ALMA MATER STUDIORUM
UNIVERSITÀ DI BOLOGNA

DESIGN ROXTAG 2.0
INTEGRATED CHIP ANTENNA
V1

Originator:

Function	Name
Project Developer	Leo Torchia

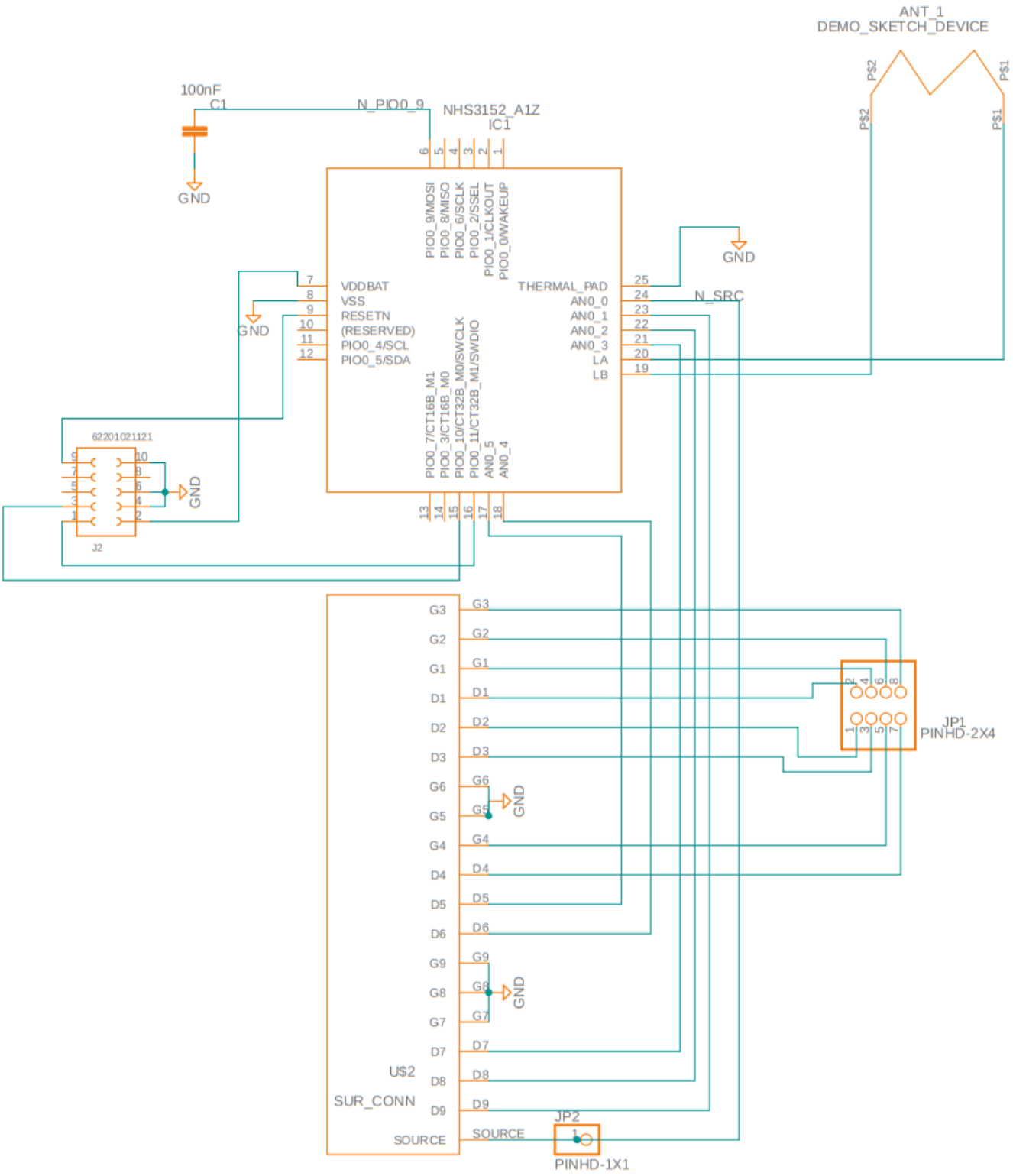
Version:

Person	Date	NOTE
Leo Torchia	2021-01-05	In this Version Antenna/chip **work** (code uploaded successfully. Issue integrating PCB with Sensor (pads not conducting), and antenna is on top layer of PCB vs bottom of board (as in DEMOBOARD)
Leo Torchia	2021-01-10	Added the capacitor connection to the pin connection list (was forgotten). Updated list of useful files.
Leo Torchia	2022-08-26	Minor note – Except fro GND, there is no crossover between pads connected to chip and pads connected to external pins --> so you cannot test a device connected to chip with external device. This isn't an issue as anyhow the internal resistance of NHS3152 would not allow for good testing.

