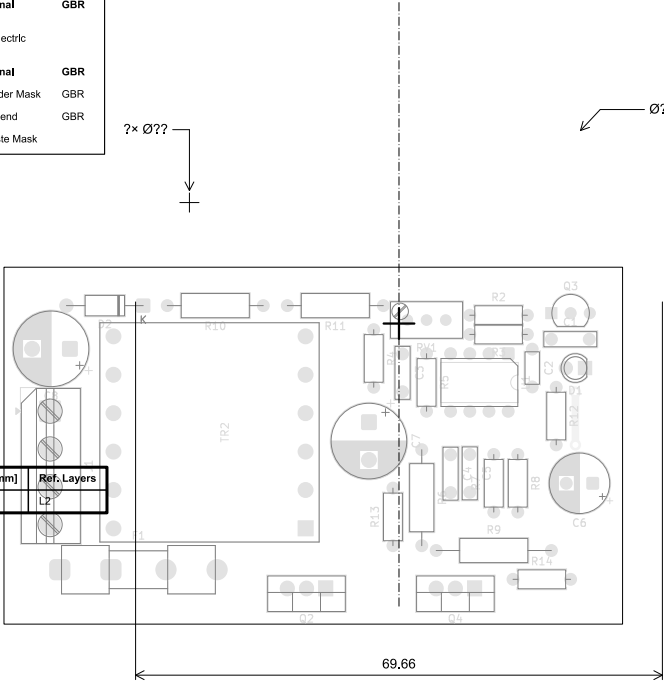


Module Fabrication Document

Layer Stack Legend

	Material	Layer	Thickness	Dielectric	Type	Gerber
		F,Paste			Paste Mask	
		F,Silkscreen			Legend	GBR
		F,Mask	0,02mm	Solder Resist	Solder Mask	GBR
	Copper	L1 (Sig, PWR)	0,07mm (2,00oz)		Signal	GBR
	Core		1,48mm	FR4_7628	Dielectric	
	Copper	L6 (Sig, PWR)	0,07mm (2,00oz)		Signal	GBR
		B,Mask	0,02mm	Solder Resist	Solder Mask	GBR
		B,Silkscreen			Legend	GBR
		B,Paste			Paste Mask	
Total thickness: 1,66mm						
Note: external layer thicknesses are specified after plating						

Top Fabrication (Scale 1:1)



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	L1

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 FR.
- 10) DESIGN GEOMETRY MINIMUM FEATURE SIZES:

