

# Board Name

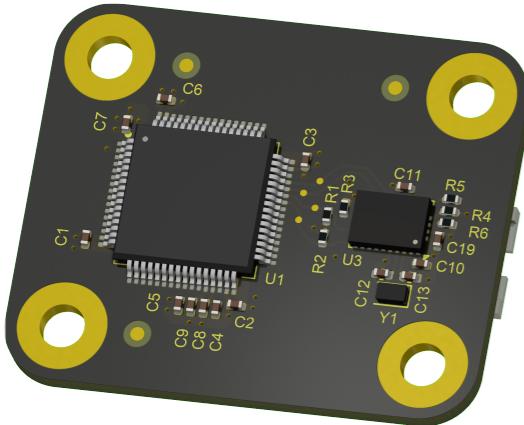
## Variant: CHECKED

2025-03-02

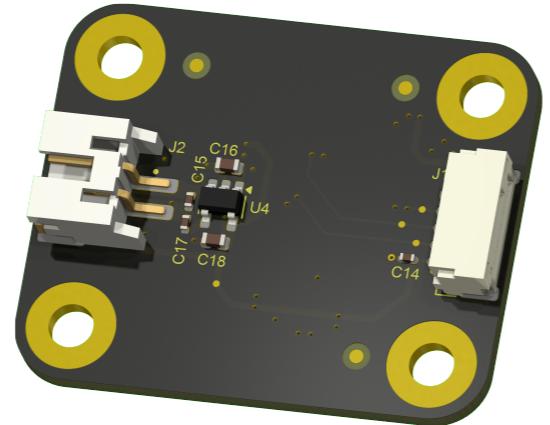
Rev 1.0.0+ (Unreleased)

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

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.

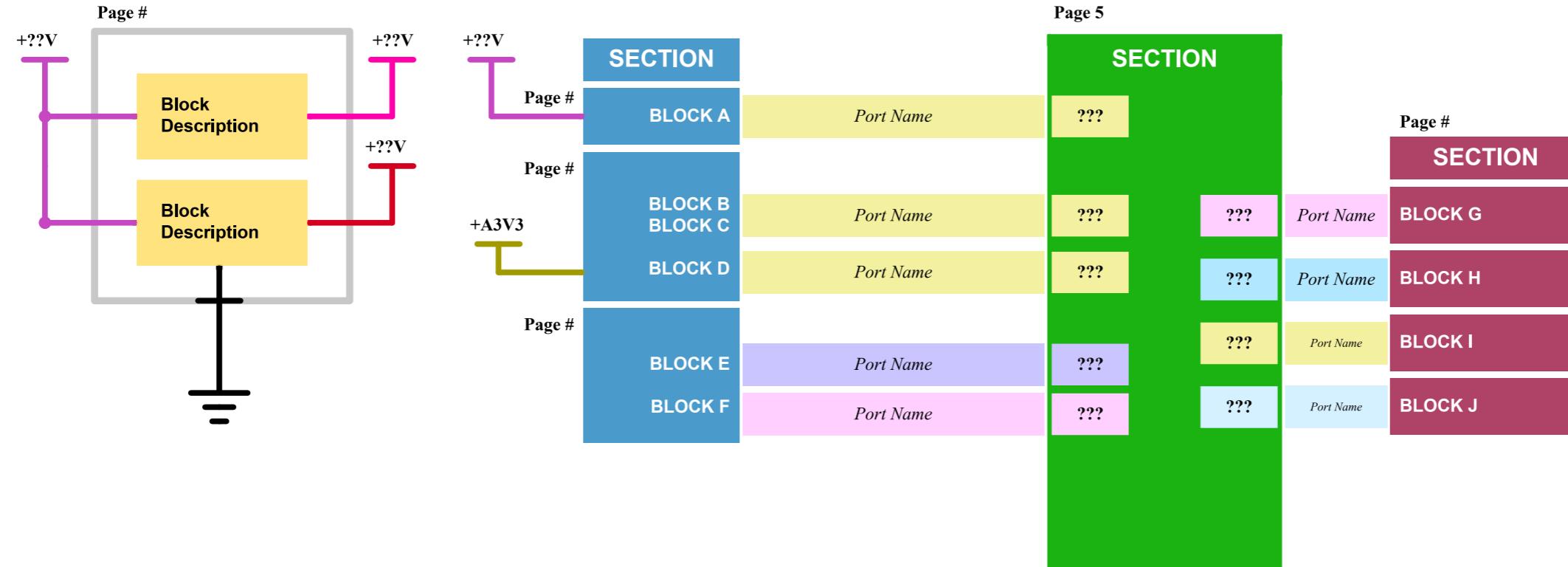
DESIGN NOTE:  
Example text for cautionary design notes.

