

# Board Name

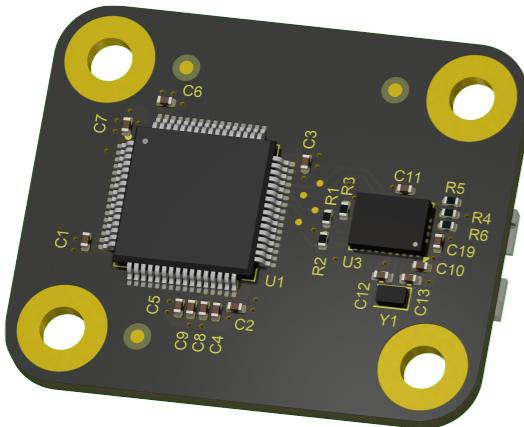
## Variant: CHECKED

2025-03-02

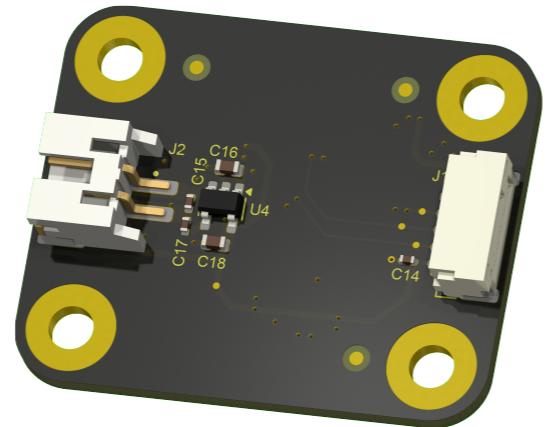
Rev + (Unreleased)

Page	Index	Page	Index	Page	Index	Page	Index
1	Cover Page	11	.....	21	.....	31	.....
2	Block Diagram	12	.....	22	.....	32	.....
3	Project Architecture	13	.....	23	.....	33	.....
4	MCU	14	.....	24	.....	34	.....
5	IMU	15	.....	25	.....	35	.....
6	Connectors	16	.....	26	.....	36	.....
7	Holes, Fiducials	17	.....	27	.....	37	.....
8	Revision History	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 - Very early stage of schematic, ignore details.

PRELIMINARY - Close to final schematic.

CHECKED - There shouldn't be any mistakes. Contact the engineer if you find any.

RELEASED - A board with this schematic has been sent to production.

Date: 02-Mar-2025

### DESIGN CONSIDERATIONS

DESIGN NOTE:  
Example text for informational design notes.

DESIGN NOTE:  
Example text for debug notes.

