Module

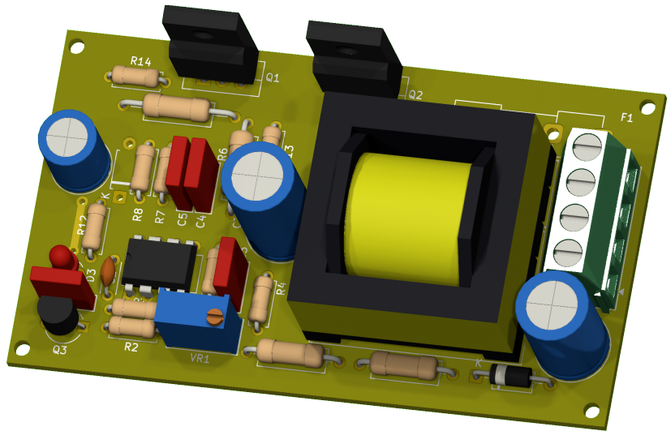
Rev + (Unreleased)

Variant: CHECKED

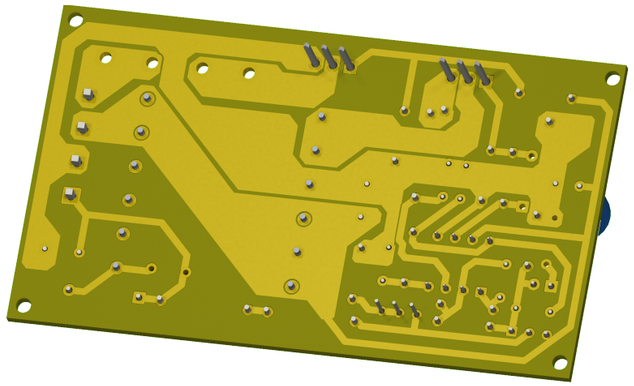
2025-12-20

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

TOP VIEW



BOTTOM VIEW



NOTES

Comment

Not fitted components are marked as ✗

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.

DESIGN CONSIDERATIONS

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.

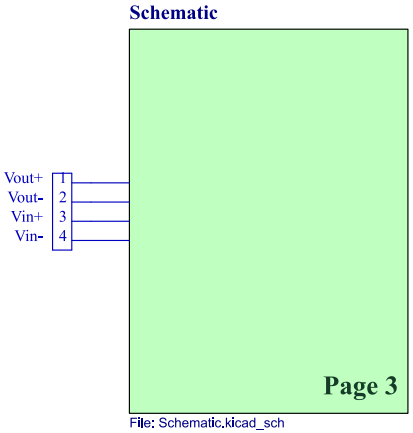
LAYOUT NOTE: Example text for critical layout guidelines.

Date: 20-Dec-2025

Comments:	Company: FR		Variant: CHECKED	Git Hash: 4fc064d
	Board Name: Module		Project Name: Step-up module 12:450 V	
Sheet Title:	File Name: StepUp_module_12to450V.kicad_sch	Designer: FR	Date: 2025-01-12	Revision: + (Unreleased)
Sheet Path: /		Reviewer: NA	Size: A3	Sheet: 1 of 4

