ignion

Antenna Intelligence

YOUR ANTENNA DIGITAL TWIN

Antenna Intelligence Cloud™ report

COMPANY NAME	DATE	SERVICE REPORT NUMBER	IGNION SUPPORT
HEVS Sion	24/10/2022	1666600867927	marc.munoz@ignion.io

Congratulations on making a step forward in your design

Most of the engineers like you, that design with a Virtual Antenna® component, made their decision based on this trifecta:

1. Choosing an antenna versatile enough to cover any protocol or band.
2. Straightforward antenna design guidance and accessible support.
3. Predictable performance from initial concept all the way to your end-product.

Your requirements

APPLICATION
Wearables & Smartwatches

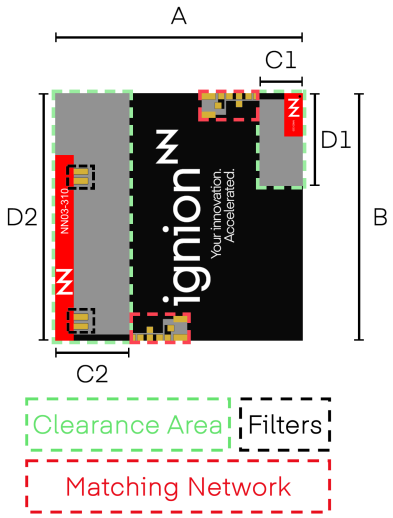
PCB DIMENSIONS
40.0 x 40.0 mm

SELECTED ANTENNA
NN03-320
NN03-310

COMMUNICATION STANDARDS & FREQUENCY RANGES
GNSS: 1559.0-1610.0 MHz
LTE-M: 791.0-960.0 MHz

Best antenna placement on your PCB

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



PCB		NN03-320		NN03-310	
Measure	mm	Measure	mm	Measure	mm
A	40.0	C1	7.0	C2	12.0
B	40.0	D1	15.0	D2	40.0

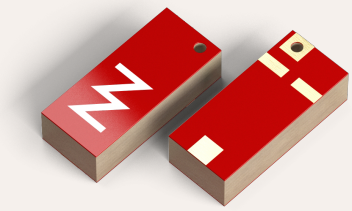
The sketch above is an approximate representation of the PCB design. The accurate model can be found in the Desing_Files_NNS1.0.zip attached in the same email where the report was received.

Your antenna design ready to use

ANTENNA 1

DUO mXTEND™ (NN03-320):

- **Application:** Wearables & Smartwatches.
- **Frequency Range(s):** 1559.0-1610.0 MHz for GNSS.
- **Tuning your antenna:** the optimized matching network is shown below.
- **Antenna footprint:** please refer to the datasheet link.



DUO mXTEND™

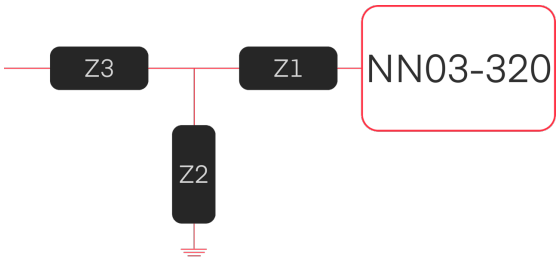
Dimensions 7.0 x 3.0 x 2.0 mm

Datasheet https://ignion.io/files/DS_NN03-320.pdf

Antenna Product Page →

Matching Network Antenna 1

GNSS Matching Network topology



Comm. Standard	Component	Value	Part Number	Manufacturer
GNSS	Z1	11nH	LQW18AN11NG80	Murata
	Z2	13nH	LQW18AN13NG80	Murata
	Z3	9.7nH	LQW15AN9N7G80	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.

If you need further assistance, please contact our antenna specialists.