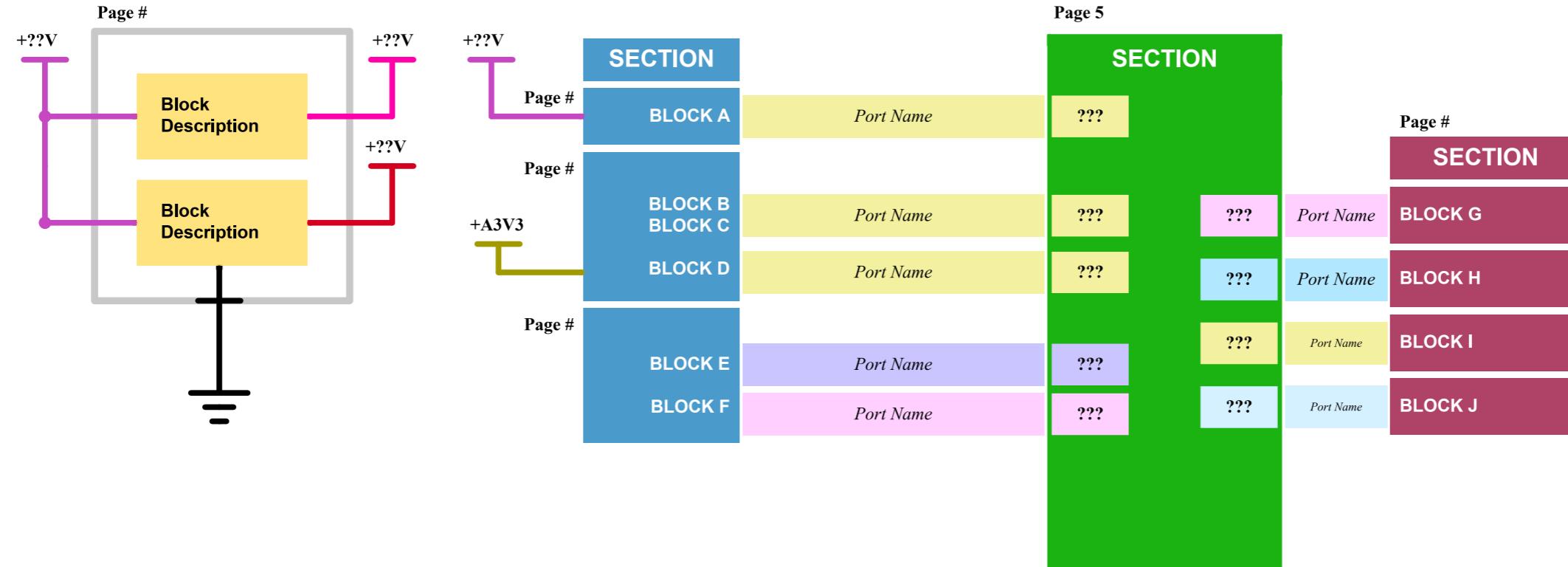
DESIGN NOTE:  
Example text for cautionary design notes.

DESIGN NOTE:  
Example text for critical design notes.

LAYOUT NOTE:  
Example text for critical layout guidelines.

	Comments:	Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
	Board Name: <b>Board Name</b>	Project Name: <b>Project Name</b>		
	Sheet Title: /	File Name: KiBot_Project_Test.kicad_sch	Designer: Author	Date: 2025-01-12 Revision: + (Unreleased)
	Sheet Path: /	Reviewer:	Size: <b>A3</b>	Sheet: <b>1</b> of <b>8</b>

# [2] Block Diagram

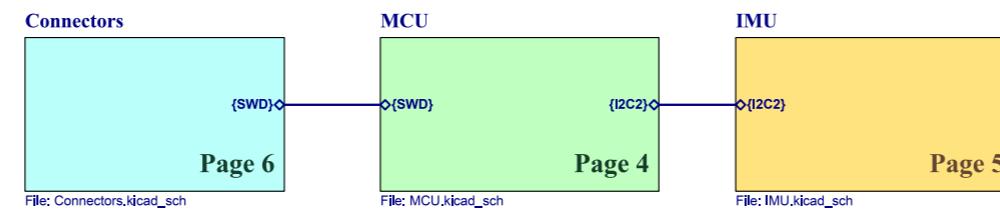


## Target specifications:

Input voltage:	?? - ?? V
Spec 2	??
Spec 3	??
Spec 4	??

	Comments:  	Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
		Board Name: Board Name	Project Name: Project Name	
	Sheet Title: Block Diagram	File Name: Block Diagram.kicad_sch	Designer: Author	Date: 2025-01-12 + (Unreleased)
	Sheet Path: /Block Diagram/	Reviewer:	Size: A3	Revision: Sheet: 2 of 8