BOARD SIZE81.800 × 47.200 mm

BOARD THICKNESS1.660 mm

TRACE WIDTH1.000 mm

TRACE TO TRACE0.200 mm

MIN. HOLE (PTH)0.800 mm

MIN. HOLE (NPTH)N/A mm

ANNULAR RING0.320 mm

COPPER TO HOLE0.254 mm

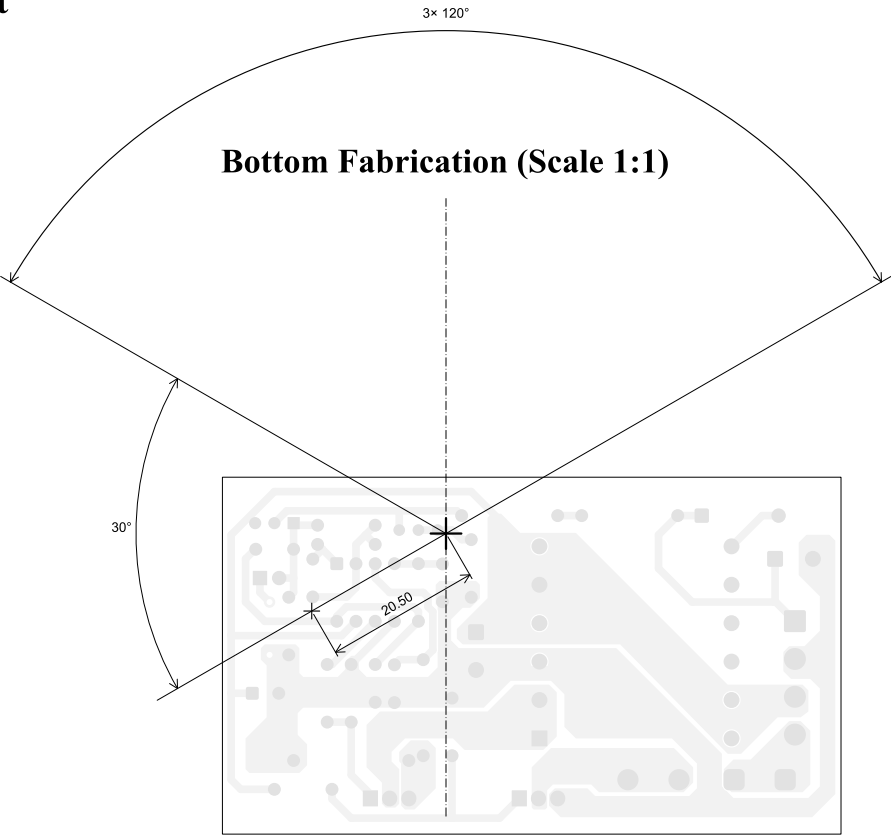
COPPER TO EDGE0.250 mm

HOLE TO HOLE0.254 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: FR	Variant: PRELIMINARY	Git Hash: 0ceba30
	Sheet Title: Top Fabrication (Scale 1:1)	Board Name: Module	Project Name: Alixp Step up module 12 to 450 V	
	Sheet Path:	File Name: StepUp_module_12to450V.kicad_pcb	Designer: FR	Date: 2024-04-13
			Reviewer:	Revision: + (Unreleased)
			Size: A4	Sheet: 1 of 7

Module Fabrication Document



All dimensions are in millimeters unless otherwise specified.

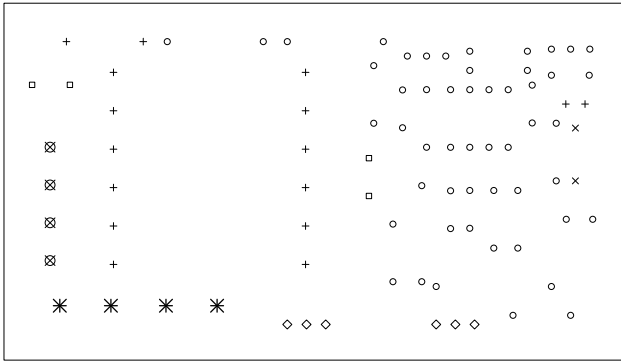
	Comments:	Company: FR		Variant: PRELIMINARY	Git Hash: 0ceba30
		Board Name: Module		Project Name: Alixp Step up module 12 to 450 V	
	Sheet Title: Bottom Fabrication (Scale 1:1)	File Name: StepUp_module_12to450V.kicad_pcb	Designer: FR	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: A4	Sheet: 2 of 7

Module Fabrication Document

Drill Table

Symbol	Count	Hole Size	Plated	Hole Shape	Drill Layer Pair	Hole Type
×	2	0,80mm (31,50mils)	PTH	Round	L1 (Sig, PWR) - L6 (Sig, PWR)	Via
○	52	0,80mm (31,50mils)	PTH	Round	L1 (Sig, PWR) - L6 (Sig, PWR)	Pad
+	16	0,90mm (35,43mils)	PTH	Round	L1 (Sig, PWR) - L6 (Sig, PWR)	Pad
□	4	1,00mm (39,37mils)	PTH	Round	L1 (Sig, PWR) - L6 (Sig, PWR)	Pad
◇	6	1,20mm (47,24mils)	PTH	Round	L1 (Sig, PWR) - L6 (Sig, PWR)	Pad
⊠	4	1,30mm (51,18mils)	PTH	Round	L1 (Sig, PWR) - L6 (Sig, PWR)	Pad
✱	4	1,70mm (66,93mils)	PTH	Round	L1 (Sig, PWR) - L6 (Sig, PWR)	Pad
Total 88						

Drill Drawing L1 - L2 (Scale 1:1)



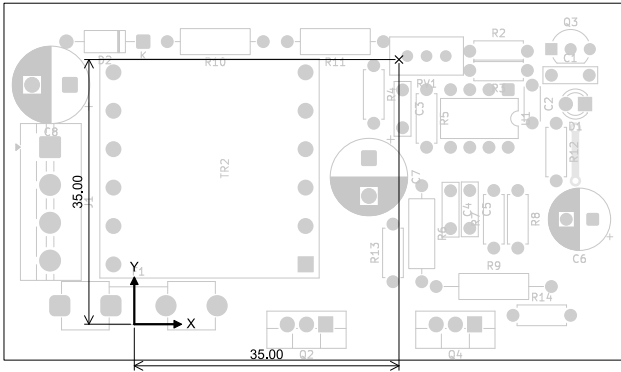
	Comments:	Company:		Variant:		Git Hash:		
		FR		PRELIMINARY		0ceba30		
	Sheet Title:	Board Name:			Project Name:			
		Module			Alixp Step up module 12 to 450 V			
	Drill Drawing (L1 - L2)	File Name:		Designer:		Date:		Revision:
	StepUp_module_12to450V.kicad_pcb		FR		2024-04-13		+ (Unreleased)	
Sheet Path:			Reviewer:		Size:		Sheet:	
					A4		3 of 7	

