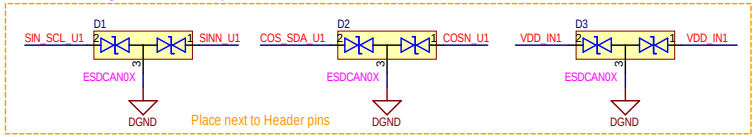
A_28_IPS2550_4x90_OD32_ID15

Date:	09.05.2022
Project:	A_28_V10_IPS2550_4x90_OD32_ID15.PrjPcb
Version	1.0
Content:	Schematic
	Sensor PCB Layout
	Sensor PCB 3D Model
	Bill of Materials
	PCB Manifacturing Requirements
	Sensor PCB Layer Stack
	Sensor PCB Placement
	Target PCB Layout
	Target PCB 3D Model

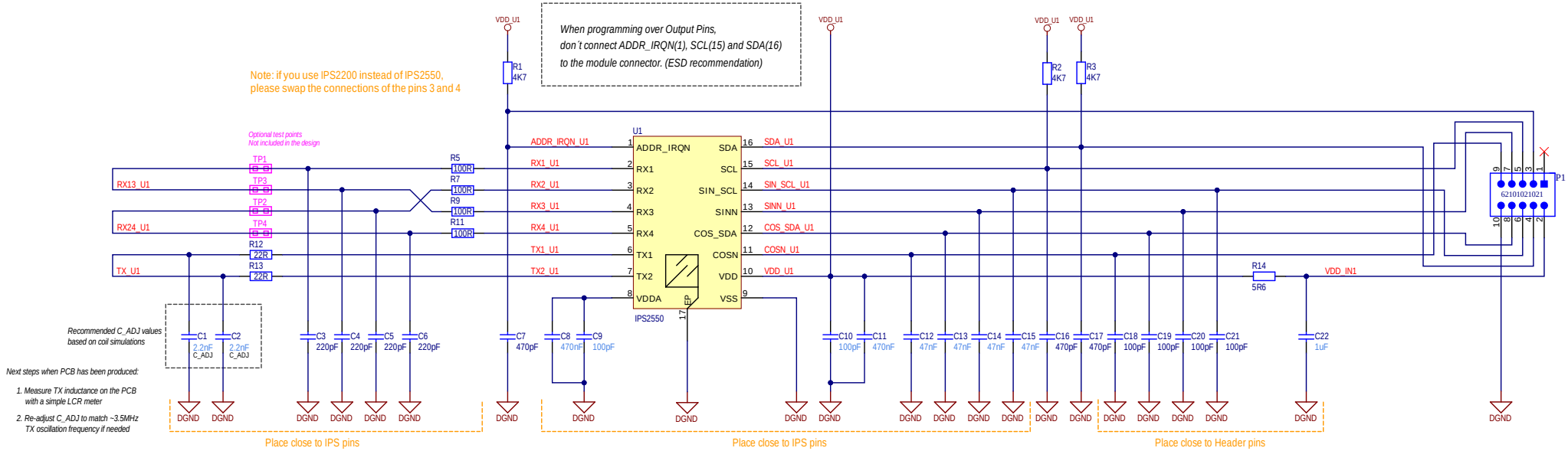
ESD protection diodes are recommended for voltage levels over 20kV.
E.g. using 150pF / 330Ohm ESD gun for test

Optional ESD protection suggestion
Not included in the design due to small available placement area



When programming over Output Pins, don´t connect ADDR_IRQN(1), SCL(15) and SDA(16) to the module connector. (ESD recommendation)

