

Module

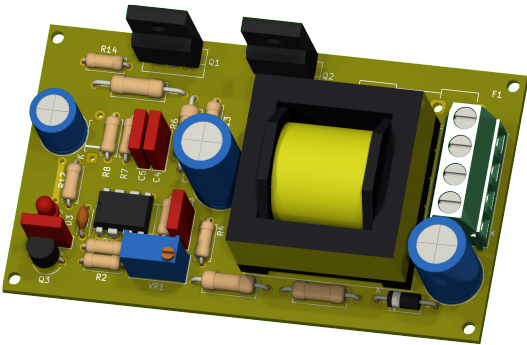
Rev + (Unreleased)

Variant: PRELIMINARY

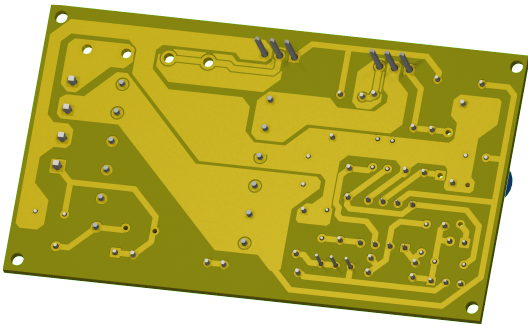
2025-12-14

Page	Index	Page	Index	Page	Index	Page	Index
1	Cover Page	11	.....	21	.....	31	.....
2	Block Diagram	12	.....	22	.....	32	.....
3	Schematic	13	.....	23	.....	33	.....
4	Revision History	14	.....	24	.....	34	.....
5	.....	15	.....	25	.....	35	.....
6	.....	16	.....	26	.....	36	.....
7	.....	17	.....	27	.....	37	.....
8	.....	18	.....	28	.....	38	.....
9	.....	19	.....	29	.....	39	.....
10	.....	20	.....	30	.....	40	.....

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: 14-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: PRELIMINARY	Git Hash: 95cd2d0
		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

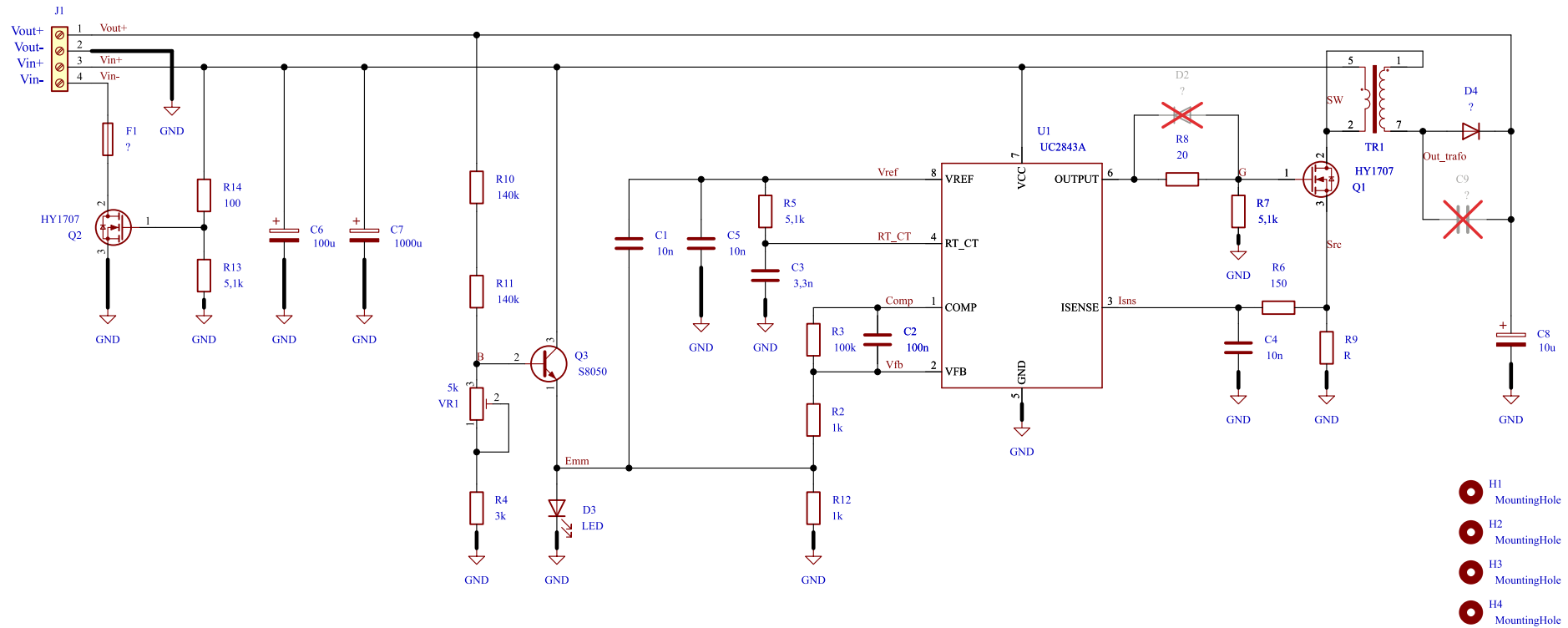
# [2] Block Diagram

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



	<b>Comments:</b> AliExpress module	<b>Company:</b> FR		<b>Variant:</b> PRELIMINARY	<b>Git Hash:</b> 95cd2d0
		<b>Board Name:</b> Module		<b>Project Name:</b> Step-up module 12:450 V	
	<b>Sheet Title:</b> Block Diagram	<b>File Name:</b> Block Diagram.kicad_sch	<b>Designer:</b> FR	<b>Date:</b> 2025-01-12	<b>Revision:</b> + (Unreleased)
	<b>Sheet Path:</b> /Block Diagram/		<b>Reviewer:</b> NA	<b>Size:</b> A4	<b>Sheet:</b> 2 of 4

# [3] Schematic



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

Comments:

Company:

FR

Variant:

PRELIMINARY

Git Hash:

95cd2d0

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

[illegible]