# [3] Project Architecture



File: Connectors.kicad\_sch

File: MCU.kicad\_sch

File: IMU.kicad\_sch

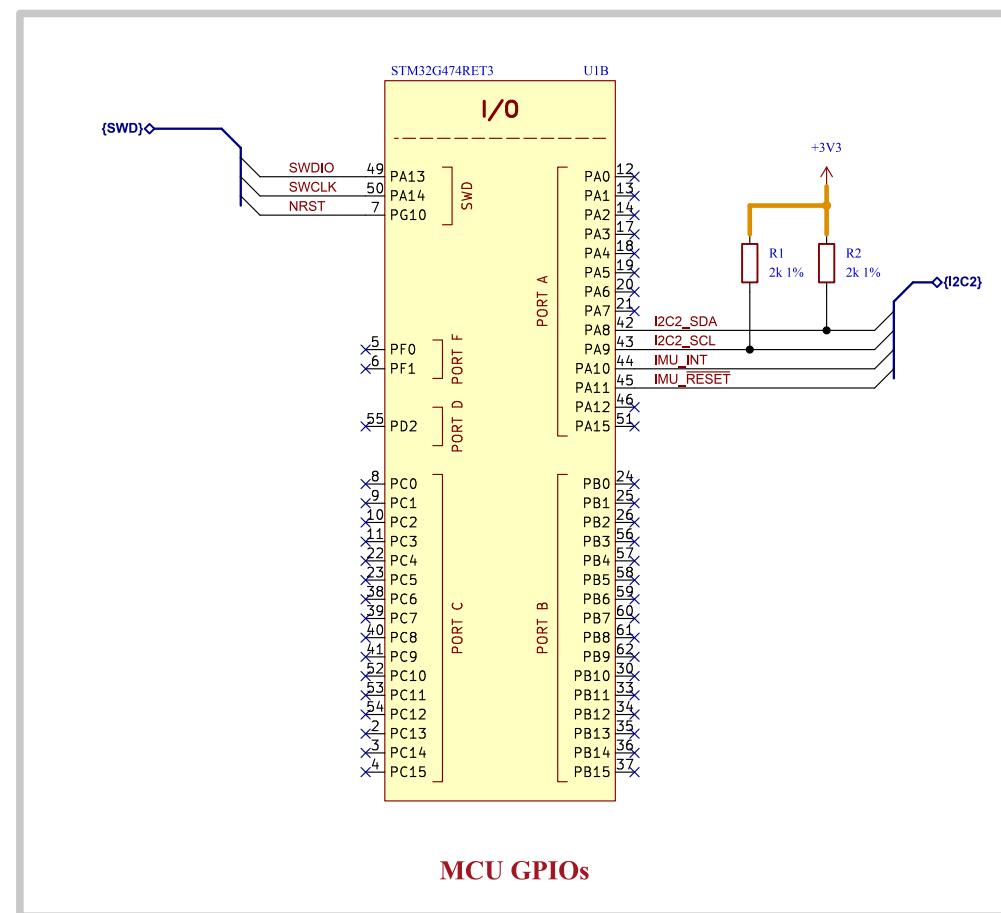
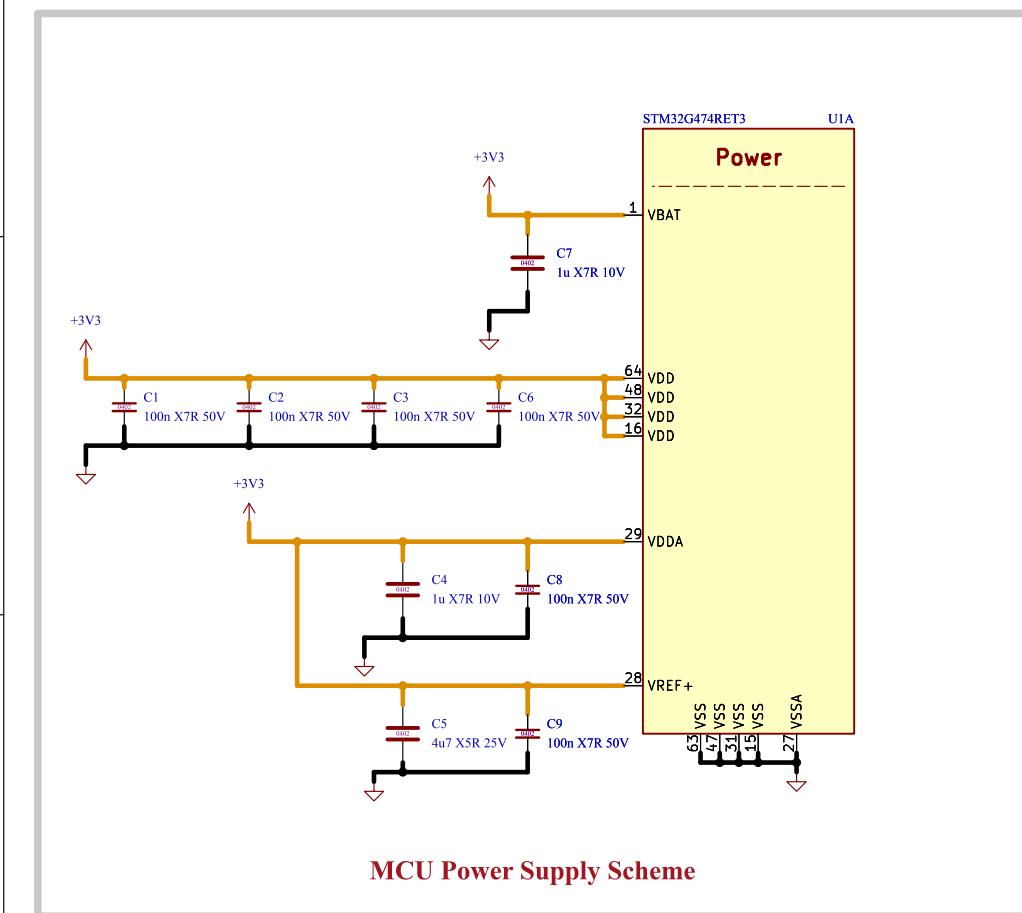
**Holes, Fiducials**

