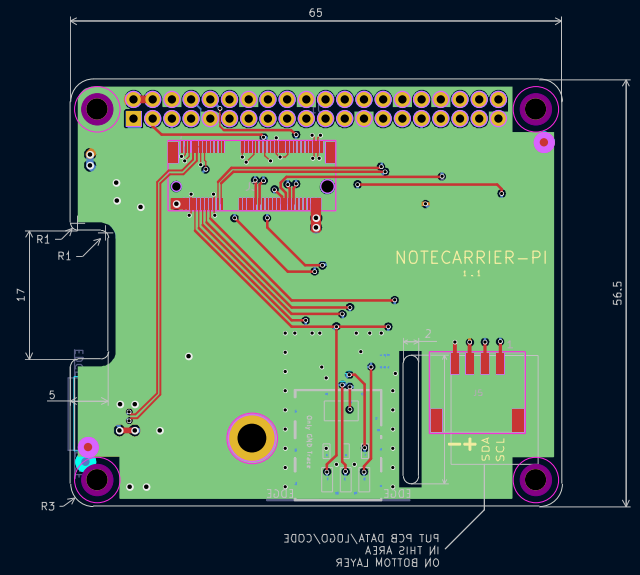
Layer Name	Type	Material	Thickness (mm)	Color
F.Silkscreen	Top Silk Screen	Not specified	0 mm	White
F.Paste	Top Solder Paste		0 mm	
F.Mask	Top Solder Mask	Not specified	0.02 mm	Green
F.Cu	copper		0.035 mm	
Dielectric 1	prepreg	FR4-TG150	0.2 mm	Not specified
In1.Cu	copper		0.035 mm	
Dielectric 2	core	FR4-TG150	1 mm	Not specified
In2.Cu	copper		0.035 mm	
Dielectric 3	prepreg	FR4-TG150	0.2 mm	Not specified
B.Cu	copper		0.035 mm	
B.Mask	Bottom Solder Mask	Not specified	0.02 mm	Green
B.Paste	Bottom Solder Paste		0 mm	
B.Silkscreen	Bottom Silk Screen	Not specified	0 mm	White

F.Cu and B.Cu thickness layer shown in the table above contains +20um (from IPC-A-600 Class 2) of plating



IMPEDANCE CONTROL TABLE					
LAYER	TRACE [MM]	SPACING [MM]	IMPEDANCE SINGLE-ENDED	IMPEDANCE DIFFERENTIAL	TOLLERANCE
F.Cu	0.26	0.2	NA	90 ohm	+/- 10%
B.Cu	0.26	0.2	NA	90 ohm	+/- 10%

BOARD CHARACTERISTICS

Copper Layer Count:	4	Board Thickness:	1.5800 mm
Board overall dimensions:	65.0000 mm x 56.5000 mm		
Min track/spacing:	0.2000 mm / 0.2000 mm	Min hole diameter:	0.2540 mm
Copper Finish:	ENIG	Impedance Control:	Yes
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

Drawn by: Heath Raftery
Company: Empirical.EE

Blues Inc

Sheet:
File: Notecarrier-Pi.kicad_pcb

Title: Notecarrier-Pi

Size: A4	Date: 2023-05-29	**Rev: A**
KiCad E.D.A. kicad 7.0.2-0		Id: 1/1