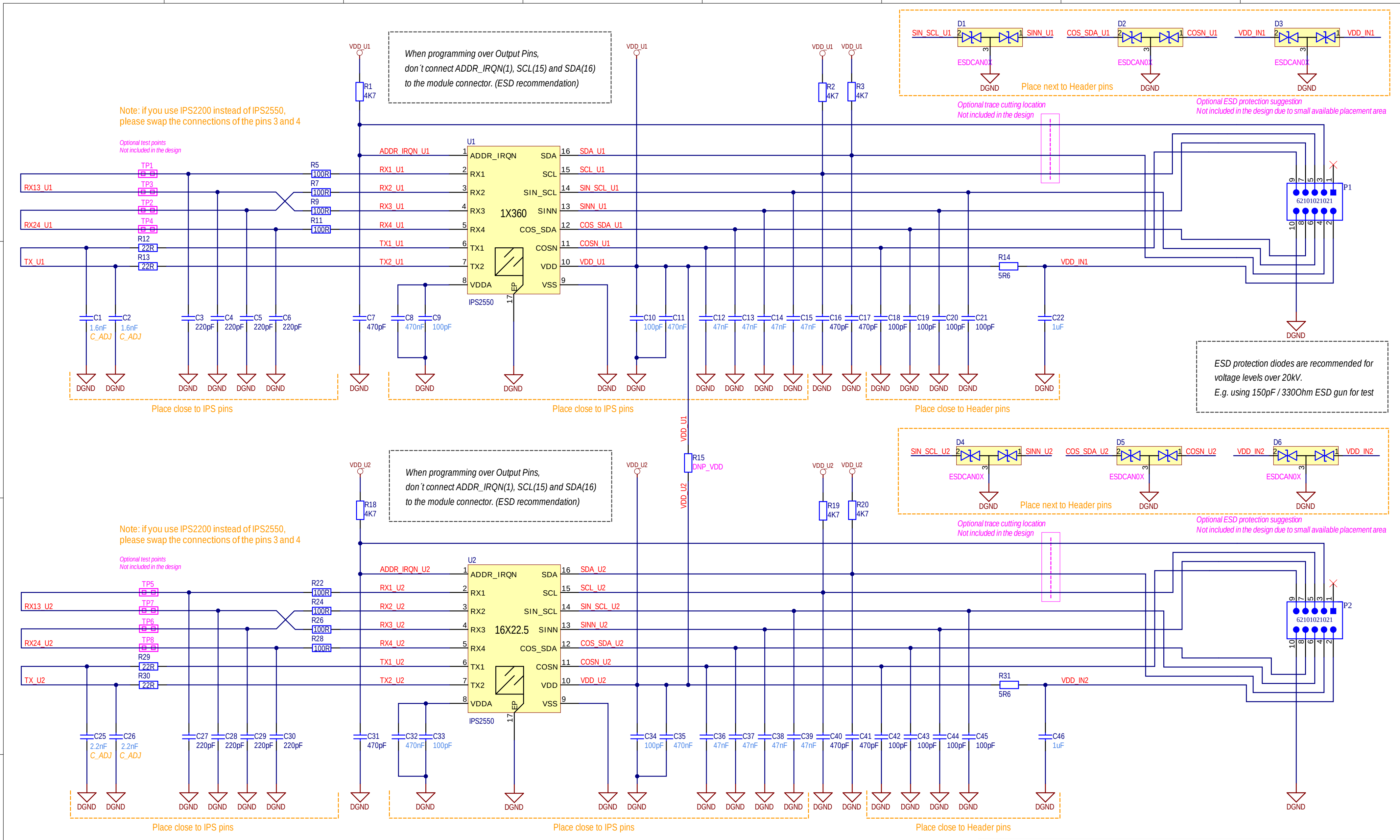
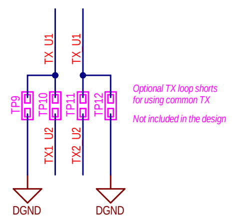
BIG IDEAS FOR EVERY SPACE

