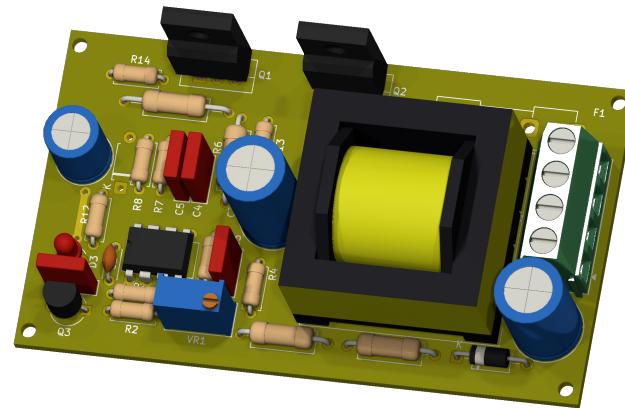
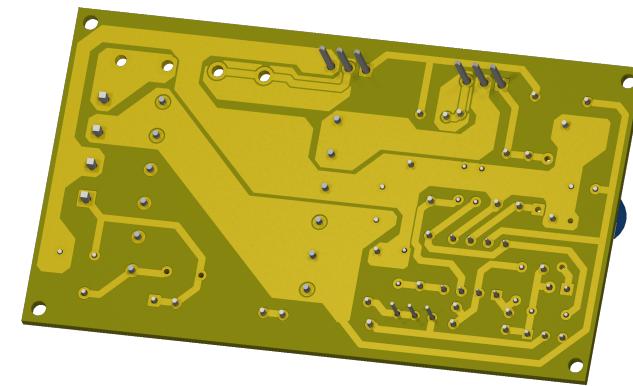


# Module Assembly Document

A

**Top View**

B

**Bottom View**

C

D

1

	Comments: Comment 1 Comment 2 Comment 3 Comment 4	Company: <b>FR</b>	Variant: CHECKED	Git Hash: 917e012
	Board Name: <b>Module</b>	Project Name: <b>Step-up module 12:450 V</b>		
	Sheet Title: Top/Bottom View	File Name: StepUp_module_12to450V.kicad_pcb	Designer: FR	Date: 2024-04-13      Revision: + (Unreleased)
	Sheet Path:		Reviewer: NA	Size: <b>A4</b> Sheet: <b>1</b> of <b>3</b>

2

3

4

5

6

# **Module Assembly Document**

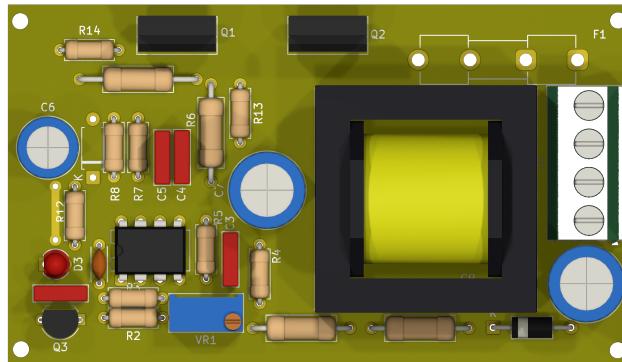
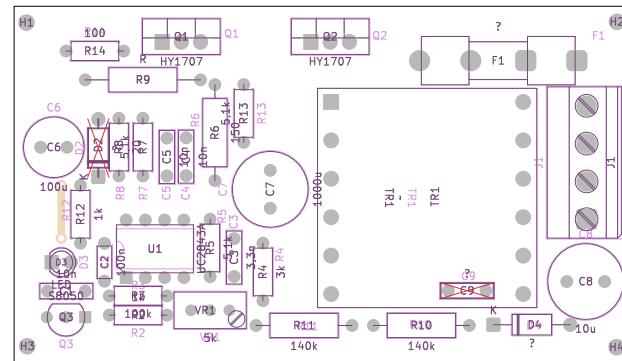
## Components

Type	Front Side	Back Side	Total
THT	31	0	31
SMD	0	0	0
<b>Total</b>	<b>31</b>	<b>0</b>	<b>31</b>

## ASSEMBLY NOTES (UNLESS OTHERWISE SPECIFIED)

- 1) DO NOT POPULATE PARTS MARKED WITH A RED CROSS.
  - 2) IF CONFLICTING INFORMATION IS FOUND BETWEEN THE ASSEMBLY FILE AND BOM, BOM SHOULD BE USED AS THE MAIN SOURCE.
  - 3) DOT IDENTIFIES PIN #1 LOCATION AND DEVICE ORIENTATION WHEN VIEWED FROM THE TOP.

## Top Assembly (Scale 1:1)



	Comments: Comment 1 Comment 2 Comment 3 Comment 4	Company: <b>FR</b>	Variant: CHECKED	Git Hash: 917e012
		Board Name: <b>Module</b>	Project Name: <b>Step-up module 12:450 V</b>	
	<b>Sheet Title:</b> Top Assembly (Scale 1:1)	<b>File Name:</b> StepUp_module_12to450V.kicad_pcb	<b>Designer:</b> FR	<b>Date:</b> 2024-04-13 <b>Revision:</b> + (Unreleased)
	<b>Sheet Path:</b>		<b>Reviewer:</b> NA	<b>Size:</b> <b>A4</b> <b>Sheet:</b> <b>2 of 3</b>

# Module Assembly Document

A

A

B

B

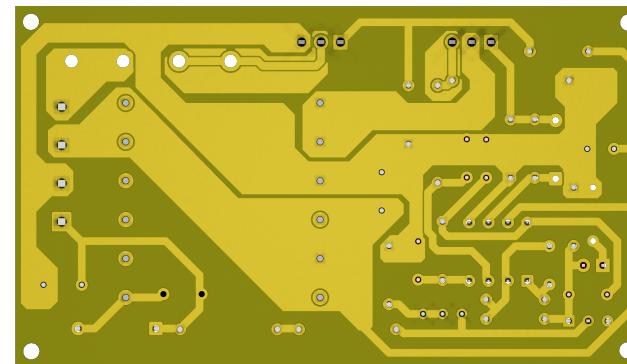
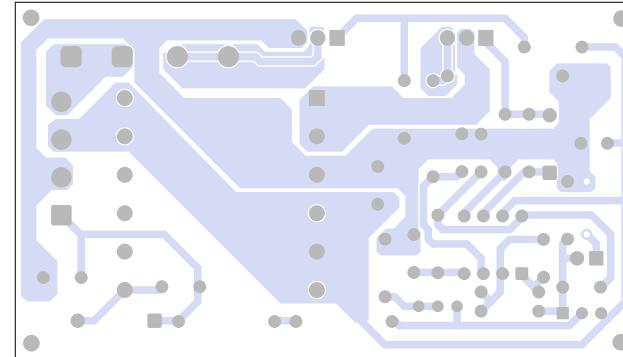
C

C

D

D

**Bottom Assembly (Scale 1:1)**



	Comments: Comment 1 Comment 2 Comment 3 Comment 4	Company: <b>FR</b>	Variant: <b>CHECKED</b>	Git Hash: <b>917e012</b>
	Board Name: <b>Module</b>	Project Name: <b>Step-up module 12:450 V</b>		
	Sheet Title: Bottom Assembly (Scale 1:1)	File Name: StepUp_module_12to450V.kicad_pcb	Designer: FR	Date: 2024-04-13      Revision: + (Unreleased)
	Sheet Path:		Reviewer: NA	Size: <b>A4</b> Sheet: <b>3</b> of <b>3</b>