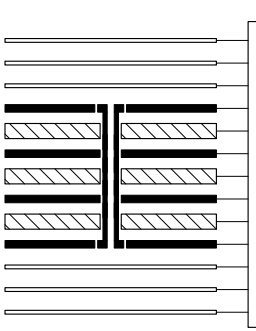
Board Name Fabrication Document

Layer Stack Legend



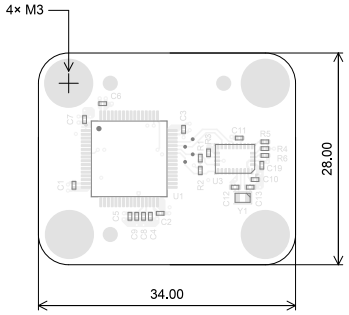
Material	Layer	Thickness	Dielectric	Type	Gerber
	F.Paste			Paste Mask	
	F.Silkscreen		Direct Printing	Legend	GBR
	F.Mask	0.02mm	Solder Resist	Solder Mask	GBR
Copper	**L1 (Sig, PWR)**	**0.035mm (1oz)**		**Signal**	**GBR**
Prepreg		0.1855mm	FR4_7628	Dielectric	
Copper	**L2 (GND)**	**0.035mm (1oz)**		**Plane**	**GBR**
Prepreg		1.1mm	FR4	Dielectric	
Copper	**L3 (GND)**	**0.035mm (1oz)**		**Signal**	**GBR**
Prepreg		0.1855mm	FR4_7628	Dielectric	
Copper	**L4 (Sig, PWR)**	**0.035mm (1oz)**		**Signal**	**GBR**
	B.Mask	0.02mm	Solder Resist	Solder Mask	GBR
	B.Silkscreen		Direct Printing	Legend	GBR
	B.Paste			Paste Mask	

Total thickness: 1.651mm
Note: external layer thicknesses are specified after plating

Impedance Table

Transmission Line	Impedance [ohms]	Tolerance [ohms]	Layer	Trace Width [mm]	Gap [mm]	Ref. Layers
Edge-Coupled Coated Microstrip	100	±10 %	L1	0.2032	0.28	L2

Top Fabrication (Scale 1:1)



FABRICATION NOTES (UNLESS OTHERWISE SPECIFIED)

1) FABRICATE PER IPC-6012A CLASS 2.

2) OUTLINE DEFINED IN SEPARATE GERBER FILE WITH "Edge_Cuts.GBR" SUFFIX.

   DIMENSIONS OF CIRCUMSIZED RECTANGLE SHOWN ON THIS DRAWING FOR REFERENCE ONLY.

3) SEE SEPARATE DRILL FILES WITH ".DRL" SUFFIX FOR HOLE LOCATIONS.

   SELECTED HOLE LOCATIONS SHOWN ON THIS DRAWING FOR REFERENCE ONLY.

4) SURFACE FINISH: IMMERSION GOLD

5) SOLDERMASK ON BOTH SIDES OF THE BOARD SHALL BE LPI, COLOR BLACK.

6) SILK SCREEN LEGEND TO BE APPLIED PER LAYER STACKUP USING YELLOW NON-CONDUCTIVE EPOXY INK.

7) ALL VIAS ARE TENTED ON BOTH SIDES UNLESS SOLDERMASK OPENED IN GERBER.

8) VENDOR SHOULD FOLLOW ROHS COMPLIANT PROCESS AND Pb FREE FOR MANUFACTURING

9) PCB MATERIAL REQUIREMENTS:

   A. FLAMMABILITY RATING MUST MEET OR EXCEED UL94V-0 REQUIREMENTS.
   B. Tg 170 C OR EQUIVALENT.
   C. EQUIVALENT MATERIAL SHALL BE RoHS COMPLIANT, HALOGEN FREE AND APPROVED BY COMPANY.

10) DESIGN GEOMETRY MINIMUM FEATURE SIZES:

BOARD SIZE	34.000 × 28.000 mm
BOARD THICKNESS	1.651 mm
TRACE WIDTH	0.200 mm
TRACE TO TRACE	0.200 mm
MIN. HOLE (PTH)	0.250 mm
MIN. HOLE (NPTH)	N/A mm
ANNULAR RING	0.150 mm
COPPER TO HOLE	0.254 mm
COPPER TO EDGE	0.250 mm
HOLE TO HOLE	0.254 mm

11) REFER TO IMPEDANCE TABLE FOR IMPEDANCE CONTROL REQUIREMENTS.

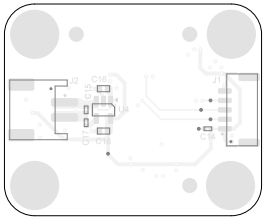
12) CONFIRM SPACE WIDTHS AND SPACINGS.

