



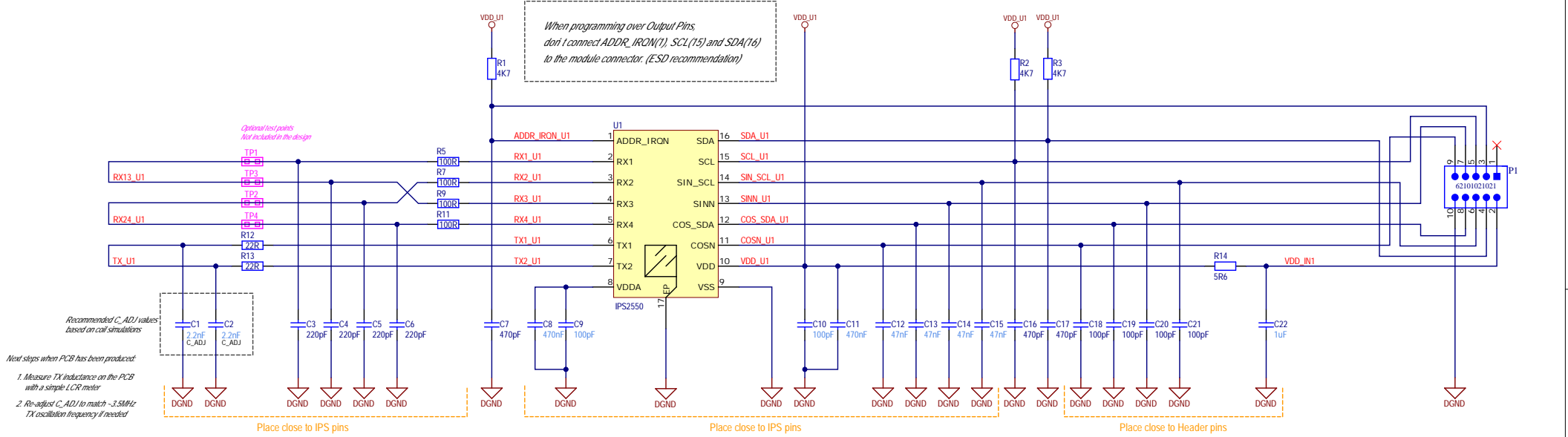
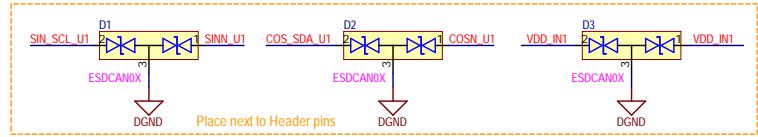
R_153_IPS2550_4x90_OD62_ID38

Date: 16.02.2022
Project: R_153_V10_IPS2550_4x90_OD62_ID38.PrjPcb
Version: 1.0
Content: Schematic
Sensor PCB Layout
Sensor PCB 3D Model
Bill of Materials
PCB Manufacturing Requirements
Sensor PCB Layer Stack
Sensor PCB Placement
Target PCB Layout
Target PCB 3D Model

Note: if you use **IPS2200** instead of **IPS2550**, please swap the connections of the pins 3 and 4

ESD protection diodes are recommended for voltage levels over 20kV.
E.g. using 150pF / 3300hm ESD gun for test

When programming over Output Pins, don't connect ADDR_IRON(1), SCL(15) and SDA(16) to the module connector. (ESD recommendation)



Recommended C_ADJ values based on coil simulations

Next steps when PCB has been produced

1. Measure TX inductance on the PCB with a simple LCR meter
2. Re-adjust C_ADJ to match ~3.5MHz TX oscillation frequency if needed

$$f_{TX} = \frac{1}{2\pi\sqrt{L_{TX}C_{TX}}} \rightarrow C_{TX} = \frac{1}{(2\pi f_{TX})^2 L_{TX}}$$