GET HELP

Your antenna design ready to use

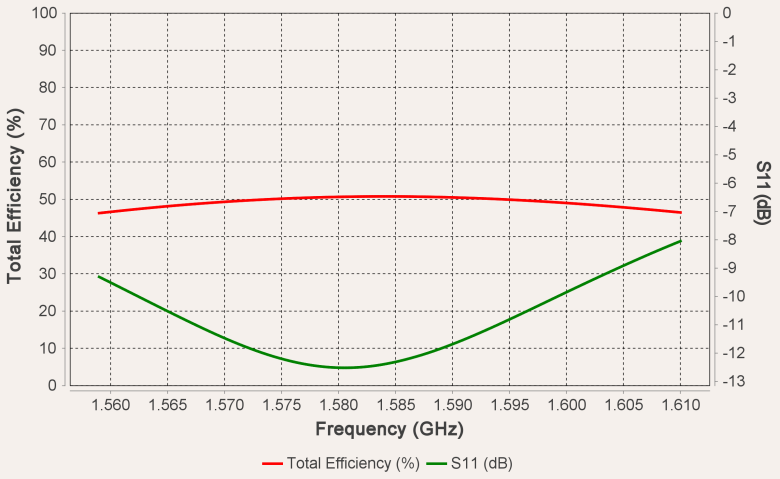
ANTENNA 1

Your Design overall Performance

Expected device performance with antenna 1

Your prototype using the NN03-320 antenna is expected to achieve the reflection coefficient (in dB) and total efficiency (in %) as shown in this graph.

Rule of thumb: it is desirable to have a reflection coefficient below -6 dB, ensuring proper impedance matching of the antenna component and optimized total efficiency.



INCREASE YOUR PERFORMANCE: to increase the performance we recommend evaluating your PCB again with increased dimensions (increasing length by 10 mm typically results in a total efficiency improvement of 0.5 dB).

	DUO mXTEND™ (NN03-320) for GNSS		
Frequency (MHz)	1559	1610	Avg 1559 - 1610
Total efficiency (%)	46.3	46.5	49.2

If you need further assistance, please contact our antenna specialists.

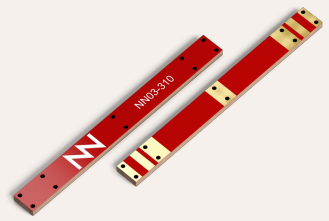
GET HELP

Your antenna design ready to use

ANTENNA 2

TRIO mXTEND™ (NN03-310):

- **Application:** Wearables & Smartwatches.
- **Frequency Range(s):** 791.0-960.0 MHz for LTE-M.
- **Tuning your antenna:** the optimized matching network is shown below.
- **Antenna footprint:** please refer to the datasheet link.



TRIO mXTEND™

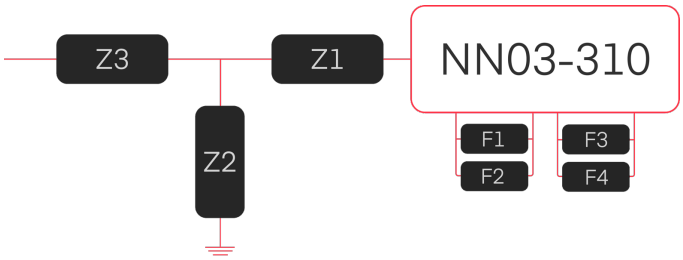
Dimensions 30.0 x 3.0 x 1.0 mm

Datasheet https://ignion.io/files/DS_NN03-310.pdf

Antenna Product Page →

Matching Network Antenna 2

LTE-M Matching Network topology



Comm. Standard	Component	Value	Part Number	Manufacturer
LTE-M	F1	0 Ohm		
	F2	0 Ohm		
	F3	0 Ohm		
	F4	0 Ohm		
	Z1	19nH	LQW18AN19NG80	Murata
	Z2	8.6nH	LQW15AN8N6G80	Murata
	Z3	3.7pF	GJM1555C1H3R7WB01	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.

If you need further assistance, please contact our antenna specialists.