Module Fabrication Document

Top Test Points (Scale 1:1)

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

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



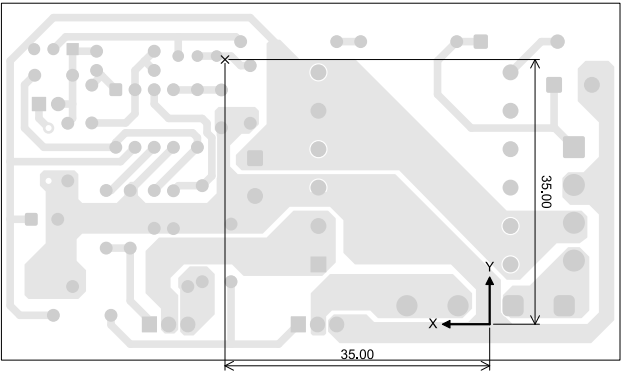
All dimensions are in millimeters unless otherwise specified.

	Comments:	Company: FR		Variant: PRELIMINARY	Git Hash: 0ceba30
		Board Name: Module		Project Name: Alixp Step up module 12 to 450 V	
	Sheet Title: Top Test Points (Scale 1:1)	File Name: StepUp_module_12to450V.kicad_pcb	Designer: FR	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: A4	Sheet: 4 of 7

Module Fabrication Document

Bottom Test Points (Scale 1:1)

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

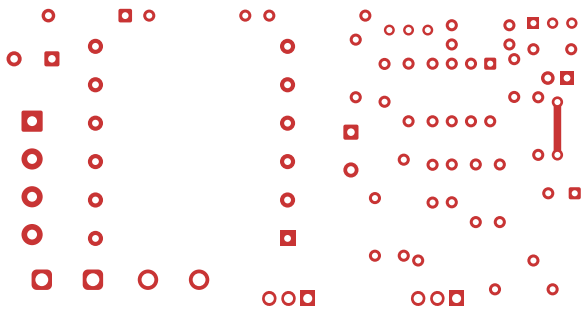


All dimensions are in millimeters unless otherwise specified.

	Comments:	Company: FR		Variant: PRELIMINARY	Git Hash: 0ceba30
		Board Name: Module		Project Name: Alixp Step up module 12 to 450 V	
	Sheet Title: Bottom Test Points (Scale 1:1)	File Name: StepUp_module_12to450V.kicad_pcb	Designer: FR	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: A4	Sheet: 5 of 7

Module Fabrication Document

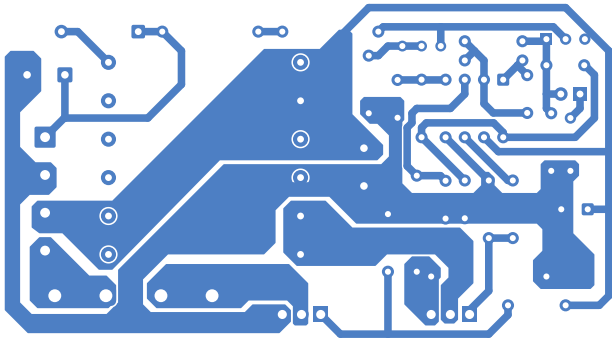
L1 (Sig, PWR) (Scale 1:1)



	Comments:	Company:		Variant:	Git Hash:
		FR		PRELIMINARY	0ceba30
	Sheet Title:	Board Name:		Project Name:	
		Module		Alixp Step up module 12 to 450 V	
	L1 (Sig, PWR) (Scale 1:1)	File Name:	Designer:	Date:	Revision:
	Sheet Path:	Reviewer:	Size:	Sheet:	
				A4	6 of 7

Module Fabrication Document

L6 (Sig, PWR) (Scale 1:1)



	Comments:	Company: FR		Variant: PRELIMINARY	Git Hash: 0ceba30
		Board Name: Module		Project Name: Alixp Step up module 12 to 450 V	
	Sheet Title: L6 (Sig, PWR) (Scale 1:1)	File Name: StepUp_module_12to450V.kicad_pcb	Designer: FR	Date: 2024-04-13	Revision: + (Unreleased)
	Sheet Path:		Reviewer:	Size: A4	Sheet: 7 of 7