oxion

Created by ignion

AI-POWERED ANTENNA INTEGRATION

Project Report

COMPANY NAME	DATE	SERVICE REPORT NUMBER	IGNION SUPPORT
NMBU	08/08/2024	cdf083a8_3648_411a_b696_670df31aeb64	gjo@indesmatech.com

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

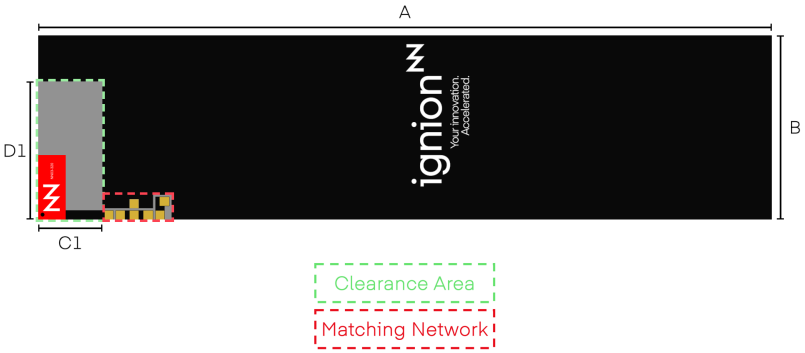
PCB DIMENSIONS
90.0 x 20.0 mm

RECOMMENDED ANTENNA(S)
Ant. 1: NN03-320

COMMUNICATION STANDARD(S) & FREQUENCY RANGE(S)
Bluetooth: 2400 - 2484 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: NN03-320	
Measure	mm	Measure	mm
A	90.0	C1	7.0
B	20.0	D1	15.0

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:** Educational & Research.
- **Frequency range(s):** 2400.0-2484.0 MHz for Bluetooth
- **Tuning your antenna:** the optimized matching network is shown below.
- **Antenna footprint:** please refer to the datasheet link.

DUO mXTEND™

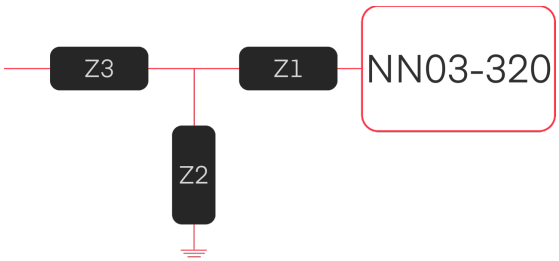
Dimensions 7.0 x 3.0 x 2.0 mm

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

Antenna product page →

Matching network antenna 1

Bluetooth matching network topology



Comm. Standard	Component	Value	Part Number	Manufacturer
Bluetooth	Z1	3.6nH	LQW15AN3N6G80	Murata
	Z2	1.6nH	LQW15AN1N6C80	Murata
	Z3	1.2pF	GJM1555C1H1R2WB01	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

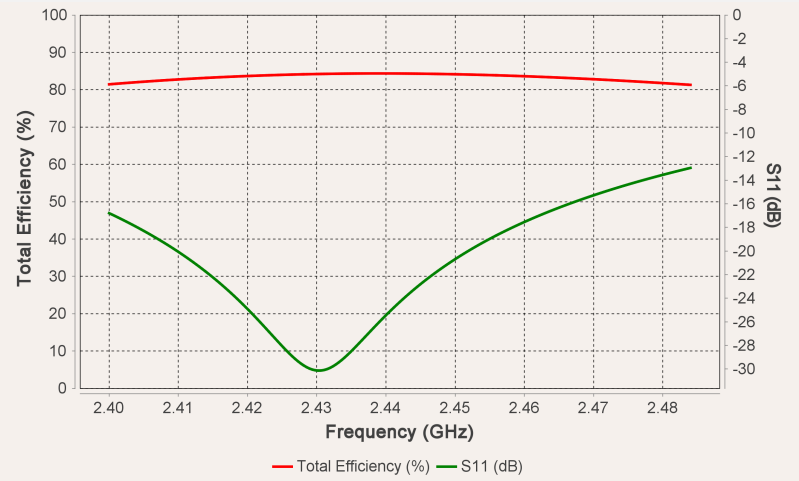
ANTENNA 1

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 Bluetooth		
Frequency (MHz)	2400	2484	Avg 2400 - 2484
Total efficiency (%)	81.4	81.3	83.3

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.