ignion

Your innovation.
Accelerated.

Smart Metering Design Guide

STEP-BY-STEP GUIDE TO INTEGRATE AND OPTIMIZE THE ANTENNA IN YOUR SMART METER.

SMART METERING DESIGN GUIDE

CHOOSE VIRTUAL ANTENNA® TECHNOLOGY FOR SUPERIOR CONNECTIVITY AND RELIABLE DATA ACCURACY.

Why is Virtual Antenna® technology your best choice for smart metering devices?

- Robust and reliable connectivity even in harsh environments.
- Easy prediction of final performance including the influence of materials in proximity.
- Easy to integrate multiple radios and multi-board designs.
- Tunable single SKU antenna solution for full smart meter portfolio.
- Low maintenance through the long device and battery lifetime.
- Cost-effective in a small package enabling space-optimized design.

What is in this Design Guide?

- **Step-by-step** walkthrough of the recommended antenna design process for a typical smart meter device using low-power cellular technologies (LTE-M and NB-IoT).
- How to achieve the best antenna performance in **multi-board PCB designs**, including the importance of connection location between the communication board and the main meter board.
- Guide on how to **combine an LTE-M and an LTE 450** antenna solution in one board using a single wireless module output.

Frequency band configurations used in this design guide:

- In this guide TRIO mXTEND™ (NN03-310) and ALL mXTEND™ (NN02-220) are configured for the following cellular bands:
    - LTE450 band 31, 72: 450 - 470 MHz.
    - LTE-M/NB-IoT band 3, 8, 12, 13, 14, 17, 20, 28: 698 - 960 MHz.
    - LTE-M/NB-IoT band 2, 4, 7, 11, 17, 18, 19, 20, 21, 22, 23, 25, 26, 27, 28, 29, 30, 32, 34, 35, 36, 37, 38, 39: 1710 - 2200 MHz.

Every design is unique, to evaluate the performance in any frequency and PCB size combination use our free online tool **Antenna Intelligence Cloud™:** https://ignion.io/antenna-intelligence/.

IGNION CUSTOMER STORY

COST-EFFECTIVE AND SPACE-EFFICIENT ANTENNA FOR SMART METERS

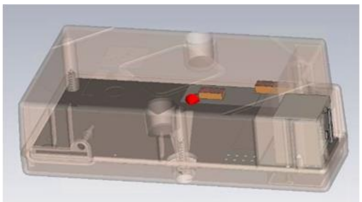
THE COMPANY. EDMI, a global leader in smart metering solutions, committed to high-quality, reliable products, headquartered in Singapore.

THE CHALLENGE. Integrating a robust RF solution inside a smart meter with all the high voltage, high current electronics surrounding it is typically not a trivial task. In addition, the electro-mechanical emission of the RF device must not impact the accuracy of the meter. Using the Virtual Antenna® technology, EDMI, was able to address these challenges through the following simple steps.

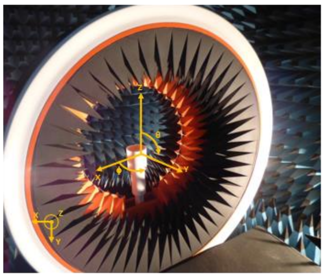
THE SIMPLIFIED JOURNEY – Design steps for the EDMI smart meter.

1. FEASIBILITY Antenna Intelligence Cloud™	2. DESIGN 3D Simulation	3. TESTING Anechoic chamber

Fast feasibility and design on bare PCB



Full device design verified in EM simulation



Finished prototype fine-tuned and tested

THE SOLUTION. EDMI integrated the DUO mXTEND™ Virtual Antenna® component on the RF Mesh Network Interface Card operating in the 900 MHz and 2.4 GHz frequency bands. The measured performance was as high as an external SMA connector paddle antenna but with the additional advantages of space-saving, cost-effectiveness, and secure robustness against external damages as it cannot be broken off nor de-tuned.

"Ignion's antenna technology is groundbreaking; couple that with their exemplary technical support provided, EDMI will look to Ignion for our future RF solutions".
Mark Foo, Assistant Vice President, EDMI Limited.

ACHIEVEMENTS

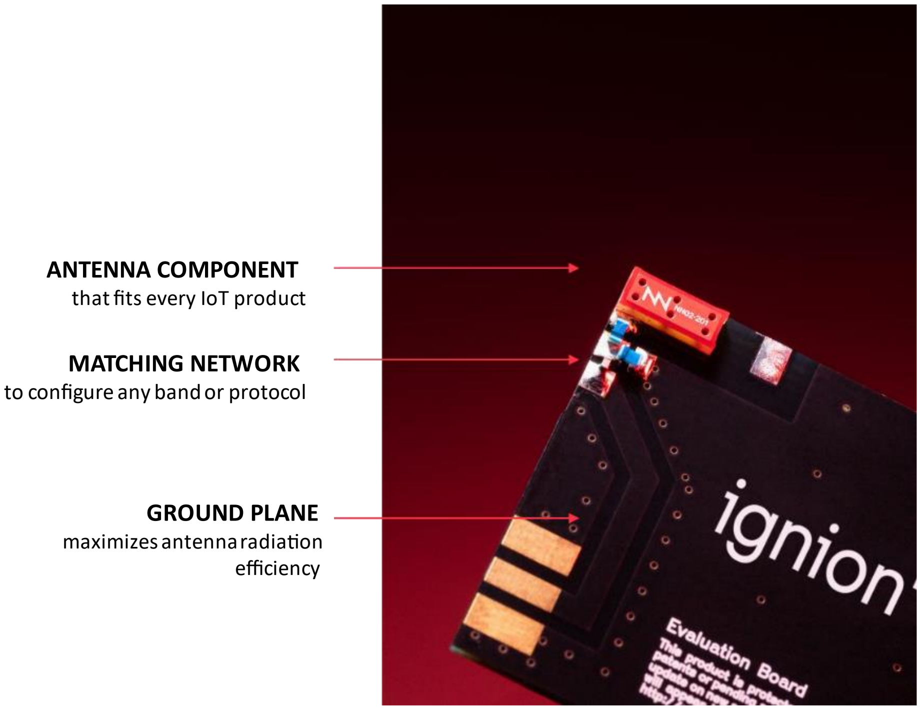
- More robust, cost-effective and compact than external antenna.
- Fast one-week turnaround on fine tuning RF matching circuit.
- 2 frequency bands covered with 1 antenna component.
- Optimal RF performance in 1st iteration.

1. SMART METERING DESIGN JOURNEY WITH VIRTUAL ANTENNA®

1.1. VIRTUAL ANTENNA® TECHNOLOGY INTRO

The Virtual Antenna® technology is the smallest and most versatile antenna option available on the market, delivering robust high performance.

The Virtual Antenna® technology consists of 3 main elements:



Antenna component: The Virtual Antenna® component is a small, wideband and/or multiband antenna component, also called antenna booster. This non-resonant 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® components to support a frequency range from 400 MHz to 10 000 MHz.

Matching network: A series of small low-cost capacitors and inductors placed between radio and antenna component to select frequency bands of operation and maximize RF energy transfer. The frequency band(s) can easily be changed at any stage, just by modifying the matching network component.

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

1.2. OVERVIEW OF DESIGN STEPS WITH VIRTUAL ANTENNA® TECHNOLOGY

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 Ignion customer services where an experienced technical team is ready to assist in all stages of the design.

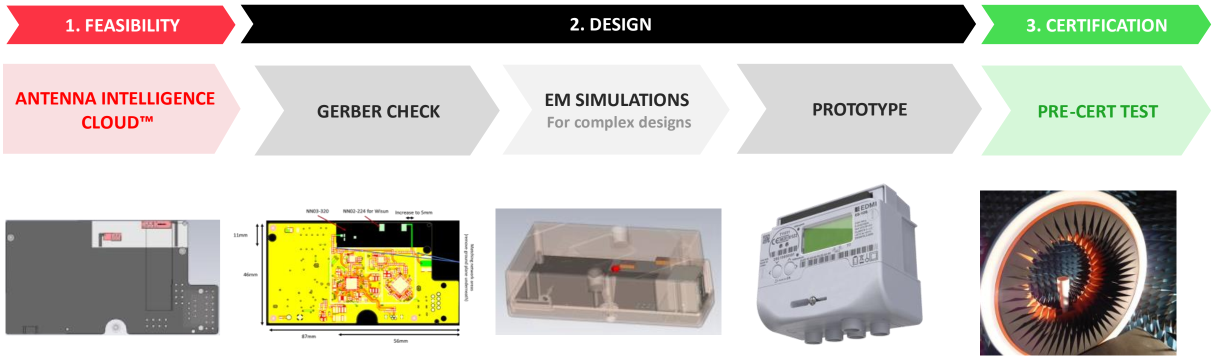


Figure 1 – Virtual Antenna® design journey steps for successful antenna integration.

Step 1 – Assess Feasibility (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, and design recommendations such as antenna placement and clearance area on a bare PCB. In addition, tailored antenna design files are provided that can simply be imported into any PCB design tool (Altium, Solidworks, Cadence, etc.). By using the Antenna Intelligence Cloud™ from the beginning, designers will know if their devices are going to meet their RF requirements before building any physical hardware. In this way, designers can take go or no-go decisions in the early stages of the design cycle minimizing cost, risks, and time to market. Often changes in antenna component placement, clearance area size or PCB dimensions can easily solve performance issues, which will be difficult to adjust later as the design hardens.

Step 2 - Build Gerber files: Building the PCB design files (Gerbers) with optimal antenna performance from the get-go is easy due to the tailored templates automatically provided by Antenna Intelligence Cloud™ along with the tailored design recommendations.

Step 3 - EM simulation: Design file validation with an Electro-Magnetic (EM) simulation of the full device considering every component influencing RF performance such as objects or different materials near the antenna (onboard components, batteries, casing, surrounding materials, etc.) and ensuring RF performance requirements are met. The simulation allows the designer to evaluate potential design changes/requests and their impact on the antenna performance.

Step 4 - Final Gerber sanity check (free Ignion service): Before initiating the physical prototype production a fast (and free) check is done ensuring that the antenna, matching network layout, and key design recommendations are correctly followed. A quick extra set of experienced eyes can catch mistakes and save an extra prototype iteration. The Gerber file is simply sent to the Ignion team which will respond with a short Gerber review report.

Step 5 - Produce prototype, test and fine-tune: Verifying that prototype performance results are aligned with expectations and fine-tuned if necessary. If needed, a VNA (Vector Network Analyzer) is used to verify the resonance frequency/S1 parameters of the antenna system, and simple adjustments in the matching network values can be done.

Step 6 - Certification pre-test: Performing over-the-air (OTA) tests with the antenna system and radio module to ensure the device is meeting certification requirements.

1.3. ANTENNA COMPONENTS FOR SMART METERING

The TRIO mXTEND™ (NN03-310) and the ALL mXTEND™ (NN02-220) Virtual Antenna® components are popular in smart metering systems due to their reliable performance in the low frequency bands. Using separate antennas configured for separate frequency bands maximizes performance while minimizing interference, resulting in robust RF performance.

TRIO mXTEND™ (NN03-310)
Component dimensions: 30.0 x 3.0 x 1.0 mm
Can be configured for: 400 – 8000 MHz.
More info.

ALL mXTEND™ (NN02-220)
Component dimensions: 24.0 x 12.0 x 2.0 mm
Can be configured for: 400 – 8000 MHz.
More info.

Virtual Antenna® components used in this design guide have the following characteristics when implemented on a PCB:

- Linearly polarized.
- Omnidirectional radiation patterns.
- Withstanding temperatures from -40 to + 125 ºC.
- Impedance of antenna solution: 50Ω.

1.3.1. BOARD CONFIGURATIONS USED ALONG THE GUIDE

When designing a smart meter, various types of designs are commonly used to meet specific requirements, such as single-board designs, or multi-board/modular designs. A separate board for communications in smart meters enables modular scalability, flexibility, and interference mitigation. It enables easy upgrading or swapping of communication modules without affecting the core metering functions. This multi-board configuration provides flexibility to adapt to changing communication standards or specific regional requirements. Meanwhile, a single-board integrated architecture simplifies overall system design, streamlines production, and lowers costs.

Consider your project's individual needs to find the best way to attain flexibility, performance, and cost-efficiency in your smart meter design.

PCB dimensions covered in the single-board design:

- Ground plane size analysis: from 200 mm x 200 mm to 100 mm x 100 mm.

PCB dimensions covered for multi-board design:

- Communication board: 60 mm x 90 mm with clearance area of 14 mm x 60 mm.
- Main board: 125 mm x 165 mm. Connecting the ground of this board extends the ground plane and the overall antenna performance.

PCB dimensions covered in surrounding materials impact analysis:

- Evaluation board size of 50 mm x 107 mm and clearance area of 16 mm x 45 mm.

