top Solder	Solder Mask	0.0200
1	Top Layer	Signal	0.0350
-	Dielectric 1	Prepreg	0.1400
2	Layer 2	Signal	0.0350
-	Dielectric 2	Core	1.2000
3	Layer 3	Signal	0.0350
-	Dielectric 3	Prepreg	0.1400
4	Bottom Layer	Signal	0.0350
-	Bottom Solder	Solder Mask	0.0200
-	Bottom Overlay	Overlay	
		Total	1.6600

Figure 8: EVK PCB stack-up

4.2 Coplanar microstrip design

Figure 1 shows a cross section of a coplanar microstrip implemented on a PCB. The top layer shows the copper layer microstrip with the width marked “W” surrounded by GND separated with the distance “S” from the microstrip. The bottom layer represents the microstrip’s GND reference. Between the top and bottom layer is the dielectric substrate with a thickness of “H” and dielectric constant “ε_r”. The width of a 50 Ω microstrip depends on mainly “ε_r” and “H” and must be calculated for each PCB layer stack-up.

“T” is the thickness of the copper layer. This can also be represented by “Base Copper Weight”, which is commonly used as the parameter for PCB stack-up.

For acceptable microstrip losses it is advised to implement a reasonably wide trace and with the GND reference on e.g., the second inner layer with the copper GND on the first inner layer removed underneath the microstrip.

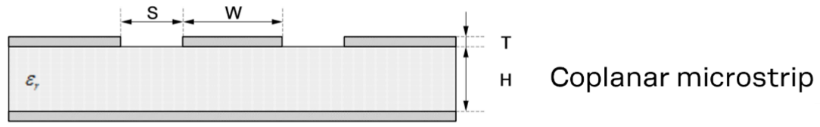


Figure 9: Coplanar microstrip structure

Appendix

A Glossary

Abbreviation	Definition
ECC	Envelope Correlation Coefficient
FCC	Federal Communications Commission (US)
ISED	Innovation, Science and Economic Development (Canada)
MIMO	Multiple-Input and Multiple-Output
RF	Radio Frequency
SMA	SubMiniature version A (connector)
VSWR	Voltage Standing Wave Ratio

Table 8: Explanation of the abbreviations and terms used

Related documentation

[1] MAYA-W2 antenna reference designsystem integration manual, UBX-21010495
[2] MAYA-W2 series data sheet, UBX-21006380
[3] FCC Permissive Change Policy, 178919

☞ 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	29-Dec-2023	Iber	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.