

A 1 2 3 4 5 6 7 8
A Module

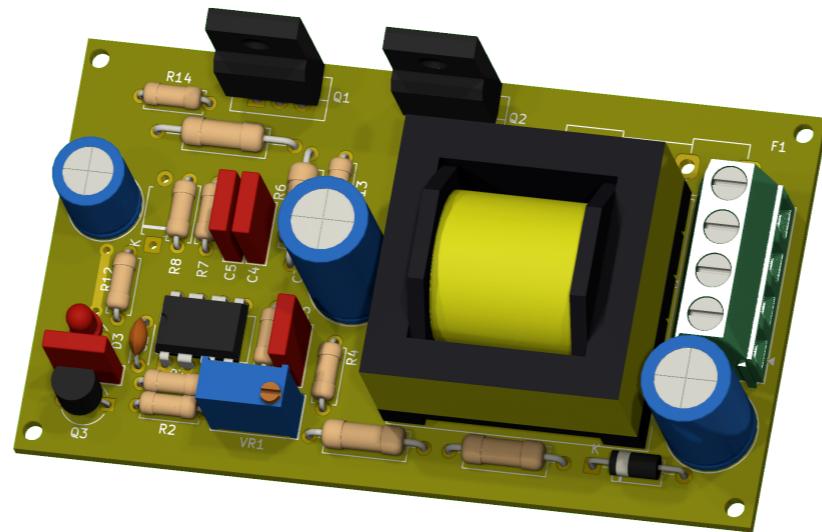
B Variant: CHECKED

A Rev + (Unreleased)

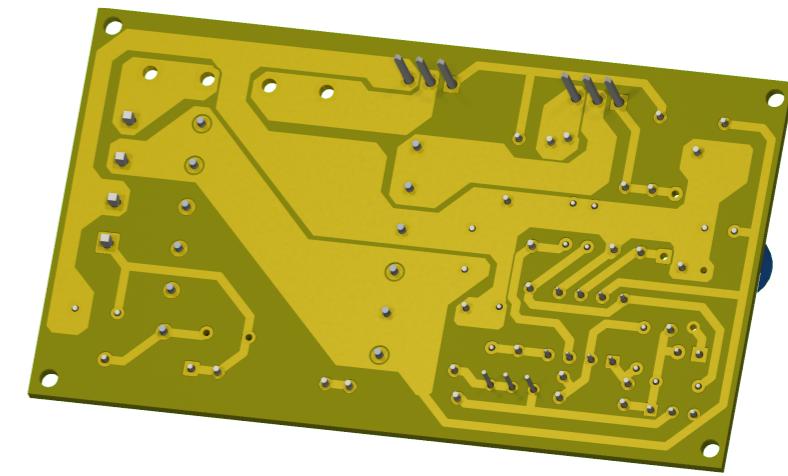
B 2025-12-19

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

C TOP VIEW



D BOTTOM VIEW



NOTES

E 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.

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.

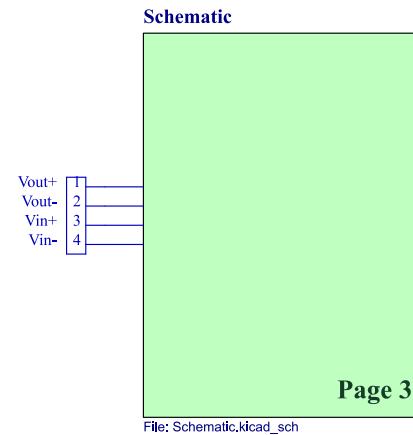
DESIGN NOTE:
Example text for critical design notes.

LAYOUT NOTE:
Example text for critical layout guidelines.