2. DETAILED STEP-BY-STEP GUIDE

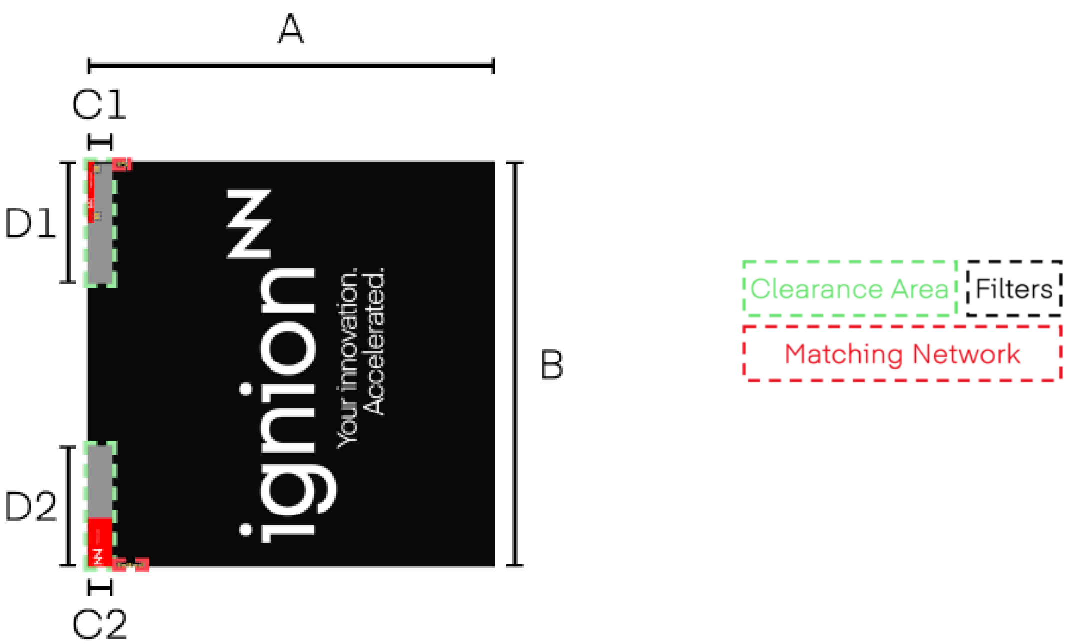
2.1. STEP 1: FEASIBILITY – USING ANTENNA INTELLIGENCE CLOUD™

The feasibility evaluation starts with a simple and fast performance estimation on a bare PCB without any other components, casings, batteries, etc. These results are provided by Ignion's Antenna Intelligence Cloud™, a unique online tool that delivers antenna performance results, suggested matching network topologies, and BoM (Bill of Materials). By simply filling in the on-line form and choosing the frequency band of operation and desired PCB size, the results are guaranteed within 24 hours, but typically are generated in minutes.

Below are the results extracted from the Antenna Intelligence Cloud™ submitted for this example project using a 200 mm x 200 mm size PCB board with operation in LTE 450 and LTE-M bands. The solution proposed uses two separate antenna components, TRIO mXTEND™ (NN03-310) for LTE 450 and ALL mXTEND™ (NN02-220) for LTE-M. The antenna components are separated to minimize the coupling and maximize the performance of both.

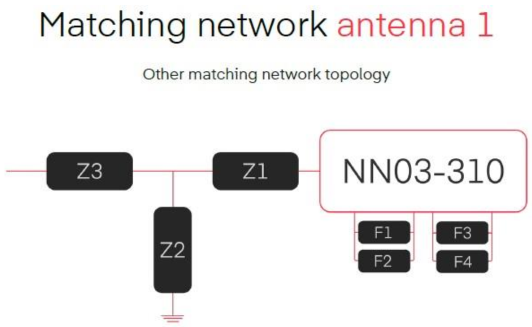
Best antenna placement on your PCB

Sketch of the proposed antenna placement and the recommended clearance area for the Virtual Antenna® component.



PCB		1: NN03-310		2: NN02-220	
Measure	mm	Measure	mm	Measure	mm
A	200.0	C1	12.0	C2	12.0
B	200.0	D1	60.0	D2	60.0

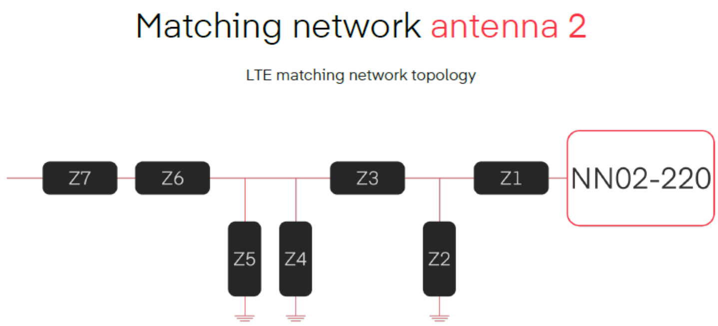
Figure 2 – PCB layout from the Antenna Intelligence Cloud™. Delivering coverage at LTE 450 with the TRIO mXTEND™ (NN03-310) and LTE-M with the ALL mXTEND™ (NN02-220).



Comm. Standard	Component	Value	Part Number	Manufacturer
LTE450	F1	0 Ohm		
	F2	0 Ohm		
	F3	0 Ohm		
	F4	0 Ohm		
	Z1	75nH	LQW18AN75NG80	Murata
	Z2	open		
	Z3	24nH	LQW18AN24NG80	Murata

The electronic component values correspond with the matching network when implemented on a bare PCB. These values may need further tuning and optimization when additional elements such as batteries, plastic covers, connectors, displays, etc. are added to your final device.

Figure 3 – Matching network topology and BoM for LTE 450 from the Antenna Intelligence Cloud™.



Comm. Standard	Component	Value	Part Number	Manufacturer
LTE	Z1	3.8nH	LQW15AN3N8G80	Murata
	Z2	11nH	LQW18AN11NG80	Murata
	Z3	1.5pF	GJM1555C1H1R5WB01	Murata
	Z4	7.2nH	LQW18AN7N2G80	Murata
	Z5	1.3pF	GJM1555C1H1R3WB01	Murata
	Z6	2.2pF	GJM1555C1H2R2WB01	Murata
	Z7	1.6nH	LQW15AN1N6C80	Murata

The electronic component values correspond with the matching network when implemented on a bare PCB. These values may need further tuning and optimization when additional elements such as batteries, plastic covers, connectors, displays, etc. are added to your final device.

Figure 4 - Matching network topology and BoM for LTE-M from the Antenna Intelligence Cloud™.

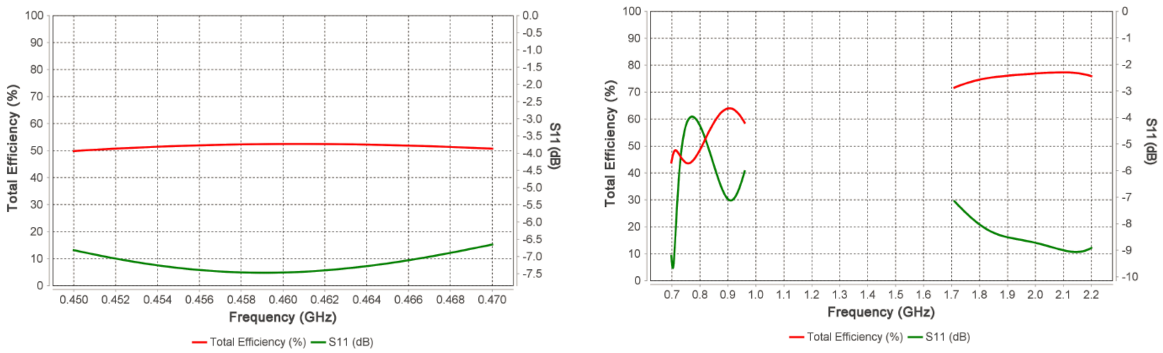


Figure 5 – Reflection Coefficient (dB) and Total Efficiency (%) of TRIO mXTEND™ (NN03-310) and the ALL mXTEND™ (NN02-220).

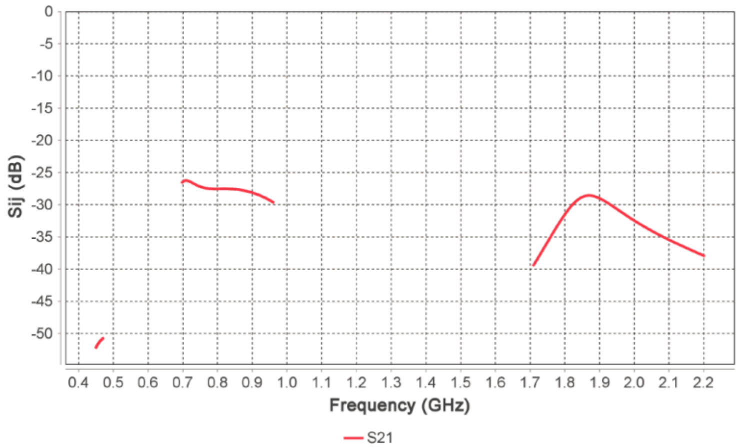


Figure 6 – Coupling effect between TRIO mXTEND™ (NN03-310) and the ALL mXTEND™ (NN02-220).

The results obtained for this first performance assessment show a good starting point on a bare PCB, covering the LTE 450 bands from 450 MHz to 470 MHz and LTE-M bands from 698 MHz to 960 MHz and 1710 MHz to 2200 MHz with the results shown in Table 1 and Table 2 below.

	TRIO mXTEND™ (NN03-310)		
Frequency (MHz)	450	470	Avg 450 - 470
Total efficiency (%)	49.8	50.7	51.7

Table 1 – Total efficiency simulated for the LTE 450 band with the TRIO mXTEND™ (NN03-310) of a PCB of 200 mm x 200 mm (Figure 2).

	ALL mXTEND™ (NN02-220)		
Frequency (MHz)	698	960	Avg 698 - 960
Total efficiency (%)	43.9	58.6	54.1
Frequency (MHz)	1710	2200	Avg 1710 - 2200
Total efficiency (%)	71.7	75.9	75.9

Table 2 - Total efficiency simulated for LTE-M with the ALL mXTEND™ (NN02-220) of a PCB of 200 mm x 200 mm (Figure 2).

Note that both the ALL mXTEND™ (NN02-220) and the TRIO mXTEND™ (NN03-310) can be used to cover all the bands. It means that you can use the same part number to cover both LTE 450 (450 – 470 MHz) and LTE-M (698 – 960 MHz and 1710 – 2200 MHz). It is recommended to evaluate the specific performance targets compared to the cost of a design with a single or two separate antenna components.

The Antenna Intelligence Cloud™ results enable evaluation of the antenna performance and whether the design is able to meet the device requirements or if the antenna solution needs adjustments. The results obtained can be utilized to assess the feasibility of meeting the Total Radiated Power (TRP) requirements set by the cellular operator certification requirements. TRP requirements may differ depending on the network carrier and operational region. Employ **Equation 1** described in **section 2.6.2**, to estimate the TRP based on the antenna's total efficiency and the output power of the RF module. Ensure that the calculated TRP exceeds the minimum TRP requirements specified by the cellular operator.

2.2. STEP 2: BUILD YOUR DESIGN FILE

When starting the device design process, carefully consider the design recommendations outlined in the Antenna Intelligence Cloud™ report. Furthermore, do consult the accompanying design files and customized templates included with each Antenna Intelligence Cloud™ report. These are specifically tailored to your design requirements and can easily be copied into your PCB design tool.

Later on, to ensure that the simulated antenna performance results are preserved in the physical prototype, it is important to adhere to the following design recommendations.

1. **Antenna placement**: Place the antenna as far as possible from other components, such as LCDs, batteries, and connectors, especially those components and covers or housings with metallic characteristics (see suitable placement in Figure 2).

2. **Clearance area:** Keep the clearance area around the antenna component as recommended in Figure 2. The clearance area must be free from electronic components, traces and ground plane in all PCB layers including the underside of the PCB directly underneath the mounted antenna area. As a general rule of thumb, the larger the clearance area the better the performance.

3. **PCB layers**: Ensure a continuous ground plane layer in at least one layer of your PCB design. Avoid any ground plane or conductive trace underneath the matching network pads area.

4. **Matching network**: Arrange pads for all the matching networks to host 0402/0603 SMD components if possible. Place pads as close as possible to the antenna feed point and within the ground plane area to enable an effective tuning of the matching network components throughout your design. Use **high Q and tight tolerance** matching network components.

5. **Transmission line and RF module**: Design your transmission line connecting the matching network to your RF module (see scheme connecting in Figure 3) so that its characteristic impedance is 50Ω. The output impedance of your RF module must be 50Ω as well. Place your RF module as close as possible to the matching network to reduce the losses introduced by the transmission line.

2.3. STEP 3: SIMULATIONS OF DIFFERENT DESIGNS

In this section, the typical most influential factors are simulated to show examples of the impact on the antenna performance. Key elements analyzed are:

- Ground plane size.
- Multi-board connection location.
- Multi-radio design in one board.
- Distance to surrounding materials (metal, concrete, and wood).

2.3.1. LTE 450 GROUND PLANE OPTIMIZATION

In this section, a parametric analysis study is presented to show the impact of changing the board length dimension. The resonant frequency of any radiation system is directly related to its size in terms of operating wavelength. Six different PCB lengths are analyzed (Figure 9).

This section shows the impact of changing the board length when the operation in the LTE 450 band (450 MHz to 470 MHz).

The first boards studied are single PCB configurations with the following characteristics:

- The PCB width dimension is 100 mm.
- The PCB length dimension changes in steps of 20 mm (between 100 - 200 mm).
- The clearance area dimensions are 40 mm x 25 mm.

The design includes the TRIO mXTEND™ (NN03-310) antenna component located in the corner covering 450 – 470 MHz.

