ignion

Your innovation.
Accelerated.

DESIGN GUIDE Wi-Fi ANTENNA INTEGRATION

UNLOCK THE POTENTIAL OF WI-FI WITH VIRTUAL ANTENNA® TECHNOLOGY.

Wi-Fi ANTENNA INTEGRATION DESIGN GUIDE

Wi-Fi (IEEE 802.11) has become a cornerstone technology for IoT applications, catering to a wide array of use cases thanks to its high data rates, already deployed infrastructure, and interoperability. From asset tracking and home automation to smart lighting and beyond, Ignion provides robust and easy to integrate Wi-Fi antenna solutions, often combined with other wireless technologies such as BLE, Cellular IoT, and GNSS. Use this design guide to prepare for your next Wi-Fi integration, either standalone or when adding Wi-Fi in co-existence with other wireless technologies. Using Virtual Antenna® technology and the Antenna Intelligence Cloud™ tool for Wi-Fi ensures:

Low Power Consumption **Easy Integration**

Robust Connection **Wireless technologies coexistence**

Why is Virtual Antenna® technology your best choice for your Wi-Fi device?

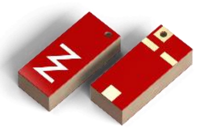
- Robust and reliable connectivity even in harsh environments.
- Easy to integrate multiple radios and even multi-board designs.
- Low maintenance over the long device and battery lifetimes.
- Cost-effective in a small package enabling space-optimized design.

What is in this Design Guide?

- Easy antenna implementation guide for any device integrating Wi-Fi. No need to be RF expert.
- How to achieve the best antenna performance. Do's, Don'ts, and Best Practices.
- Practical examples and results from the Nordic nRF7002 Wi-Fi boards.

Characteristics of the Virtual Antenna® components used in this design guide:

DUO mXTEND™ (NN03-320)
(7.0 x 3.0 x 2.0 mm)
Operating range: 1561 – 10600 MHz.
https://ignion.io/product/duo-mxtend/



NANO mXTEND™ (NN02-101)
(3.0 x 2.0 x 0.8 mm)
Operating range: 2400 – 10600 MHz.
https://ignion.io/product/nano-mxtend/

- Linearly polarized.
- Omnidirectional radiation patterns.
- Temperature range from -40 to + 125 ºC.
- Characteristic impedance 50Ω.

TABLE OF CONTENTS

1. DESIGN JOURNEY WITH VIRTUAL ANTENNA® 4

1.1. VIRTUAL ANTENNA® TECHNOLOGY INTRO 4

1.2. VIRTUAL ANTENNA® AND ANTENNA INTELLIGENCE CLOUD™ FROM SOFTWARE TO HARDWARE. 5

2. ADDING WI-FI TO YOUR DESIGN 6

2.1. WI-FI FEASIBILITY AND DIGITAL PROTOTYPE WITH ANTENNA INTELLIGENCE CLOUD™ 6

2.2. WI-FI COEXISTENCE WITH BLUETOOTH LOW ENERGY. 12

2.3. WI-FI TOGETHER WITH CELLULAR AND GPS IN A SMALL FORM FACTOR BOARD 13

2.4. PCB DESIGN RECOMMENDATIONS AND FREE GERBER REVIEW SERVICE 18

3. SUMMARY 22

4. DO YOU NEED MORE HELP? OVERVIEW OF IGNION SUPPORT SERVICES 23

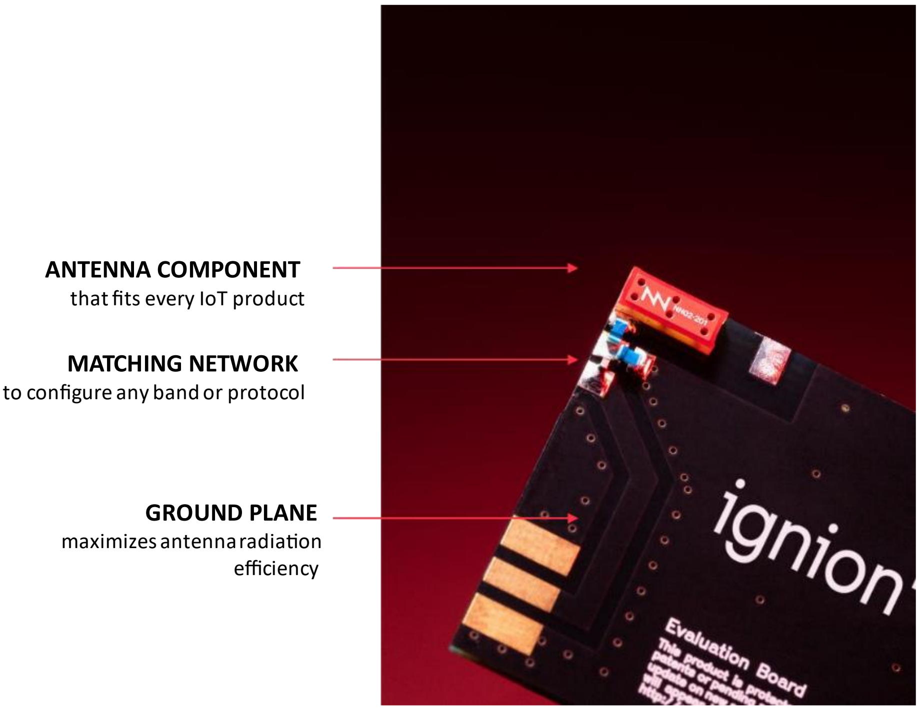
1. DESIGN JOURNEY WITH VIRTUAL ANTENNA®

With miniaturization of IoT devices and the use of many different wireless standards on the same board, the antenna integration may seem challenging, but thanks to Virtual Antenna® technology, this perception does not align with reality.

Based on our extensive experience as a pioneering antenna manufacturer, this design guide will take you through the different ways to implement a Wi-Fi antenna solution starting with a few essential do's and don'ts specifically tailored to Wi-Fi antenna integration and followed by a step-by-step guide on how to easily integrate a Wi-Fi antenna in your device.

1.1. VIRTUAL ANTENNA® TECHNOLOGY INTRO

Virtual Antenna® technology is the smallest and most versatile antenna option on the market, delivering robust high performance from a system of 3 main elements:



Antenna component: This non-resonant Virtual Antenna® component is mounted directly on the printed circuit board (PCB) and its role is to maximize the transfer of power from the radio frequency module to the ground plane, and vice versa. This unique capability enables the Virtual Antenna® portfolio to support a frequency range from 400 MHz to 10 600 MHz.

Matching network: A series of small low-cost capacitors and inductors placed between radio and antenna components for antenna frequency response tuning. The frequency band(s) can easily be modified at any stage by retuning the matching network component.

Ground plane: A Virtual Antenna® solution achieves resonance through the combination of the antenna component, matching network, and the PCB ground plane and it transmits/receives radio waves for communication through the ground plane. The ground plane dimensions have a direct influence on RF performance.

1.2. VIRTUAL ANTENNA® AND ANTENNA INTELLIGENCE CLOUD™ FROM SOFTWARE TO HARDWARE.

All the steps in this design guide can be done by any IoT device designer or hardware engineer with the right tools. Additionally, the Virtual Antenna® technology is backed by the Ignion customer services technical team ready to assist in all stages of the design.

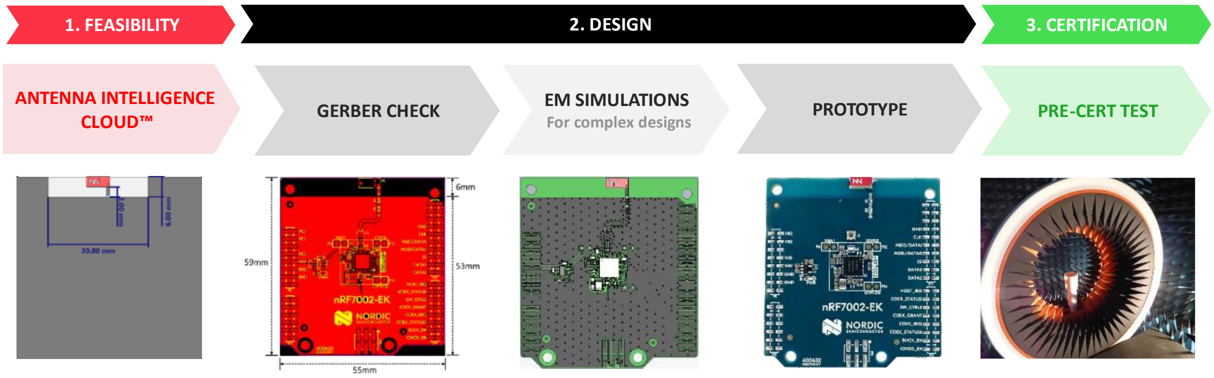


Figure 1 – Virtual Antenna® design journey steps for successful antenna integration.

