oxion

Created by ignion

AI-POWERED ANTENNA INTEGRATION

Project Report

COMPANY NAME	DATE	SERVICE REPORT NUMBER	IGNION SUPPORT
NMBU	30/07/2024	bbf5c00d_8674_4c09_b461_86c8bd5d4d13	support@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
Educational & Research

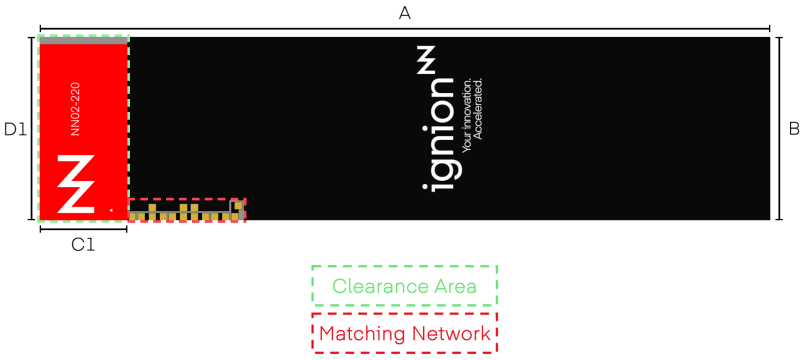
PCB DIMENSIONS
120.0 x 25.0 mm

RECOMMENDED ANTENNA(S)
Ant. 1: NN02-220

COMMUNICATION STANDARD(S) & FREQUENCY RANGE(S)
LTE-M: 791 - 960 MHz; 1710 - 2200 MHz

Best antenna placement on your PCB

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



PCB		1: NN02-220	
Measure	mm	Measure	mm
A	120.0	C1	12.0
B	25.0	D1	25.0

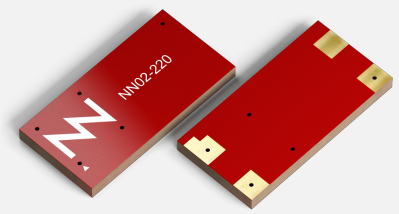
The sketch above is an approximate representation of the PCB design. The accurate model can be found in the Desing_Files_NNS1.0.zip attached in the same email where the report was received.

Your antenna design ready to use

ANTENNA 1

ALL mXTEND™ (NN02-220):

- **Application:** Educational & Research.
- **Frequency range(s):** 791.0-960.0 MHz, 1710.0-2200.0 MHz for LTE-M
- **Tuning your antenna:** the optimized matching network is shown below.
- **Antenna footprint:** please refer to the datasheet link.



ALL mXTEND™

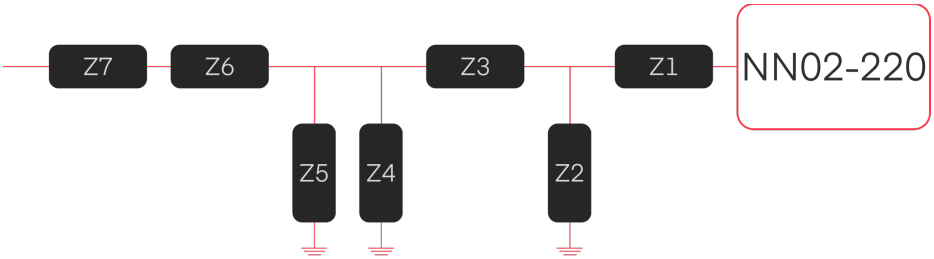
Dimensions 24.0 x 12.0 x 2.0 mm

Datasheet https://ignion.io/files/DS_NN02-220.pdf

Antenna product page →

Matching network antenna 1

LTE-M matching network topology



Comm. Standard	Component	Value	Part Number	Manufacturer
LTE-M	Z1	1.6nH	LQW15AN1N6C80	Murata
	Z2	7.5nH	LQW15AN7N5G80	Murata
	Z3	8.5pF	GJM1555C1H8R5WB01	Murata
	Z4	open		
	Z5	1.5pF	GJM1555C1H1R5WB01	Murata
	Z6	2.3nH	LQW15AN2N3G80	Murata
	Z7	1.7pF	GJM1555C1H1R7WB01	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