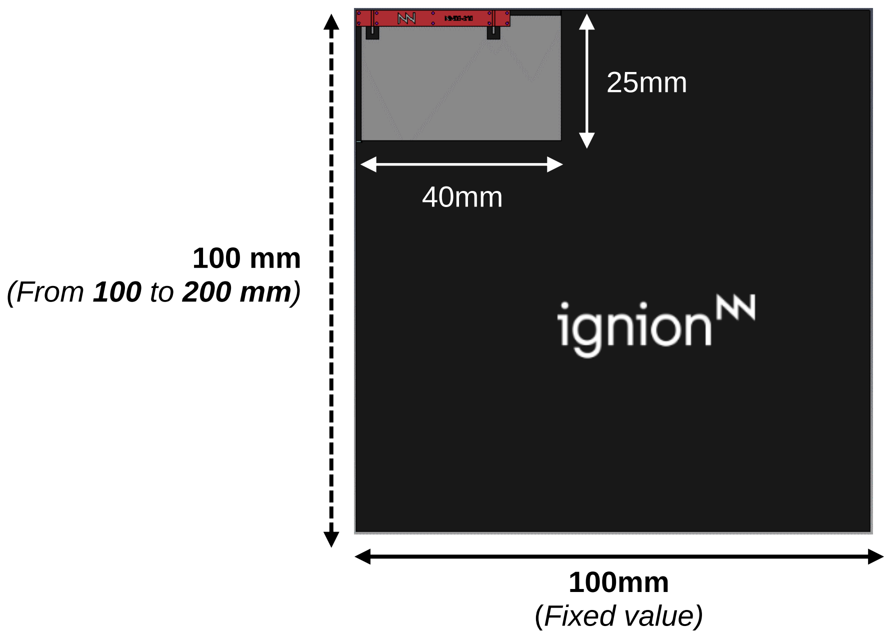


Figure 7 – Initial board configuration under analysis.

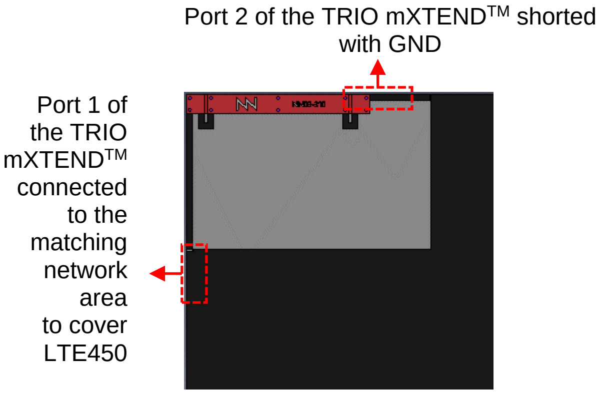


Figure 8 – PCB clearance area zoom in.

Note that the second port of the TRIO mXTEND™ (NN03-310) antenna component relates to the ground plane to provide a better performance 450 – 470 MHz.

On the other side of the antenna component, the matching network is located tuning the response to 450-470MHz.

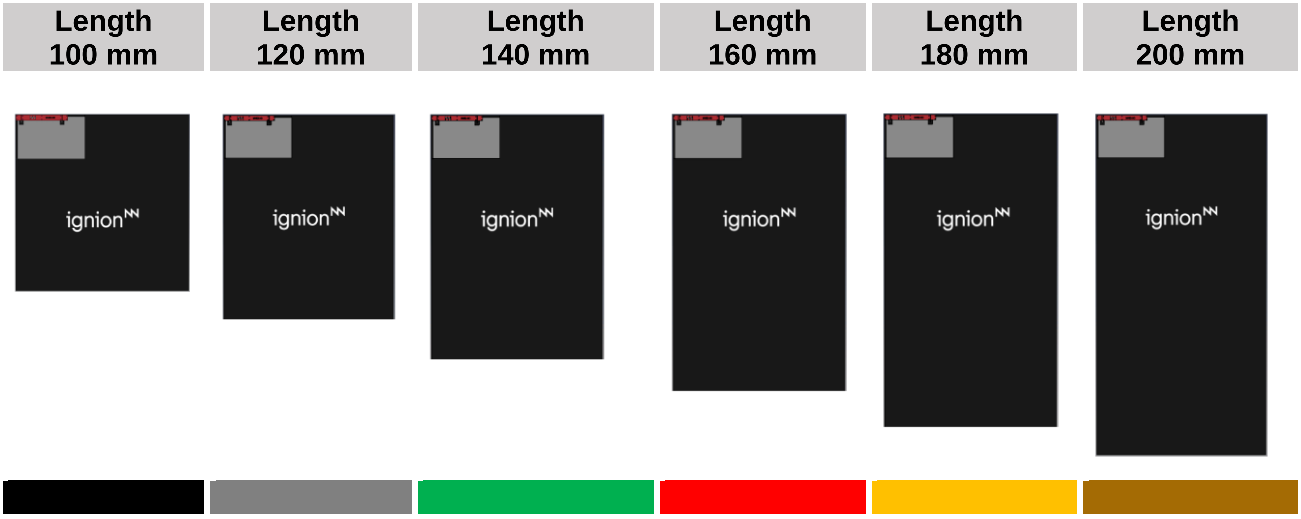


Figure 9 – All the PCB lengths analyzed (100 mm x 100 mm, 120 mm x 100 mm, 140 mm x 100 mm, 160 mm x 100 mm, 180 mm x 100 mm and 200 mm x 100 mm).

The S-parameters and the total efficiency of each of the PCB's shown in Figure 9 are evaluated in Figure 10 - Figure 11 and Table 3. Note that the total efficiency for LTE450 is considerably higher when the PCB is longer.

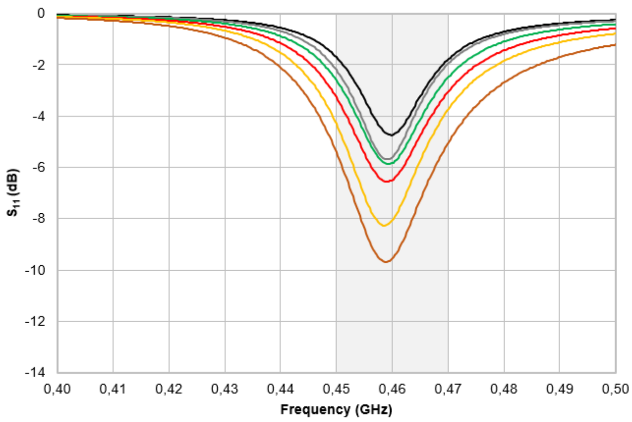


Figure 10 – S-parameters of the different evaluation boards ranging from PCB 100 mm x 100 mm to PCB 200 mm x 100 mm for LTE 450.

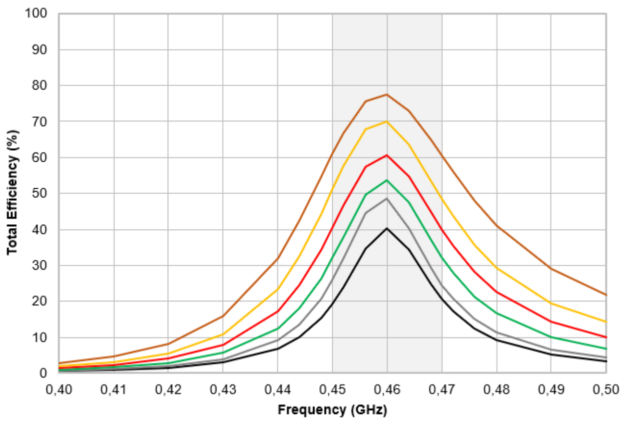


Figure 11 – Total efficiency of the different evaluation boards ranging from PCB 100 mm x 100 mm to PCB 200 mm x 100 mm for LTE 450.

	Total Efficiency @450 MHz [%]	Total Efficiency @470 MHz [%]	Total Efficiency Average (450-470 MHz) [%]
PCB 100 x 100	19.1	20.7	28.2
PCB 120 x 100	25.9	24.5	35.0
PCB 140 x 100	31.8	32.3	41.4
PCB 160 x 100	40.3	40.0	49.2
PCB 180 x 100	51.0	48.4	58.8
PCB 200 x 100	60.9	60.5	68.4

Table 3 – Total efficiency for the LTE 450 bands with the TRIO mXTEND™ (NN03-310).

There is a direct relation between ground plane length and antenna performance. The longer the ground plane is, the higher the achieved performance.

2.3.2. RESOURCES FOR GROUND PLANE DIMENSION INFLUENCE IN LTE-M/NB-IOT BANDS

The same ground plane dimensions influence is seen when operating in the cellular bands like LTE-M and NB-IoT.

Two recommended application notes, that show the PCB dimension influence in the cellular LTE-M/NB-IoT bands:

TRIO mXTEND™ (NN03-310)

Application note:
PCB size influence on performance with TRIO mXTEND™

Application Note Link

ALL mXTEND™ (NN02-220)

Application note:
PCB size influence on performance with ALL mXTEND™

Application Note Link

2.3.3. LTE 450 MULTI-BOARD DESIGN – OPTIMAL CONNECTION POINT LOCATION

In this section, we focus on the multi-board configuration that allows flexibility for enhanced communication performance in smart metering. In a multi-board smart meter design, the location of the ground plane connection between both boards is crucial. The connections are directly related to the enlargement of the ground plane and the connection location directly impacts the antenna performance.

In this section, a parametric analysis study is presented to show the impact of changing the location point of the connection between two boards. One board is working as a communication board and includes the TRIO mXTEND™ (NN03-310) antenna component. The second board is the main board (typically with the metering functionality). Nine different locations of the connection point are analyzed. The locations are organized in three sections, each related to one side of the board.

The two PCB examples and their characteristics:

- Communication board: 60 mm x 90 mm.
- Clearance area on the communication board: 60 mm x 14 mm.
- Main board: 125 mm x 165 mm.
- 14 Pin connector between communication and main board.

Communication Board

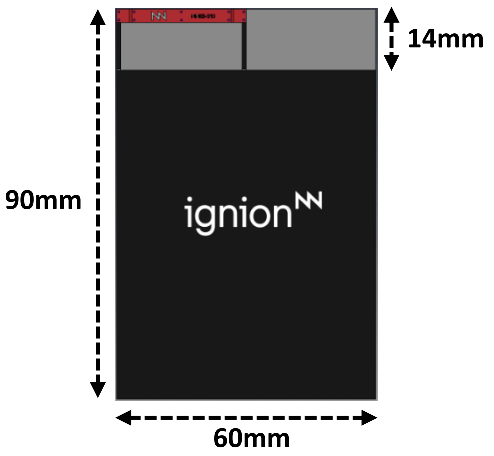


Figure 12 - Communication board with dimensions of 60 mm x 90 mm.

Main Board

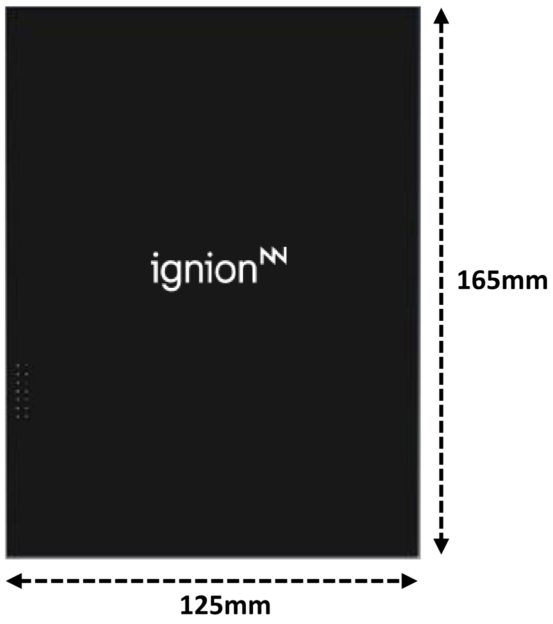


Figure 13 - Main board with dimensions of 125 mm x 165 mm.

The grounding connection between the communication and main boards has been made using two of fourteen pins.

Connection between boards

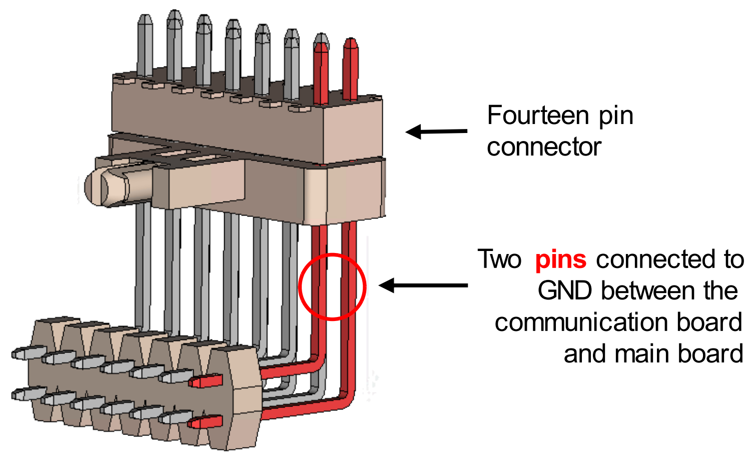


Figure 14 – Pin connector used for connecting the communication and main boards.

Multi-board Configuration

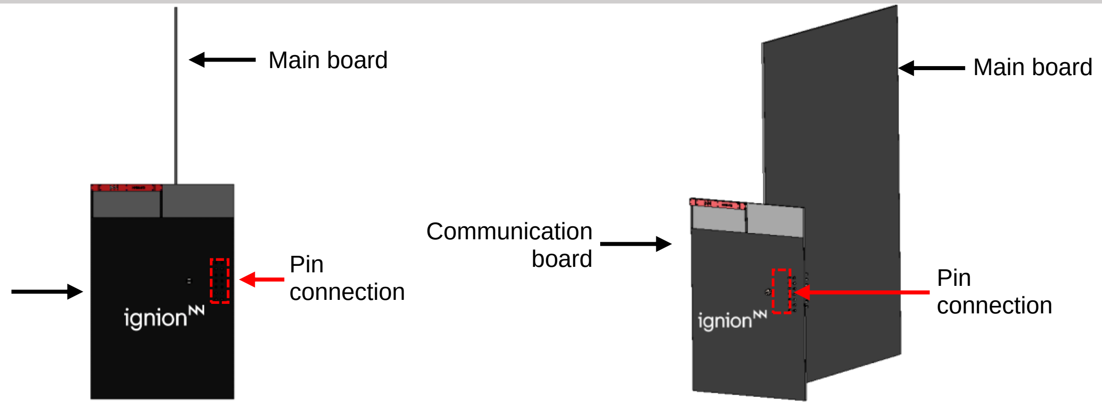


Figure 15 - Configuration of communication and main boards connected through a pin connector located at the communication right side.