Step 1 – Assess Feasibility with Antenna Intelligence Cloud™ (free Ignion service): Through the Antenna Intelligence Cloud™ the developer is provided with tailored guidance and digital prototype performance results, including S-parameters, total efficiency, design recommendations and tailored antenna design files (Altium, SolidWorks, Cadence, etc.). By simply completing the online form and selecting the frequency band of operation and desired PCB size, the results are guaranteed within 24 hours, but typically, they are generated in minutes.

Step 2 - Build Gerber files and sanity check (free Ignion service): Gerber files can be built and reviewed with the Ignion team which will respond with a short Gerber review report.

Step 3 - EM simulation: The simulation considering every detail allows the designer to evaluate potential design changes/requests and their impact on the antenna performance.

Step 5 - Produce prototype, test, and fine-tune: Verifying that prototype performance results are aligned with expectations and fine-tuned if necessary.

Step 6 - Certification pre-test: If needed, over-the-air (OTA) tests with the antenna system and radio module can be done to ensure the device meets certification requirements.

2. ADDING WI-FI TO YOUR DESIGN

2.1. WI-FI FEASIBILITY AND DIGITAL PROTOTYPE WITH ANTENNA INTELLIGENCE CLOUD™

Adding Wi-Fi to your design is easier than ever, thanks to Virtual Antenna® technology. In the following sections, we present examples of how to start with a new device design as well as how to expand existing designs connectivity with Wi-Fi.

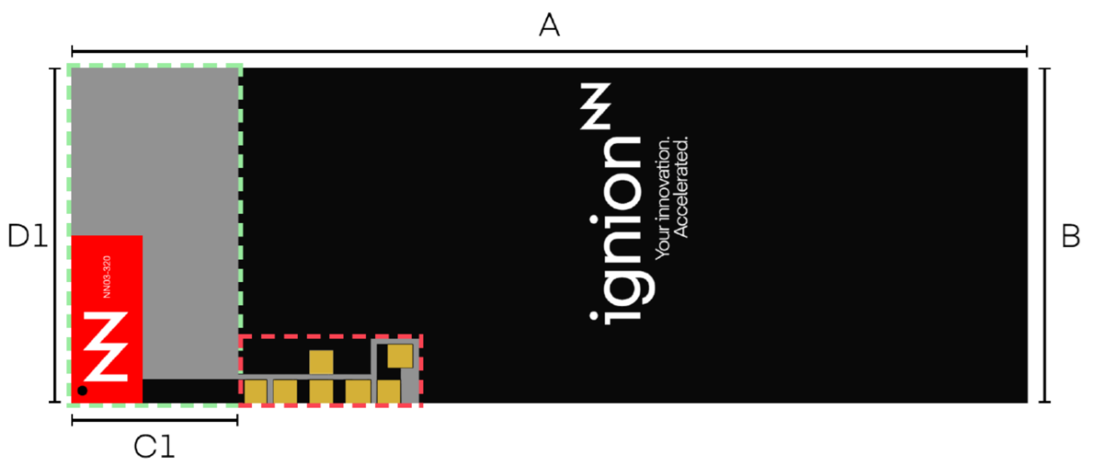
2.1.1. FIRST STEPS – DIGITAL PROTOTYPE

It is highly recommended that any IoT device starts with a feasibility analysis, which consists of a simple and rapid performance estimation on a bare PCB without any additional components, casings, batteries, etc. These results can be generated by Ignion's Antenna Intelligence Cloud™, an exclusive online tool that allows any user to pre-assess the performance of their device antenna performance results, suggested matching network topologies, and a Bill of Materials (BoM), all together with automated design files containing all the antenna recommendations needed for starting your board design.

Example project:

Initial requirements for device size: 14 mm x 50 mm
Wi-Fi bands: 2.4 – 2.485 GHz

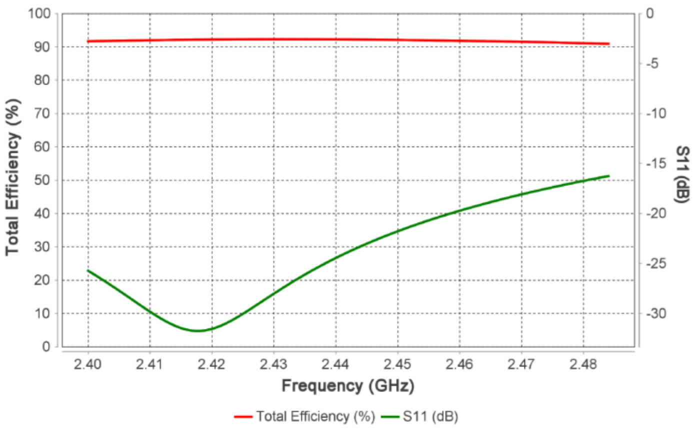
Below are the results extracted from the Antenna Intelligence Cloud™ submitted for this example project using a 14 mm x 50 mm size PCB board for Wi-Fi (2.4 – 2.485 GHz)

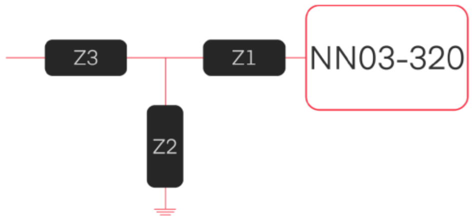


PCB		1: NN03-320	
Measure	mm	Measure	mm
A	40.0	C1	7.0
B	14.0	D1	14.0

Clearance Area
Matching Network

Figure 2 – Antenna and clearance area recommended for the example above from the Antenna Intelligence Cloud™.





	Value	Part Number
Z_1	3.3 nH	LQW15AN3N3G80
Z_2	4.3 nH	LQW18AN4N3G80
Z_3	0Ω	Resistor

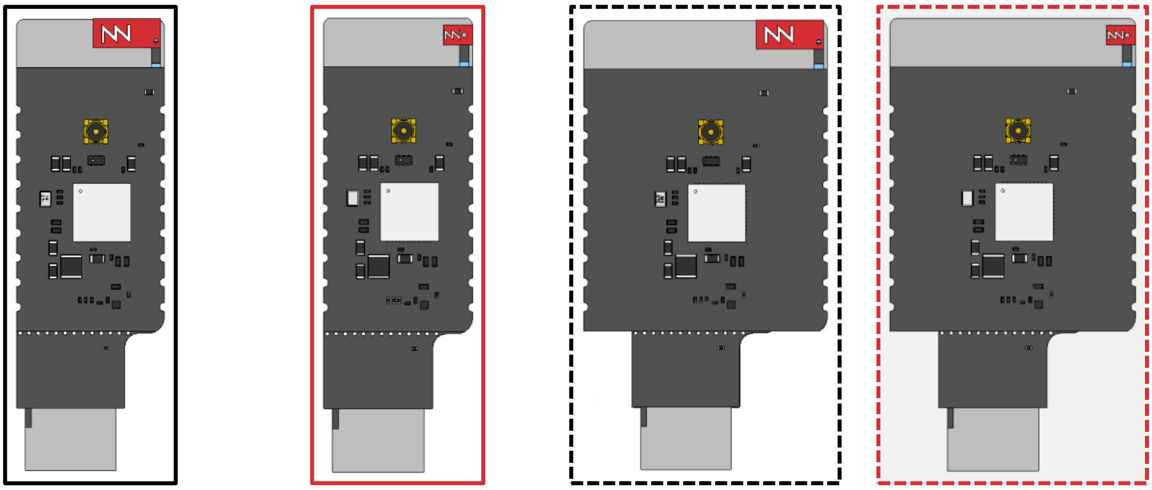
Figure 3 – Antenna performance (Voltage Standing Wave Ratio (VSWR) and Total efficiency), matching network topology, and BoM for the example above from the Antenna Intelligence Cloud™.

When assessing the feasibility, there are several pathways available to adjust the antenna design, depending on the importance of antenna performance, device dimension constraints and cost. It is recommended to run a few different design ideas through the Antenna Intelligence Cloud™ to decide on the optimal balance between performance, size and cost.

2.1.2. OPTIMIZING FOR SMALL SIZE/COST OR PERFORMANCE SIMULATIONS

In this section, we analyze an example with Nordic Semiconductor. nRF7002 Plugin Board using dual-band Wi-Fi 2.4 GHz to 2.485 GHz and 5.150 GHz to 5.875 GHz. As PCB dimensions, clearance area and the antenna component chosen all have an impact on the wireless performance, it is important to understand how these design choices influence your final device.

For this example, two different antennas are compared, the DUO mXTEND™ (7.0 mm x 3.0 mm x 2.0 mm) and the NANO mXTEND™ (3.0 mm x 2.0 mm x 0.8 mm) across different PCB sizes, 14 mm x 40 mm and 24 mm x 40 mm. Depending on your design constraints, the larger component, DUO mXTEND™, will deliver higher performance compared to NANO, however, NANO is a more compact and lower cost component.