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



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

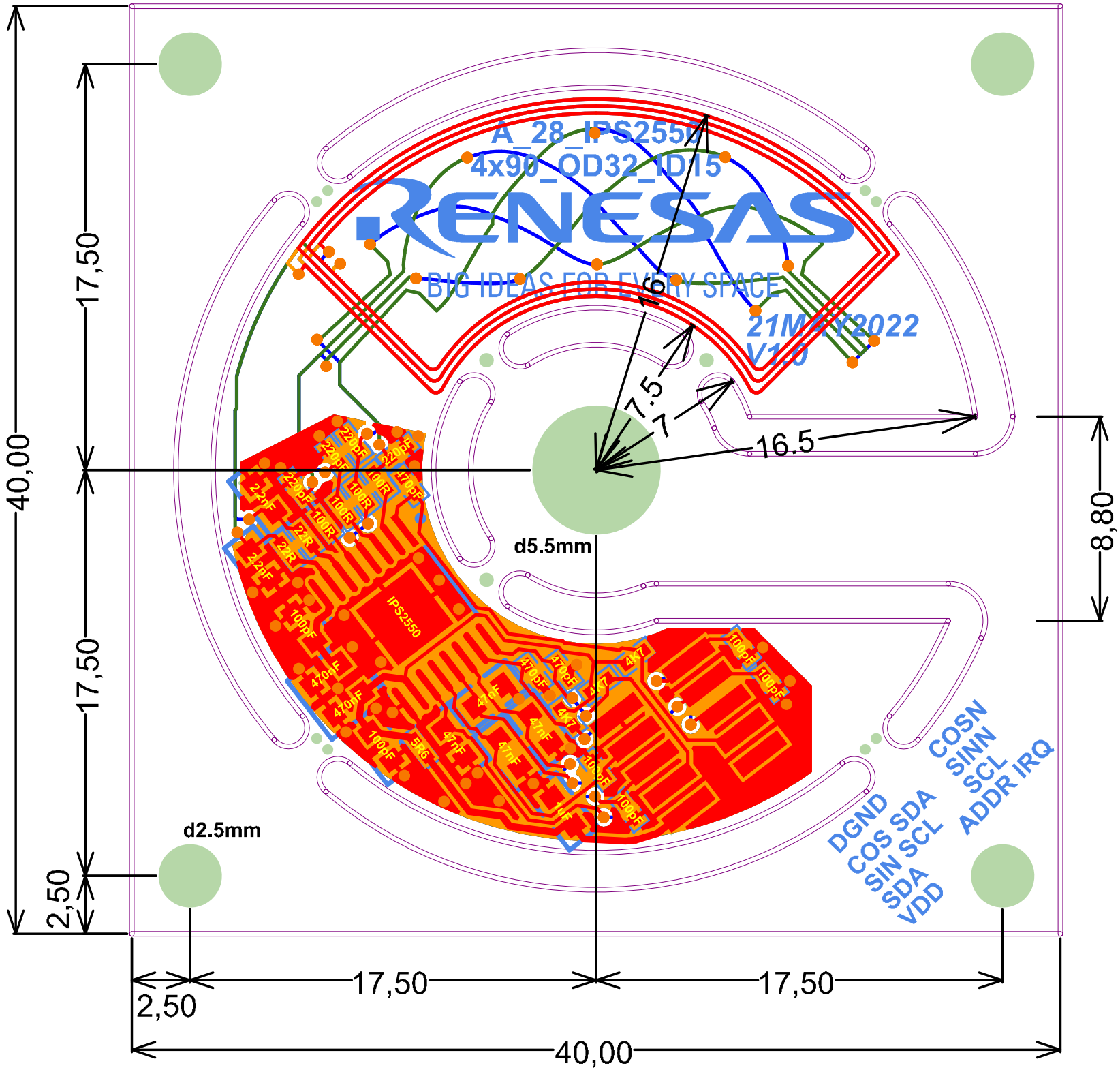
C_ADJ = To be adjusted according to coil incductivity. Calculate C_TX with the given formula.
C_ADJ = 2 x C_TX
It´s recommended to use C0G or NP0 ceramic capacitors