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: 1.0.0+ (Unreleased)
	Sheet Path: /	Reviewer:	Size: <b>A3</b>	Sheet: <b>1 of 8</b>

# [2] Block Diagram

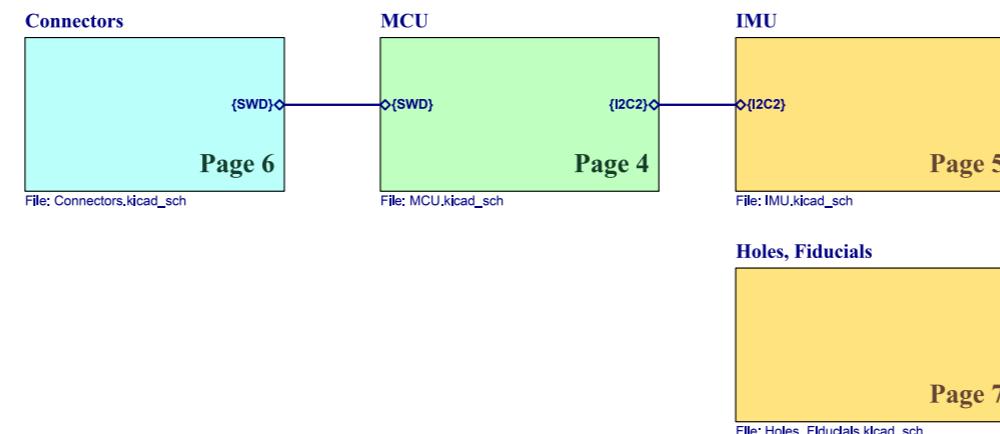


## Target specifications:

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

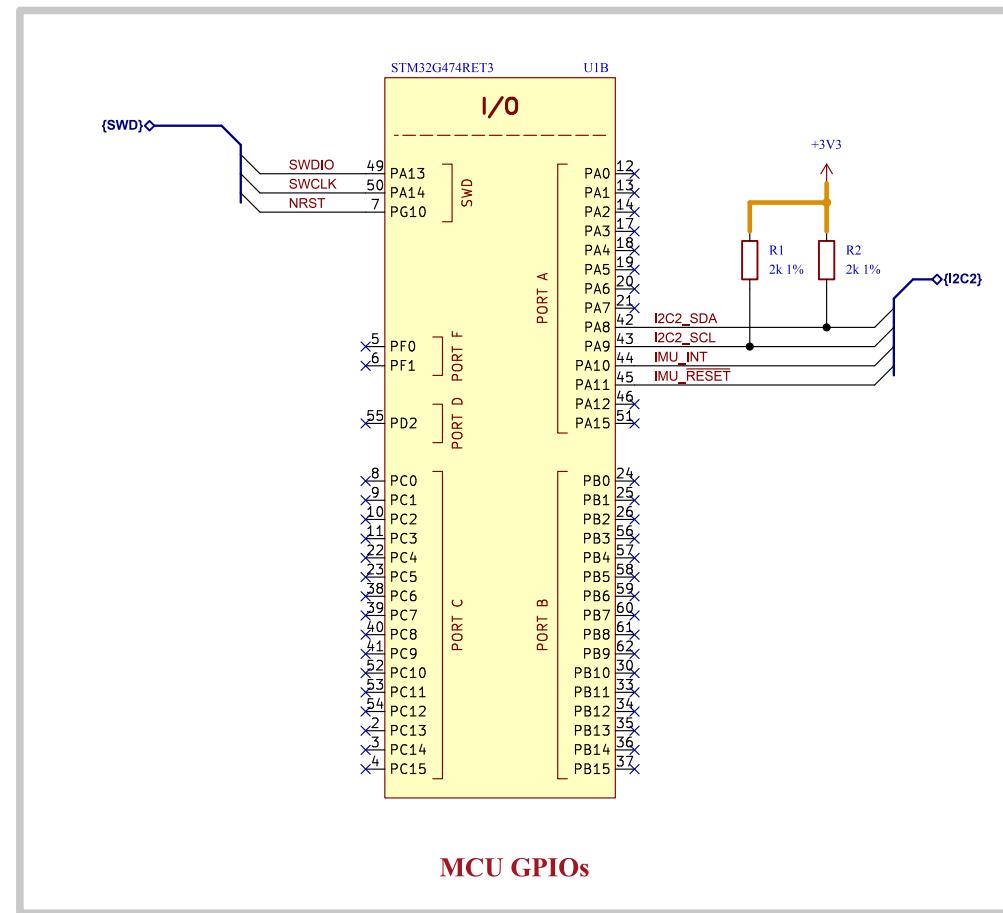
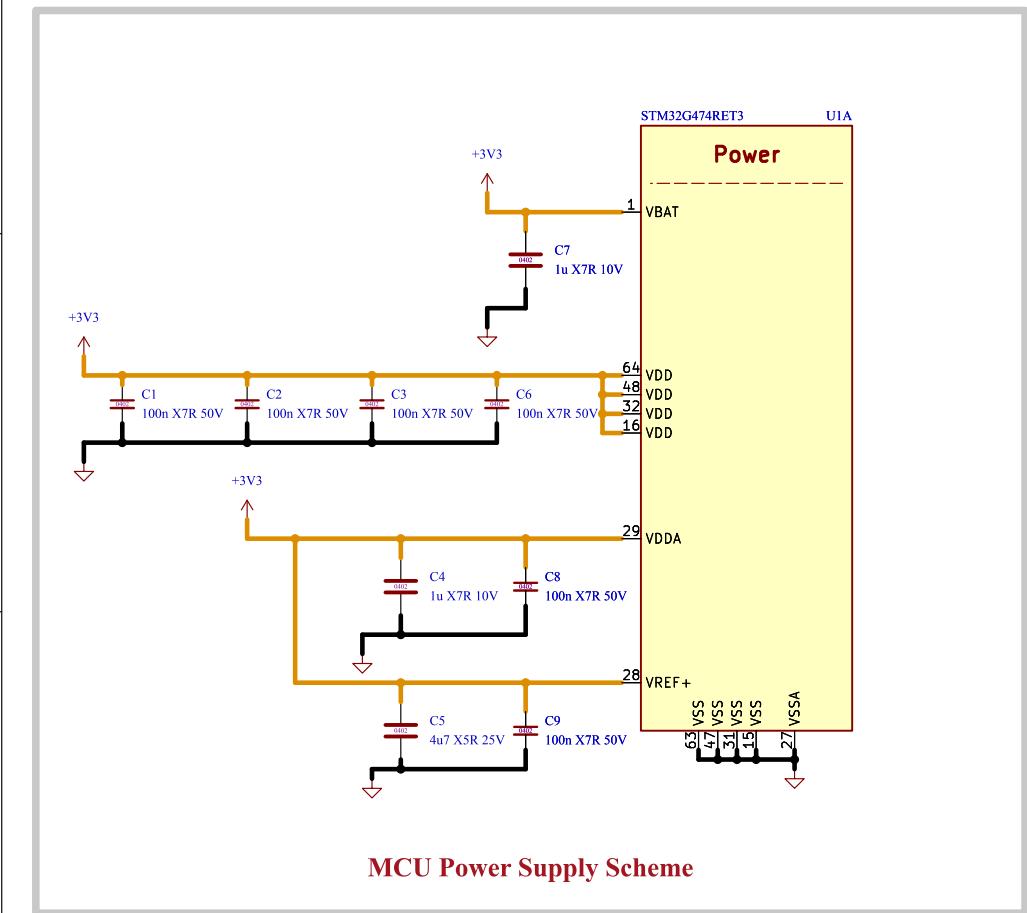
	Comments:  	Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
		Board Name: Board Name	Project Name: Project Name	
	Sheet Title: Block Diagram	File Name: Block Diagram.kicad_sch	Designer: Author	Date: 2025-01-12
	Sheet Path: /Block Diagram/		Reviewer:	Revision: 1.0.0+ (Unreleased)
			Size: <b>A3</b>	Sheet: <b>2 of 8</b>

# [3] Project Architecture



	Comments:	Company: Company Name	Variant: CHECKED	Git Hash: 77e9e33
	Board Name: <b>Board Name</b>		Project Name: <b>Project Name</b>	
	Sheet Title: Project Architecture	File Name: Project Architecture.kicad_sch	Designer: Author	Date: 2025-01-12
	Sheet Path: /Project Architecture/		Reviewer:	Revision: 1.0.0+ (Unreleased)
			Size: <b>A3</b>	Sheet: <b>3 of 8</b>