DUO mXTEND™ (PCB 40mm x 14mm)	**NANO mXTEND™ (PCB 40mm x 14mm)**	**DUO mXTEND™ (PCB 40mm x 24mm)**	**NANO mXTEND™ (PCB 40mm x 24mm)**

Figure 4 – PCB form factors and antennas compared for a Wi-Fi USB plugin board.

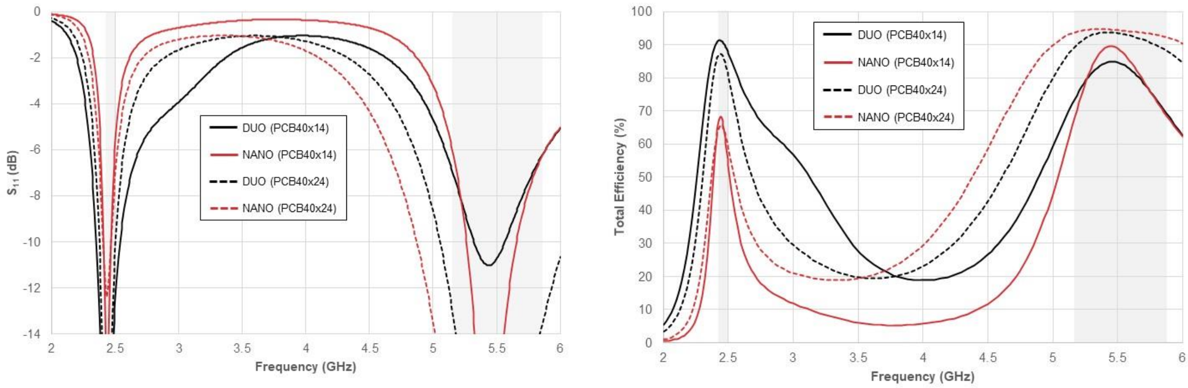


Figure 5 - S-parameters (left) and Total Efficiency (right) responses of plugin boards introduced in Figure 4.

	Total Efficiency Average (2400-2500 MHz) [%]	**Total Efficiency Average (5150-5875 MHz) [%]**
DUO, PCB 40 mm x 14 mm	90.2	79.5
NANO, PCB 40 mm x 14 mm	63.0	80.6
DUO, PCB 40 mm x 24 mm	85.2	91.8
NANO, PCB 40 mm x 24 mm	63.2	94.0

Table 1 - Total efficiency simulated for Wi-Fi for DUO mXTEND™ (NN03-320) and NANO mXTEND™ (NN02-101) with PCBs 40 mm x 14 mm and 40 mm x 24 mm.

Results shown in Figure 5 and Table 1 demonstrate that both DUO mXTEND™ (NN03-320) and NANO mXTEND™ (NN02-101) can provide competent solutions despite having different PCB dimensions.

After completing the preliminary simulated analysis, developers can make informed decisions about which PCB to implement based on size and the space for the antenna, and which antenna is more suitable for their needs based on cost/performance. Subsequently, it is advisable to conduct PCB simulations that consider matching network layout, electronics, casings, and other elements to account for their impact on the final performance.

2.1.3. PHYSICAL DEVICE PERFORMANCE COMPARED WITH SIMULATIONS.

In this example the PCB of 49 mm x 17 mm with DUO mXTEND™ was chosen to be built in a physical prototype. Below a direct comparison between measurements and simulations is shown. Unlike the previous simulations, this time, a comprehensive simulation is performed considering full device implementation. To perform the measurements, a cable test setup must be assembled on the PCB, which essentially involves removing the RF module and soldering a coaxial cable with properly grounded edges. One end of the cable is soldered to the input/output Wi-Fi port of the module, while the other end features an SMA connector. Parameters such as S-parameters and Total efficiency are then accurately measured.

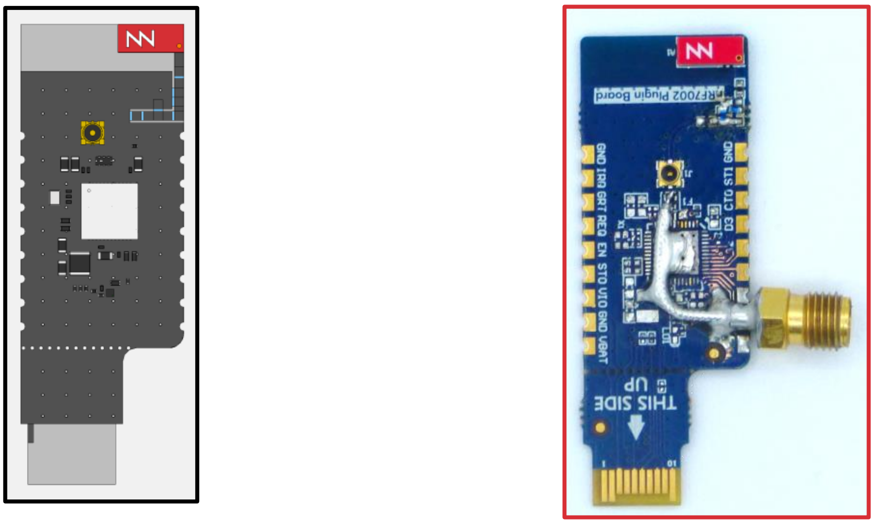


Figure 6 – Nordic Semiconductor nRF7002 Plugin Board PCB of 49 mm x 17.5 mm with DUO mXTEND™ (NN03-320) simulated (left) and measured (right).

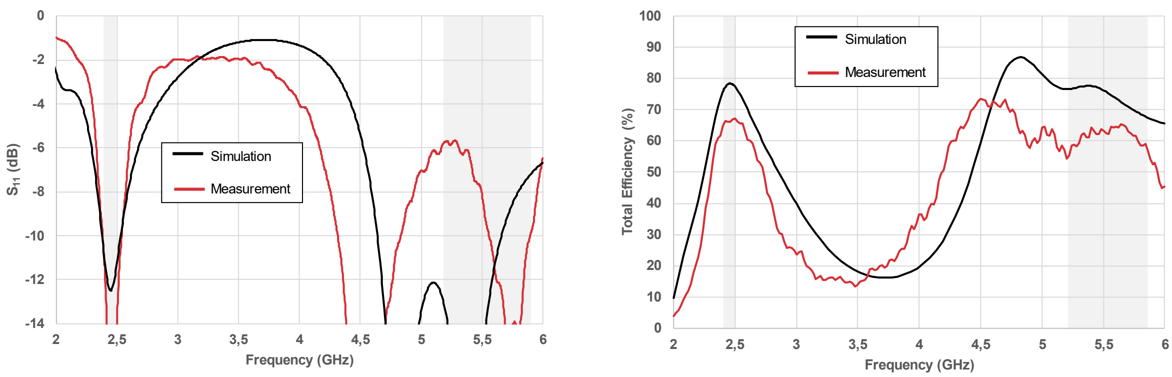
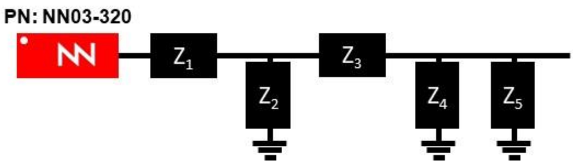


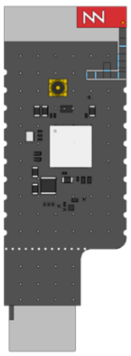
Figure 7 – S-parameters (left) and Total Efficiency (right) responses of plugin boards introduced in Figure 6.

	Total Efficiency Average (2400-2500 MHz) [%]	Total Efficiency Average (5150-5875 MHz) [%]
DUO mXTEND™, PCB 49 mm x 17.5 mm (simulations)	77.7	74.1
DUO mXTEND™, PCB 49 mm x 17.5 mm (measurements)	66.3	60.9

Table 2 - Total efficiency simulated and measured for Wi-Fi for DUO mXTEND™ (NN03-320) in a PCBs 49 mm x 17.5 mm.

Wi-Fi Dual Band (2400 MHz – 2500 MHz, 5150 MHz – 5875 MHz), DUO mXTEND™





	Value	Part Number
Z_1	0Ω	Resistor
Z_2	0.6 pF	GJM1555C1HR60WB01
Z_3	2.2 nH	LQW15AN2N2G80
Z_4	0.5 pF	GJM1555C1HR50WB01
Z_5	3.3 nH	LQW15AN3N3G80
Z_6	0.7 pF	GJM1555C1HR70WB01

Table 3 - BoM PCB 49 mm x 17.5 mm, NN03-320 simulated.

	Value	Part Number
Z_1	0Ω	Resistor
Z_2	Open	-
Z_3	2.6 nH	LQW15AN2N6G80
Z_4	0.5 pF	GJM1555C1HR50WB01
Z_5	3.5 nH	LQW15AN3N5G80
Z_6	0Ω	Resistor

Table 4 - BoM PCB 49 mm x 17.5 mm, NN03-320 measured.