Comments:	Company:	Variant:	Git Hash:
FR		CHECKED	df2262b
Board Name:	Project Name:		
Module	Step-up module 12:450 V		
Sheet Title:	File Name:	Designer:	Date: Revision:
	StepUp_module_12to450V.kicad_sch	FR	2025-01-12 + (Unreleased)
Sheet Path:	/	Reviewer:	Size: Sheet:
		NA	A3 1 of 4

[2] Block Diagram

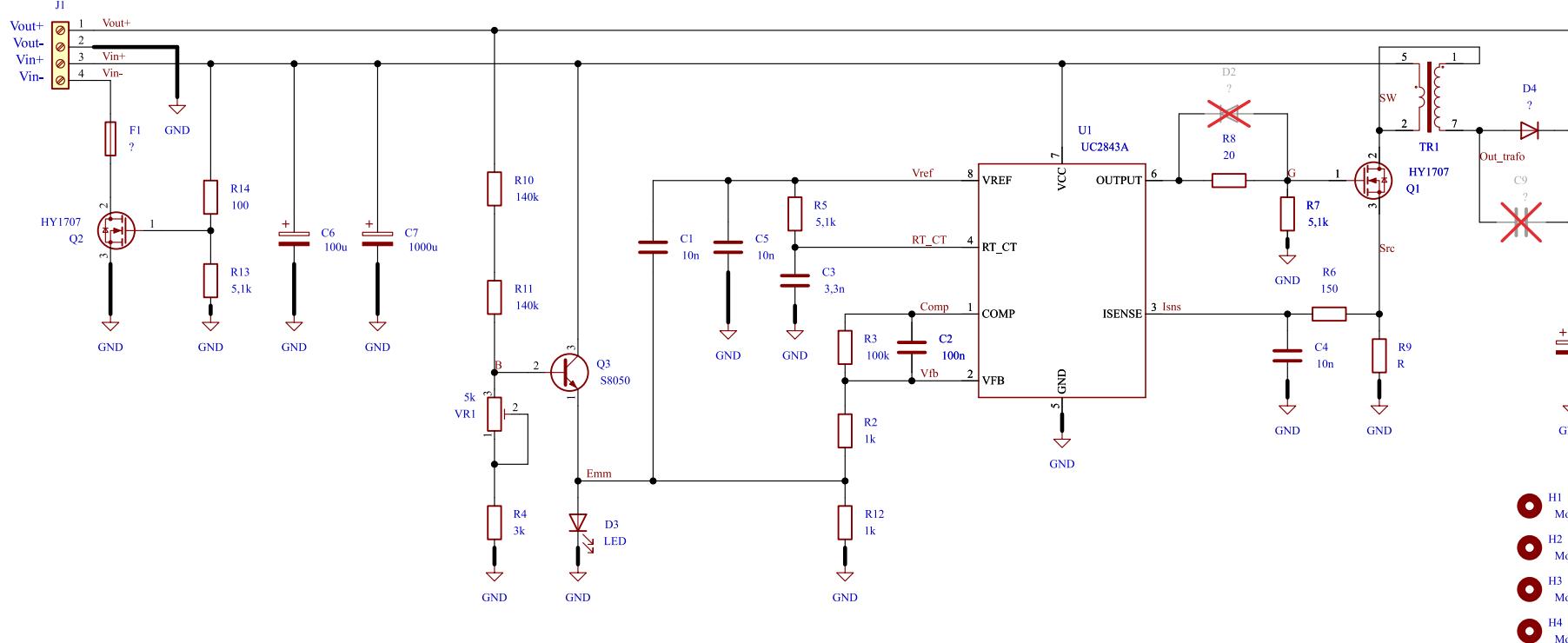
MODULE INTERFACE:
 Vin: 9 V to 2 V according to
 AliExpress product page
 Vout: 200 V to 450 V,
 adjustable through VR1



Page 3

	Comments: AliExpress module	Company: FR	Variant: CHECKED	Git Hash: df2262b
	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:	Variant:	Git Hash:
FR	Module		
Board Name:	Project Name:		
Sheet Title:	File Name:	Designer:	Date: Revision:
Schematic	Schematic.kicad_sch	FR	2025-01-12 + (Unreleased)
Sheet Path:	Reviewer:		
/Block Diagram/Schematic/	NA		
	Size:	Sheet:	
	A4	3 of 4	

[4] Revision History

	Comments: "Comment 1" "Comment 2" "Comment 3" "Comment 4"	Company: FR	Variant: CHECKED	Git Hash: df2262b
		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 + (Unreleased)
	Sheet Path: /Revision History/	Reviewer: NA	Size: A4	Sheet: 4 of 4