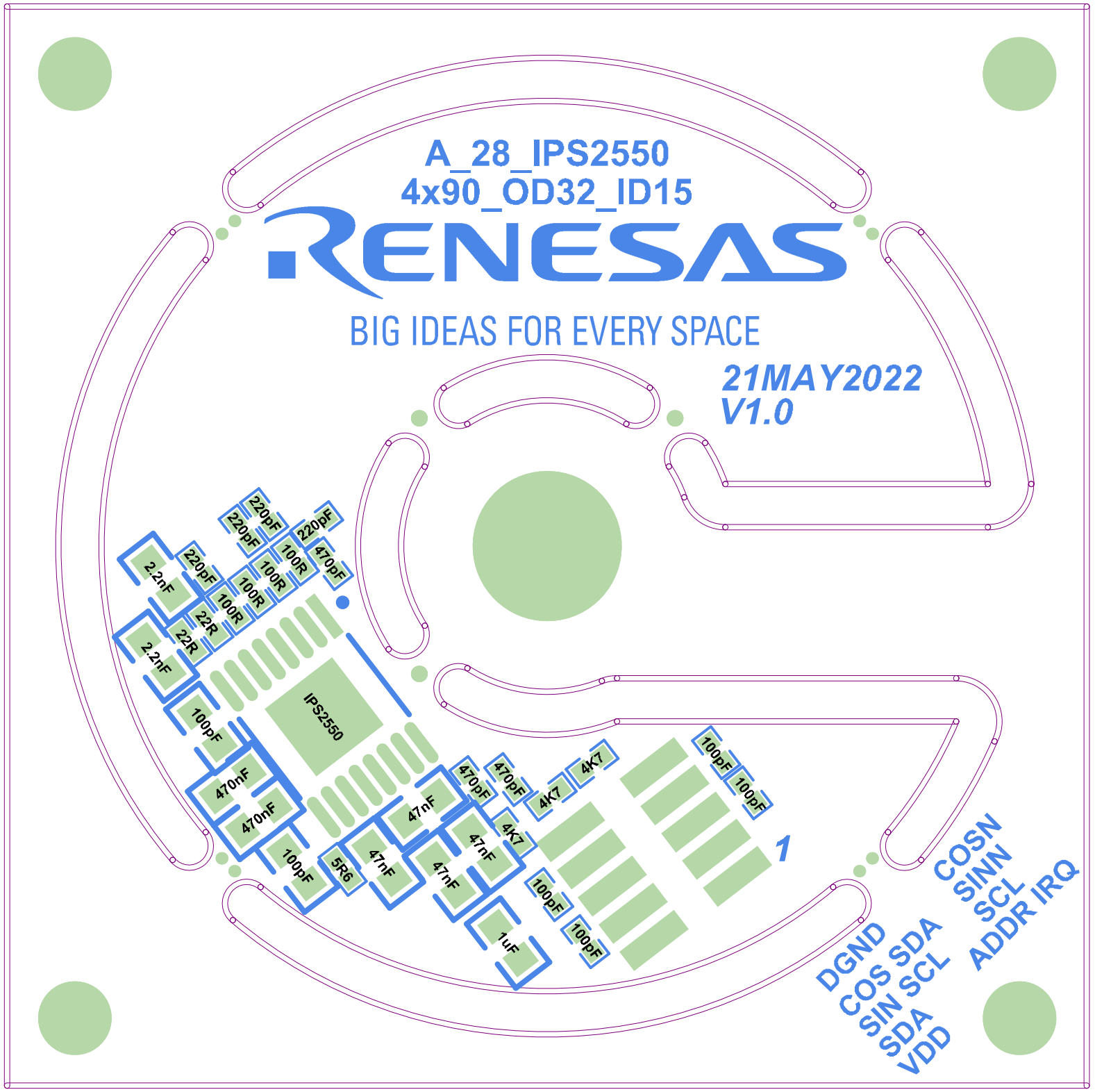
Use dielectric strength Vr >= 50V X7R capacitors for C7 + (C10 .. C22)

LEGEND

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

Project name	A_28_V10_IPS2550_4x90_OD32_ID15.PrjPcb		Version	1.0	RENESAS BIG IDEAS FOR EVERY SPACE
Sheet title	A_28_V10_IPS2550_4x90_OD32_ID15		Size	A3	
File name	A_28_V10_IPS2550_4x90_OD32_ID15.SchDoc				
Date	26.07.2022	SYS.APP.ENG.TEAM	Drawn by	B.GOMBOR	Sheet 1 of 1