The results shown above demonstrate that the simulated results are well aligned with the physical device measurements. Initial antenna response on the physical device might be shifted compared to simulations but can easily be fine-tuned through the matching network.

Further several variables can affect the antenna response when comparing simulation and measurement for instance in terms of antenna impedance and radiation efficiency, including:

- **Copper trace:** Changes in copper trace thickness can affect radiation and impedance characteristics.
- **Nearby components:** The radiation pattern, impedance and resonant frequency of the antenna can be altered by nearby components.
- **Manufacturing process and tolerances:** Variations in the manufacturing process and tolerances can result in differences between the characteristics of simulated and real antennas.
- **Mounting technique:** How the antenna is mounted or placed in the test setup can affect the electromagnetic environment and the performance of the antenna.
- **Measurement equipment calibration**: Instrument calibration is important because it introduces a degree of tolerance error that can affect the accuracy of measurement results.

Above aspects must be carefully considered to ensure optimized performance in the physical device as these are not included in simulations.

2.2. WI-FI COEXISTENCE WITH BLUETOOTH LOW ENERGY.

In a solution where Wi-Fi (802.11) and Bluetooth Low Energy (BLE) must coexist, the choice of using a single or separate antennas depend on various factors, including the specific application requirements, available PCB layout space, and desired RF performance. Here are a few topics for when to consider separate antennas or a single antenna:

Separate antennas:

1. **High-performance requirements**: If your application demands the highest possible performance for both Wi-Fi and BLE simultaneously, using separate antennas is often the preferred choice.

2. **Avoiding interference**: Separate antennas can help minimize interference between Wi-Fi and BLE, especially if they operate in the same frequency band (2.4 GHz). Isolation between antennas can help maintain signal integrity for both technologies.

Single antenna:

1. **Space Constraints**: In compact devices with limited space, using a single, shared antenna can be a practical solution to save space and reduce the complexity of the design.

2. **Cost considerations**: Using a single antenna can be more cost-effective in terms of components and manufacturing, as it eliminates the need for multiple antennas and additional connectors.

3. **Moderate performance requirements**: If the application's performance requirements are moderate, a single antenna may be a fit for both Wi-Fi and BLE wireless technologies.

Ultimately, the decision to use separate antennas or a single antenna should be based on a careful assessment of the specific requirements and constraints of your application. It's also important to consider antenna placement, radiation patterns, and potential interference sources to optimize the coexistence of Wi-Fi and BLE in your solution. Ignion offers simulation and testing that can be valuable tools in evaluating the performance of different antenna configurations. For more info about it, see the following link: https://ignion.io/resources-support/technical-center/engineering-support/.

2.3. WI-FI TOGETHER WITH CELLULAR AND GPS IN A SMALL FORM FACTOR BOARD

Wi-Fi can easily be added to cellular designs even in small form factors PCBconductor , both for single, dual, and triple band Wi-Fi. Here we show an example of NANO mXTEND™ (NN02-101) for Wi-Fi dual band integrated in a 50 mm by 50 mm PCB together with TRIO mXTEND™ (NN03-310) for LTE-M/NB-IoT and the DUO mXTEND™ (NN03-320) for GNSS. Due to the flexibility of the NANO mXTEND™, the Wi-Fi chip antenna can be placed in the middle of the board instead of the corner, enabling the three antennas to perform without suffering from significant coupling effects.

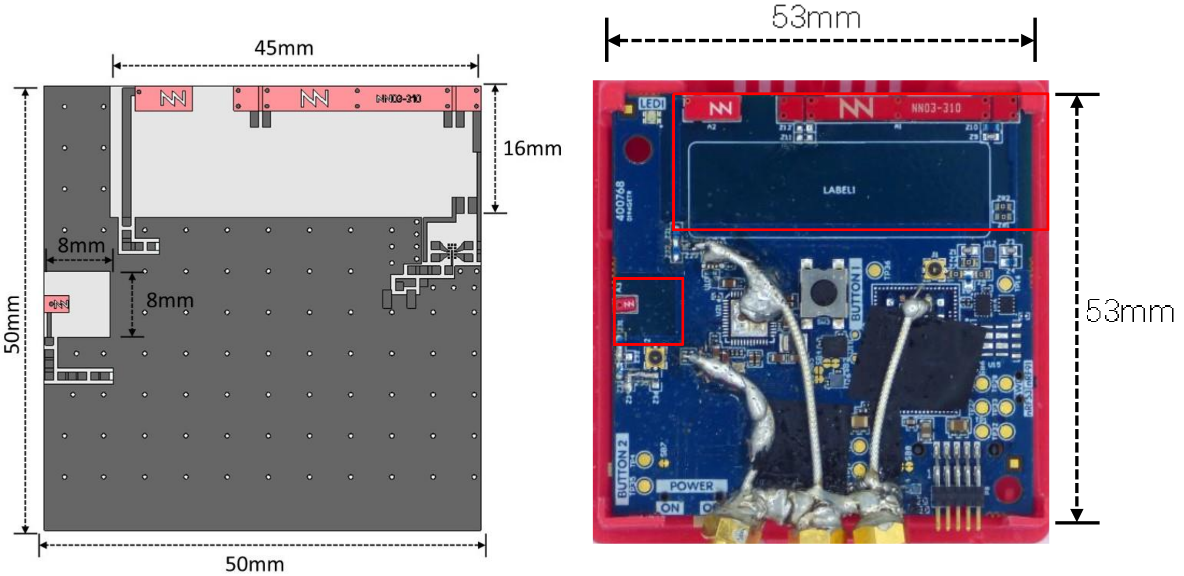


Figure 8 – A small form factor PCB with TRIO mXTEND™ (NN03-310) for LTE-M/NB-IoT, DUO mXTEND™ (NN03-320) for GNSS and NANO mXTEND™ (NN02-101) for Wi-Fi.

Results demonstrate that Virtual Antenna® technology provides low S-parameters and high efficiency for LTE-M/NB-IoT (values shown are an enveloping of all states of the Qorvo switch), GNSS, and Wi-Fi (Figure 9 and Table 5 - Table 7). Additionally, coupling values of the proposed design are always below -13dB (coupling values LTE-GNSS and LTE - Wi-Fi are enveloping all states of the cellular matching network switch).

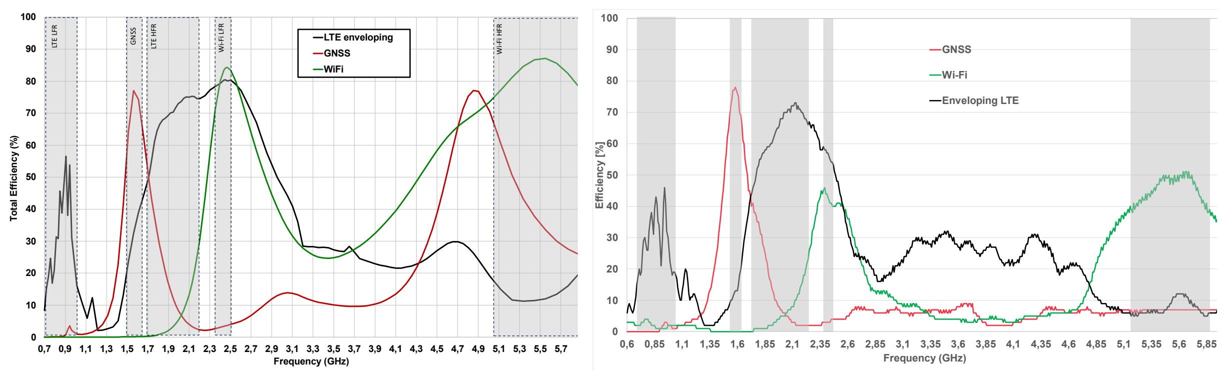


Figure 9 – Performance of a small form factor PCB with TRIO mXTEND™ (NN03-310) for LTE-M/NB-IoT (enveloping), DUO mXTEND™ (NN03-320) for GNSS and NANO mXTEND™ (NN02-101) for Wi-Fi.

	Total Efficiency NN03-310					
	@698 MHz [%]	@960 MHz [%]	Average (698 - 960 MHz) [%]	@1710 MHz [%]	@2200 MHz [%]	Average (1710 - 2200 MHz) [%]
Sim	8.3	31.2	31.9	50.5	74.5	69.3
Meas	18.0	32.0	31.2	41.0	69.0	62.3

Table 5 - Total efficiency for LTE with the TRIO mXTEND™ (NN03-310) of a small form factor PCB (Figure 8).

	Total Efficiency NN03-320		
	@1561 MHz [%]	@1606 MHz [%]	Average (1561 - 1606 MHz) [%]
Sim	77.1	74.2	75.7
Meas	78.1	75.0	76.7

Table 6 - Total efficiency for GNSS with the TRIO mXTEND™ (NN03-310) of a small form factor PCB (Figure 8).

