

REVISION HISTORY

REV	DESCRIPTION	ZONE

FABRICATION NOTES (UNLESS OTHERWISE SPECIFIED)

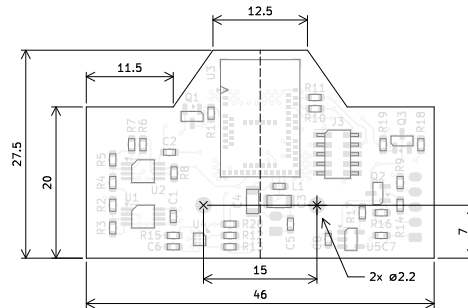
- 1) FABRICATE PER IPC-6012A CLASS 2.
- 2) OUTLINE DEFINED IN SEPARATE GERBER FILE WITH "Edge_Cuts.GBR" SUFFIX.
- 3) SEE SEPARATE DRILL FILES WITH ".DRL" SUFFIX FOR HOLE LOCATIONS.

SELECTED HOLE LOCATIONS SHOWN ON THIS DRAWING FOR REFERENCE ONLY.
- 4) SURFACE FINISH: HAL SNPB
- 5) SOLDERMASK ON BOTH SIDES OF THE BOARD SHALL BE LPI, COLOR GREEN.
- 6) SILK SCREEN LEGEND TO BE APPLIED PER LAYER STACKUP USING WHITE NON-CONDUCTIVE EPOXY INK.
- 7) ALL VIAS ARE TENTED ON BOTH SIDES UNLESS SOLDERMASK OPENED IN GERBER.
- 8) RESERVED
- 9) PCB MATERIAL REQUIREMENTS:
 - A. FLAMMABILITY RATING MUST MEET OR EXCEED UL94V-0 REQUIREMENTS.
 - B. Tg 135 C OR EQUIVALENT.
- 10) DESIGN GEOMETRY MINIMUM FEATURE SIZES:

BOARD SIZE	46.000 × 27.500 mm
BOARD THICKNESS	1.647 mm
TRACE WIDTH	0.100 mm
TRACE TO TRACE	0.200 mm
MIN. HOLE (PTH)	0.300 mm
MIN. HOLE (NPTH)	2.200 mm
ANNULAR RING	0.150 mm
COPPER TO HOLE	0.250 mm
COPPER TO EDGE	0.500 mm
HOLE TO HOLE	0.250 mm

- 11) ALL DIMENSIONS ARE IN MILLIMETERS UNLESS OTHERWISE SPECIFIED.
- 12) FOR REFERENCE ONLY THE STACKUP CORRESPONDS TO JLCPCB STACKUP JLC04161H-7628.
FOR REFERENCE ONLY. IMPEDANCE IS THE CONTROLLING PARAMETER OVER LAYER STACKUP. GERBER DATA MAY NOT BE ALTERED EXCEPT FOR STANDARD FABRICATION ALLOWANCES.

THE SUPPLIED ARTWORK MAY OR MAY NOT CONTAIN THE SPECIFIED TRACE GEOMETRIES ON EVERY LAYERS SPECIFIED.



All dimensions are in millimeters unless otherwise specified.

Stackup Table

Material	Layer	Thickness	Dielectric	Type
	F.Paste			Paste Mask
	F.Silkscreen			Legend
	F.Mask	0.0305mm		Solder Mask
Copper	F.Cu	0.035mm (1.00oz)		Signal
Prepreg		0.2104mm	FR4	Dielectric
Copper	In1.Cu	0.0152mm (0.43oz)		Signal
Core		1.065mm	FR4	Dielectric
Copper	In2.Cu	0.0152mm (0.43oz)		Plane
Prepreg		0.2104mm	FR4	Dielectric
Copper	B.Cu	0.035mm (1.00oz)		Plane
	B.Mask	0.0305mm		Solder Mask
	B.Silkscreen			Legend
	B.Paste			Paste Mask