R_144_IPS2550_1x16x_OD37_ID13_DUAL

Date:	21.10.2022
Project:	R_144_V10_IPS2550_1x16x_OD37_ID13_DUAL.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





DNP_VDD: Populate R15 = 0E to short VDD_U1 + VDD_U2

C_ADJ = To be adjusted according to coil inductivity. Calculate C_TX with the given formula.
C_ADJ = 2 x C_TX when separate transmitters are in use
Calculate C_ADJ individually for each TX
It´s recommended to use C0G or NP0 ceramic capacitors

Use dielectric strength Vr >= 50V capacitors for C7 + (C10 .. C22) and C31 + (C34 .. C46)

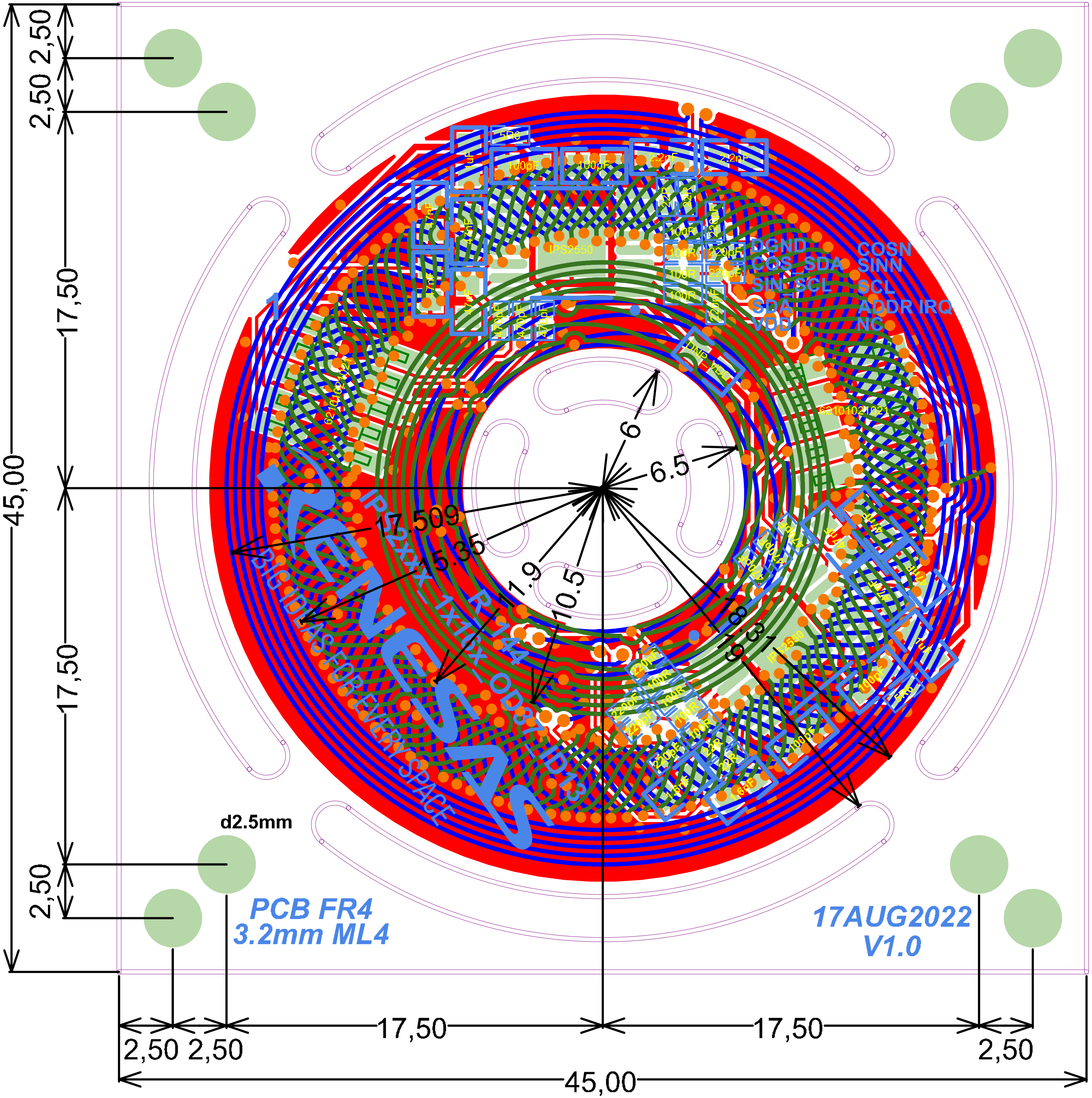
LEGEND

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

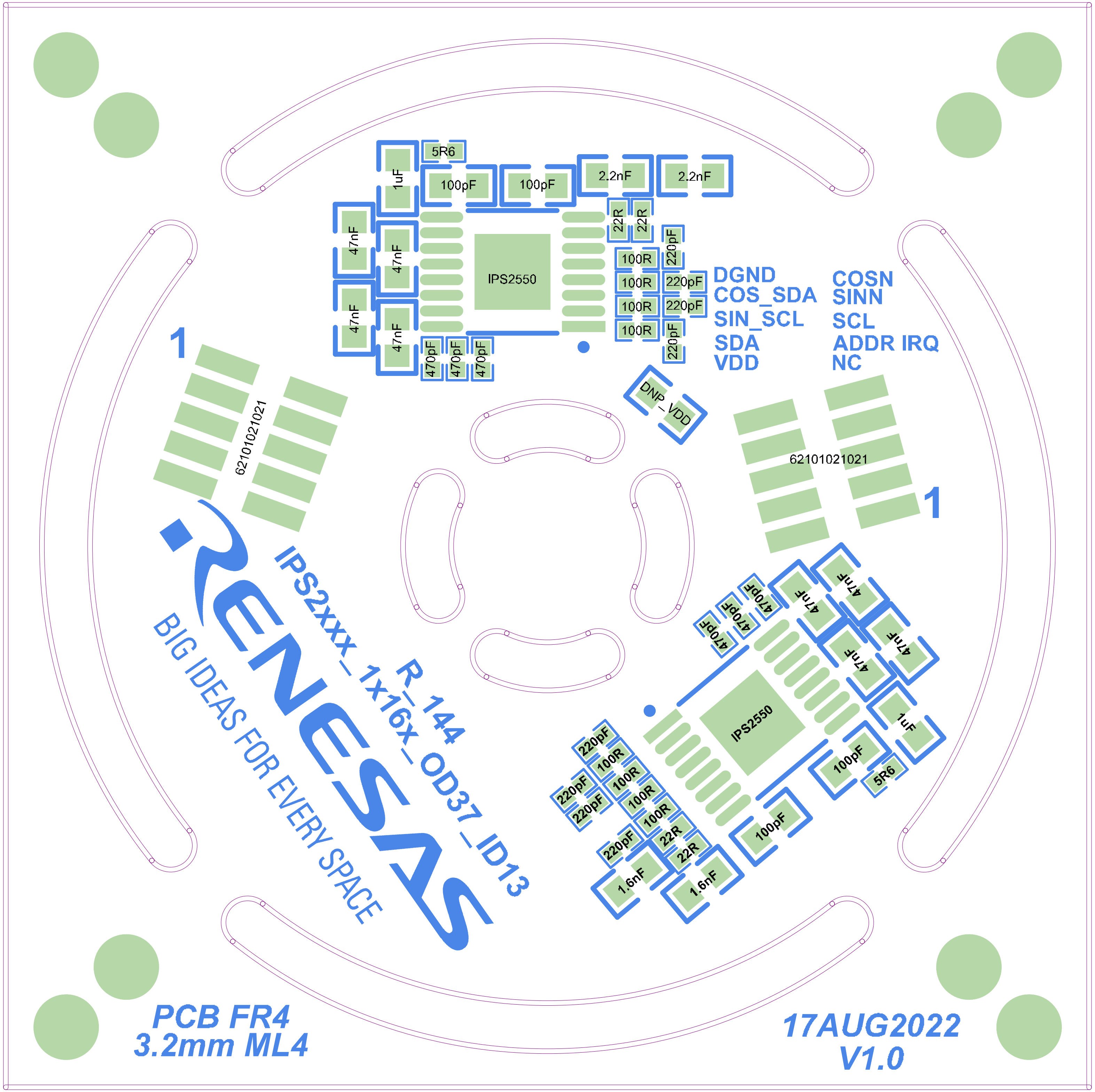
$$f_{TX} = \frac{1}{2\pi\sqrt{L_{TX}C_{TX}}} \rightarrow C_{TX} = \frac{1}{(2\pi f_{TX})^2 L_{TX}}$$