	Total Efficiency NN02-101					
	@2400 MHz [%]	@2500 MHz [%]	Average (2400 - 2500 MHz) [%]	@5150 MHz [%]	@5875 MHz [%]	Average (5150 - 5875 MHz) [%]
Sim	80.2	83.5	83.1	77.3	78.2	82.9
Meas	44.0	41.0	41.5	41.0	40.2	45.5

Table 7 - Total efficiency for Wi-Fi with the TRIO mXTEND™ (NN03-310) of a small form factor PCB (Figure 8).

Bear in mind that, for the Wi-Fi specific case the differences in antenna performance are mainly caused by the antenna location being underneath the batteries, coupling effects, and other RF components that are not typically considered in simulations. For this reason, it is important to analyze measurements in the same scenario as it is done in simulations and always expect a drop when manufacturing.

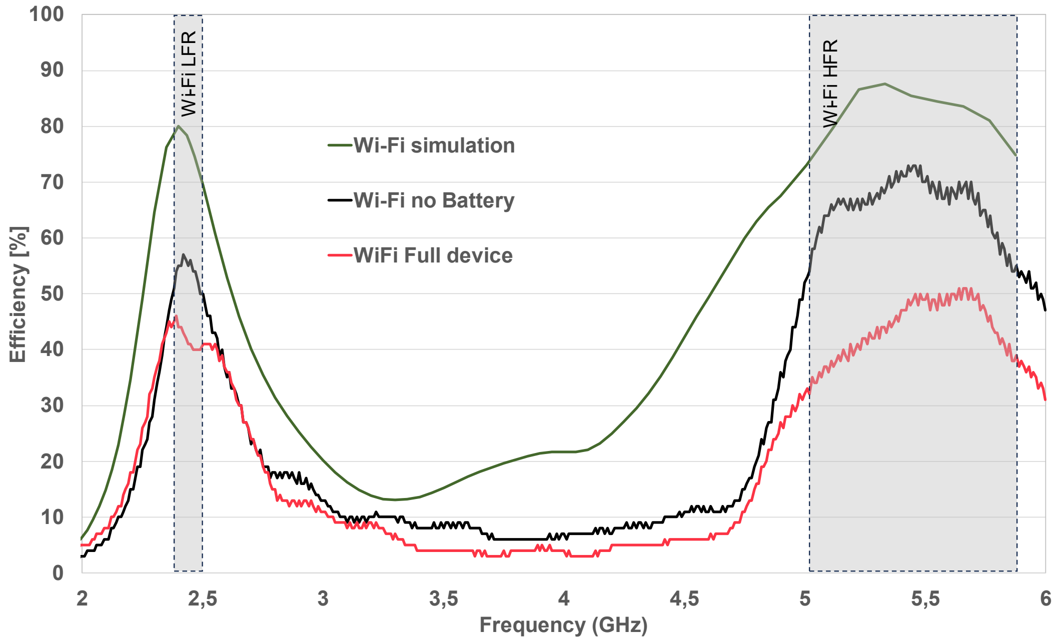


Figure 10 – Performance comparison for Wi-Fi between simulation on a bare PCB (green trace), full device without batteries (black trace) and full device with batteries (red trace). Each of them implemented with the NANO mXTEND™ (NN02-101).

2.3.1. WI-FI TOGETHER WITH CELLULAR AND GPS: HOW TO USE ANTENNA INTELLIGENCE CLOUD™ IN ANY DESIGN STAGE.

The digital twin tool, the Antenna Intelligence Cloud™, may be perceived as only suitable for feasibility analysis at the start of a project. However, it should be noted that it can also be applied to existing designs. In this example, a larger device is considered so the antenna components can be separated from each other to minimize the coupling and maximize the performance.

To show how close the Antenna Intelligence Cloud™ performance estimations are to real practical measurements, an example comparison is shown below. Find the measurement results from TRIO mXTEND™ in the nRF9160-DK board from Nordic Semiconductor compared with the results extracted from the Antenna Intelligence Cloud™ submitted for this example project using a 155 mm x 63 mm size PCB board covering LTE-M (699 – 960 MHz and 1710 – 2200 MHz).

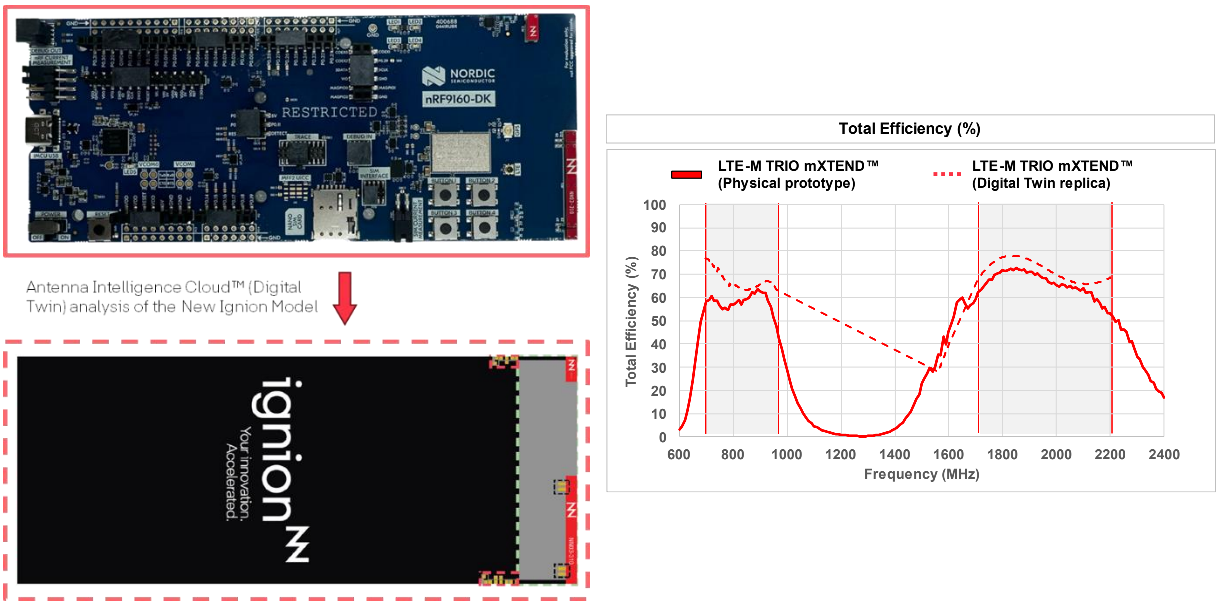
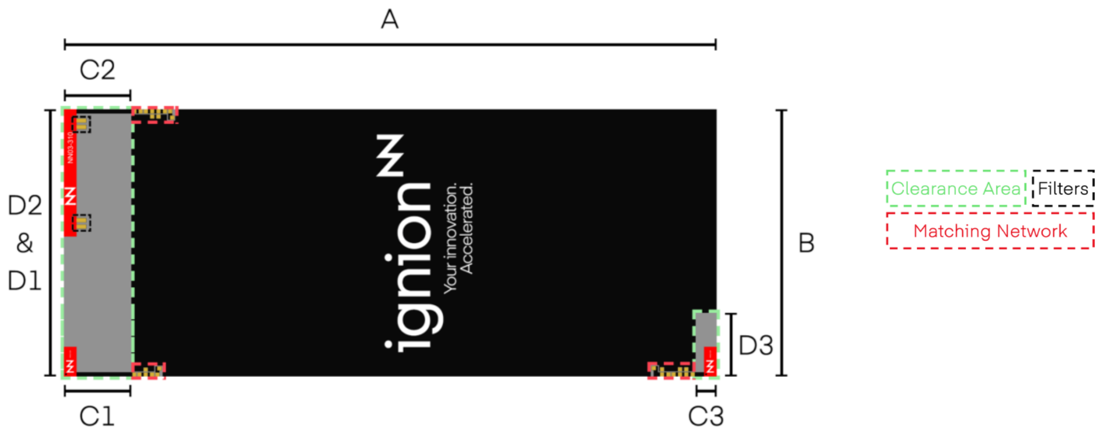


Figure 11 – nRF9160-DK a 155 mm by 63 mm board and its digital twin by the Antenna Intelligence Cloud™ tool (left). The total efficiency for LTE-M with the TRIO mXTEND™ (NN03-310) in the nRF9160-DK (red solid trace) and the total efficiency for LTE-M extracted from the Antenna Intelligence Cloud™ tool (red dashed trace).

This example demonstrates the accuracy of the Antenna Intelligence Cloud™ tool. It can be used to make fair estimation on future updates/addition of Wi-Fi in an existing device.



PCB		1: NN03-320		2: NN03-310		3: NN03-320	
Measure	mm	Measure	mm	Measure	mm	Measure	mm
A	155.0	C1	16.0	C2	16.0	C3	5.0
B	63.0	D1	63.0	D2	63.0	D3	15.0

Figure 12 – PCB layout extracted from the Antenna Intelligence Cloud™ delivering coverage at LTE-M with the TRIO mXTEND™ (NN03-310) (top-left), GNSS (bottom-left), and Wi-Fi dual band (right) with the DUO mXTEND™ (NN03-320).