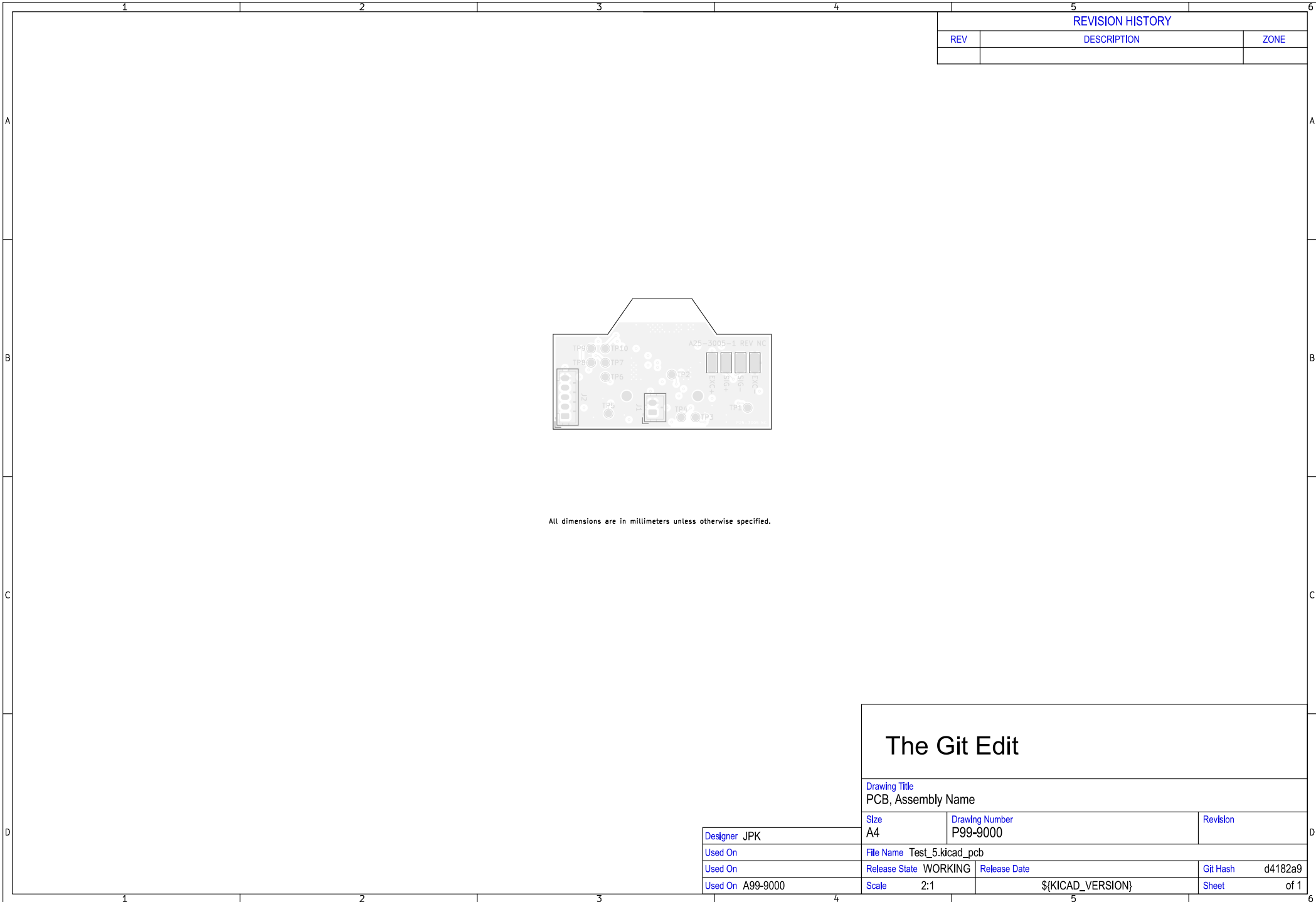
Total thickness: 1.6472mm
Note: external layer thicknesses are specified after plating

Impedance Table

Layer	Ref1	Ref2	Type	Impedance [ohms]	Width [mil]	Space [mil]	Tolerance
L1	AIR	L2	SE	50	13,75	---	10%
L1	AIR	L2	DIFF	90	11,26	8,00	10%
L1	AIR	L2	DIFF	100	8,68	8,00	10%

The Git Edit

Drawing Title PCB, Assembly Name			
Size A4	Drawing Number P99-9000		Revision
Designer JPK	File Name Test_5.kicad_pcb		
Used On	Release State WORKING	Release Date	Git Hash d4182a9
Used On A99-9000	Scale 2:1	Sheet of 1	



All dimensions are in millimeters unless otherwise specified.