Page 7

File: Holes, Fiducials.kicad\_sch

	Comments:	Company:	Variant:	Git Hash:
		Company Name		
	Board Name:	Board Name	Project Name:	Project Name
	Sheet Title:			
	Project Architecture	File Name: Project Architecture.kicad_sch	Designer: Author	Date: 2025-01-12 + (Unreleased)
	Sheet Path: /Project Architecture/		Reviewer:	Size: <b>A3</b> Sheet: <b>3 of 8</b>

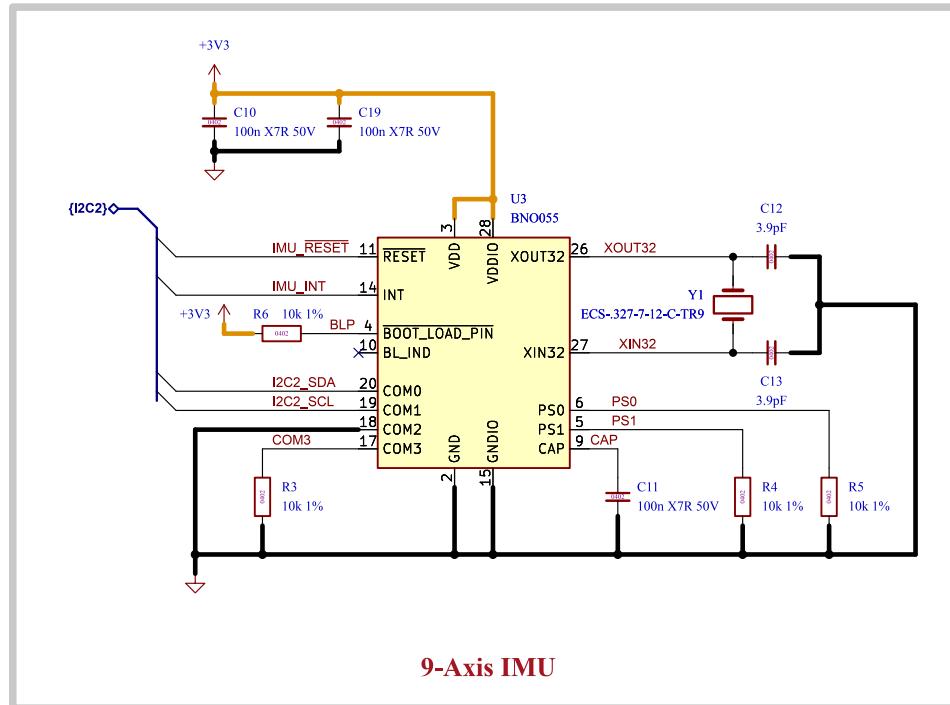
# [4] MCU



Comments: AN5346 STM32G474 Datasheet p.81 J. Pieper ADC investigation	Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
Board Name: <b>Board Name</b>	Project Name: <b>Project Name</b>		
Sheet Title: MCU	File Name: MCU.kicad_sch	Designer: Author	Date: 2025-01-12
Sheet Path: /Project Architecture/MCU/	Reviewer:	Size: <b>A4</b>	Sheet: <b>4</b> of 8

# [5] IMU

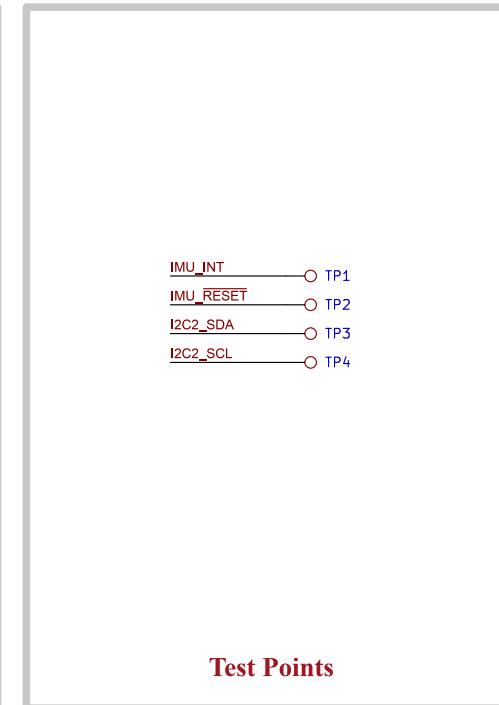
A



IMU_INT	TP1
IMU_RESET	TP2
I2C2_SDA	TP3
I2C2_SCL	TP4

**Test Points**

B



C

1

	Comments:  Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
	Board Name: <b>Board Name</b>	Project Name: <b>Project Name</b>	
	Sheet Title: IMU	File Name: IMU.kicad_sch	Designer: Author
	Sheet Path: /Project Architecture/IMU/		Reviewer: Size: <b>A4</b>
			Sheet: <b>5</b> of 8