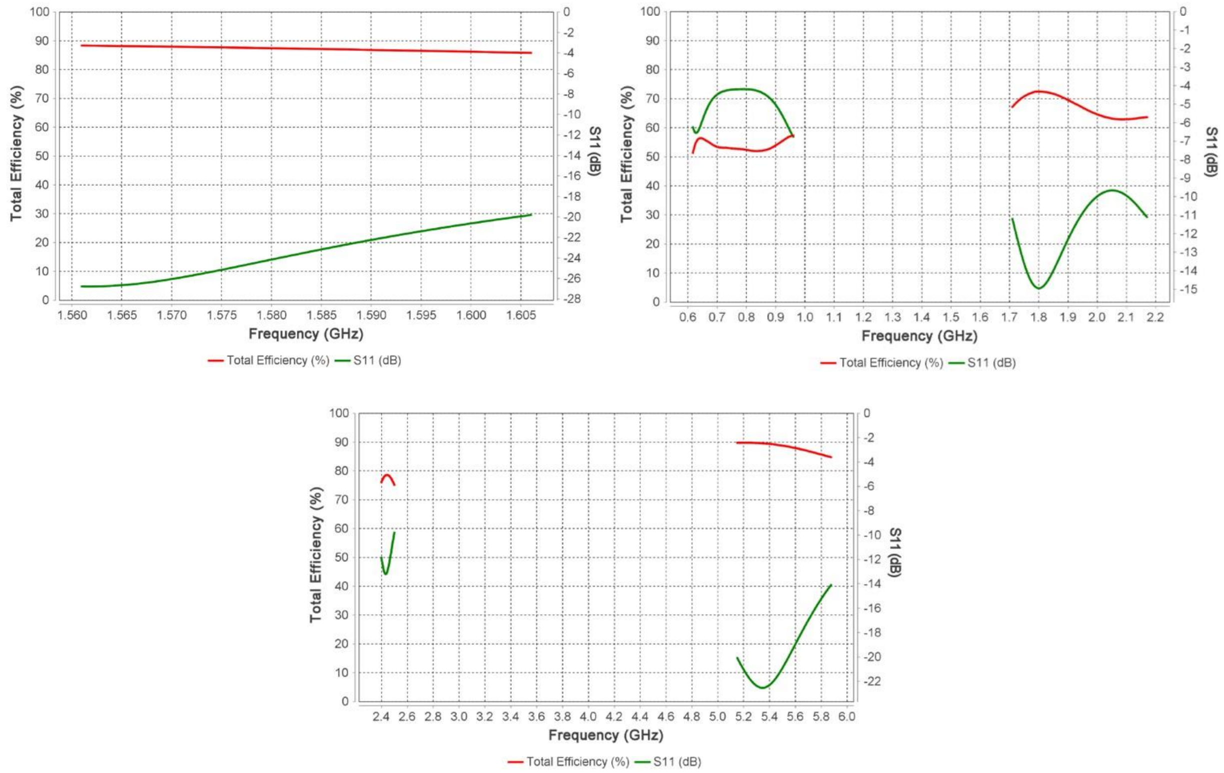


Figure 13 - Reflection Coefficient (dB) and Total Efficiency (%) of TRIO mXTEND™ (NN03-310) covering LTE-M (top-right), the DUO mXTEND™ (NN03-320) for GNSS (top-left), and the DUO mXTEND™ (NN03-320) for Wi-Fi dual band (bottom).

The results obtained for this first performance assessment show a good starting point on a bare PCB, covering the LTE-M bands from 698 MHz to 960 MHz and 1710 MHz to 2200 MHz (NN03-310), the GNSS bands from 1561 MHz to 1606MHz (NN03-320) and Wi-Fi Dual Band from 2400 MHz to 2500MHz and from 5150 MHz to 5875MHz (NN03-320) with the results shown in Figure 13.

2.3.2. RESULTS: S-PARAMETERS AND EFFICIENCY – MULTIBOARD EXAMPLE

Wi-Fi integration can be achieved in an IoT device with multiple boards connected, here is an example with the expansion board nRF7002-EK m mounted on a bigger Nordic Semiconductor. DK board.
Once the Gerber is verified (See section 2.4) and physical prototypes are produced, it is time for passive antenna measurements. In the measurement setup, the antenna boards remain unpopulated involving a SMA connector and a coaxial cable performing passive measurements. In this example the nRF7002-EK is assembled on different main boards (nRF9160-DK, the nRF21540-DK, and the nRF5340-DK), as illustrated in Figure 14. The results demonstrate that the DUO mXTEND™ (NN03-320) consistently delivers high performance for Wi-Fi dual band, as highlighted in Figure 15, Table 8. The matching network is easily optimized for each scenario.

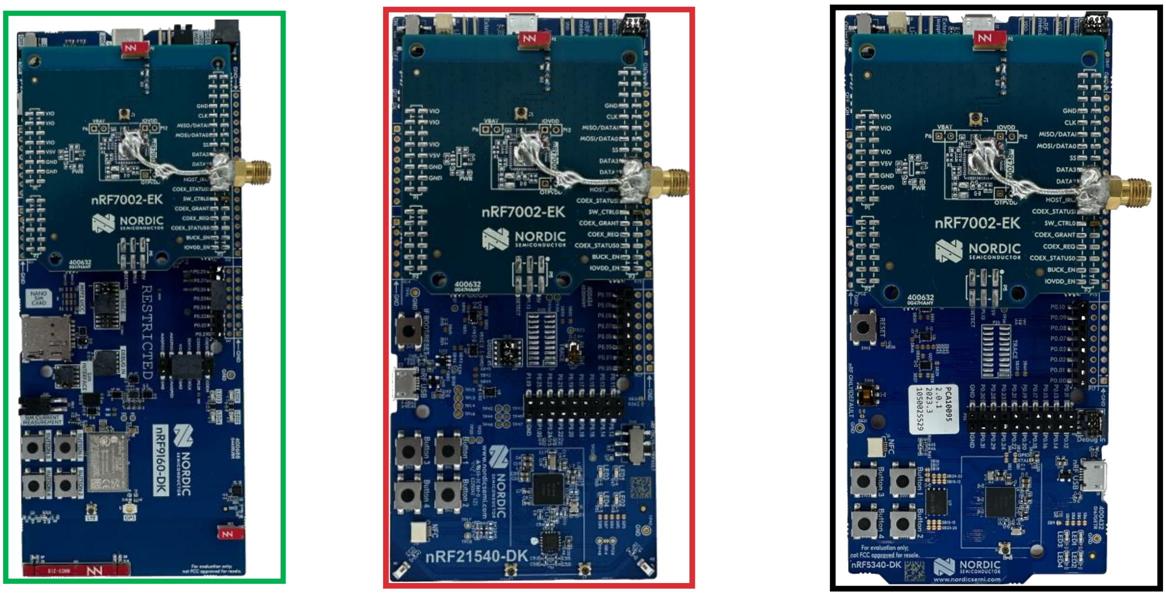


Figure 14 - Different set-up measured: Arduino + nRF9160-DK (left), Arduino + nRF21540-DK (middle), Arduino + nRF5340-DK (right).

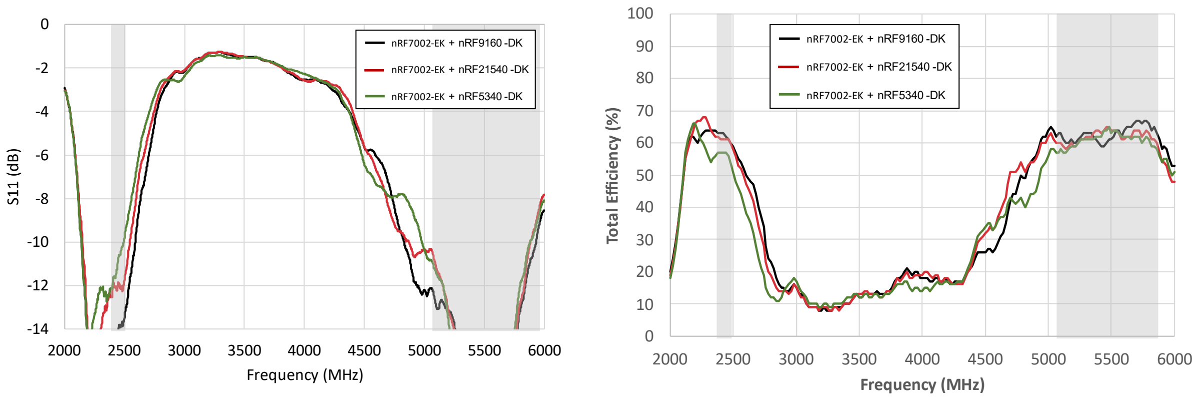


Figure 15 - S-parameters (left) and Total Efficiency (right) responses of set-ups introduced in Figure 14.