2.3.4. RECOMMENDED CONNECTION LOCATION BETWEEN COMMUNICATION AND MAIN BOARD

Why does the location of the connection point between both boards matter?

When dealing with a design that integrates two boards, the location of the connection point between them is critical in optimizing antenna performance. The propagation of simulated electrical surface currents guides the selection of where to place the connecting point. Choosing a location that maximizes these electrical surface currents assures their propagation across the entirety of the ground plane, resulting in increased overall performance. The placement of this connection point is important since it affects the efficiency and effectiveness of the smart metering system.

Note that the more ground plane surface currents can propagate, the greater total efficiency can be obtained.

For the simulated environment, the best position area for locating the optimized connection point between the communication and main board is the area shown in the following figure.

This is a pin connection solution specifically for the simulated environment. Ignion can help you with your specific design starting with the design recommendations gathered in the Antenna Intelligence Cloud™ report.

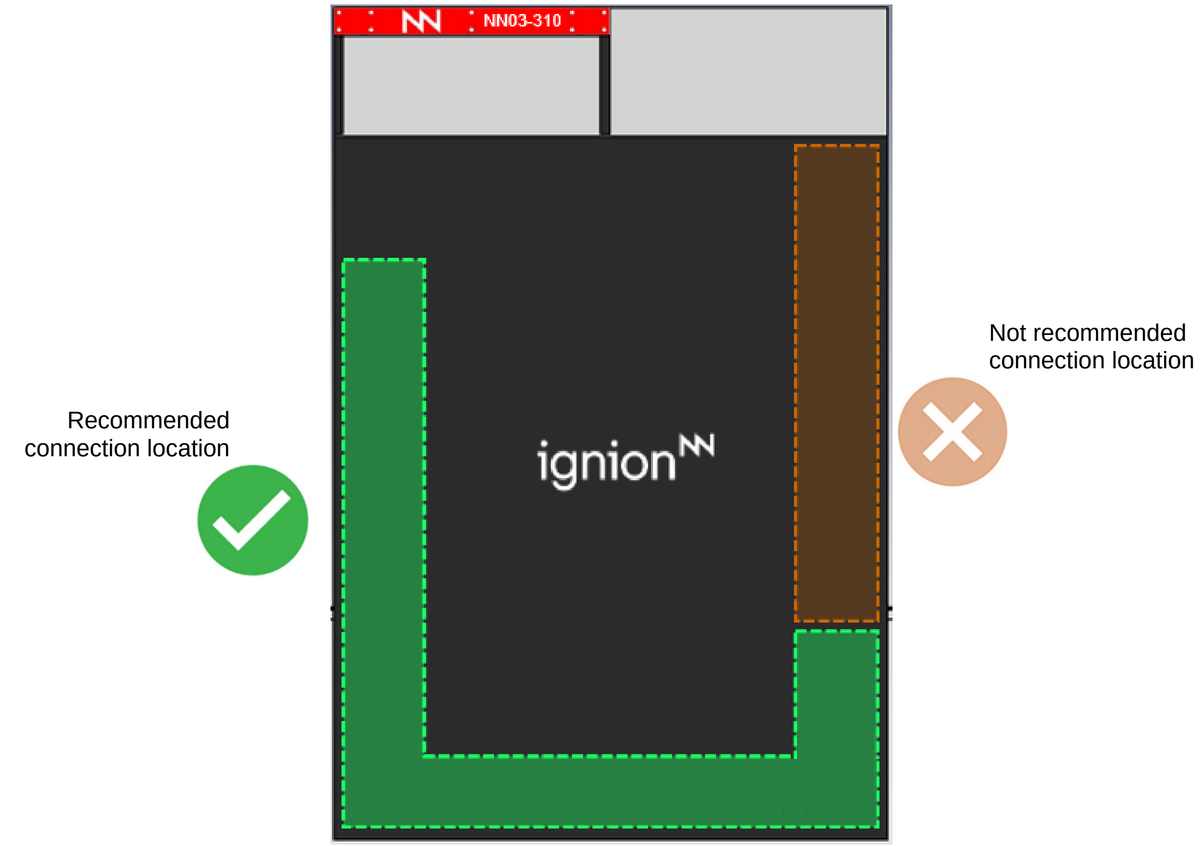


Figure 16 – Recommended locations between communication and main boards.

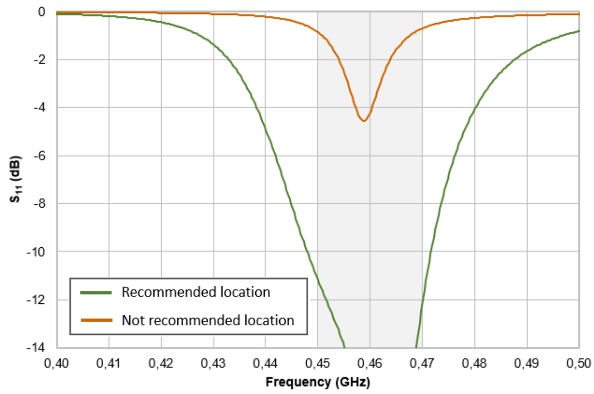


Figure 17 – S-parameters comparison of recommended (green trace) and not recommended (brown trace) connection locations.

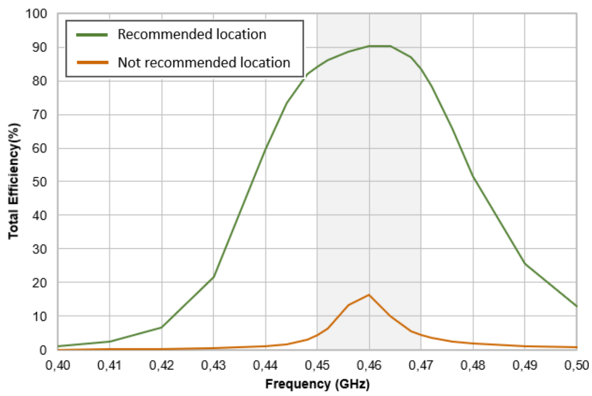


Figure 18 – Total efficiency comparison of recommended (green trace) and not recommended (brown trace) connection locations.

In the image shown above, the location of the connection between communication and main boards in a best position (green trace), maximizes the efficiency drastically compared to a worst position (brown trace), having a difference up to 15 dB.

	Total Efficiency Average (450-470 MHz) [%]	Total Efficiency Maximum Peak (450-470 MHz) [%]	Total Efficiency @450 MHz [%]	Total Efficiency @470 MHz [%]
Recommended location	87.1	90.3	84.3	83.4
Not recommended location	8.5	16.4	4.3	4.3

Table 4 – Total efficiency results of recommended and not recommended location.

At low-frequency ranges, large ground planes guarantee higher antenna efficiencies. The ground plane can be increased by choosing an optimal position for the connection point between both boards.

In the appendix (section 5) of this document, a deeper analysis of the antenna performance based on the areas available for pin connection is shown. These results are taken from a simulated environment and might change for your specific case. Ignion recommends you evaluate the pin location in your own design. Take advantage of the Ignion services to ensure the optimized antenna performance of your smart meter. For further information, check the engineering support services on the Ignion website: https://ignion.io/resources-support/technical-center/engineering-support/.

2.3.5. LTE 450 AND LTE-M MULTI-BOARD DESIGN EXAMPLE

When designing multi-board devices, the ground plane enlargement through the pin connection location influences especially the lower frequencies as seen above in the LTE 450 (450-470 MHz). See also appendix in section 5. Adding an LTE-M (698-960 MHz and 1710-2200 MHz) connection to the same communication board is easy, and below an example shows the achievable performance using a pin connection location optimized for the lower bands.

The smart meter design analyzed consists of two boards. One board is working as a communication board with the TRIO mXTEND™ (NN03-310) antenna component configured for LTE 450 (450-470 MHz) and the ALL mXTEND™ (NN02-220) antenna component configured for LTE-M (698-960 MHz and 1710-2200 MHz). The second board is the typical smart meter main board.

This multi-board design has the following characteristics:

- Communication board: 60 mm x 90 mm.
- Clearance area on communication board: 60 mm x 14 mm.
- Main board: 165 mm x 125 mm.
- The connection between communication and main board is made through pin connectors.

Communication Board + Main Board

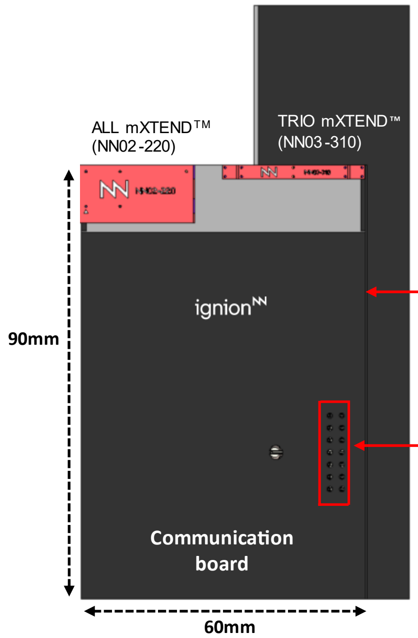


Figure 19 - Communication board with dimensions of 60 mm x 90 mm.

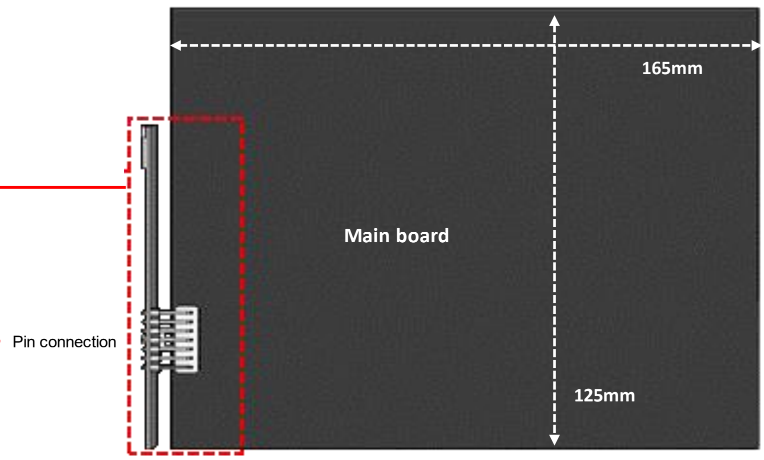


Figure 20 - Main board with dimensions of 125 mm x 165 mm.

The connection between the communication and main board has been made using the same pin connector from the previous sections.

2.3.6. S-PARAMETERS AND TOTAL EFFICIENCY PERFORMANCE

The S-parameters and the total efficiency of the PCB shown at Figure 15 are evaluated in the Figure 21, Table 5, and Table 6.

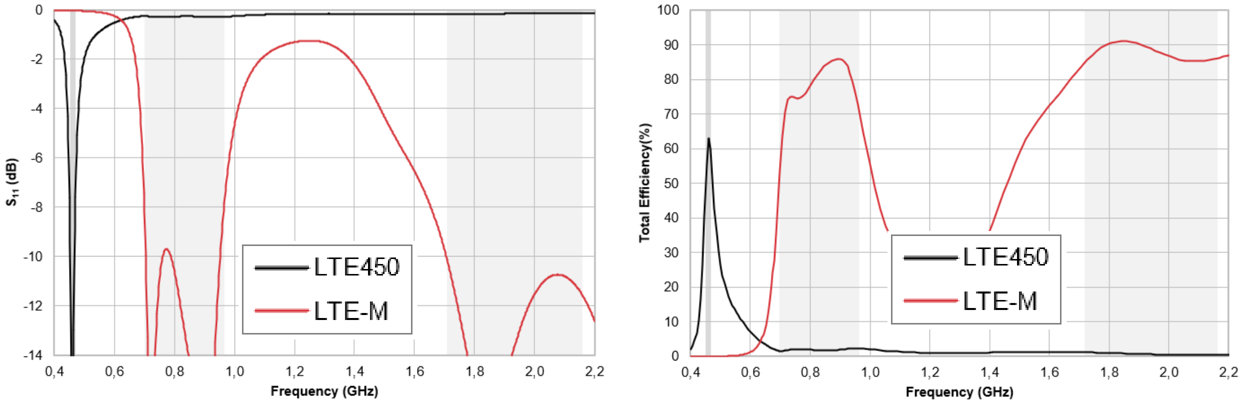


Figure 21 – S-parameters (left) and total efficiency (right) responses of LTE450 and LTE-M solution.

	VSWR / S11(dB) @450MHz		**VSWR / S11(dB)** @470MHz	
LTE 450	2.4 / -7.7		2.4 / -7.7	
	VSWR / S11(dB) @698MHz	**VSWR / S11(dB)** @960MHz	**VSWR / S11(dB)** @1710MHz	**VSWR / S11(dB)** @2200MHz
LTE-M	2.5 / -7.5	2.2 / -8.5	1.9 / -10.2	1.7 / -11.7

Table 5 - VSWR results summary of LTE-M and LTE 450 band.