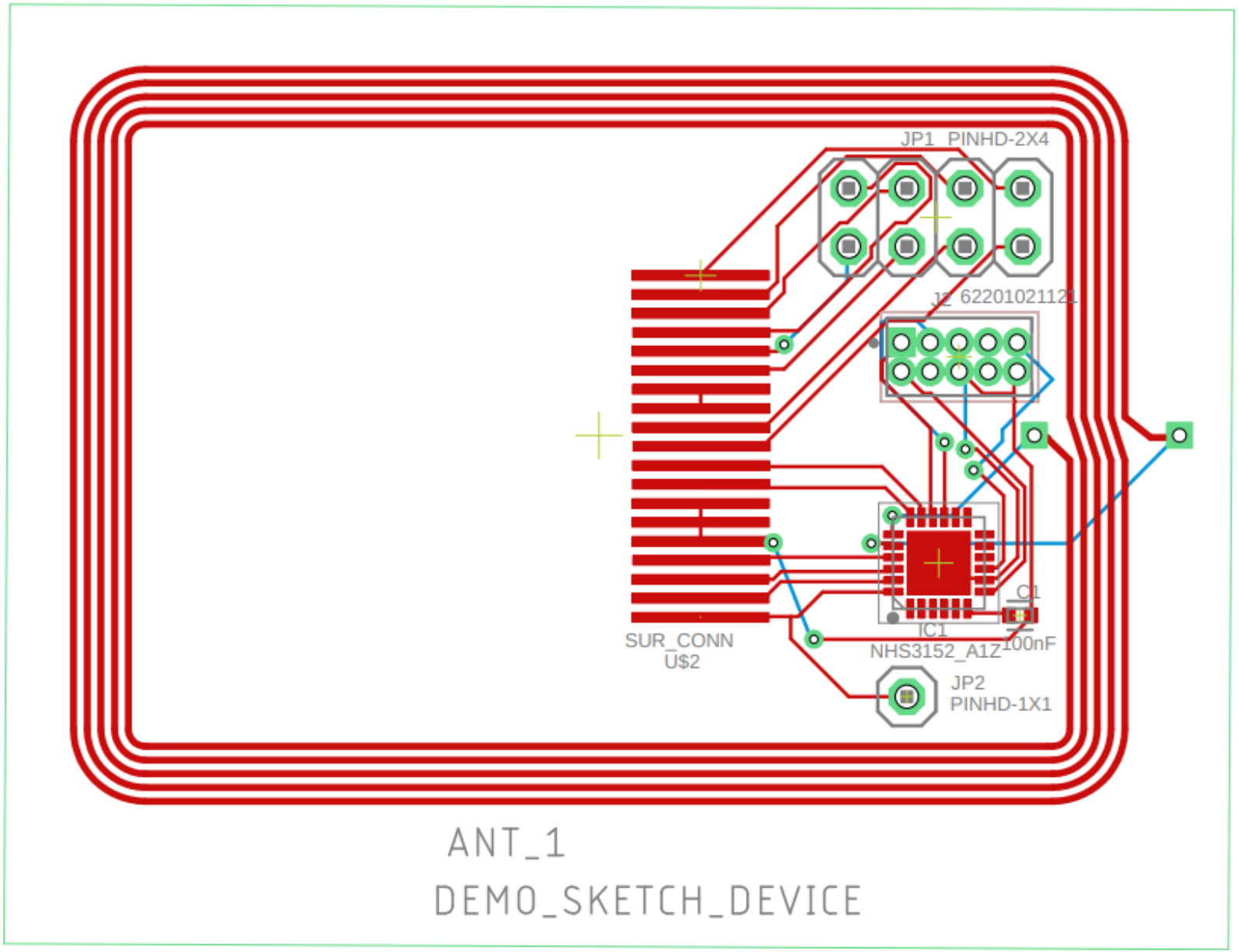
Contents

SCHEMATIC V6

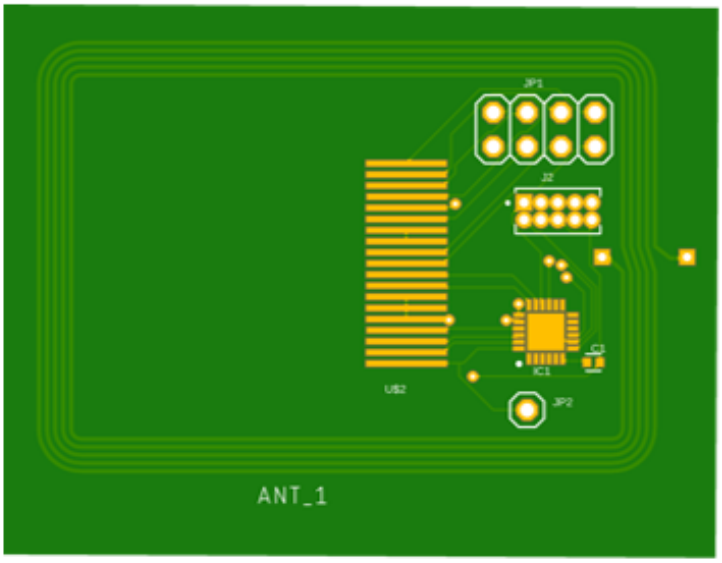
NHS3152 connections for: sensor - antenna DEMO -JPLINK - cap & test pins