REVISION HISTORY		
REV	DESCRIPTION	ZONE

The Git Edit

Drawing Title
PCB, Assembly Name

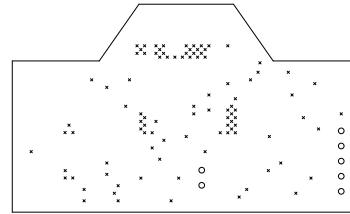
Designer JPK	Size A4	Drawing Number P99-9000	Revision
Used On	File Name Test_5.kicad_pcb	Release State WORKING	Release Date
Used On A99-9000	Scale 2:1	Git Hash d4182a9	Sheet of 1

REVISION HISTORY

REV	DESCRIPTION	ZONE

Drill Table

Symbol	Count	Hole Size	Plated	Hole Shape	Drill Layer Pair	Hole Type
×	97	0.30mm (11.81mils)	PTH	Round	F,Cu - B,Cu	Via
○	7	0.75mm (29.53mils)	PTH	Round	F,Cu - B,Cu	Pad
	Total 104					



The Git Edit

Drawing Title

PCB, Assembly Name

Size	Drawing Number	Revision
A4	P99-9000	

Designer JPK

Used On File Name Test_5.kicad_pcb

Used On Release State WORKING Release Date Git Hash d4182a9

Used On A99-9000 Scale 2:1 Sheet of 1

Scale 2:1

Release Date \${KICAD_VERSION}

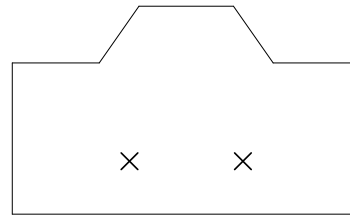
Sheet of 1

REVISION HISTORY

REV	DESCRIPTION	ZONE

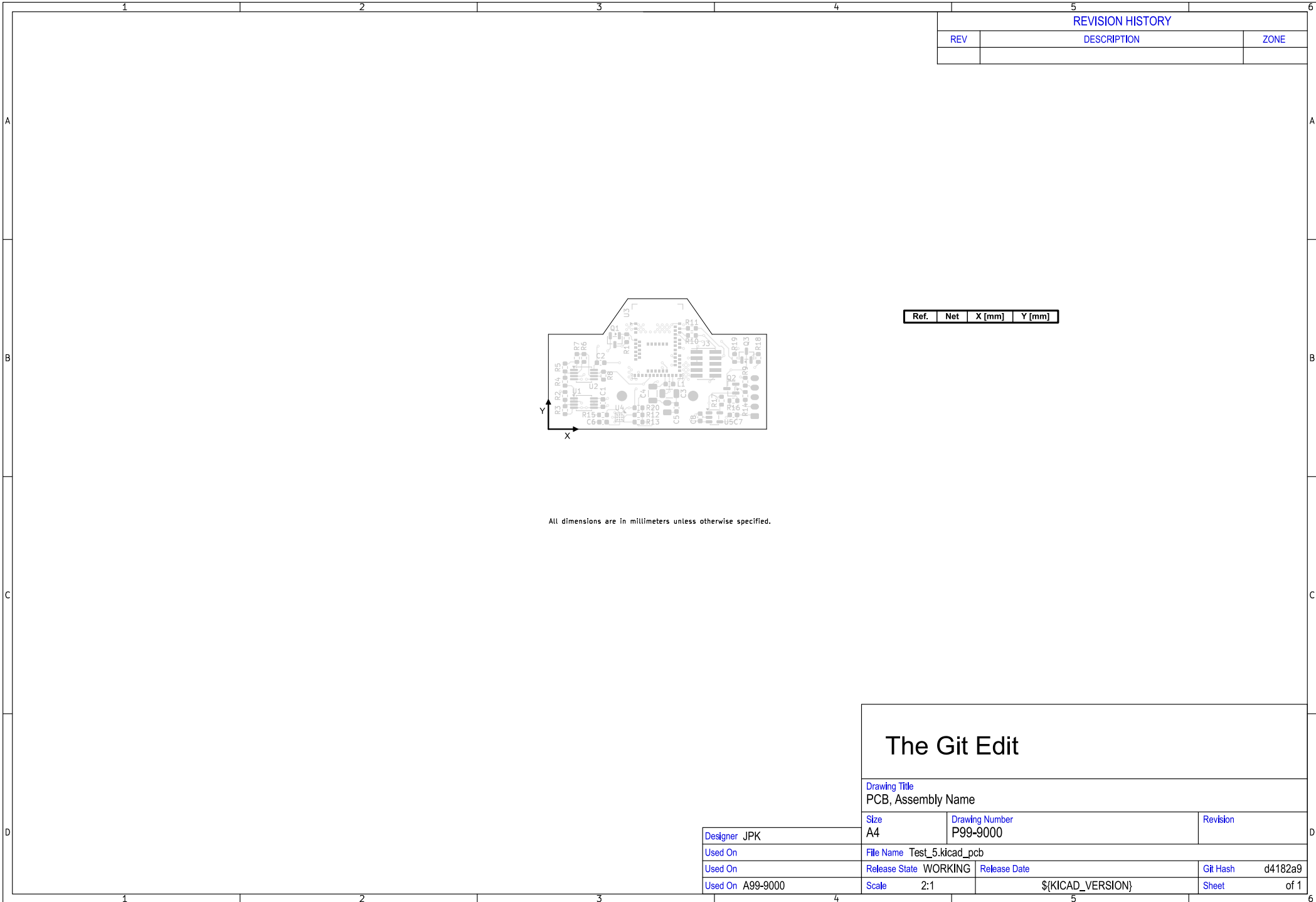
Drill Table