For LTE450 band the solution provides a **VSWR<2.4** between **450-470 MHz**. As well for LTE-M band that has a **VSWR<2.5** between **698-960MHz** and a **VSWR<1.9** between **1710-2200 MHz.**

	Total Efficiency @450 MHz [%]		**Total Efficiency @470 MHz [%]**		**Total Efficiency Average (450-470 MHz) [%]**	
LTE 450	48.2		55.8		57.1	
	Total Efficiency @698 MHz [%]	**Total Efficiency @960 MHz [%]**	**Total Efficiency Average [%]**	**Total Efficiency @1710 MHz [%]**	**Total Efficiency @2200 MHz [%]**	**Total Efficiency Average [%]**
LTE-M	53.1	72.9	77.6	84.0	86.3	87.6

Table 6 - Total efficiency results summary of LTE-M and LTE 450 band.

For both combined cellular technology solution shown above the best position area for locating the optimized connection point between the communication and main board has been implemented always following all the design recommendations specifically for the simulated environment. Ignion can help you with your specific design starting with the design recommendations gathered in the Antenna Intelligence Cloud™ report.

2.3.7. BILL OF MATERIALS FOR THE LTE 450 and LTE-M SOLUTION

Here we can find the bill of materials associated with the matching network needed to tune each antenna component to the desired bands.

LTE 450
(450 – 470 MHz)

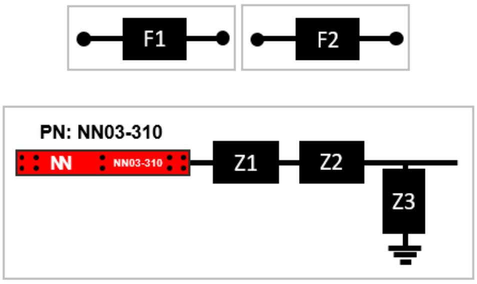


Figure 22 – LTE 450 matching network topology.

	Value	Part Number
F1	0Ω	-
F2	0Ω	-
Z1	72 nH	LQW18AN72NG80
Z2	7.5 nH	LQW18AN7N5G80
Z3	13 nH	LQW18AN13NG80

Table 7 – LTE450 bill of materials for matching network.

LTE-M
(698 – 960 MHz and 1710 – 2200 MHz)

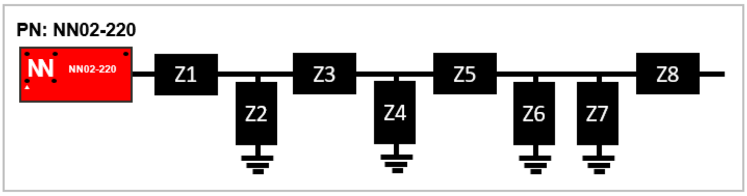


Figure 23 – LTE-M matching network topology.

	Value	Part Number
Z1	0Ω	-
Z2	0.6 pF	GJM1555C1HR60WB01
Z3	4.1 nH	LQW15AN4N1G80
Z4	7.9 nH	LQW15AN7N9G80
Z5	1.5 pF	GJM1555C1H1R5WB01
Z6	9.5 nH	LQW15AN9N5G80
Z7	0.4 pF	GJM1555C1HR40WB01
Z8	2.1 pF	GJM1555C1H2R1WB01

Table 8 – LTE-M bill of materials for matching network.

2.3.8. LTE 450 AND LTE-M IN SINGLE RF MODULE OUTPUT

Currently, most of the chipsets in the market have a single pin out for all LTE bands (LTE 450: 72 and 31, LTE-M: 5, 8, 12, 13, 20, 26, 28, 2, 3, 4 and 66). Thus, smart meter designs supporting both LTE 450 and LTE-M, typically would require antenna solutions where both are connected to the same LTE output pin.

The schematic below illustrates one of the recommended solutions for smart meter designs with a single output pin solution covering LTE 450 (450 - 470MHz) and LTE-M (698 – 960 MHz and 1710 – 2200 MHz). The solution includes a 2-states switch between the LTE module output pin and each matching network branch (LTE 450 and LTE-M).

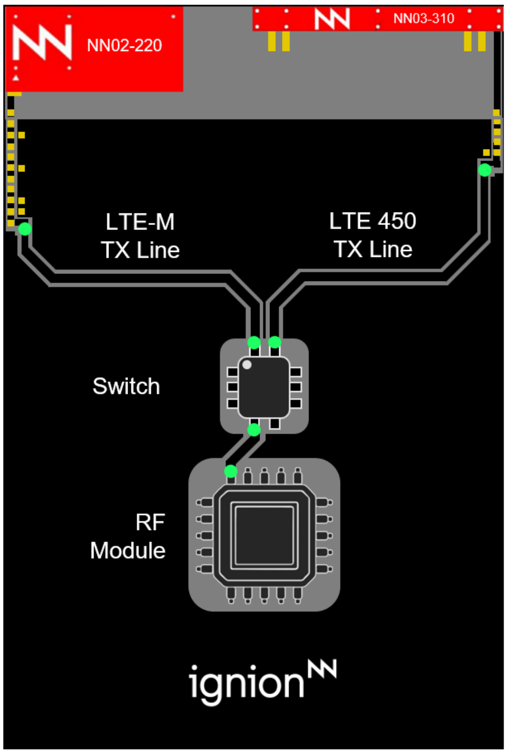


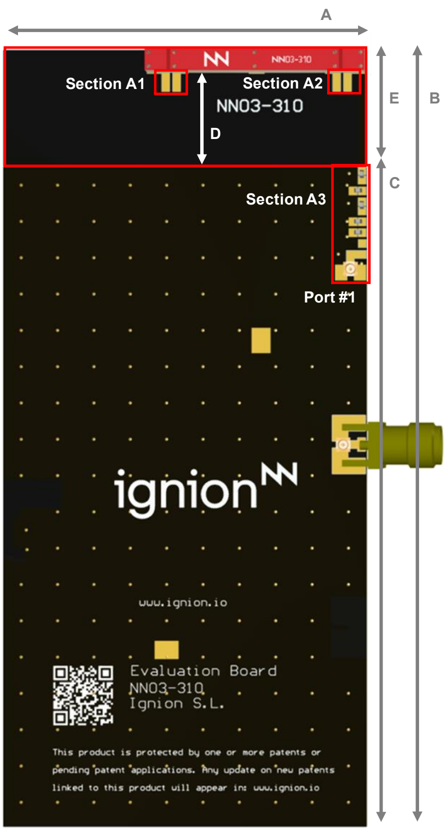
Figure 24 – Schematic with recommended antenna design including an RF switch when covering LTE 450 + LTE-M in a single RF Module.

The schematic shown above is just for illustration purposes. It is strongly recommended that the switch and the RF module be placed as close as possible to the I/O port of the matching networks to minimize losses. Note that the transmission line and the switch can introduce additional losses compared to the results shown in section Figure 21. It is recommended to use low-loss switches from e.g., Skyworks or Qorvo.

Other combinations of wireless technologies and sizes can easily be evaluated such as adding other sub-GHz technologies using either one or multiple antennas. The custom capabilities of the Virtual Antenna® technology provide limitless possibilities and tailored performance.

2.3.9. INFLUENCE OF SURROUNDING MATERIALS

This section explores the impact of different environmental conditions on the performance of NB-IoT and LTE/CAT-M in **smart metering** applications, such as being in close proximity to a metal pipe/cabinet or close to a concrete wall. The study uses a reference PCB with dimensions of 107 mm x 50 mm and clearance area of 16 mm x 45 mm for TRIO mXTEND™ (**Figure 25**). The PCB is positioned at distances of 20 mm and 0 mm from wood, concrete and metal surfaces (**Figure 26**). These PCB dimensions are commonly used in smart meters, and similar results can be expected in PCBs of different dimensions. Note that the results presented here are based on direct measurements performed in a laboratory setting.



Measure	mm
A	50
B	107
C	91
D	13
E	16

D: Distance between the TRIO mXTEND™ antenna component and the ground plane.

Material: The evaluation board is simulated on FR4 substrate. Thickness is 1 mm.

Figure 25 – PCB used to evaluate the impact of materials in proximity. NB-IoT and LTE/CAT-M (698 MHz – 960 MHz, 1710 MHz – 2200 MHz).

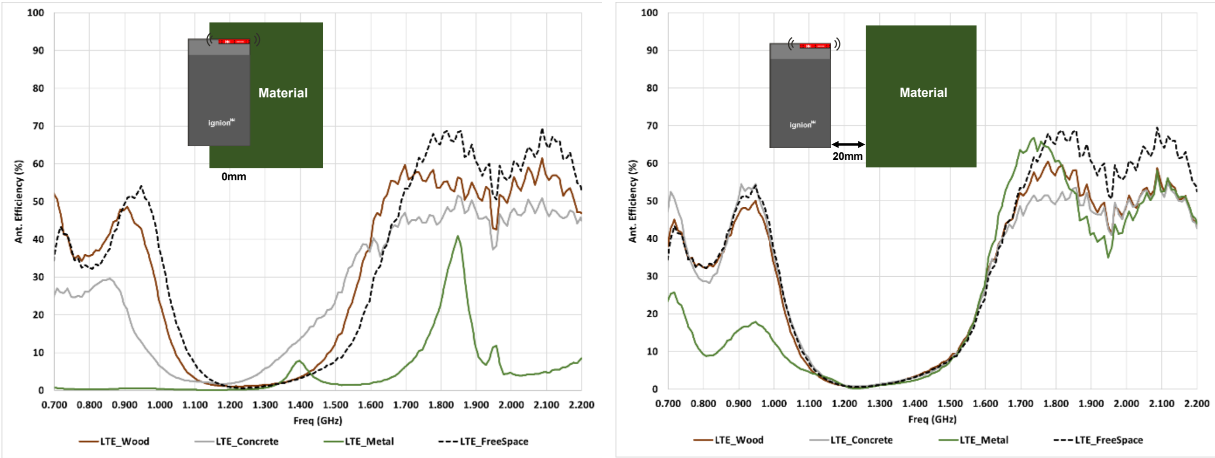


Figure 26 – Summary of antenna performance variation with different materials in proximity. NB-IoT and LTE/CAT-M with the TRIO mXTEND™ (NN03-310) deployed at 20 mm **(right)** distance and at 0 mm **(left)** from the analyzed materials (wood, concrete, and metal) with a PCB of 107 mm x 50 mm (**Figure 25**).

		Total Efficiency (%) of LTE-M/NB-IoT	
		Avg (698 – 960 MHz)	**Avg (1710 – 2200 MHz)**
Free-Space		39.6	62.1
Wood	**0mm**	41.0	54.3
	20mm	38.8	52.8
Concrete	**0mm**	25.2	46.5
	20mm	39.9	49.9
Metal	**0mm**	<1.0	12.4
	20mm	14.9	51.5

Table 9 – Summary of antenna performance variation in different environmental conditions for NB-IoT and LTE/CAT-M with the TRIO mXTEND™ (NN03-310) on a PCB of 107 mm x 50 mm.

This section shows the impact of proximity to various materials on antenna performance. Materials commonly encountered in smart metering can affect wireless performance.

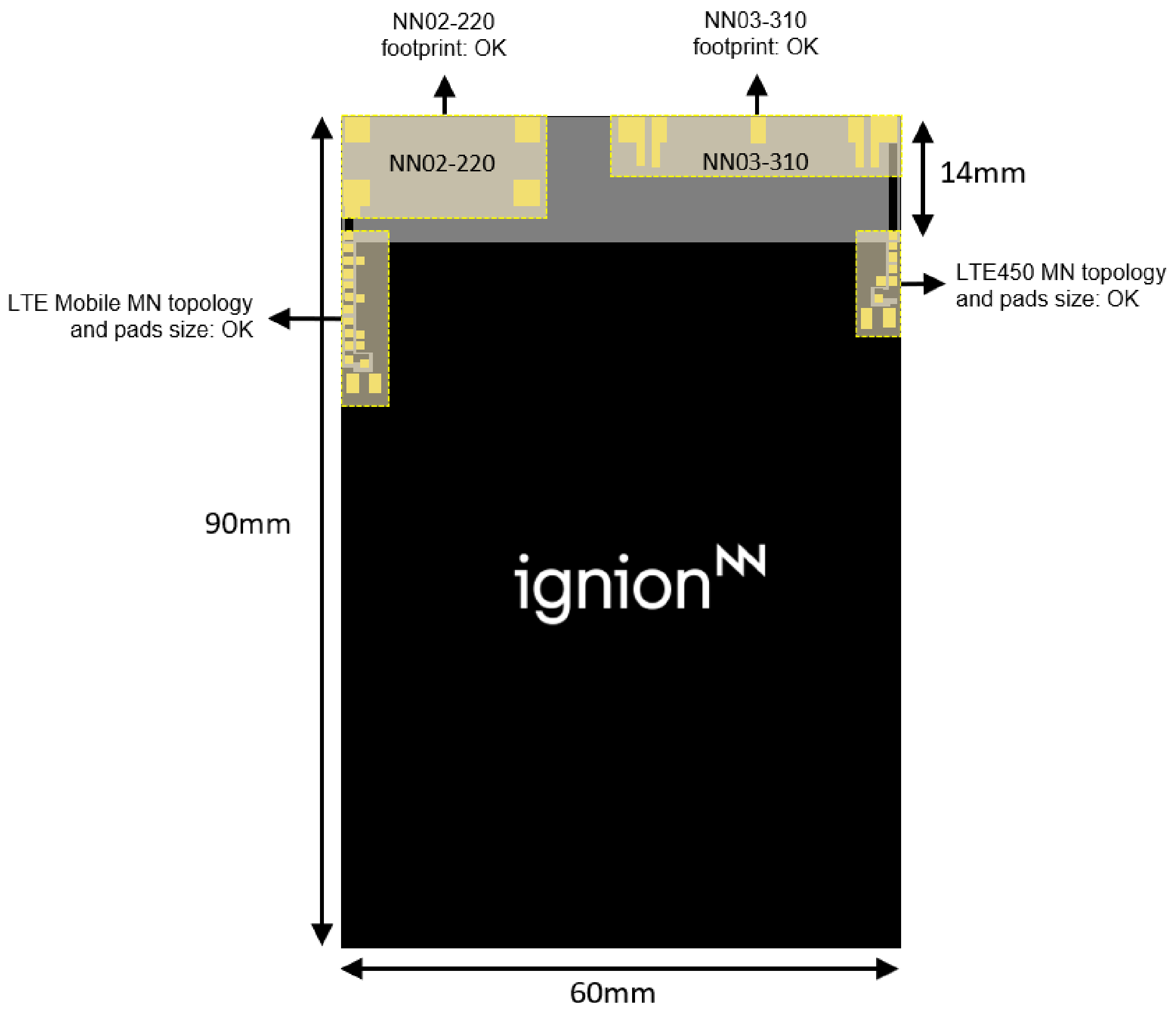
- The antenna performance hardly changes in the presence of **wood**.
- **Concrete** in proximity causes a drop in efficiency, especially at high bands, but no shifting in frequency, so there is no need to readjust the matching network.
- **Metal** is a challenging material when placed close to an antenna. A reconfiguration/retuning of the matching network is recommended to adjust for the specific distance between device and metal.
- Note that surrounding materials impact does not affect most certifications as these are usually done in free-space.

