

FABRICATION NOTES (UNLESS OTHERWISE SPECIFIED)

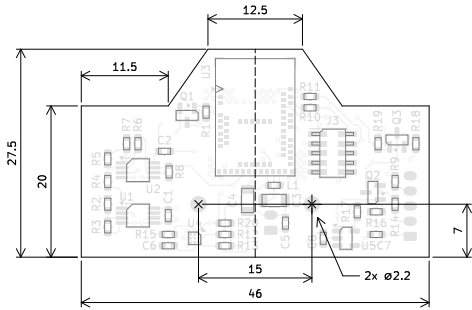
- 1) FABRICATE PER IPC-6012A CLASS 2.
- 2) OUTLINE DEFINED IN SEPARATE GERBER FILE WITH "Edge\_Cuts.GBR" SUFFIX.
- 3) SEE SEPARATE DRILL FILES WITH ".DRL" SUFFIX FOR HOLE LOCATIONS.  
  
SELECTED HOLE LOCATIONS SHOWN ON THIS DRAWING FOR REFERENCE ONLY.
- 4) SURFACE FINISH: HAL SNPB
- 5) SOLDERMASK ON 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.



All dimensions are in millimeters unless otherwise specified.

REVISION HISTORY		
REV	DESCRIPTION	ZONE

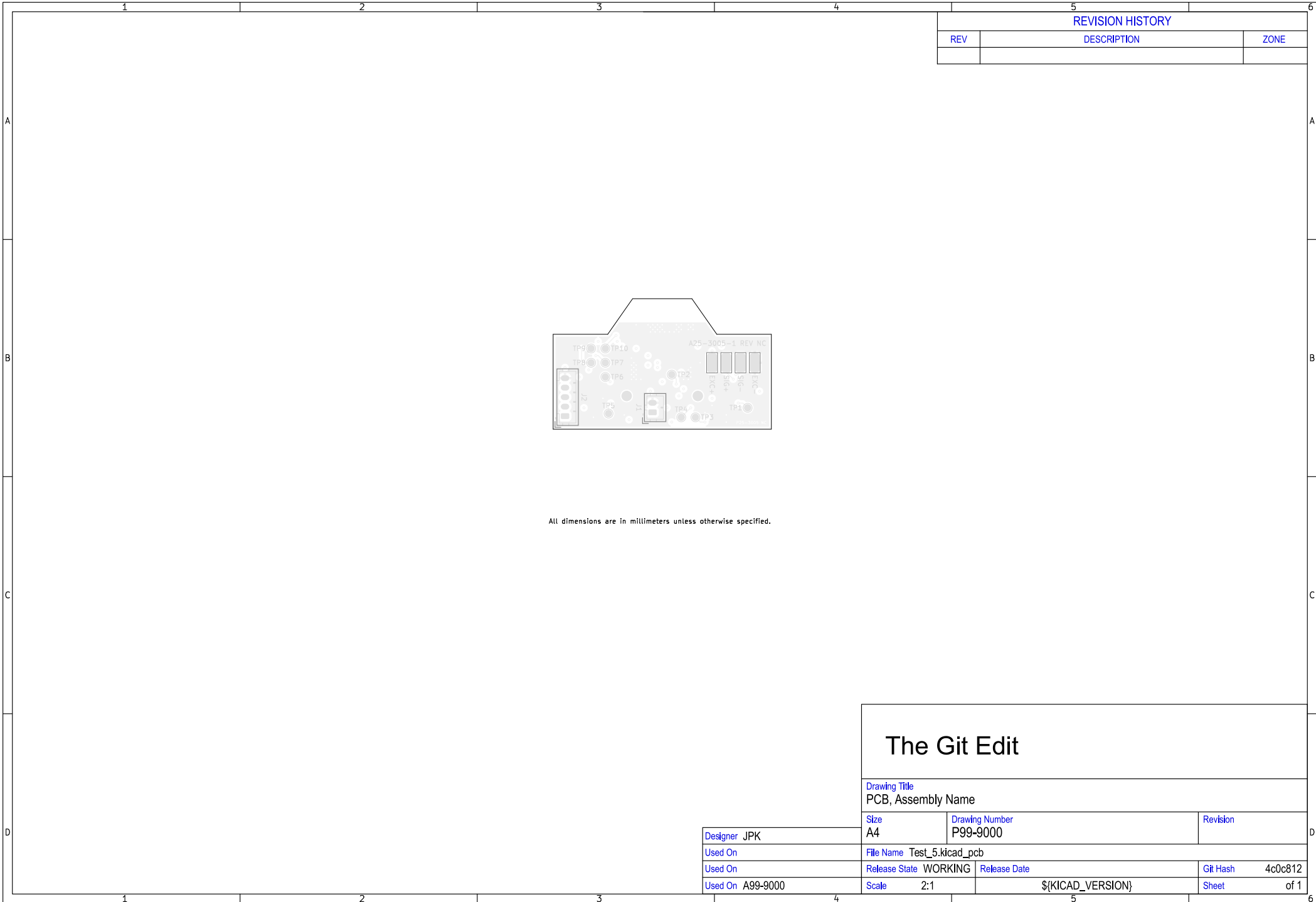
**Stackup Table**

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

# 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
			Git Hash 4c0c812
	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

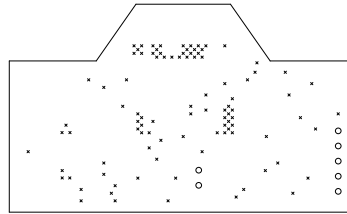
Designer JPK	Size A4	Drawing Number P99-9000	Revision
Used On	File Name Test_5.kicad_pcb	Release State WORKING	Release Date
Used On A99-9000	Scale 2:1	Git Hash 4c0c812	Sheet of 1