Symbol	Count	Hole Size	Plated	Hole Shape	Drill Layer Pair	Hole Type
×	2	2.20mm (86,61mils)	NPTH	Round	F,Cu - B,Cu	Mechanical
	Total 2					



The Git Edit

Drawing Title PCB, Assembly Name			
Size A4	Drawing Number P99-9000	Revision	
Designer JPK	File Name Test_5.kicad_pcb		
Used On	Release State WORKING	Release Date	Git Hash d4182a9
Used On A99-9000	Scale 2:1		Sheet of 1

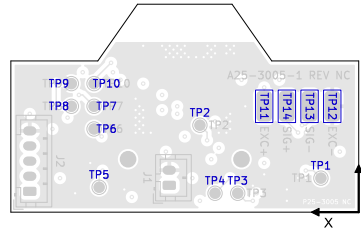


REVISION HISTORY		
REV	DESCRIPTION	ZONE

Ref.	Net	X [mm]	Y [mm]
------	-----	--------	--------

All dimensions are in millimeters unless otherwise specified.

REVISION HISTORY		
REV	DESCRIPTION	ZONE



Ref.	Net	X [mm]	Y [mm]
TP1	Net-(U1A-)	5.00	4.50
TP2	LOAD_CELL	21.00	11.50
TP3	Net-(J1-Pin_1)	16.00	2.50
TP4	BATT_INT	19.00	2.50
TP5	STAT	34.32	3.29
TP6	RESET	35.00	11.00
TP7	SWDCLK	35.00	14.00
TP8	+3.3V	38.00	14.00
TP9	GND	38.00	17.00
TP10	SWDIO	35.00	17.00
TP11	EXC+	12.50	14.00
TP12	GND	3.50	14.00
TP13	SIG+	6.50	14.00
TP14	SIG-	9.50	14.00

All dimensions are in millimeters unless otherwise specified.

