Module

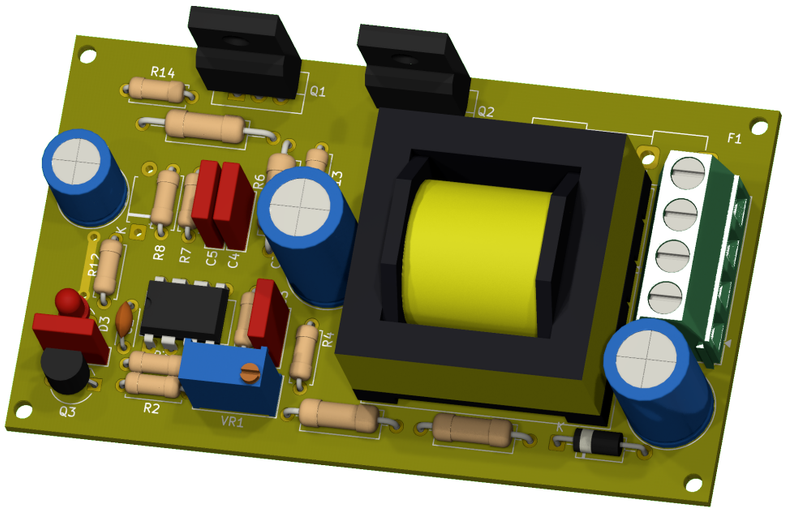
Rev 1.0.1

Variant: RELEASED

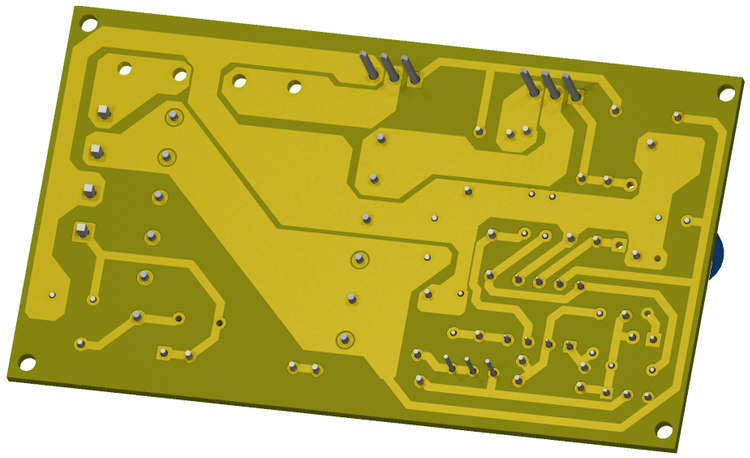
2025-12-21

Page	Index
1	Cover Page
2	Block Diagram
3	Schematic
4	Revision History
5	.....
6	.....
7	.....
8	.....
9	.....
10	.....

TOP VIEW



BOTTOM VIEW



DRAFT - Early concept stage, schematic drafted.

PRELIMINARY - Minor schematic changes are expected, ready for layout planning.

CHECKED - DRC completed, schematic and layout ready for review.

RELEASED - Design ready for manufacturing.

NOTE STYLING CONVENTION

DESIGN NOTE: Example text for informational design notes.

DESIGN NOTE: Example text for debug notes.

DESIGN NOTE: Example text for cautionary design notes.

DESIGN NOTE: Example text for critical design notes.

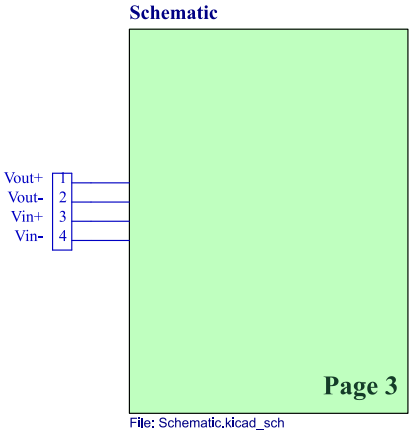
LAYOUT NOTE: Example text for critical layout guidelines.

Date: 21-Dec-2025

	Comments:	Company: FR	Variant: RELEASED	Git Hash: 9cbd478
		Board Name: Module	Project Name: Step-up module 12:450 V	
	Sheet Title:	File Name: StepUp_module_12to450V.kicad_sch	Designer: FR	Date: 2025-12-20 / Revision: 1.0.1
	Sheet Path: /		Reviewer: NA	Size: A3 / Sheet: 1 of 4