GET HELP

Your antenna design ready to use

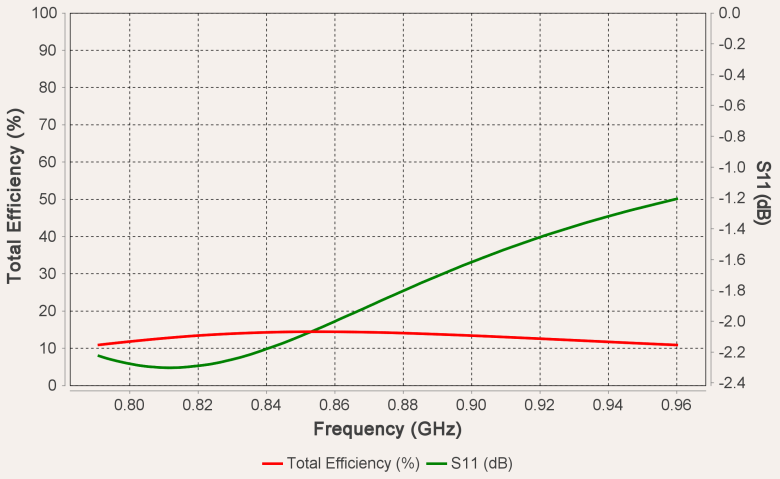
ANTENNA 2

Your Design overall Performance

Expected device performance with antenna 2

Your prototype using the NN03-310 antenna is expected to achieve the reflection coefficient (in dB) and total efficiency (in %) as shown in this graph.

Rule of thumb: it is desirable to have a reflection coefficient below -6 dB, ensuring proper impedance matching of the antenna component and optimized total efficiency.



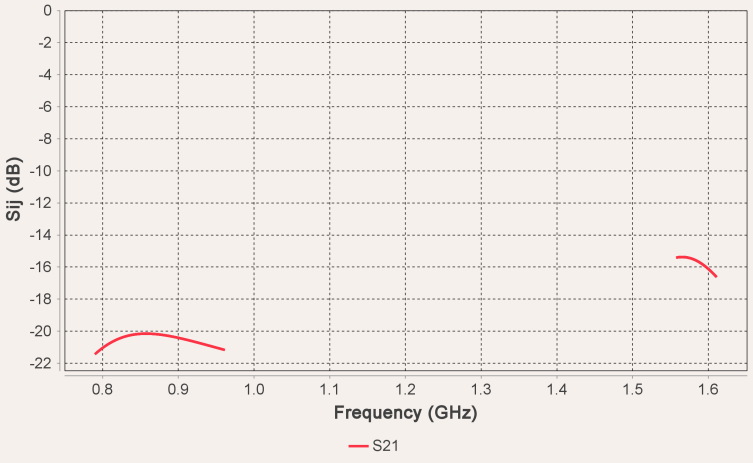
INCREASE YOUR PERFORMANCE: to increase the performance we recommend evaluating your PCB again with increased dimensions (increasing length by 10 mm typically results in a total efficiency improvement of 0.5 dB).

	TRIO mXTEND™ (NN03-310) for LTE-M		
Frequency (MHz)	791	960	Avg 791 - 960
Total efficiency (%)	10.9	10.9	13.1

Antenna Coupling

In multiantenna devices coupling coefficients indicate whether the antennas are electromagnetically isolated or not. For each antenna "i" to antenna "j", there is a Sij coupling trace that indicates the isolation levels between the antennas in the studied frequencies.

Rule of thumb: it is desirable to have antenna couplings below -10 dB, ensuring proper isolation between antenna components and optimized total efficiency.



TECHNICAL INFORMATION: coupling between two antennas can be significantly improved by increasing the distance between them.

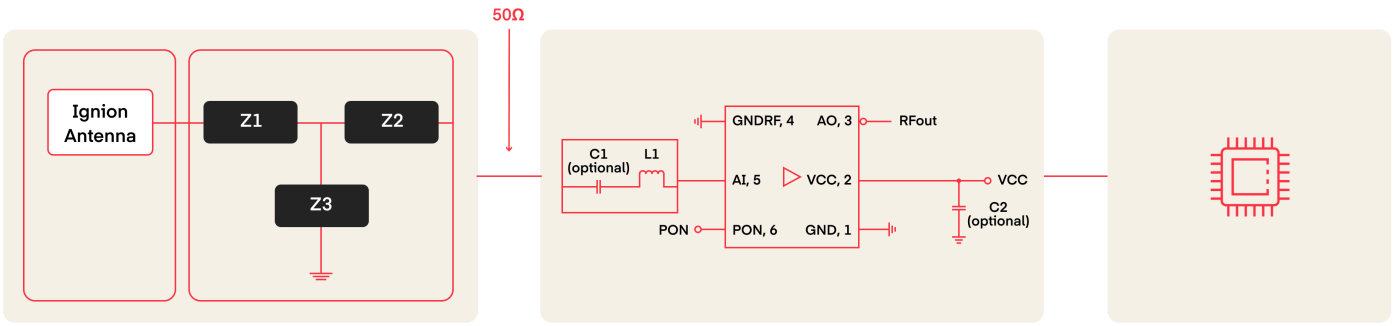
If you need further assistance, please contact our antenna specialists.