	Total Efficiency Average (2400-2500 MHz) [%]	Total Efficiency Average (5150-5875 MHz) [%]
nRF7002-EK + nRF5340-DK	61.3	62.8
nRF7002-EK + nRF9160-DK	60.3	62.0
nRF7002-EK + nRF21540-DK	55.3	61.1

Table 8 - Total efficiency at 2400MHz – 2500MHz and at 5150MHz – 5875MHz results of set-ups introduced in Figure 14.

2.4. PCB DESIGN RECOMMENDATIONS AND FREE GERBER REVIEW SERVICE

This section covers the main PCB design recommendations to guarantee a successful antenna integration.

2.4.1. GENERAL BEST PRACTICES FOR A SUCCESFUL ANTENNA INTEGRATION

By following these general hardware layout do's and don'ts, you'll ensure to exploit the full RF performance potential of your device.

Do’s:

Ensure proper clearance area: Virtual Antenna® components require a ground plane clearance under them on all layers. This will allow the Virtual Antenna® system to radiate at maximum capacity.

Size of the PCB ground plane matters: The size of the ground plane matters when using Virtual Antenna® components, especially for high performance communication use cases. A continuous ground plane will enhance the performance of the antenna system. Optimizing the size of the ground plane per your design constraints will be beneficial for the antenna performance.

50 Ω transmission line: The transmission line is the RF power conduit between your radio and your antenna. All the energy you generate travels to the antenna through the transmission line. Ensuring a 50 Ω transmission line characteristic impedance will avoid unnecessary losses in the system caused by impedance mismatches.

Don’ts:

Place parts over or under the antenna: such as metallic enclosure, battery, etc.

Nearby components on the PCB: Components placed too close together could cause problems for automated pick-and-place machines during assembly and testing.**Nearby Components on the PCB**: Components placed too close together could cause problems for automated pick-and-place machines during assembly and testing.

Improper via stitching: Improper via stitching can cause interference with electronic components on board and might shift the antenna resonance frequency.

Ignion offers a free-of-charge service in which all these points are checked in a Gerber file review. Below we explain the details of this sanity check procedure. As an example, a new antenna project is presented in which DUO mXTEND™ (NN03-320) is integrated for Wi-Fi dual band into the PCB nRF7002 Wi-Fi expansion kit from Nordic Semiconductor.

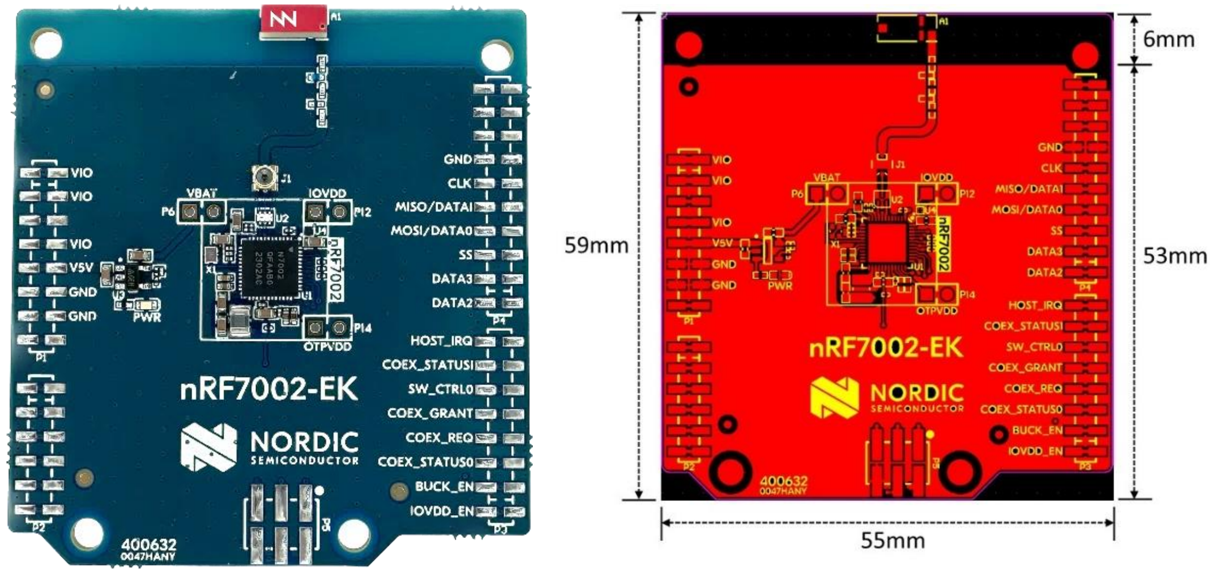
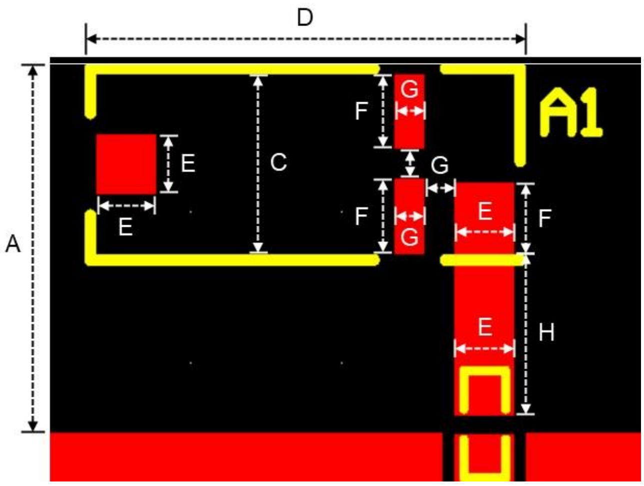


Figure 16 – Example of Gerber check - Original PCB and Gerber of nRF7002-EK

2.4.2. ANTENNA FOOTPRINT AND CLEARANCE AREA

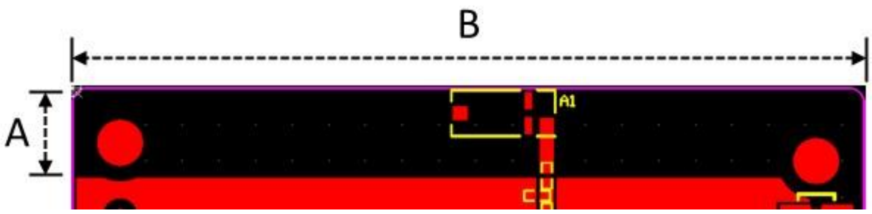
All Virtual Antenna® technology part numbers have a certain footprint, which can be found in the corresponding datasheets on the Ignion website. It is critical to measure all dimensions of pads and the distances between them so that the antenna can be soldered properly (Figure 17). An error in this procedure might cause electrical or mechanical problems.



	Measured [mm]
A	6.0
B	55.0
C	3.0
D	7.0
E	1.0
F	1.25
G	0.5
H	2.7

Figure 17 – Sanity check of antenna footprint

In addition, it is essential to leave a minimum clearance area around the antenna to optimize performance. This area should be free of copper, as illustrated in Figure 17. The significance of this area's impact on performance is demonstrated in several application notes available on Ignion's website (https://ignion.io/resources-support/technical-center/application-notes/).

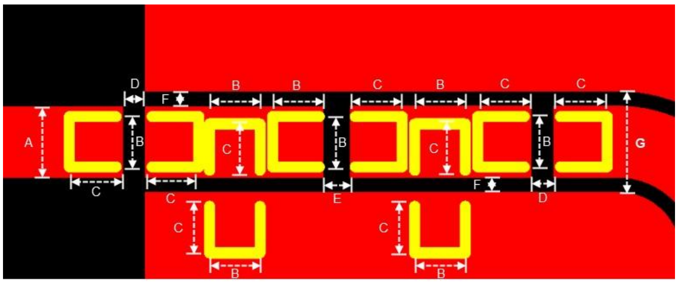


	Measured [mm]
A	6.0
B	55.0

Figure 18 - Sanity check of the antenna clearance area

2.4.3. MATCHING NETWORK FOOTPRINT

Ignion’s Antenna Intelligence Cloud™ reports contain the matching network topologies most convenient to match your antennas. Then, developers must draw a pad layout compatible with components 0402 and 0603 (Imperial code). Figure 19 shows the matching network and pad size defined in the Gerber of nRF7002-EK (Figure 16). It is also advisable to eliminate the ground plane beneath the area of the matching network pads. If there is a ground plane located less than 1mm below these pads, it can negatively impact performance. If you have any uncertainties, please feel free to contact Ignion engineers for additional recommendations.