Scale 2:1      \${KICAD\_VERSION}

REVISION HISTORY		
REV	DESCRIPTION	ZONE

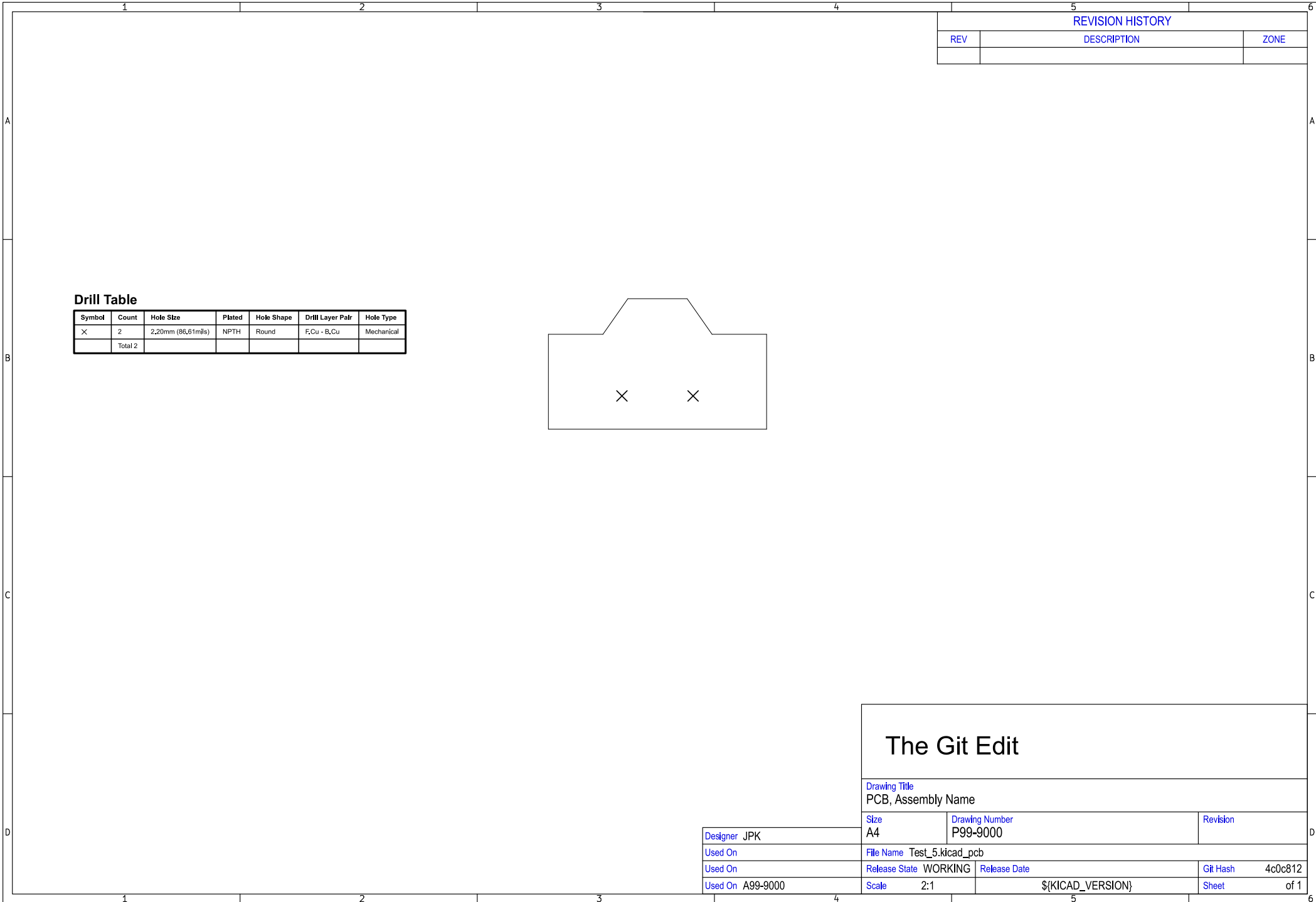
### Drill Table

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

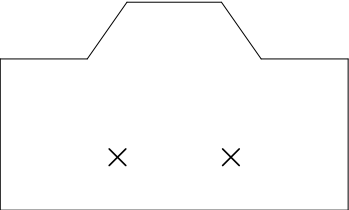
Drawing Title PCB, Assembly Name			
Size A4	Drawing Number P99-9000	Revision	
Designer JPK	File Name Test_5.kicad_pcb	Used On	
Used On A99-9000	Release State WORKING	Release Date	Git Hash 4c0c812
Scale 2:1	\${KICAD_VERSION}		Sheet of 1