The Git Edit

Drawing Title

PCB, Assembly Name

Size

A4

Drawing Number

P99-9000

Revision

Designer JPK

Used On

File Name Test_5.kicad_pcb

Used On

Release State WORKING

Release Date

Git Hash d4182a9

Used On A99-9000

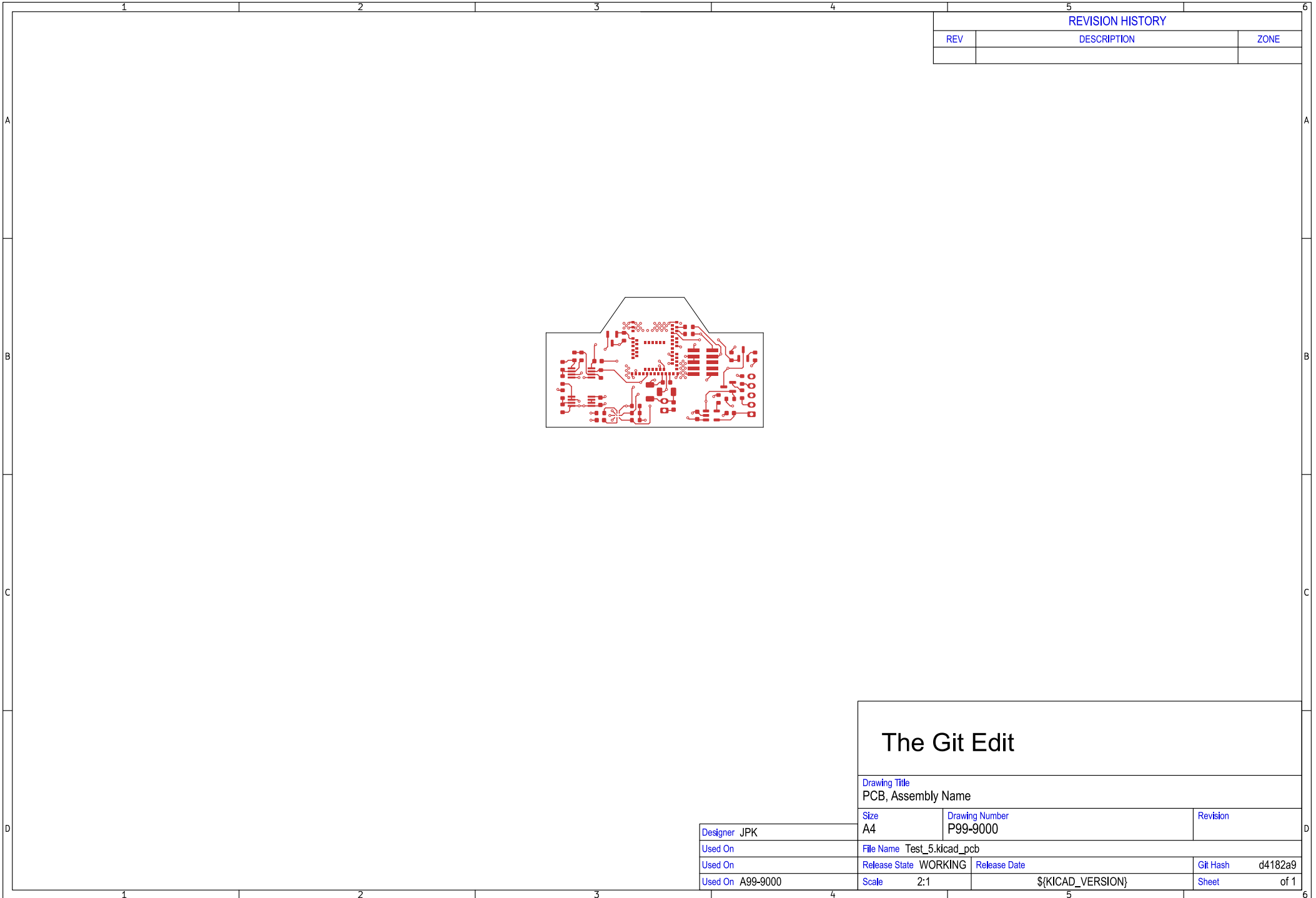
Scale

2:1

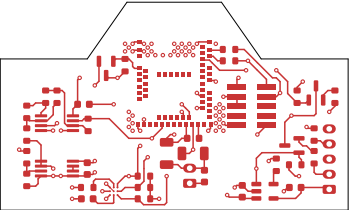
\${KICAD_VERSION}

Sheet

of 1



REVISION HISTORY		
REV	DESCRIPTION	ZONE



<h1>The Git Edit</h1>			
Drawing Title PCB, Assembly Name			
Size A4	Drawing Number P99-9000	Revision	
Used On A99-9000	File Name Test_5.kicad_pcb	Release State WORKING	Release Date
Used On A99-9000	Scale 2:1	Git Hash d4182a9	
Used On A99-9000	Scale 2:1	Sheet of 1	



REVISION HISTORY		
REV	DESCRIPTION	ZONE

The Git Edit

Drawing Title PCB, Assembly Name			
Size A4	Drawing Number P99-9000	Revision	
Used On A99-9000	File Name Test_5.kicad_pcb	Release State WORKING	Release Date
Used On A99-9000	Scale 2:1	Git Hash d4182a9	
Used On A99-9000	Scale 2:1	Sheet of 1	



REVISION HISTORY		
REV	DESCRIPTION	ZONE

The Git Edit

Drawing Title
PCB, Assembly Name

Designer JPK	Size A4	Drawing Number P99-9000	Revision
Used On	File Name Test_5.kicad_pcb		
Used On	Release State WORKING	Release Date	Git Hash d4182a9
Used On A99-9000	Scale 2:1	\${KICAD_VERSION}	Sheet of 1



REVISION HISTORY		
REV	DESCRIPTION	ZONE

The Git Edit

Drawing Title PCB, Assembly Name			
Size A4	Drawing Number P99-9000	Revision	
Used On A99-9000	File Name Test_5.kicad_pcb	Release State WORKING	Release Date
Used On A99-9000	Scale 2:1	Git Hash d4182a9	Sheet of 1

Designer JPK
Used On A99-9000