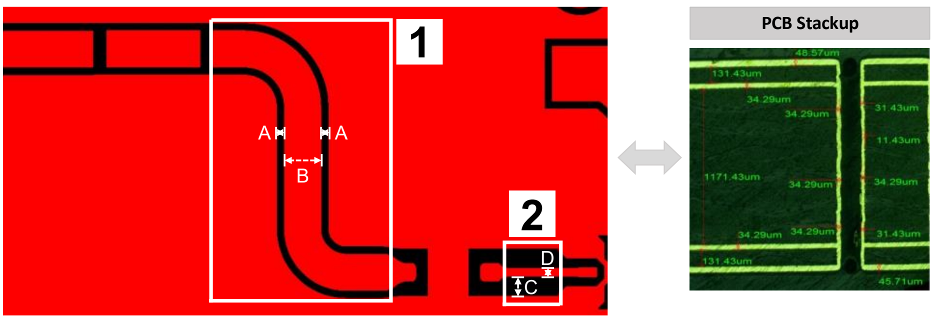


	Measured [mm]
A	1.0
B	0.7
C	0.75
D	0.3
E	0.37
F	0.2
G	1.4

Figure 19 – Matching network topology and pad size

2.4.4. CHARACTERISTIC IMPEDANCE

It is recommended to design the transmission line that connects the RF module input/output port to the matching network in such a way that it maintains a characteristic impedance as close to 50 Ω as possible. This helps minimize the insertion losses that may be introduced. Below, you can find an example of a transmission line as it appears in the nRF7002-EK (Figure 20).



	Measured [mm]
A	0.2
B	1.0
C	0.5
D	0.2
E	1.4
F	1.6

(no ground plane below the transmission line)

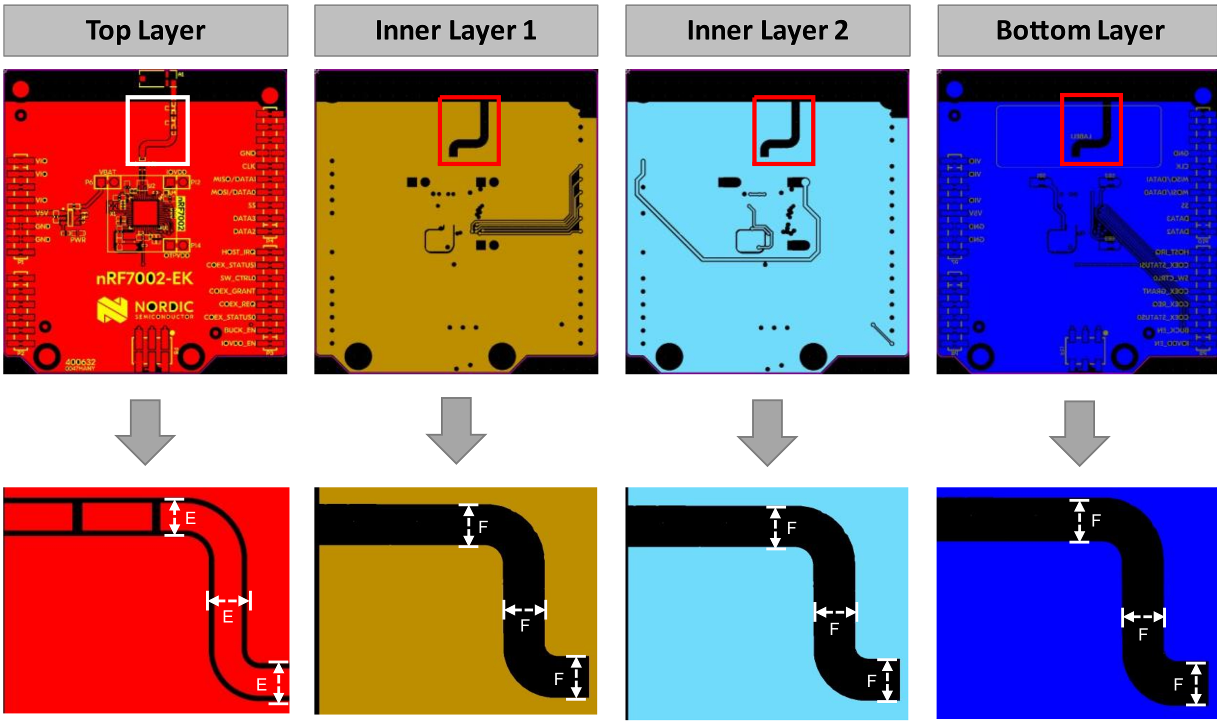


Figure 20 – Transmission line dimensions of nRF7002-EK.

3. SUMMARY

To summarize, Wi-Fi can easily be added to both new and existing device designs, even together with cellular & GNSS in small form factors. To ensure high RF performance in your Wi-Fi design, it is essential to adhere to the steps and design recommendations shown in this design guide. By following these guidelines, designers can ensure optimal antenna integration, reliable wireless communication, and efficient power management in their solutions. To recap the steps that should be followed:

1. **Understand the dependency between device board dimensions, clearance area and wireless performance.**

    - Utilize the Antenna Intelligence Cloud™ as a starting point to gain insights into the expected performance, while getting a head start with tailored design files and choosing the most suitable Virtual Antenna® component that aligns with the design requirements, considering factors such as frequency range, antenna performance, and size constraints.

    - If there is a complex design (either small form factor or with several wireless standards), it is recommended to conduct a simulation of the fully populated PCB, considering all integrated elements of the final device.

2. **Share your Gerber file with Ignion for a free review ensuring the device is following the design recommendations provided by the Antenna Intelligence Cloud™ tool.**

3. **Produce hardware prototype and compare with simulated results to assess the need for fine-tuning of the matching network for potential further maximizing of the antenna performance.**

4. **Final device optimization through the adjustment of the matching network**.

By following these steps, designers can optimize the antenna integration, resulting in enhanced performance, reliable communication, and successful deployment of Wi-Fi systems.

4. DO YOU NEED MORE HELP? OVERVIEW OF IGNION SUPPORT SERVICES

Virtual Antenna® technology makes antenna optimization accessible for any electronics engineer, however if additional support is needed during product development, the Ignion team is ready to help in every step of the design journey.

https://ignion.io/resources-support/technical-center/engineering-support/.

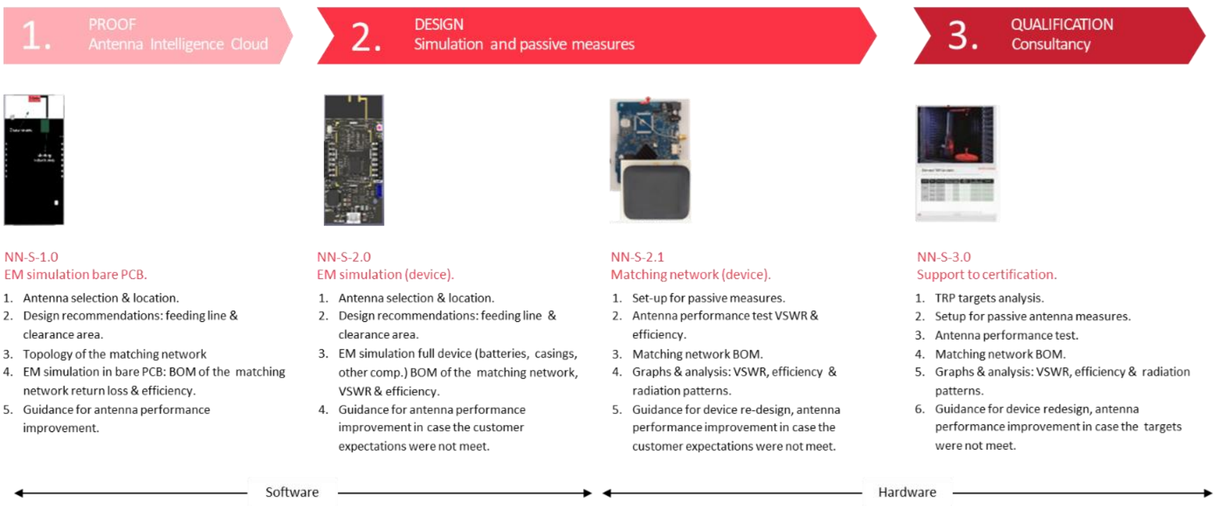


Figure 21 - Ignion services for antenna integration.

The Ignion products based on its proprietary Virtual Antenna® technology are protected by one or more of the following patents: https://ignion.io/files/Patent-list-NN.pdf within this document which is property of Ignion and is subject to change without prior notice. Information is provided "as is" and without warranties. It is prohibited to copy or reproduce this information without prior approval.

Ignion is an ISO 9001:2015 certified company. All our antennas are lead-free and RoHS and REACH compliant.

ISO 9001: 2015 Certified

RoHS 2002/95/EC

ignion

Your innovation.
Accelerated.

Contact:
support@ignion.io
+34 935 660 710

Barcelona

Av. Alcalde Barnils, 64-68 Modul C, 3a pl.
Sant Cugat del Vallés
08174 Barcelona
Spain

Shenzhen

Topway Information Building, Binhai Avenue,
Nanshan District, Nº 3369 – Room 2303
Shenzhen, 518000
China

+86 13826538470

Tampa

8875 Hidden River Parkway
Suite 300
Tampa, FL 33637
USA