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

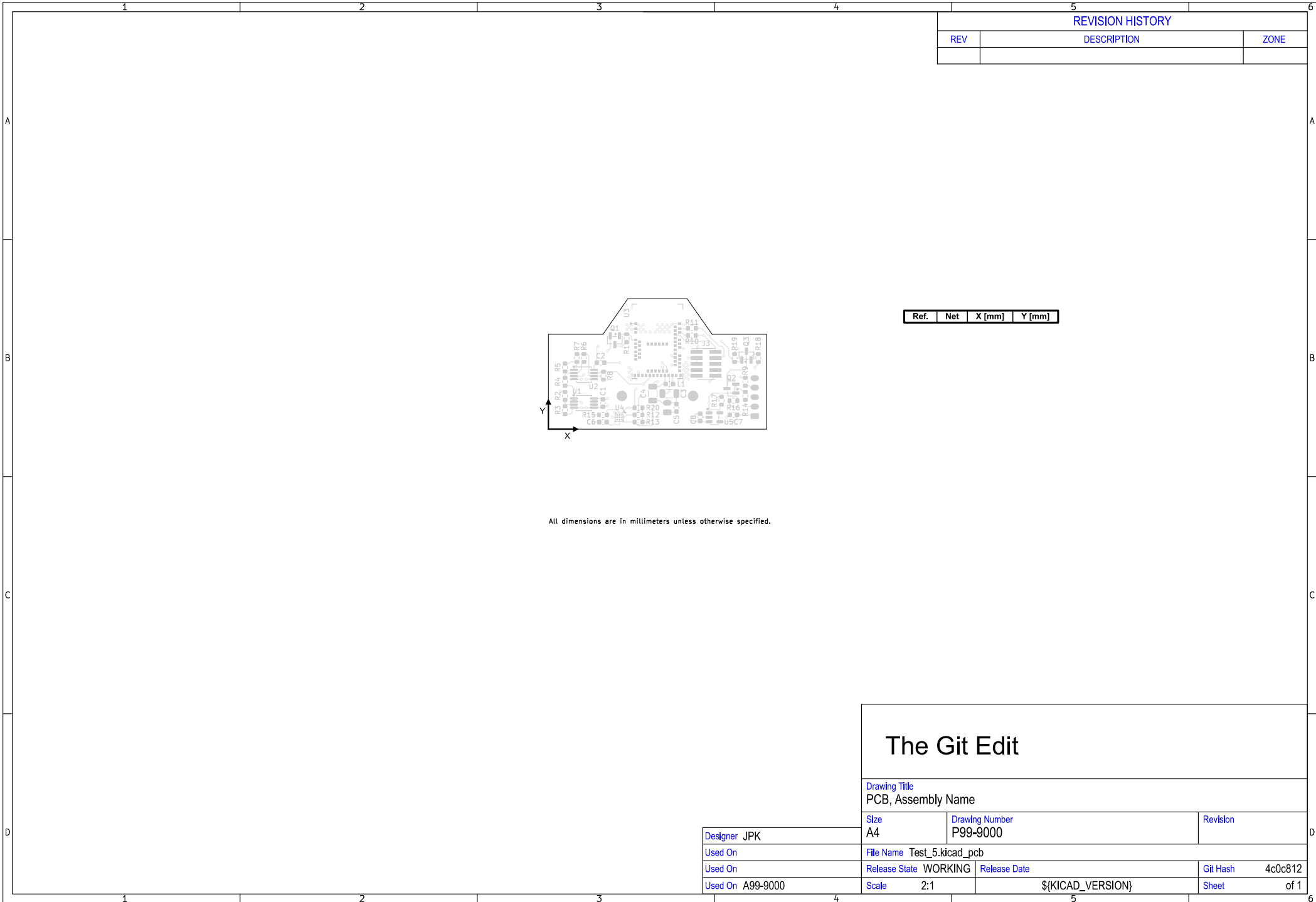
File Name Test\_5.kicad\_pcb

Used On

Release State WORKING	Release Date	Git Hash 4c0c812
--------------------------	--------------	---------------------

Used On A99-9000

Scale 2:1	Sheet of 1	Revision \${KICAD_VERSION}
--------------	---------------	-------------------------------