All dimensions are in millimeters unless otherwise specified.

Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
	Board Name: **Board Name**		Project Name: **Project Name**	
Sheet Title: Top Fabrication (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
Sheet Path:		Reviewer:	Size: **A4**	Sheet: **1** of **9**

Board Name Fabrication Document

Bottom Fabrication (Scale 1:1)



All dimensions are in millimeters unless otherwise specified.

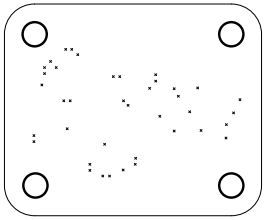
	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: Bottom Fabrication (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **2** of **9**

Board Name Fabrication Document

Drill Table

Symbol	Count	Hole Size	Plated	Hole Shape	Drill Layer Pair	Hole Type
×	39	0.25mm (9.84mils)	PTH	Round	L1 (Sig, PWR) - L4 (Sig, PWR)	Via
○	4	3.20mm (125.98mils)	PTH	Round	L1 (Sig, PWR) - L4 (Sig, PWR)	Pad
	Total 43					

Drill Drawing L1 - L4 (Scale 1:1)

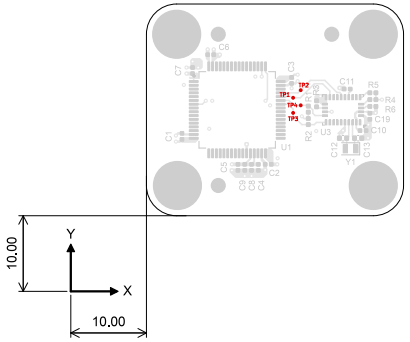


	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: Drill Drawing (L1 - L4)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **3** of **9**

Board Name Fabrication Document

Top Test Points (Scale 1:1)

Ref.	Net	X [mm]	Y [mm]
TP1	IMU_INT	29.40	25.60
TP2	IMU_RESET	30.40	26.60
TP3	I2C2_SDA	29.40	23.60
TP4	I2C2_SCL	30.40	24.60



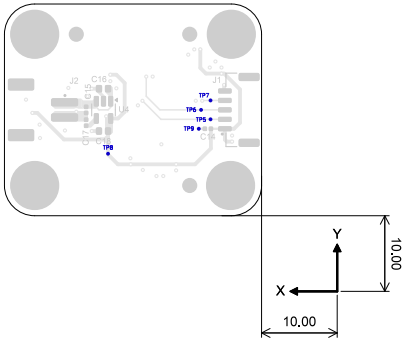
All dimensions are in millimeters unless otherwise specified.

	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: Top Test Points (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **4** of **9**

Board Name Fabrication Document

Bottom Test Points (Scale 1:1)

Ref.	Net	X [mm]	Y [mm]
TP5	SWDIO	16.75	22.75
TP6	SWCLK	18.00	24.00
TP7	NRST	16.75	25.25
TP8	+3V3	30.30	18.20
TP9	GND	18.30	21.50

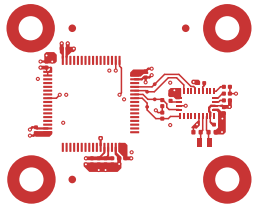


All dimensions are in millimeters unless otherwise specified.

	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: Bottom Test Points (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **5** of **9**

Board Name Fabrication Document

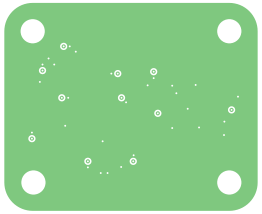
L1 (Sig, PWR) (Scale 1:1)



	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: L1 (Sig, PWR) (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **6** of **9**

Board Name Fabrication Document

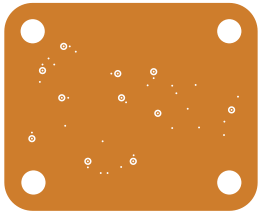
L2 (GND) (Scale 1:1)



	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: L2 (GND) (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **7** of **9**

Board Name Fabrication Document

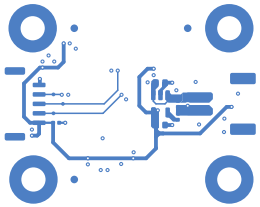
L3 (GND) (Scale 1:1)



	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: L3 (GND) (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **8** of **9**

Board Name Fabrication Document

L4 (Sig, PWR) (Scale 1:1)



	Comments:	Company: **Company Name**		Variant: CHECKED	Git Hash: ace2c6d
		Board Name: **Board Name**		Project Name: **Project Name**	
	Sheet Title: L4 (Sig, PWR) (Scale 1:1)	File Name: KiBot_Project_Test.kicad_pcb	Designer: Author	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: **A4**	Sheet: **9** of **9**