2.4. STEP 4: GERBER CHECK

Before producing the physical prototype, a Gerber check is recommended ensuring that the design files meet the design recommendations provided. Ignion provides the Gerber review service free of charge, ensuring that the prototype performance will be aligned with the simulated result.

In the free Gerber review service, an Ignion engineer will review key elements including antenna location, clearance area, matching network topology, feeding lines, transmission lines, and other critical design considerations. This review helps identify and mitigate potential oversights and provides valuable feedback to optimize the design for maximum efficiency and performance while avoiding costly hardware iterations.

GERBER CHECK OF LTE 450 AND LTE-M SOLUTION BOARD



Measure	mm
A	12.0
B	23.0
C	3.0
D	0.5
E	1.0
F	8.5
G	2.0
H	2.5
I	10.0
J	7.0

Tolerance: ±0.05mm

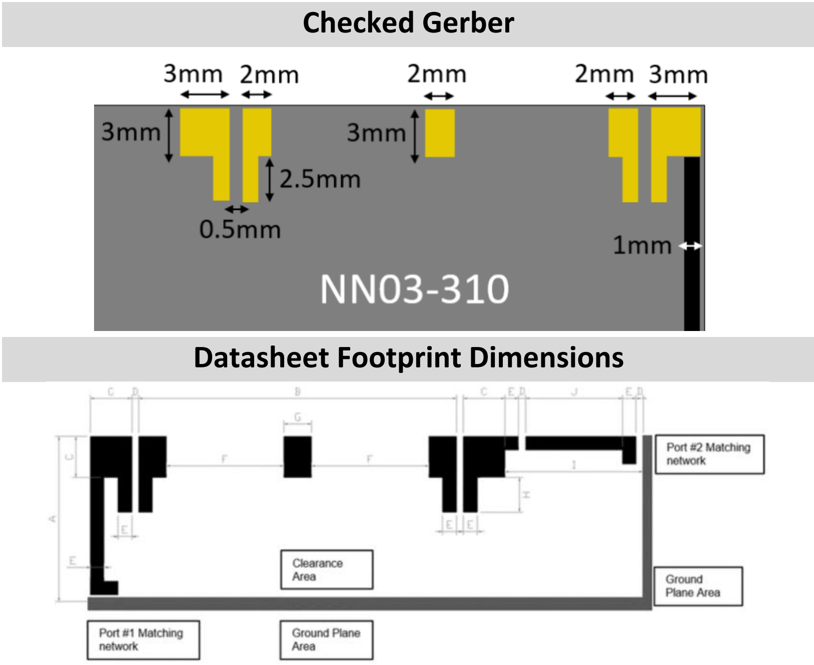


Figure 27 – Example of Gerber check review report.

2.5. STEP 5: PROTOTYPING

Once all the previous steps of this guide have been implemented throughout the design journey and with the Gerber file checked by the Ignion team, the prototype is built.
If performance does not meet the simulated efficiency, further improvements can typically be achieved by fine-tuning the overall system.

During the prototype testing, the flexibility to easily retune the antenna system in response to any hardware design changes is a big advantage and reduces risk of re-designing.

2.6. STEP 6: MEETING CERTIFICATION REQUIREMENTS

The Virtual Antenna® technology enables developers to ensure their design is capable of meeting certification requirements from the beginning of the IoT device design journey. Additionally, if any changes in certification requirements appear after the initial design, the antenna solution can easily be tuned to the new frequency band without needing to change the antenna component. Important recommendations:

- **Critical band selection:**

Certification requirements for some smart metering devices are obtained from cellular operators, and vary by region, frequency band, and device operation characteristics. When choosing LTE bands for your smart meter, make sure to consider:

  - Future compatibility assures that your smart meter will be able to quickly adjust to future LTE improvements and/or additional bands in the deployment region. Be aware of the deployment bands needed for your smart meter device. Ensure compliance with the certification regulations of the region (refer to section 2.6.1. for example).

  - Make sure to consider efficient use of the needed bands, covering wider than needed often results in unnecessary congestion and lower antenna performance. Spectrum efficiency is often overlooked, and can have a big influence on final performance, and device design.

- **Perform OTA tests before going through certification.**

To prepare for the accredited lab certification tests, it is recommended that a range of pre-certification tests are performed, including OTA (Over-the-Air) test measurements on the final physical prototype. These measurements assess various parameters, including conducted power, conducted sensitivity, Total Radiated Power (TRP), and Total Isotropic Sensitivity (TIS). By conducting these measurements, any potential performance issues can be identified and fine-tuning of the antenna solution can be performed to meet the certification requirements effectively. At Ignion we offer pre-certification tests and consultancy services to ensure that your device meets the required targets.

2.6.1. EVALUATE CERTIFICATION REQUIREMENTS

The certification requirements are obtained from the cellular operator and vary both by region as well as per device operation characteristics. For this example, we consider TRP requirements from AT&T for a device operating on LTE/Cat-M bands (Source: AT&T: Radiated Performance Requirements version 1.6).

Band	Minimum TRP Requirement Power Class 3	Minimum TRP Requirement Power Class 5
2	+20.0 dBm	+17.0 dBm
4	+20.0 dBm	+17.0 dBm
12	+18.0 dBm	+15.0 dBm

Figure 28 – LTE/Cat-M requirements for large form factors **above** 107 mm in the longest direction: Free-Space.

2.6.2. ESTIMATE TOTAL RADIATED POWER (TRP) AND COMPARE TO CERTIFICATION REQUIREMENTS

The estimated TRP results are calculated using Equation 1 below considering 23 dBm (Power class 3) of output power from the RF module for bands 2, 4, and 12, and 26 dBm for bands 31 and 72. Power injected values taken from TX62 Global MTC Module (LTE-M/NB-IoT) datasheet from Telit Cinterion.

Estimated TRP (dBm) = Output power (dBm) + Total Efficiency (dB)

Equation 1 - Estimated TRP (dBm) calculated from RF Module output power (dBm) combined with antenna solution total efficiency (dB).

The following table shows the TRP assessment from the simulated results taken from section 2.3.5 and how pre-certification values can be obtained even before having a physical prototype.

Europe			Uplink Average Antenna Efficiency [%]	Nominal power injected (dBm)	Calculated TRP based on Antenna Efficiency [dBm]
Band	**Downlink (MHz)**	**Uplink (MHz)**			
31	452.5 - 457.5	462.5 - 467.5	59.8	26	23.8
72	461 - 466	451 - 456	58.5	26	23.7
North America					
2	1930 - 1990	1850 - 1910	90.5	23	22.6
4	2110 - 2155	1710 - 1755	84.0	23	22.2
12	729 - 746	699 - 716	62.2	23	20.9

Table 10 – TRP values obtained from antenna efficiency based on optimal antenna integration in a simulated environment. Power injected values taken from TX62 Global MTC Module (LTE-M/NB-IoT) datasheet from Telit Cinterion.

The current design example has demonstrated remarkable performance by exceeding the certification targets with a substantial margin, meeting the requirements. This achievement strongly indicates that it is anticipated to effortlessly pass the certification tests.

2.6.3. PRE-CERTIFICATION LAB TEST - VERIFY YOUR DEVICE TRP PERFORMANCE

On the final physical prototype, perform OTA (Over-the-Air) test measurements of conducted power, conducted sensitivity, Total Radiated Power (TRP), and Total Isotropic Sensitivity (TIS) to prepare and ensure that the device will pass the official certification tests.

3. SUMMARY

To maximize RF performance in your smart meter 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 smart meter solutions. Here are the steps that should be followed:

1. Understand the dependency between device board dimensions and the product requirements including communication protocols, and frequency bands ensuring spectrum efficiency, future compatibility, and regulatory compliances.

2. 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 smart metering requirements, considering factors such as frequency range, antenna performance, and size constraints.

3. Evaluate the impact of PCB size and clearance area on the overall performance and adjust if needed. In the case of a multi-board solution, optimize the ground pin location to enhance overall antenna performance by performing EM simulations. You can follow the design recommendations from the section 2.3.3 and the recommendations gathered in the Antenna Intelligence Cloud™ report.

4. Evaluate surrounding materials in the smart meter ambient environment and its impact on performance. See section 2.3.9.

5. Conduct a comprehensive simulation of the fully populated PCB, considering all integrated elements of the final device, to obtain accurate RF performance results.

6. 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.

7. On the final physical prototype, perform OTA (Over-the-Air) test measurements to prepare and ensure that the device will pass the official certification tests.

By following these steps, designers can optimize the antenna integration design of smart meters, resulting in enhanced performance, reliable communication, and successful deployment of smart metering systems. For any further help during the design journey, see all our services here: https://ignion.io/resources-support/technical-center/engineering-support/.

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.

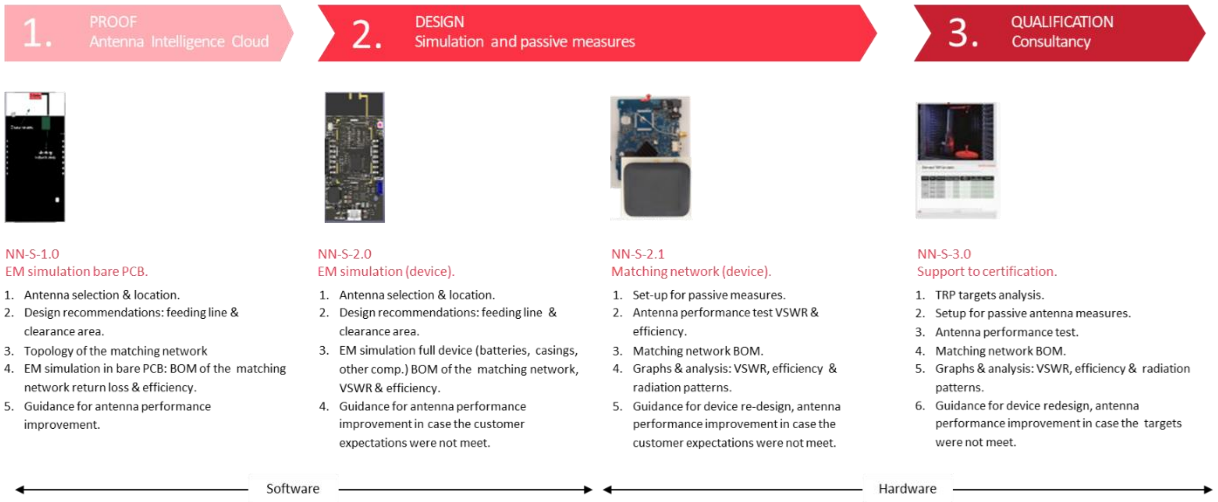


Figure 29 - Ignion services for antenna integration.

5. APPENDIX: CONNECTION POINT LOCATION PERFORMANCE ANALYSIS

5.1. CONNECTION POINT LOCATED AT THE OPPOSITE SIDE OF THE ANTENNA

In this section, we show the antenna performance results when locating the connection point at the opposite side of the antenna:

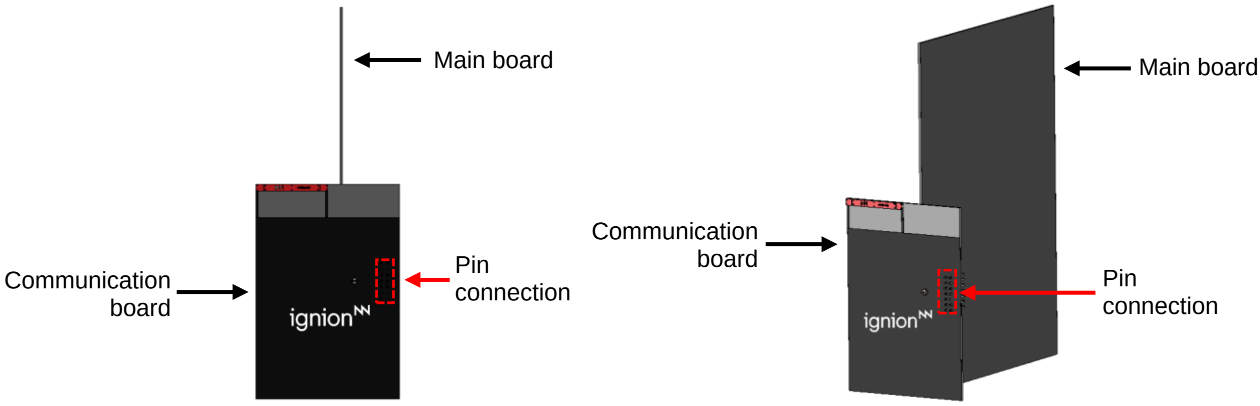


Figure 30 - Configuration set-up with the communication board connected to the main board through a pin connection located at the opposite side of the antenna.