C_ADJ = To be adjusted according to coil inductivity. Calculate C_TX with the given formula.
C_ADJ = 2 x C_TX
It's recommended to use C0G or NP0 ceramic capacitors
Use dielectric strength Vr >= 50V X7R capacitors for C7 + (C10 .. C22)

LEGEND

- Component Size 0603
- Component Size 0402
- Net Name
- Optional Placement Suggestion
- Optional Component

Project name	R_153_V10_IPS2550_4x90_OD62_ID38.PrjPcb	Version	1.0	
Sheet title	R_153_V10_IPS2550_4x90_OD62_ID38	Size	A3	
File name	R_153_V10_IPS2550_4x90_OD62_ID38.SchDoc			
Date	23.02.2022	SYS.APP.ENG.TEAM	Drawn by	
			Sheet	1 of 1

PCB: ML4
3.2mm FR4

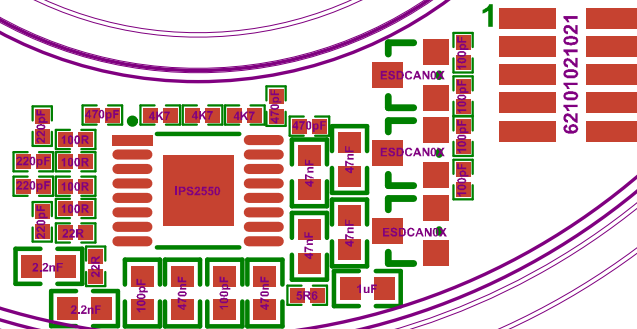
15FEB2022
V1.0

R_153_IPS2550_4x90_OD62_ID38



BIG IDEAS FOR EVERY SPACE














N/A
ADDR IRQ SDA
SCL SIN SCL
SINN COS SDA
COSN DGND



PCB Manufacturing requirements

Project Name	R_153_V10_IPS2550_4x90_OD62_ID38.PrjPcb
Number of copper layers	4
PCB Base material	FR-4
Final PCB Thickness	3.20mm +/-10%
Thickness of copper coating	35um
Final cover	ENIG
Minimal copper width	0.153mm
Minimal copper to copper distance	0.153mm
Via hole/pad diameter	Blind (1:2 & 3:4) 0.2/0.5mm and Through-hole (1:4) 0.3/0.6mm
Slotted holes	Yes, see milling paths
Panel size	70x70mm
Silk screen color	White on TOP
Solder mask color	Green

Board Stack Report

Stack Up		Layer Stack			
Layer	Board Layer Stack	Name	Material	Thickness	Constant
1		Top Paste			
2		Top Overlay			
3		Top Solder	Solder Resist	0,010mm	3,5
4		Top Layer	Copper	0,035mm	
5		Dielectric 1	PP-022	0,200mm	4,5
6		Inner Layer 1	Copper	0,035mm	
7		FR4	Core-043	2,640mm	4,3
8		Inner Layer 2	Copper	0,035mm	
9		Dielectric 2	PP-022	0,200mm	4,5
10		Bottom Layer	Copper	0,035mm	
11		Bottom Solder	Solder Resist	0,010mm	3,5
12		Bottom Overlay			
13		Bottom Paste			
	Height : 3,200mm				

Designator	Value	Quantity	Footprint
D1, D2, D3		3	SOT23-BAT54X
P1		1	62101021021
C22	1uF	1	C0603 3D
C1, C2	2.2nF	2	C0603 3D
R1, R2, R3	4K7	3	R0402 3D
R14	5R6	1	R0402 3D
R12, R13	22R	2	R0402 3D
C12, C13, C14, C15	47nF	4	C0603 3D
C9, C10	100pF	2	C0603 3D
C18, C19, C20, C21	100pF	4	C0402 3D
R5, R7, R9, R11	100R	4	R0402 3D
C3, C4, C5, C6	220pF	4	C0402 3D
C8, C11	470nF	2	C0603 3D
C7, C16, C17	470pF	3	C0402 3D
U1	IPS2xxx	1	TSOP65P640X120_HS-17N

