

Module

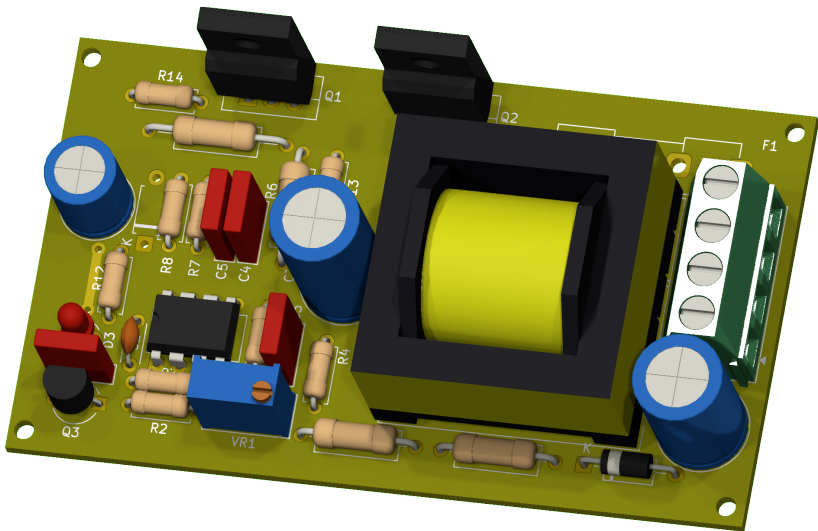
Variant: CHECKED

Rev + (Unreleased)

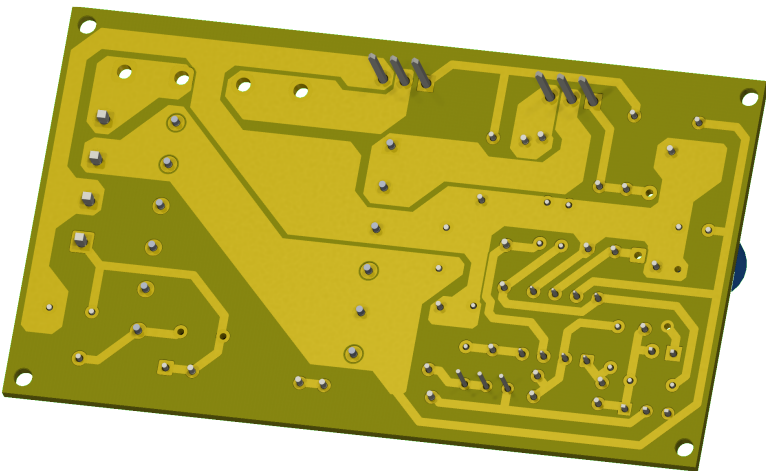
2025-12-19

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 X

- 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.

Date: 19-Dec-2025

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.

	Comments:	Company: FR	Variant: CHECKED	Git Hash: cf618eb
	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
	Sheet Path: /	Reviewer: NA	Size: A3	Revision: + (Unreleased)
				Sheet: 1 of 4

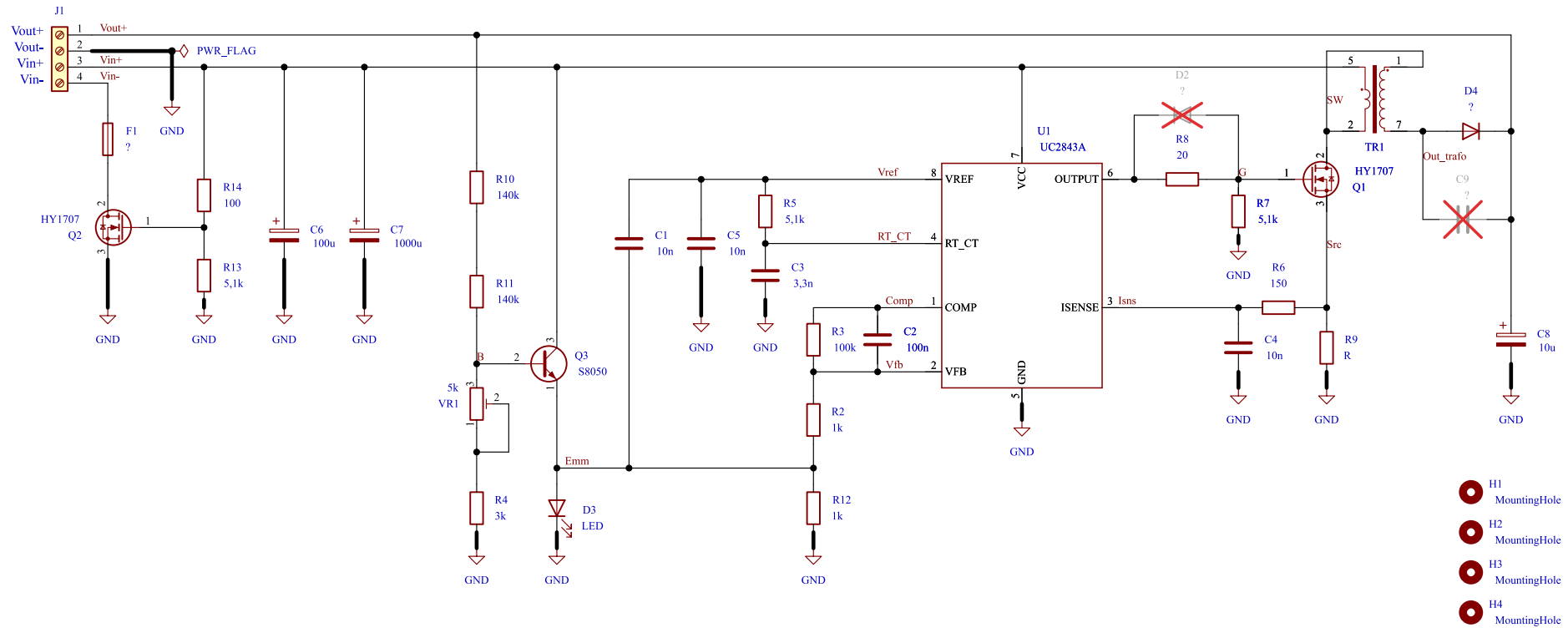
# [2] Block Diagram

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: cf618eb
		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



- H1 MountingHole
- H2 MountingHole
- H3 MountingHole
- H4 MountingHole

Comments:

Company:

FR

Variant:

CHECKED

Git Hash:

cf618eb

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:          | Company:                   |           | Variant:                | Git Hash:      |
|  | "Comment 1"        | FR                         |           | CHECKED                 | cf618eb        |
|  | "Comment 2"        | Board Name:<br>Module      |           | Project Name:           |                |
|  | "Comment 3"        |                            |           | Step-up module 12:450 V |                |
|  | Sheet Title:       | File Name:                 | Designer: | Date:                   | Revision:      |
|  | Revision History   | Revision History.kicad_sch | FR        | 2025-01-12              | + (Unreleased) |
|  | Sheet Path:        |                            | Reviewer: | Size:                   | Sheet:         |
|  | /Revision History/ |                            | NA        | A4                      | 4 of 4         |