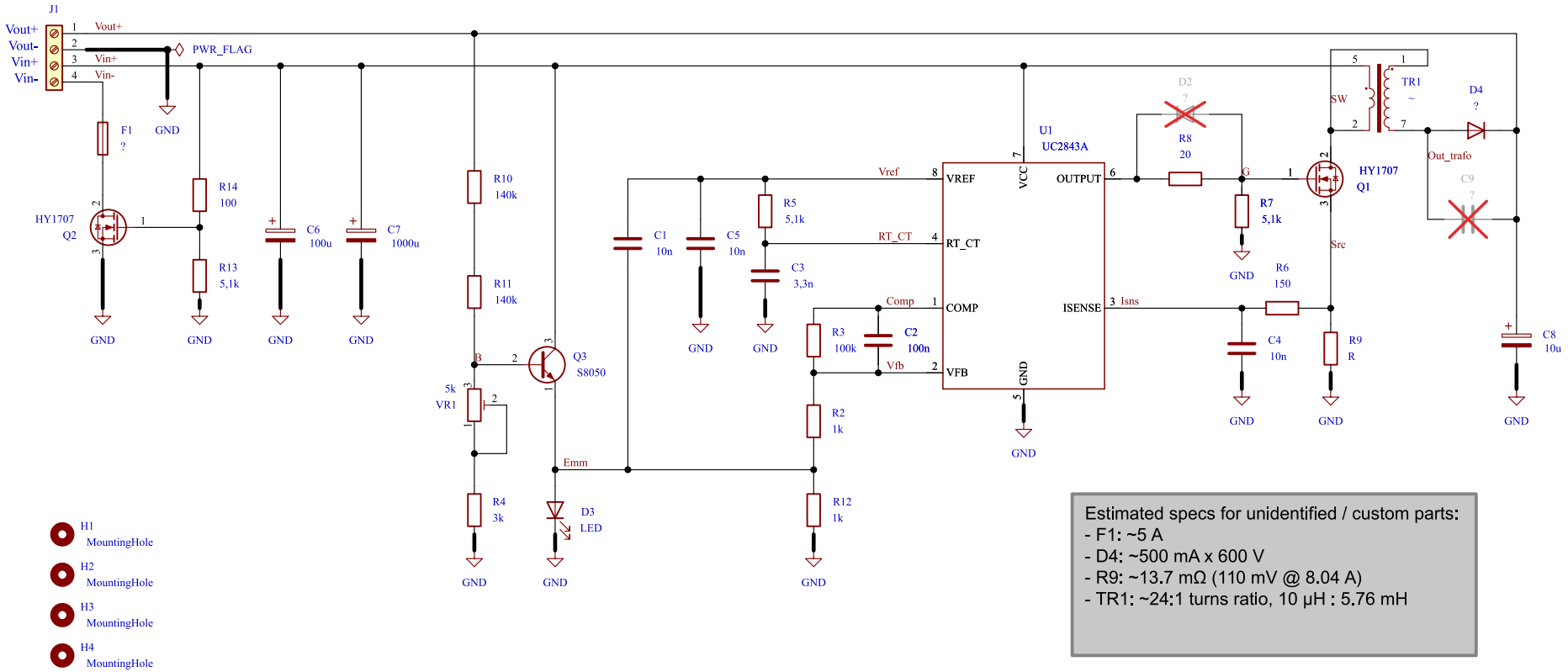
[2] Block Diagram

MODULE INTERFACE:
Vin: 9 V to 24 V according to AliExpress product page
Vout: 200 V to 450 V, adjustable through VR1



Comments: AliExpress module example (link)	Company: FR		Variant: RELEASED	Git Hash: 9cbd478	
	Board Name: Module		Project Name: Step-up module 12:450 V		
Sheet Title: Block Diagram	File Name: Block Diagram.kicad_sch	Designer: FR	Date: 2025-12-20	Revision: 1.0.1	
Sheet Path: /Block Diagram/		Reviewer: NA	Size: A4	Sheet: 2 of 4	

[3] Schematic



Estimated specs for unidentified / custom parts:
- F1: ~5 A
- D4: ~500 mA x 600 V
- R9: ~13.7 mΩ (110 mV @ 8.04 A)
- TR1: ~24:1 turns ratio, 10 µH : 5.76 mH

Comments:	Company: FR		Variant: RELEASED	Git Hash: 9cbd478
	Board Name: **Module**		Project Name: **Step-up module 12:450 V**	
Sheet Title: Schematic	File Name: Schematic.kicad_sch	Designer: FR	Date: 2025-12-20	Revision: 1.0.1
Sheet Path: /Block Diagram/Schematic/		Reviewer: NA	Size: **A4**	Sheet: **3** of **4**

[4] Revision History

Version 1.0.0 - 2025-12-19

First issue

Schematic capture complete, except for parts:

- not populated (D2, C9);

- with unreadable or absent markings (F1, D4, R9, TR1).

CI failed on release - do not use.

Version 1.0.1 - 2025-12-21

Comments:	Company: FR		Variant: RELEASED	Git Hash: 9cbd478	
	Board Name: **Module**		Project Name: **Step-up module 12:450 V**		
Sheet Title: Revision History	File Name: Revision History.kicad_sch	Designer: FR	Date: 2025-12-20	Revision: 1.0.1	
Sheet Path: /Revision History/		Reviewer: NA	Size: **A4**	Sheet: **4** of **4**	