GET HELP

GNSS design recommendations

For GNSS applications it is recommended to include a Low Noise Amplifier (LNA) component unless the RF module already has it integrated.

The LNA should be included between the Matching Network of the antenna and the in/out port of the RF module.



It is important to follow the guidance included in the datasheet of the LNA supplier to achieve 50 ohms at the input of the antenna matching network.

If you need further assistance, please contact our antenna specialists.

GET HELP

Certification targets for LTE bands

To ensure your cellular IoT project success, it is important to ensure proper antenna performance in the context of the cellular certification targets early in the design phase. Verify the operator Total Radiated Power (TRP) requirements as these will set the main boundaries for your antenna requirements.

TRP Calculation:

TRP (dBm) = $\eta a(dB) + P$ *in RF module*

For example, in the US market AT&T has a set of requirements for IoT devices. A product with LTE CAT 1 is required to deliver a minimum of 30% of total efficiency in the LFR and a minimum of 50% in the HFR to meet the TRP (Total Raidated Power) requirements. Contact Ignion support to get help identifying the minimum antenna performance needed and the resulting minimum PCB size needed.

Band	Minimum TRP Requirement	Minimum TIS Requirement (Primary Antenna)	Minimum TIS Requirement (Secondary Antenna)
2	+20.0 dBm	-91 dBm/10MHz	-87 dBm/10MHz
4	+20.0 dBm	-93 dBm/10MHz	-89 dBm/10MHz
5	+18.0 dBm	-89 dBm/10MHz	-85 dBm/10MHz
12	+18.0 dBm	-91 dBm/10MHz	-87 dBm/10MHz
14	+17.0 dBm	-87 dBm/10MHz	-83 dBm/10MHz
17	+18.0 dBm	-88 dBm/10MHz	-84 dBm/10MHz
29	-	-88 dBm/10MHz	-84 dBm/10MHz
30	+19.0 dBm	-91 dBm/10MHz	-87 dBm/10MHz
66	+20.0 dBm	-93 dBm/10MHz	-89 dBm/10MHz

If you need further assistance, please contact our antenna specialists.

GET HELP

General design recommendations

for performance optimization
with Virtual Antenna® technology

1 CLEARANCE AREA

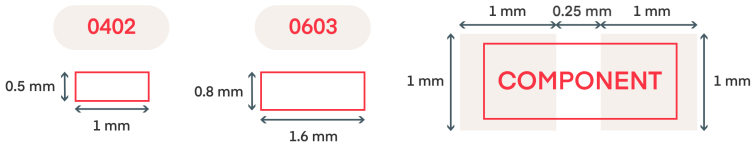
Consider the recommended clearance area in all directions around the antenna component. It must be free from electronic components, traces and ground plane in all PCB layers including the area underneath the antenna.

2 ANTENNA LOCATION

Keep the antenna in a corner of the PCB, as far as possible from other metalic components.

3 MATCHING NETWORK

Arrange pads for the Matching Network to host the 0402/0603 SMD components. Place pads as close as possible to the antenna feeding point and within the ground plane area. The Matching Network might need returning as other elements of your design are placed around the antenna. Use preferably high Q and tight tolerance components.



4 MATERIALS

Use low loss materials (i.e. PET plastic, Polyethylene Terephthalate) for the housings and enclosures.

5 MULTI-LAYER PCBs

Ensure that all the grounding sections in every PCB layer are properly connected through vias.

6 TRANSMISSION LINE AND RF CHIP

Design your transmission line connecting the Matching Network to your RF chip so that its characteristic impedance is 50 Ohms. Locate your RF chip as close as possible to the Matching Network to reduce losses.

7 GROUND PLANE LAYER

Ensure a continuous conducting ground plane in at least one layer of your PCB. Always maximize the surface of your ground area on the PCB of your device to maximize its radiation performance.

NEED MORE HELP COMPLETING YOUR DESIGN?

You are now ready to start designing your full device and building your prototype following the recommendations. Once you have designed your PCB layout you can submit the design file to Ignion for a sanity check.

SUBMIT YOUR DESIGN FILES

Ignion offers complete end-to-end design services.