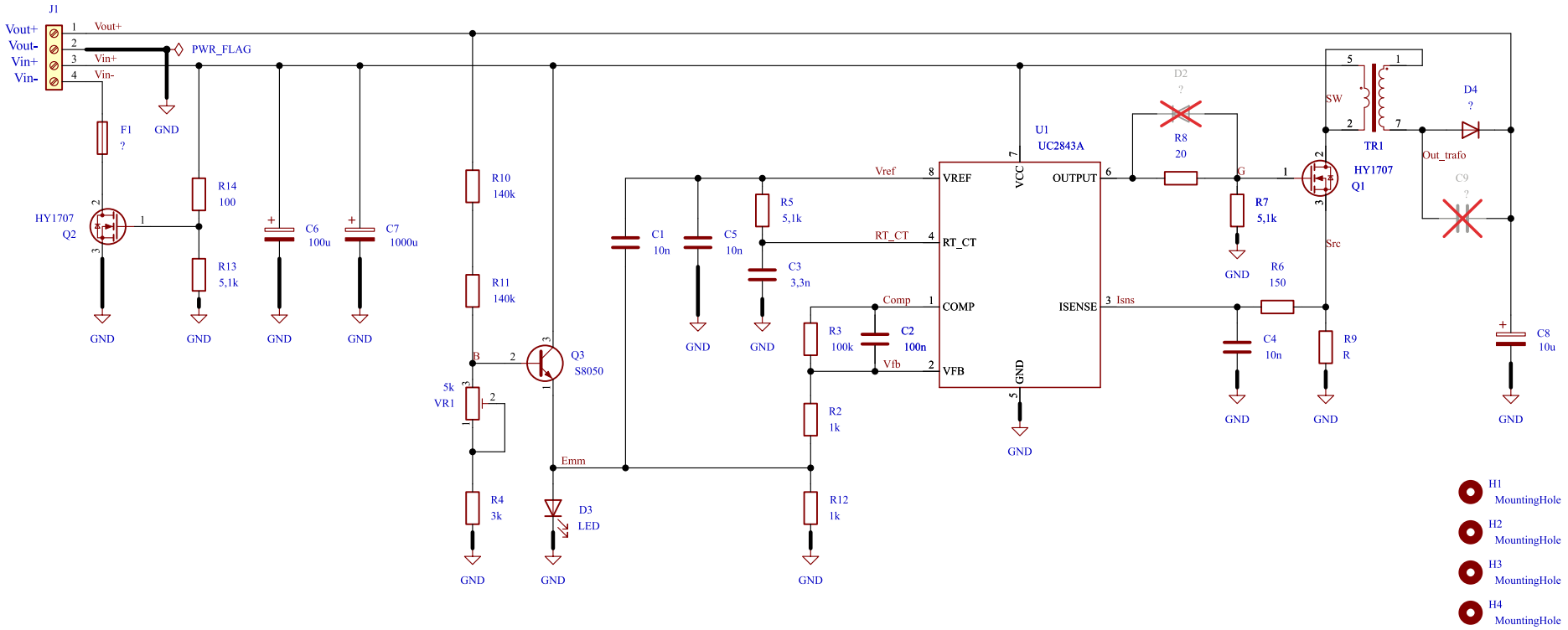
[2] Block Diagram

MODULE INTERFACE:
Vin: 9 V to 2 V according to AliExpress product page
Vout: 200 V to 450 V, adjustable through VR1



	Comments: AliExpress module	Company: FR		Variant: CHECKED	Git Hash: 4fc064d
		Board Name: **Module**		Project Name: **Step-up module 12:450 V**	
	Sheet Title: Block Diagram	File Name: Block Diagram.kicad_sch	Designer: FR	Date: 2025-01-12	Revision: + (Unreleased)
	Sheet Path: /Block Diagram/		Reviewer: NA	Size: **A4**	Sheet: **2** of **4**

[3] Schematic



	Comments:	Company: FR	Variant: CHECKED	Git Hash: 4fc064d
		Board Name: **Module**	Project Name: **Step-up module 12:450 V**	
	Sheet Title: Schematic	File Name: Schematic.kicad_sch	Designer: FR	Date: 2025-01-12	Revision: + (Unreleased)
	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);

Best guess:

- F1: ~5 A

- D4: ~500 mA x 600 V

- R9: 10-100 mΩ

Comments: "Comment 1" "Comment 2" "Comment 3" "Comment 4"	Company: **FR**		Variant: CHECKED	Git Hash: 4fc064d	
	Board Name: **Module**		Project Name: **Step-up module 12:450 V**		
Sheet Title: Revision History	File Name: Revision History.kicad_sch	Designer: FR	Date: 2025-01-12	Revision: + (Unreleased)	
Sheet Path: /Revision History/		Reviewer: NA	Size: **A4**	Sheet: **4** of **4**	