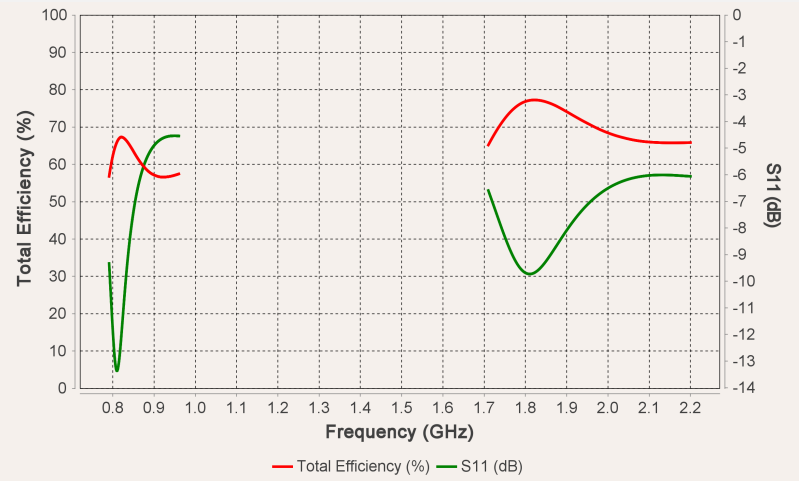
ANTENNA 1

Your design overall performance

Expected device performance with antenna 1

Your prototype using the NN02-220 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).

	ALL mXTEND™ (NN02-220) for LTE-M		
Frequency (MHz)	791	960	Avg 791 - 960
Total efficiency (%)	56.8	57.4	60.5
Frequency (MHz)	1710	2200	Avg 1710 - 2200
Total efficiency (%)	65.2	65.8	70.4

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

GET HELP

Certification targets for LTE bands

To ensure passing cellular certification in your IoT project, it is important to secure 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 requirements might vary for every network carrier and region of operation. Use this formula to calculate corresponding TRP estimation based on the total efficiency given in this report and RF module outpot power. Verify that the estimated TRP is greater than minimum TRP operator requirements.*

TRP calculation:

$$TRP = P_{RF} + 10 * \log_{10}\left(\frac{Ef}{100}\right)$$

$Ef = Min.Total\ Efficiency\ (\%)$

$P_{RF} = Power\ RF\ module\ (dBm)$

$TRP = Min.TRP\ (\mathrm{dBm})$

*Note these will only be estimations. To verify final TRP performance, physical lab tests are required.

US region example with AT&T:

This example shows the calucation of the minimum total efficiency (in %) required to pass the LTE Cat 1 TRP requirements from AT&T in the US region when using a RF module of +23 dBm (Power Class 3) output power. Contact Ignion support if you need any help in assessing your specific performance requirements.

Band	Minimum TRP Requirement	Minimum Total Efficiency	Minimum TIS Requirement (Primary Antenna)	Minimum TIS Requirement (Secondary Antenna)
2	+20.0 dBm	50%	-91 dBm/10MHz	-87 dBm/10MHz
4	+20.0 dBm	50%	-93 dBm/10MHz	-89 dBm/10MHz
5	+18.0 dBm	31%	-89 dBm/10MHz	-85 dBm/10MHz
12	+18.0 dBm	31%	-91 dBm/10MHz	-87 dBm/10MHz
14	+17.0 dBm	25%	-87 dBm/10MHz	-83 dBm/10MHz
17	+18.0 dBm	31%	-88 dBm/10MHz	-84 dBm/10MHz
29	-	-	-88 dBm/10MHz	-84 dBm/10MHz
30	+19.0 dBm	40%	-91 dBm/10MHz	-87 dBm/10MHz
66	+20.0 dBm	50%	-93 dBm/10MHz	-89 dBm/10MHz

(Source: AT&T Radiated Performance Requirements version 1.6)

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.