PCB

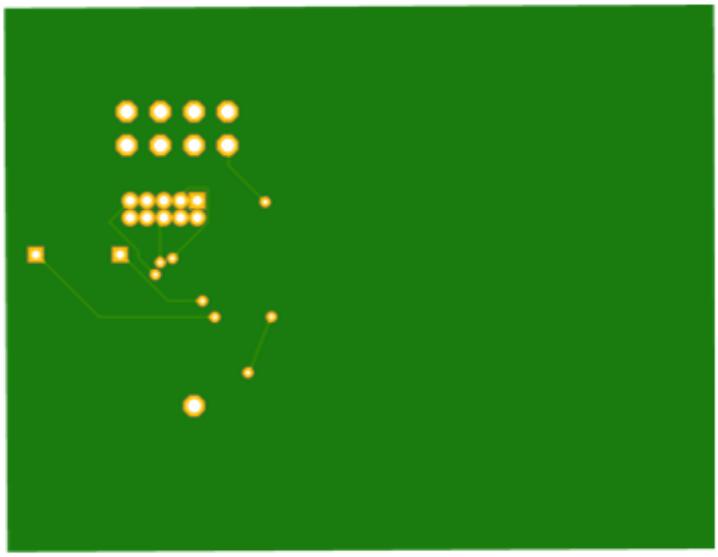

JP1 PINHD-2X4
J2 62201021121
SUR_CONN
U$2
IC1
NHS3152_A1Z
C1
100nF
JP2
PINHD-1X1
ANT_1
DEMO_SKETCH_DEVICE

TOP



TOP PCB with antenna, chip, connector to sensor and to pins

BOTTOM



Bottom PCB
Connect wires unable to connect from TOP

DRILLS

Drill holes for non-SMD components

General description

This Document contains the instructions to reproduce the Designed PCB which integrates the NFC chip NHS3152, its antenna, and pads to Connect to the Sensor. The chip/antenna/energy harvesting process was successfully tested. Minor issue is how to connect the pads to the sensor's chip.

Files/links

Files for fabrication

	FILES FOR FABRICATION: VERSON1-NXP-VALIDATION	
	Assembly	Pick&Place files: front back
	DrillFiles	drill**.XLN**
	GerberFiles	.grb – files with traces
	ODBFiles	Different format to gerber , combines gerber with pick&place
	BOM**.excel**	Bill of materials. The important is manufacturing number – then company sources it on their own.
	GerberFiles**.zip**	.zip to read with https://gerber.ucamco.com/
	Netlist**.ipc**	List of connections – I never used it.
	.pdf	PCB & SCHEMATIC.
	.png	Top – BOT- DRILLS
	V1-PCB-FILES-2021-11-02**.zip**	A zip of the whole project

Files in Fusion

	FILES IN FUSION – Not in this folder, just as reference.	
	NHS3152 Component/ NHS3152_LIBRARY_FOOTPRINT	Chip footprint
	NXP-validation/ NXP-example-Antenna-chip-connections	Actual project
	Connector_library/ Surface_connector_library	The sensor chip connection.
	Antenna_Library_comp/Antenna_Library_4-5-6-DEMO	Antenna used: DEMO_SKETCH_DEVICE

Other/useful

	OTHER FILES/USEFUL	
	release_mra2_nhs3152\release_mra2_12_4_nhs3152\docs\NHS3152 Demo PCB	NXP Gerber from which I modelled the antenna.
	https://gerber.ucamco.com/	Online gerber viewer
	https://www.nxp.com/docs/en/data-sheet/NHS3152.pdf	Datasheet with NHS3152 footprint dimensions.

Bill of Materials

COMPONENTS		
PART	**Manufacturer n.**	**NOTE**
NS3152	NHS3152/A1Z	IOT integration and readout
Antenna	From demo sketch	Energy harvest and communication
Pad Connector	Designed	Connect to sample design
Capacitor	CL05B104KO5NNNC	Power chip while Ndef communication ongoing (which interrupts energy harvesting)
PINHEAD 2*4	10129381-908002BLF	Connection for external testing
PINHEAD 1	2301-6111TG	Source connection for external testing
CONN HEADER	62201021121	Connect to JP-Link
SOFTWARE		
FUSION360		Software used for Design
geerber		See gerber files online (NHS3152 Demo PCB)

DESIGN GUIDE

The design was done Using Fusion360 and Fusion360electronics.