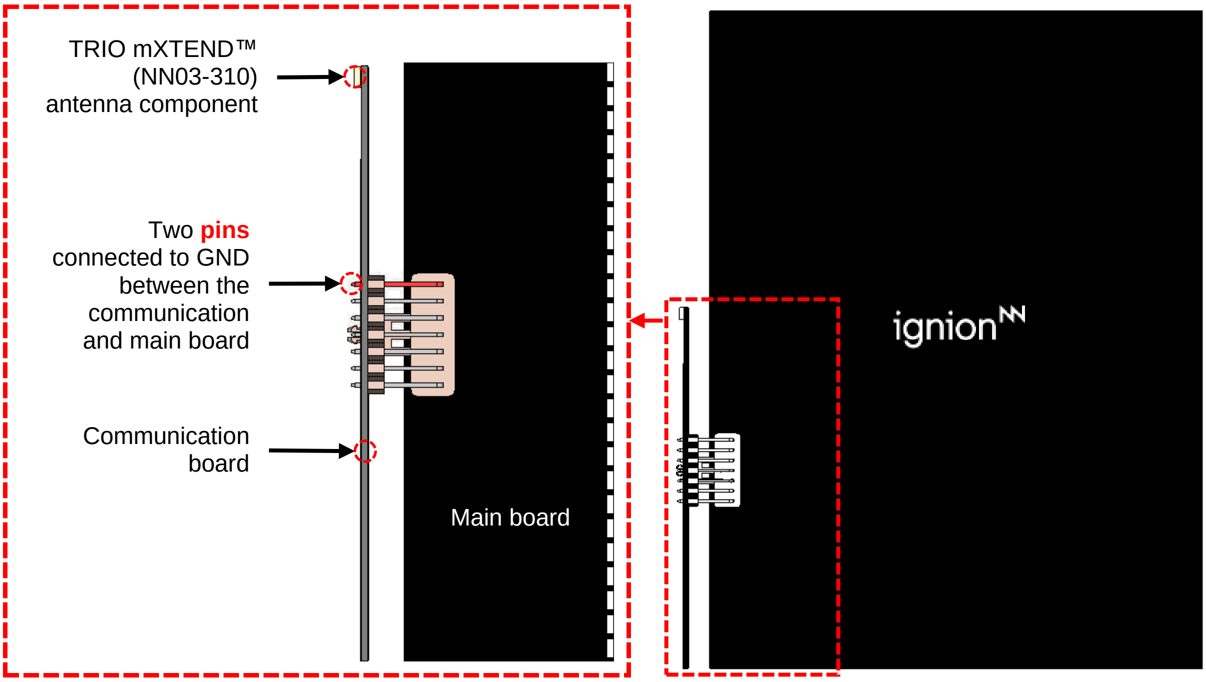


Figure 31 – Connection between communication and main board using a pin connection.

GND Point at 19 mm	GND Point at 38 mm	GND Point at 57 mm

Figure 32 - Connection at 19 mm on the opposite side of the antenna.

Figure 33 - Connection at 38 mm on the opposite side of the antenna.

Figure 34 - Connection at 57 mm on the opposite side of the antenna.

	Total Efficiency Average (450-470 MHz) [%]	Total Efficiency Maximum Peak (450-470 MHz) [%]	Total Efficiency @450 MHz [%]	Total Efficiency @470 MHz [%]
GND Point at 19 mm	8.5	16.4	4.3	4.3
GND Point at 38 mm	21.5	36.8	10.1	13.6
GND Point at 57 mm	54.6	77.4	34.0	41.1

Table 11 - Total efficiency results summary for each analyzed case.

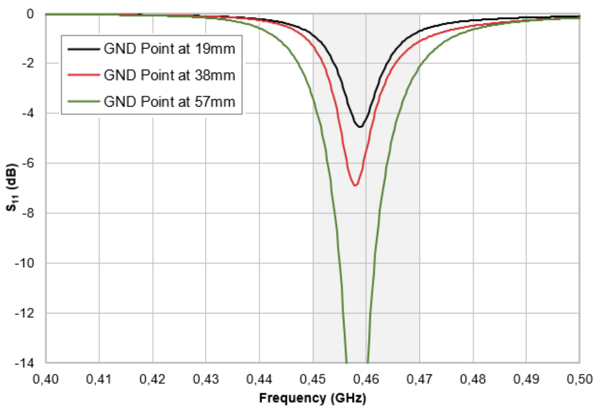


Figure 35 – S-parameters of the different connection point locations.

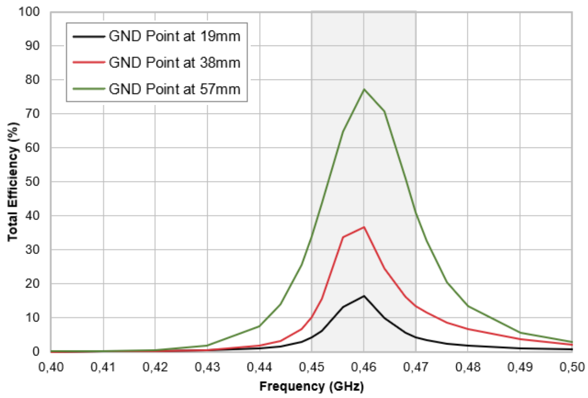


Figure 36 - Total efficiency of the different connection point locations.

As shown above, by locating the connection point between communication and main boards at 57mm (green trace) on the communication board right side, the antenna efficiency is improved by more than 3dB compared to the worst case (19mm case, black trace).

5.1.1. TOPOLOGY AND BILL OF MATERIALS USED FOR THE CONNECTION POINT LOCATED ON THE OPPOSITE SIDE OF THE ANTENNA

Here we can find the bill of materials associated with each location of the connection.

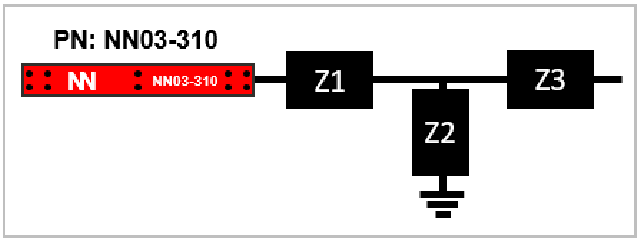


Figure 37 – Matching network topology used for each case analyzed.

Connection at 19 mm and 38 mm on the opposite side of the antenna.

	Value	Part Number
Z1	2.9 pF	GJM1555C1H2R9WB01
Z2	18 pF	GJM1555C1H180FB01
Z3	8.8 pF	GJM1555C1H8R8WB01

Table 12 – Bill of materials of the connection cases at 19 mm and 38 mm on the opposite side of the antenna.

Connection at 57 mm on the opposite side of the antenna.

	Value	Part Number
Z1	2.9 pF	GJM1555C1H2R9WB01
Z2	16 pF	GJM1555C1H160GB01
Z3	8.8 pF	GJM1555C1H8R8WB01

Table 13 – Bill of materials of the connection case at 57 mm on the opposite side of the antenna**.**

5.2. CONNECTION POINT LOCATED AT THE SAME SIDE OF THE ANTENNA

In this section, we show the results analyzing locating the connection point on the same side as the antenna:

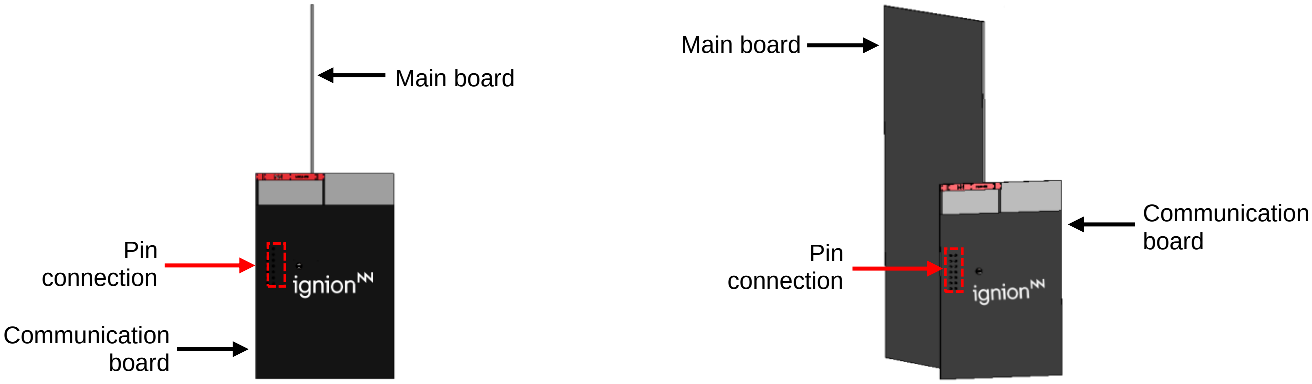


Figure 38 - Set-up with communication board connected to the main board through a pin connection located at the same side as the antenna on the communication board.

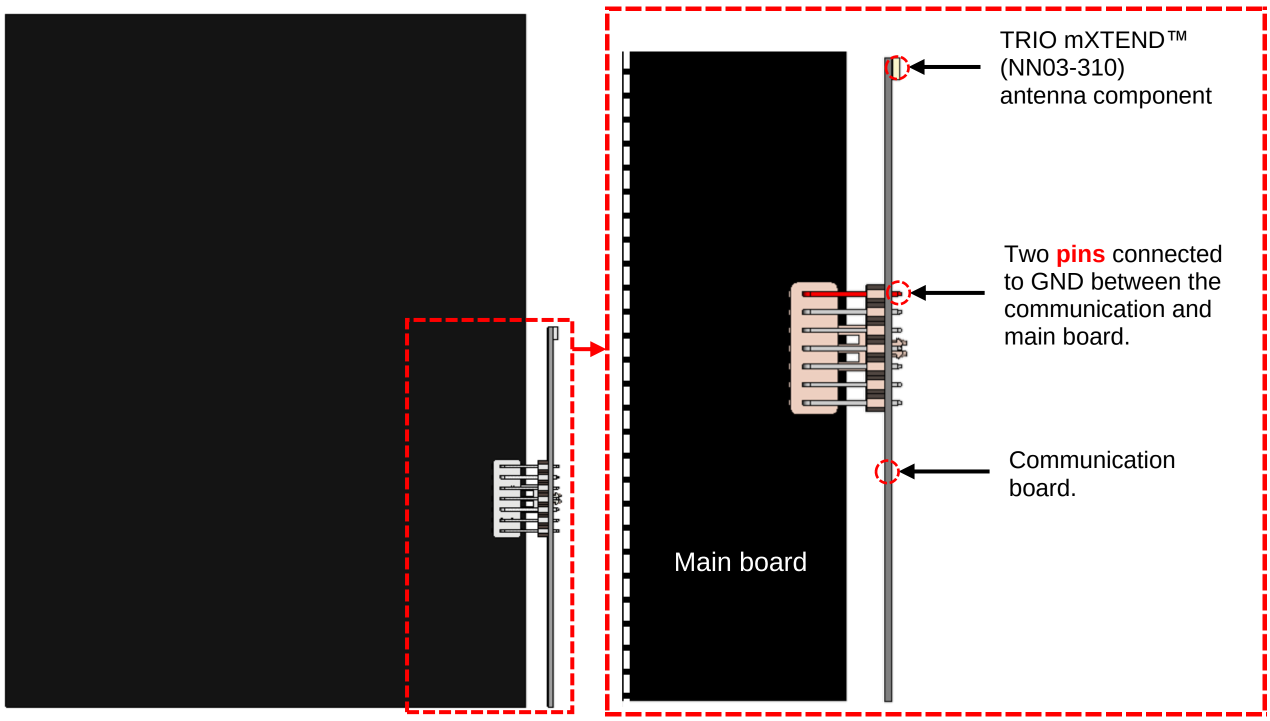


Figure 39 – Connection between communication and main board using pin connection.

GND Point at 19 mm | **GND Point at 38 mm** | **GND Point at 57 mm**

Figure 40 - Connection at 19 mm on the same side of the antenna.

Figure 41 - Connection at 38 mm on the same side of the antenna.

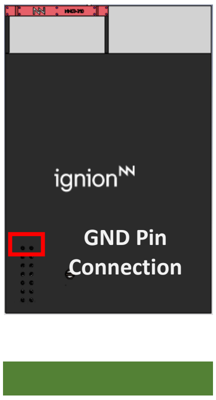


Figure 42 - Connection at 57 mm on the same side of the antenna.

	Total Efficiency Average (450-470MHz) [%]	Total Efficiency Maximum Peak (450-470MHz) [%]	Total Efficiency @450MHz [%]	Total Efficiency @470MHz [%]
GND Point at 19mm	73.4	84.7	65.6	65.3
GND Point at 38mm	74.0	84.6	58.1	70.5
GND Point at 57mm	87.1	90.3	84.3	83.4

Table 14 - Total efficiency results summary for each analyzed case.