Project name	R_144_V10_IPS2550_1x16x_OD37_ID13_DUAL.PrjPcb	Version	1.0		
Sheet title	R_144_V10_IPS2550_1x16x_OD37_ID13_DUAL	Size	A3		
File name	R_144_V10_IPS2550_1x16x_OD37_ID13_DUAL.SchDoc				
Date	21.10.2022	SYS. APP. ENG. TEAM	Drawn by	B.GOMBOR	Sheet 1 of 1

RENESAS
BIG IDEAS FOR EVERY SPACE



2,50
2,50
17,50
45,00
17,50
2,50
d2.5mm
PCB FR4
3.2mm ML4
17AUG2022
V1.0
17,50
17,50
2,50
2,50
2,50
45,00
6
6.5
17.509
15.35
11.9
10.5
18.31
19
DGND
COS SDA
SIN SCL
SDA
VDD
COSN
SINN
SCL
ADDR IRQ
NC
RENESAS
BIG IDEAS FOR EVERY SPACE


5R6
1uF
100pF
100pF
2.2nF
2.2nF
22R
22R
47nF
47nF
47nF
47nF
IPS2550
220pF
100R
100R
100R
100R
220pF
220pF
220pF
470pF
470pF
470pF
DGND
COS_SDA
SIN_SCL
SDA
VDD
COSN
SINN
SCL
ADDR IRQ
NC
DNP_VDD
1
62101021021
62101021021
1
IPS2xxx_1x16x_OD37_ID13
R_144
BIG IDEAS FOR EVERY SPACE
IPS2550
1uF
100pF
5R6
100pF
1.6nF
1.6nF
22R
22R
PCB FR4
3.2mm ML4
17AUG2022
V1.0

PCB Manufacturing requirements

Project Name	R_144_V10_IPS2550_1x16x_OD37_ID13_DUAL.PrjPcb
Number of copper layers	4
PCB Base material	FR-4
Final PCB Thickness	3.2mm +/-10%
Thickness of copper coating	35um
Final cover	ENIG
Minimal copper width	0.15mm
Minimal copper to copper distance	0.15mm
Via hole/pad diameter	Through-hole (1:4) 0.3/0.6mm, Blind (1:2) (3:4) 0.2/0.5mm
Slotted holes	Yes, see milling paths
Panel size	45x45mm
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
P1, P2		2	62101021021
C22, C46	1uF	2	C0603 3D
C1, C2	1.6nF	2	C0603 3D
C25, C26	2.2nF	2	C0603 3D
R14, R31	5R6	2	R0402 3D
R12, R13, R29, R30	22R	4	R0402 3D
C12, C13, C14, C15, C36, C37, C38, C39	47nF	8	C0603 3D
C9, C10, C33, C34	100pF	4	C0603 3D
R5, R7, R9, R11, R22, R24, R26, R28	100R	8	R0402 3D
C3, C4, C5, C6, C27, C28, C29, C30	220pF	8	C0402 3D
C7, C16, C17, C31, C40, C41	470pF	6	C0402 3D
R15	DNP_VDD	1	R0603 3D
U1, U2	IPS2xxx	2	TSOP65P640X120_HS-17N


17
6
9.5
7.5
16.5
d5.5mm
TARGET_16x22.5_OD33_ID23
TARGET_1x360_OD19_ID12