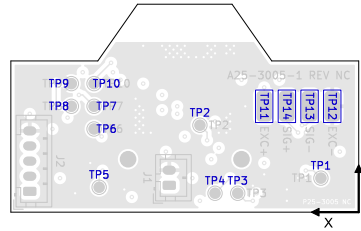


REVISION HISTORY		
REV	DESCRIPTION	ZONE

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

A99-9000

Release State WORKING

Scale

Release Date

2:1

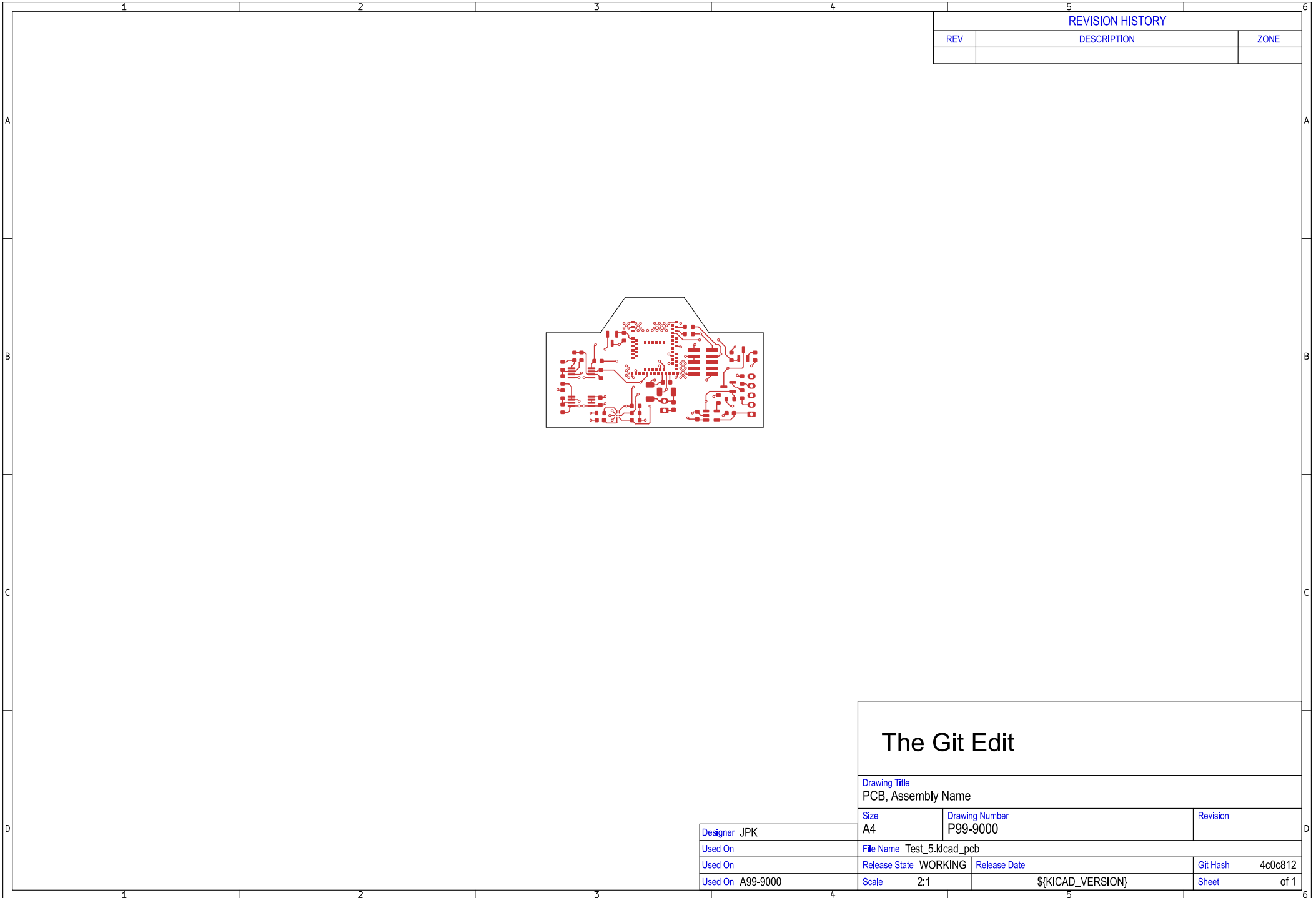
\${KICAD\_VERSION}

Git Hash

4c0c812

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 4c0c812	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	Release State WORKING	Release Date
Used On A99-9000	Scale 2:1	Git Hash 4c0c812	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	Release State WORKING	Release Date
Used On A99-9000	Scale 2:1	Git Hash 4c0c812	Sheet of 1