Antenna design

SKETCH TO DXF	
STEP	**NOTE**
Create square rectangular centered sketch to internal dimensions	
Fillet corners (arbitrary small)	
Offset (X-times) always from line 1.	
Use sketch lines and measurements to make the transform the rectangles into a single coil.	
Document settings → units : make sure units are inches	#! Important to have correct dimensions when importing. Inch to inch works. mm to mm not sure.
Click on Sketches – right click sketches – save as dxf – File is now saved as DXF	
NEW ELECTRONIC LIBRARY	
Create new electronic library → save	
NEW FOOTPRINT	
Create new footprint → save	
(in cmd line) RUN → import-dxf → ok	
Browse to dxf → set units inches → set linewidth to 0.3mm (convert to inches) → ok	
Check antenna dimension are correct	
Cancel all Sketch lines	
Attach 2 pth pads	
Cmd→RUN→ set_name_value	
NEW SYMBOL	

Create new symbol → draw symbol you like and attach 2 pins to it (here chose if pins are IO or what you want)	
Run→ set_name_value	
DEVICE	
create new device → new local package ok	Device name is what shows up in library
Add part → symbol	new component should be in your library
CHECK COMPONENT	
File – new electronic design → Trial_for_instructions (name of the library) TRIAL (name of component)	

Chip package design

STEP	NOTE
From Library → create new package → QFN	
Transpose details from Datasheet	
Package is added as footprint called QFN. Create a symbol → name the pins. I think that if you name the pins correctly, they will map easier.	
Create new device → map pins and add package etc.	

Pad connector design

STEP	NOTE
This is a new library – made a new DEVICE. The footprint is stretched surface mounts. Dimensions are a copy of the pads from the sensor footprint, just reversed (since the sensor will go upside down to connect.)	

Integration

STEP	NOTE
Create new electronic design	
Add components: NS3152; Antenna; Pad Connector; Capacitor; PINHEAD 2*4; PINHEAD 1; CONN HEADER	
CONN HEADER connects following JPLINK in DEMOSKETCH	#! this piece shows up mirrored in schematic. (mirrored in pcb after schematic). So I mirrored it in schematic so it connections are ok in pcb.
PAD connector to PINHEADs, NHS3152;	Connect PADS so that Gates to gnd or to test pinheads, Source in common to a DAC output and Drains to input pins ANA.. Bring Source to PINHEAD for testing as well.

PCB	
Add pieces onto PCB and arrange in a satisfactory geometry. Use automatic Router to make the best routing. Eliminate all routes on the left of the pads (where chip goes)	
Document using ODD+ (gerber with pick&place), gerber, pick and place, BOM	
Redo BOM so that parts have actual manufacturer number on them	

Pin connections

From		TO	NOTE
VSS		GND	The central Thermal Pad will be the common chip ground.
Thermal Pad		GND	
LA-LB		Antenna	Pads for antenna
J2 connections – Copy the J2 connection of NHS3152DB – so I can connect to LPCLINK			
9		RESTET	There are a copy of the LPC-Link connection to NHS3152DB – this allow to connect to the PC and upload code via cable vs NFC.
[10-6-4]		GND	
1		PIO0_11	
3		PIO0_10	
2		VDDBAT	
Capacitor connection			
PIO0_9	C1[100nf]	VSS (**GND**)	This connects the capacitor to the circuit – This is used to power the Circuit while the Chip is communicating with the phone, thus not harvesting energy .
Source connector to CHIP Connections			
ANA_0		Source	Measure Resistance between Source and any of D9-5 (use DAQ-ADC-12C between these)
ANA_1		D9	
ANA_2		D8	
ANA_3		D7	
ANA_4		D6	
ANA_5		D5	
Source Connector to JP1/JP2 connections			
Source		JP2	Connections to PINS so that I Can verify with an external instrument the contact pads are connecting. However since the same device isn't connected to both CHIP and external SMU, cannot verify that chip is measuring the correct resistance.
G7-8-9-7-6		GND	
JP1 connections			
D4		7	
G4		5	
D3		3	
D2		1	
D1		2	
G1		4	
G2		6	
G3		8	

Errors/improvements to design

The Design worked: Code was successfully uploaded to chip, and a resistance was measured.

Connection issue

However it isn't easy to connect the surface mount pads to the Sensor – placing them in contact doesn't actually bring contact, and Molybdenum cannot be soldered. A new design with a zif connector or something should be implemented. There are also no direct pins for testing the ADC-DAC.

Antenna position

Antenna was position on top layer – whilst in design it is on bottom layer.