

Module

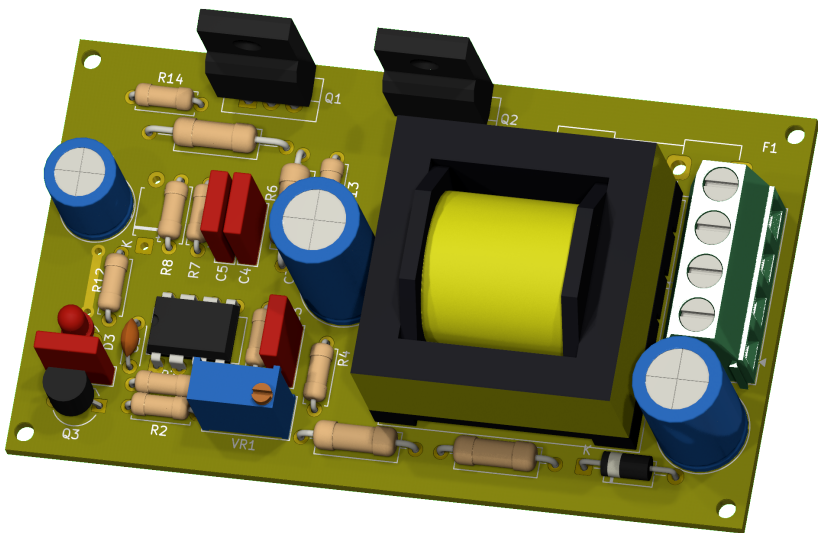
Variant: CHECKED

Rev + (Unreleased)

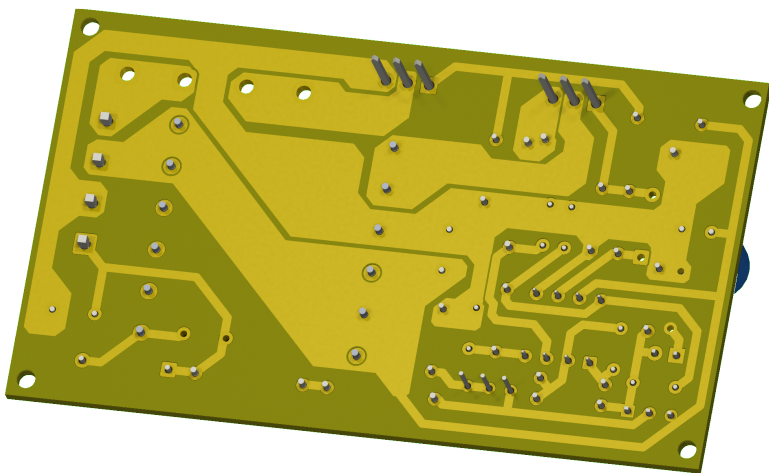
2025-12-20

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 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.

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

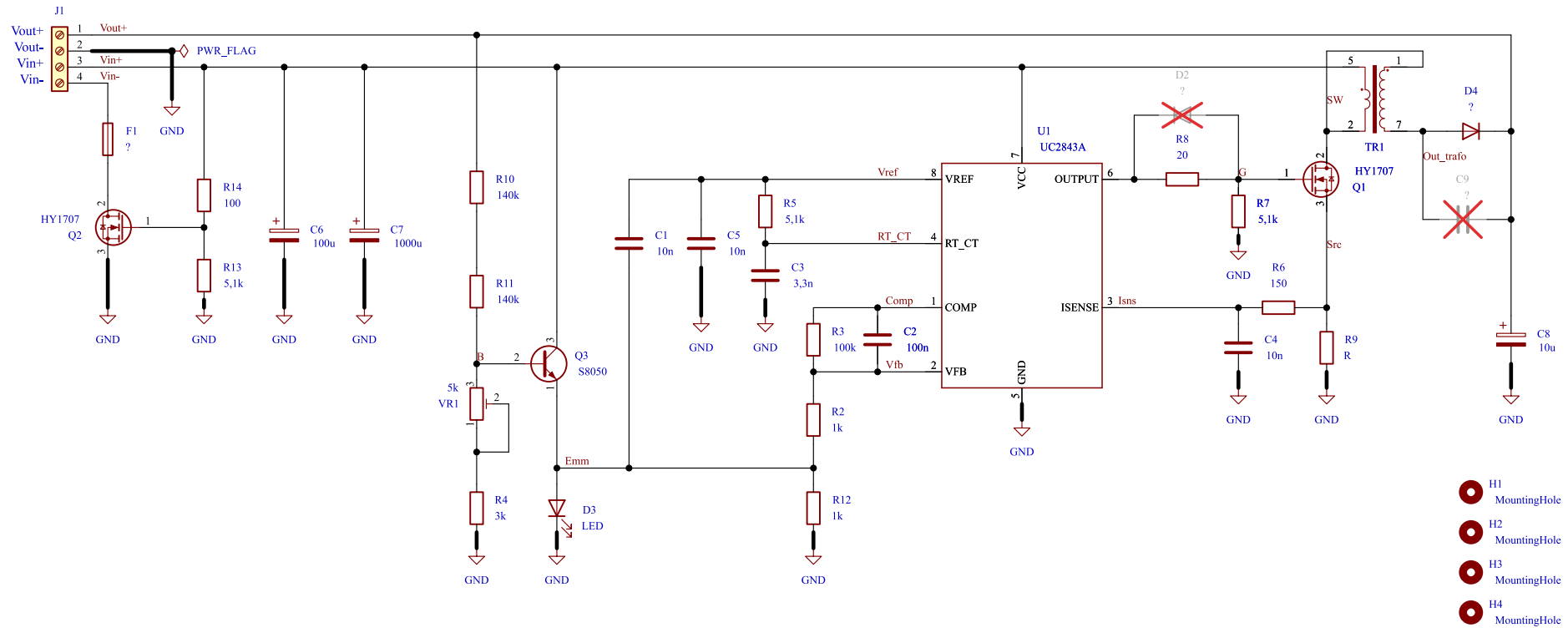
[2] Block Diagram

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



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