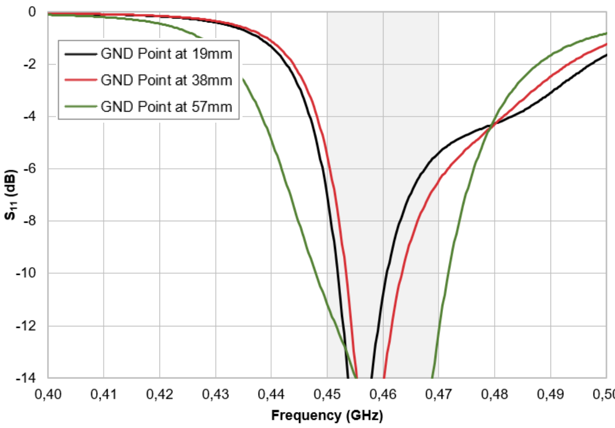


Figure 43 – S-parameters response comparison of every studied case.

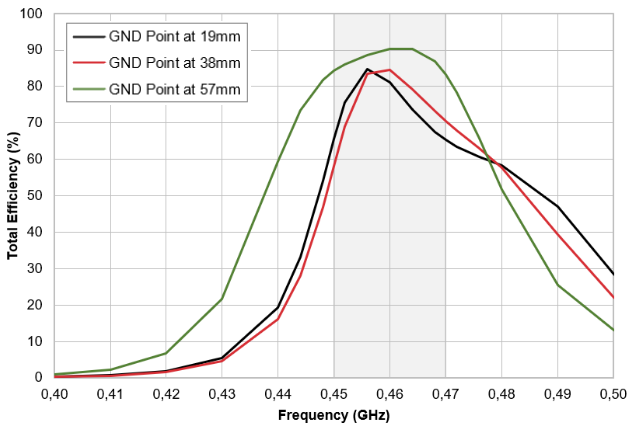


Figure 44 - Total efficiency response comparison of every studied case.

As shown on the results above, locating the connection point between communication and main boards at the same side of the antenna, maximizes the surface currents and the antenna performance in comparison to the case at the opposite side of the communication board (**section 5.1)**. This shows the importance of analyzing the simulated surface currents of the multi-board device before deciding on the connection point between both boards.

5.2.1. TOPOLOGY AND BILL OF MATERIALS USED FOR THE CONNECTION POINT LOCATED ON THE SAME SIDE AS THE ANTENNA

Here we can find the bill of materials associated to each band of the solution.

Connection at 19mm and 38mm on the same side of the antenna.

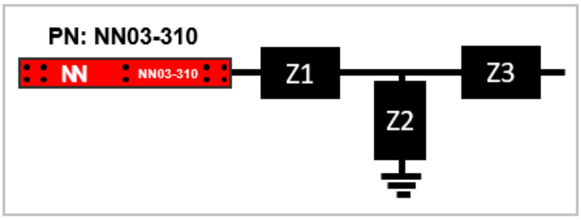


Figure 45 – Matching network topology used for the connection cases at 19mm and 38mm on the same side of the antenna.

	Value	Part Number
Z1	3.1 pF	GJM1555C1H3R1WB01
Z2	8.8 pF	GJM1555C1H8R8WB01
Z3	5.7 pF	GJM1555C1H5R7WB01

Table 15 – Bill of materials of the connection cases at 19mm and 38mm.

Connection at 57mm on the same side of the antenna.

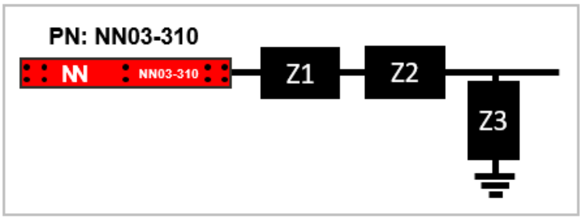


Figure 46 – Matching network topology used for the connection 57 mm on the same side of the antenna.

	Value	Part Number
Z1	18 pF	GJM1555C1H180FB01
Z2	3.5 pF	GJM1555C1H3R5WB01
Z3	15 pF	GJM1555C1H150GB01

Table 16 – Bill of materials of the connection case at 57mm.

5.3. CONNECTION POINTS LOCATED AT THE BOARD BOTTOM SIDE

In this section, we show the results after analyzing the fact of locating the connection point at the communication board bottom side:

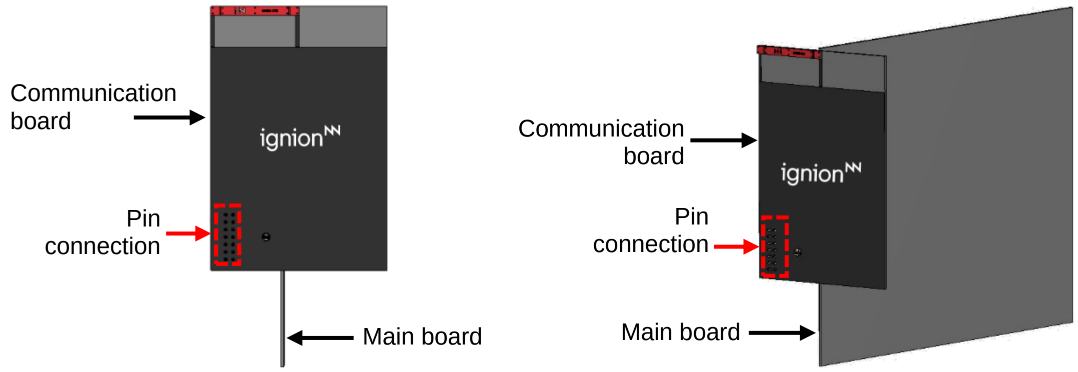


Figure 47 - Set-up configuration about communication board connected to the main board through pin connection located at the communication board bottom side.

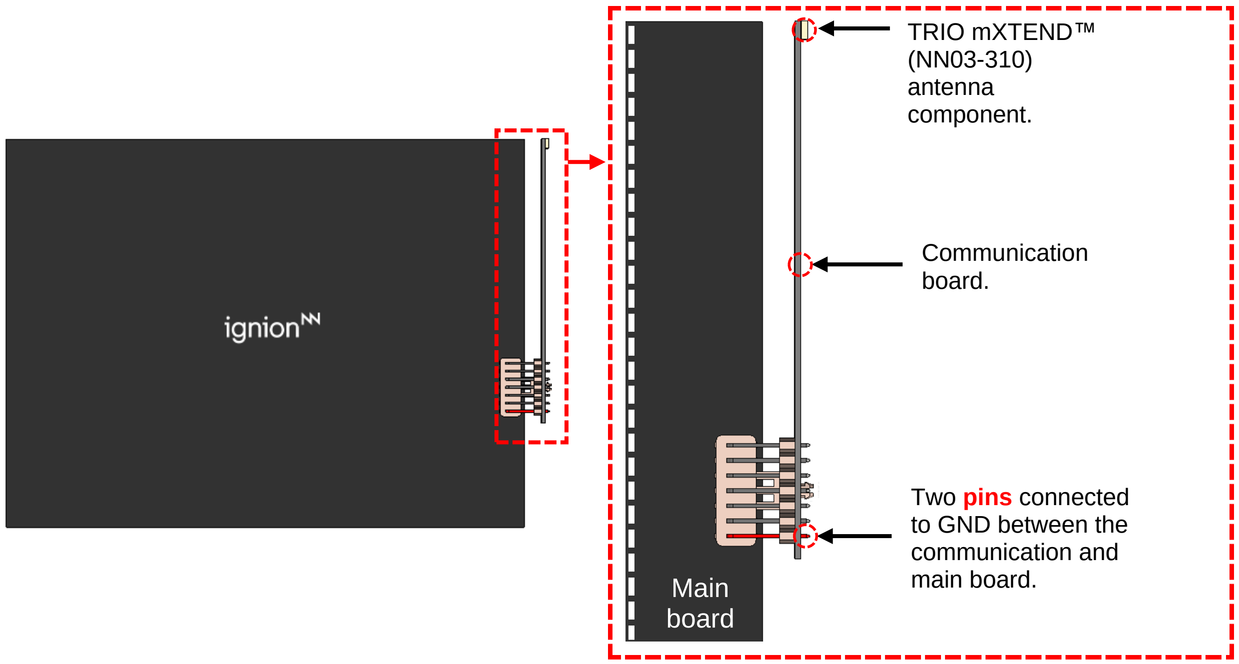


Figure 48 - Connection between communication and main boards using a pin connector.

GND Point at Bottom Left	GND Point at Bottom Middle	GND Point at Bottom Right
		
Figure 49 - Connection at the communication board bottom left.	**Figure 50** - Connection at the communication board bottom middle.	**Figure 51** - Connection at the communication board bottom right.

	Total Efficiency Average (450-470 MHz) [%]	Total Efficiency Maximum Peak (450-470 MHz) [%]	Total Efficiency @450 MHz [%]	Total Efficiency @470 MHz [%]
GND Point at Bottom Left	65.4	78.6	53.9	56.8
GND Point at Bottom Middle	62.3	79.2	46.9	49.8
GND Point at Bottom Right	50.8	65.7	39.0	39.2

Table 17 - Total efficiency results summary for each analyzed case.

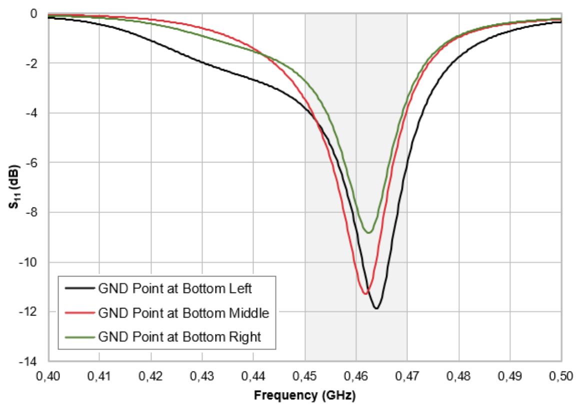


Figure 52 – S-parameters response comparison of every studied case.

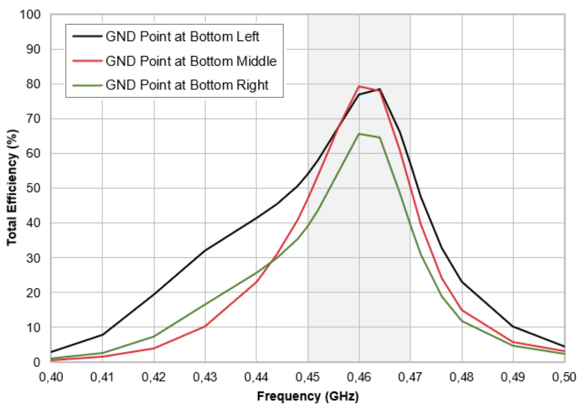


Figure 53 - Total efficiency response comparison of every studied case.

As shown on the results above, locating the connection point between communication and main boards at the bottom side of the communication board, shows a very good antenna performance meaning that the simulated electrical surface currents are strong enough to take advantage of the main board ground plane, enhancing and optimizing the overall antenna performance. Furthermore, as close as we move the pin connection to the opposite side of the antenna, the worse is the antenna performance by being closer to the not recommended pin connector location shown in section 2.3.3.. This shows the importance of analyzing the simulated surface currents of the multi-board device before deciding on the connection point between both boards.

5.3.1. TOPOLOGY AND BILL OF MATERIALS USED FOR THE CONNECTION POINT LOCATED AT THE BOARD BOTTOM SIDE

Here we can find the bill of materials associated to each band of the solution.

Connection at left bottom side of the communication board

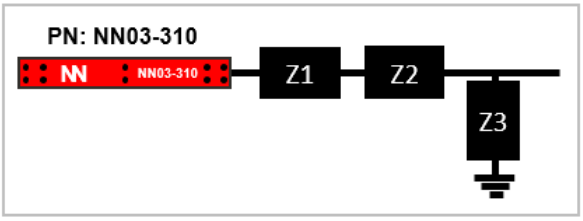


Figure 54 – Matching network topology used for the connection case at left bottom side.

	Value	Part Number
Z1	5.6 pF	GJM1555C1H5R6WB01
Z2	5.6 pF	GJM1555C1H5R6WB01
Z3	18 pF	GJM1555C1H180FB01

Table 18 – Bill of materials of the connection case at left bottom side.

Connection at middle bottom side of the communication board

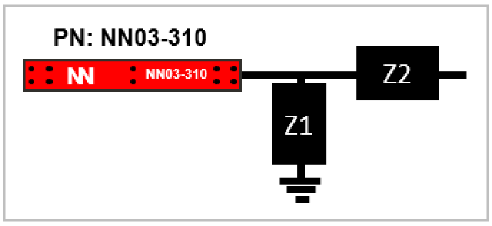


Figure 55 – Matching network topology used for the connection case at middle bottom side.

	Value	Part Number
Z1	1.7 pF	GJM1555C1H1R7WB01
Z2	0.8 pF	GJM1555C1HR80WB01

Table 19 – Bill of materials of the connection case at middle bottom side.

Connection at right bottom side of the communication board

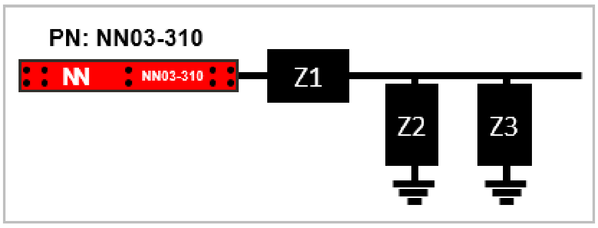


Figure 56 - Matching network topology used for the connection case at right bottom side.

	Value	Part Number
Z1	2.8 pF	GJM1555C1H2R8WB01
Z2	7.1 pF	GJM1555C1H7R1WB01
Z3	13 pF	GJM1555C1H130GB01

Table 20 - Bill of materials of the connection case at right bottom side.

The TRIO mXTEND™ and the ALL mXTEND™ antenna components and other Ignion products based on its proprietary Virtual Antenna® technology are protected by one or more of the following Ignion patents.

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