2

3

4

5

6

# [6] Connectors

A

A

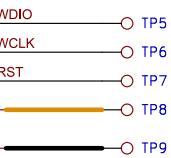
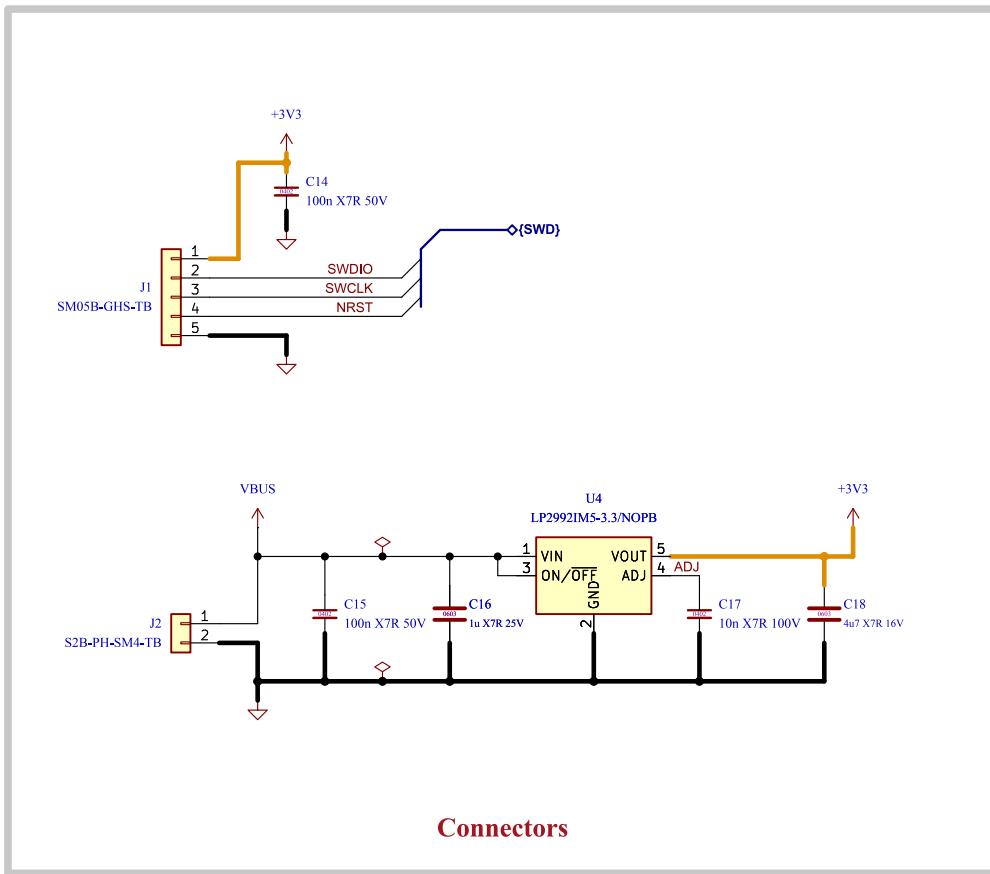
B

B

C

C

**Connectors**



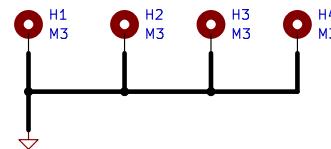
**Test Points**

D

D

	Comments: Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
	Board Name: <b>Board Name</b>	Project Name: <b>Project Name</b>	
	Sheet Title: Connectors	File Name: Connectors.kicad_sch	Designer: Author
	Sheet Path: /Project Architecture/Connectors/	Reviewer:	Size: <b>A4</b>
			Sheet: <b>6</b> of <b>8</b>

# [7] Holes, Fiducials



**Mounting Holes**

**Front**

- FD1
- FD2
- FD3

**Bottom**

- FD4
- FD5
- FD6

**Fiducial Markers**

	Comments:	Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
	Board Name: <b>Board Name</b>			Project Name: <b>Project Name</b>
	Sheet Title: Holes, Fiducials	File Name: Holes, Fiducials.kicad_sch	Designer: Author	Date: 2025-01-24 Revision: + (Unreleased)
	Sheet Path: /Project Architecture/Holes, Fiducials/		Reviewer:	Size: <b>A4</b> Sheet: <b>7</b> of <b>8</b>

## [8] Revision History

A **Version 1.0.0 not found.**

Version 1.0.0 not found.

B

C

D

A

B

C

D

	Comments:	Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
	Board Name: <b>Board Name</b>	Project Name: <b>Project Name</b>		
	Sheet Title: Revision History	File Name: Revision History.kicad_sch	Designer: Author	Date: 2025-01-12 Revision: + (Unreleased)
	Sheet Path: <a href="#">/Revision History/</a>		Reviewer:	Size: <b>A4</b> Sheet: <b>8</b> of <b>8</b>