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:

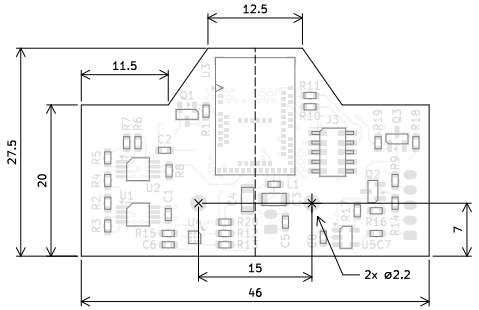
A. FLAMMABILITY RATING MUST MEET OR EXCEED
UL94V-0 REQUIREMENTS.
B. Tg 135 C OR EQUIVALENT.

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

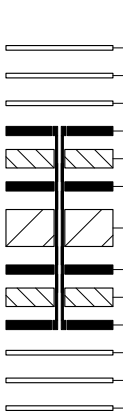
11) ALL DIMENSIONS ARE IN MILLIMETERS UNLESS OTHERWISE
SPECIFIED.

12) FOR REFERENCE ONLY THE STACKUP CORRESPONDS TO JLCPCB
STACKUP JLC04161H-7628.
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

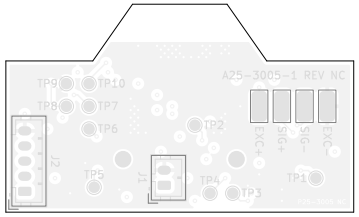
Impedance Table

Transmission Line	Impedance [ohms]	Tolerance [ohms]	Layer	Trace Width [mil]	Gap [mil]	Ref. Layers
Single-Ended Microstrip	50	±10 %	L1	13.75	N/A	L2
Coplanar Differential Pair	100	±10 %	L1	8.68	8.00	L2

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	Scale: 2:1	${KICAD_VERSION}	Sheet: of 1
Used On: A99-9000			

REVISION HISTORY		
REV	DESCRIPTION	ZONE



All dimensions are in millimeters unless otherwise specified.

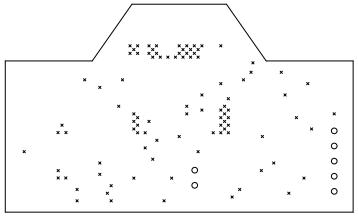
The Git Edit

Drawing Title: PCB, Assembly Name			
Size: A4	Drawing Number: P99-9000		Revision:
Designer: JPK	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

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					



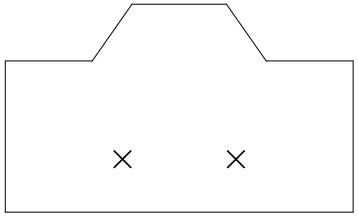
The Git Edit

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

REVISION HISTORY		
REV	DESCRIPTION	ZONE

Drill Table

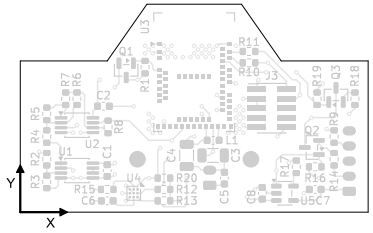
Symbol	Count	Hole Size	Plated	Hole Shape	Drill Layer Pair	Hole Type
×	2	2.20mm (86.61mils)	NPTH	Round	F.Cu - B.Cu	Mechanical
	Total 2					



The Git Edit

Drawing Title: PCB, Assembly Name			
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



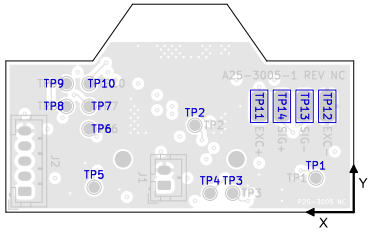
Ref.	Net	X [mm]	Y [mm]

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	File Name: Test_5.kicad_pcb		
Used On	Release State: WORKING	Release Date	Git Hash: d4182a9
Used On	Scale: 2:1	${KICAD_VERSION}	Sheet: of 1
Used On: A99-9000			

REVISION HISTORY		
REV	DESCRIPTION	ZONE



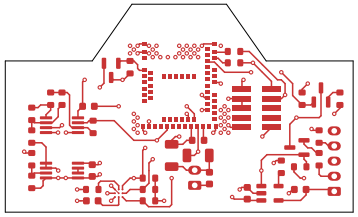
Ref.	Net	X [mm]	Y [mm]
TP1	Net-(U1A–)	5.00	4.50
TP2	LOAD_CELL	21.00	11.50
TP3	Net-(J1-Pin_1)	16.00	2.50
TP4	BATT_INT	19.00	2.50
TP5	STAT	34.32	3.29
TP6	RESET	35.00	11.00
TP7	SWDCLK	35.00	14.00
TP8	+3.3V	38.00	14.00
TP9	GND	38.00	17.00
TP10	SWDIO	35.00	17.00
TP11	EXC+	12.50	14.00
TP12	GND	3.50	14.00
TP13	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			
Designer JPK	Size A4	Drawing Number P99-9000	Revision
Used On	File Name Test_5.kicad_pcb		
Used On	Release State WORKING	Release Date ${KICAD_VERSION}	Git Hash d4182a9
Used On A99-9000	Scale 2:1		Sheet of 1

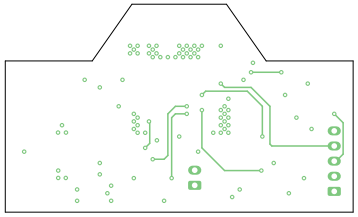
REVISION HISTORY		
REV	DESCRIPTION	ZONE



The Git Edit

Drawing Title: PCB, Assembly Name			
Size: A4	Drawing Number: P99-9000		Revision:
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	${KICAD_VERSION}	Sheet: of 1

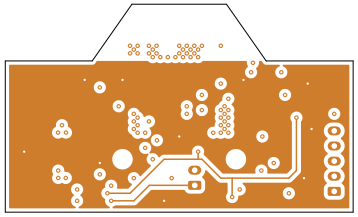
REVISION HISTORY		
REV	DESCRIPTION	ZONE



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: ${KICAD_VERSION}	Git Hash: d4182a9
Used On: A99-9000	Scale: 2:1		Sheet: of 1

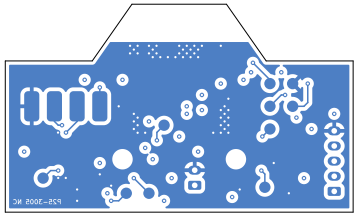
REVISION HISTORY		
REV	DESCRIPTION	ZONE



The Git Edit

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