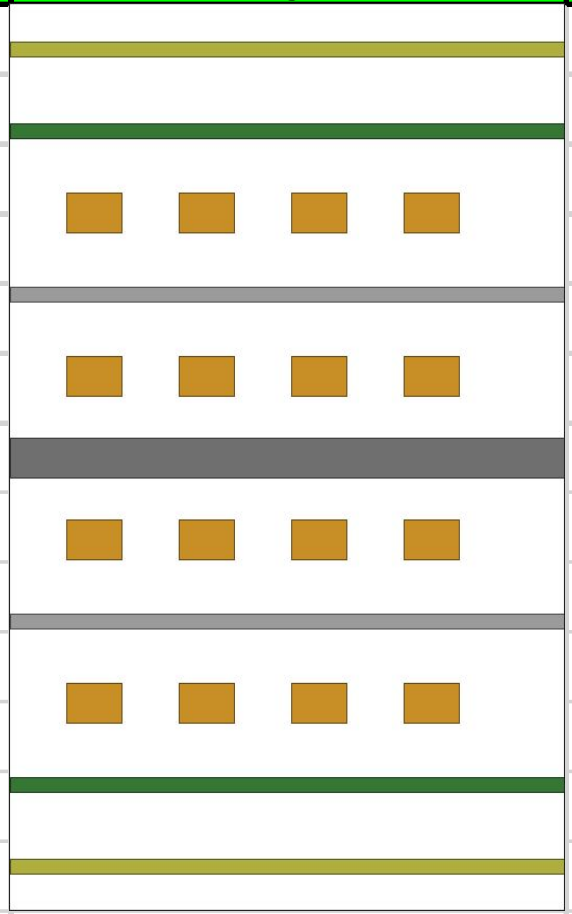

A_28_IPS2550
4x90_OD32_ID15
RENESAS
BIG IDEAS FOR EVERY SPACE
21MAY2022
V1.0
16
7.5
7
16.5
8,80
d5.5mm
d2.5mm
40,00
17,50
17,50
2,50
2,50
17,50
17,50
40,00
COSN
SINN
SCL
ADDR IRQ
DGND
COS SDA
SIN SCL
SDA
VDD


A_28_IPS2550
4x90_OD32_ID15
RENESAS
BIG IDEAS FOR EVERY SPACE
21MAY2022
V1.0
IPS2550
220pF
220pF
220pF
2.2nF
220pF
100R
470pF
100R
100R
100R
22R
22R
2.2nF
100pF
470nF
470nF
100pF
5R6
47nF
47nF
47nF
47nF
470pF
470pF
4K7
4K7
4K7
100pF
100pF
100pF
100pF
1uF
1
DGND
COS SDA
SIN SCL
SDA
VDD
COSN
SINN
SCL
ADDR IRQ

PCB Manufacturing requirements

Project Name	A_28_V10_IPS2550_4x90_OD32_ID15.PrjPcb
Number of copper layers	4
PCB Base material	FR-4
Final PCB Thickness	1.55mm +/-10%
Thickness of copper coating	35um
Final cover	ENIG
Minimal copper width	0.15
Minimal copper to copper distance	0.15mm
Via hole/pad diameter	Through-hole (1:4) 0.2/0.5mm
Slotted holes	Yes, see milling paths
Panel size	40x40mm
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		Diekectric 1	PP-022	0,208mm	4,5
6		Inner Layer 1	Copper	0,035mm	
7		Core	Core-042	0,991mm	4,6
8		Inner Layer 2	Copper	0,035mm	
9		Dielectric 2	PP-022	0,208mm	4,5
10		Bottom Layer	Copper	0,035mm	
11		Bottom Solder	Solder Resist	0,010mm	3,5
12		Bottom Overlay			
13		Bottom Paste			
	Height : 1,567mm				

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

16
15.5
7
d5.5mm
TARGET_4X90
OD31_ID14