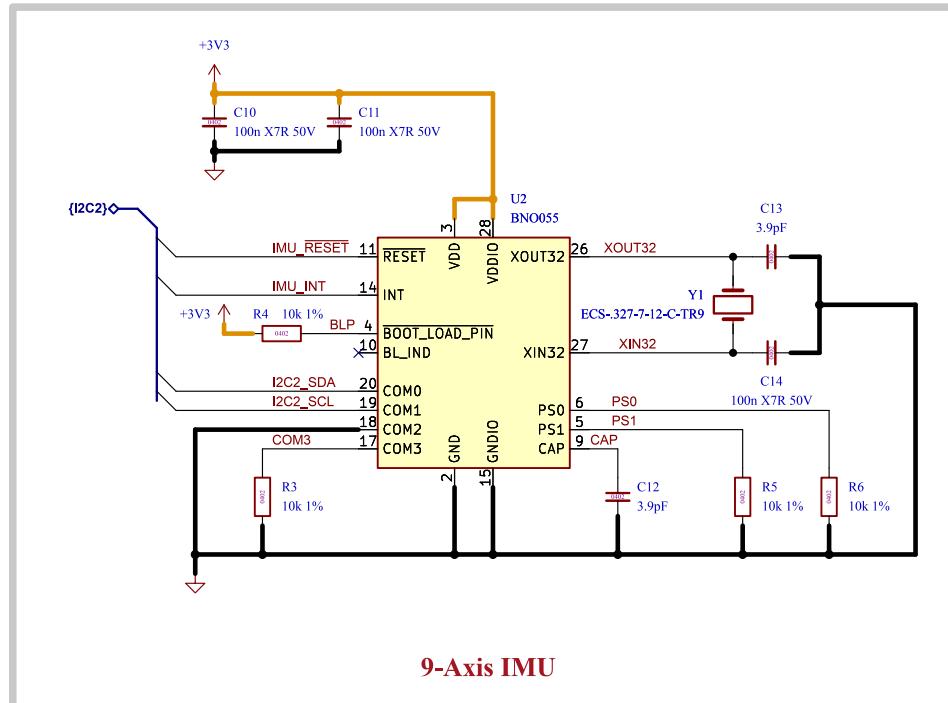
## [4] MCU



	<b>Comments:</b> AN5346 STM32G474 Datasheet p.81 J. Pieper ADC investigation	<b>Company:</b>		
		<b>Company Name</b>	Variant: CHECKED	
		<b>Board Name:</b>	<b>Project Name:</b> <b>Project Name</b>	
		<b>Board Name</b>		
	<b>Sheet Title:</b> MCU	<b>File Name:</b> MCU.kicad_sch	<b>Designer:</b> Author	<b>Date:</b> 2025-01-12 <b>Revision:</b> 1.0.0+ (Unreleased)
	<b>Sheet Path:</b> /Project Architecture/MCU/		<b>Reviewer:</b>	<b>Size:</b> <b>A4</b> <b>Sheet:</b> <b>4</b> of <b>8</b>

# [5] IMU

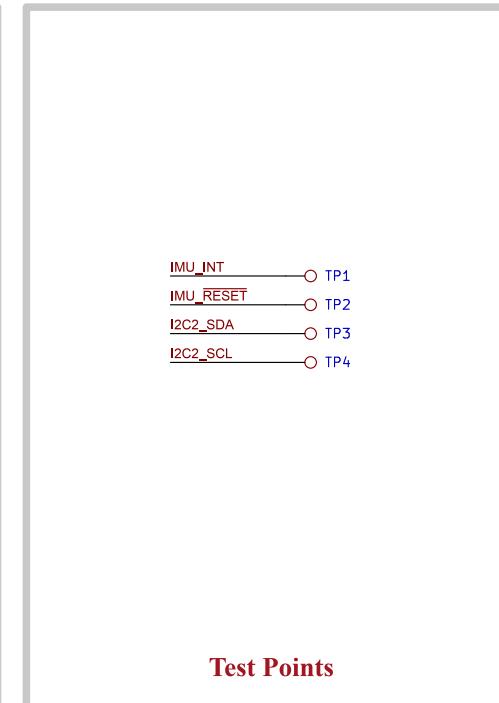
A



**IMU\_INT** — TP1  
**IMU\_RESET** — TP2  
**I2C2\_SDA** — TP3  
**I2C2\_SCL** — TP4

**Test Points**

B

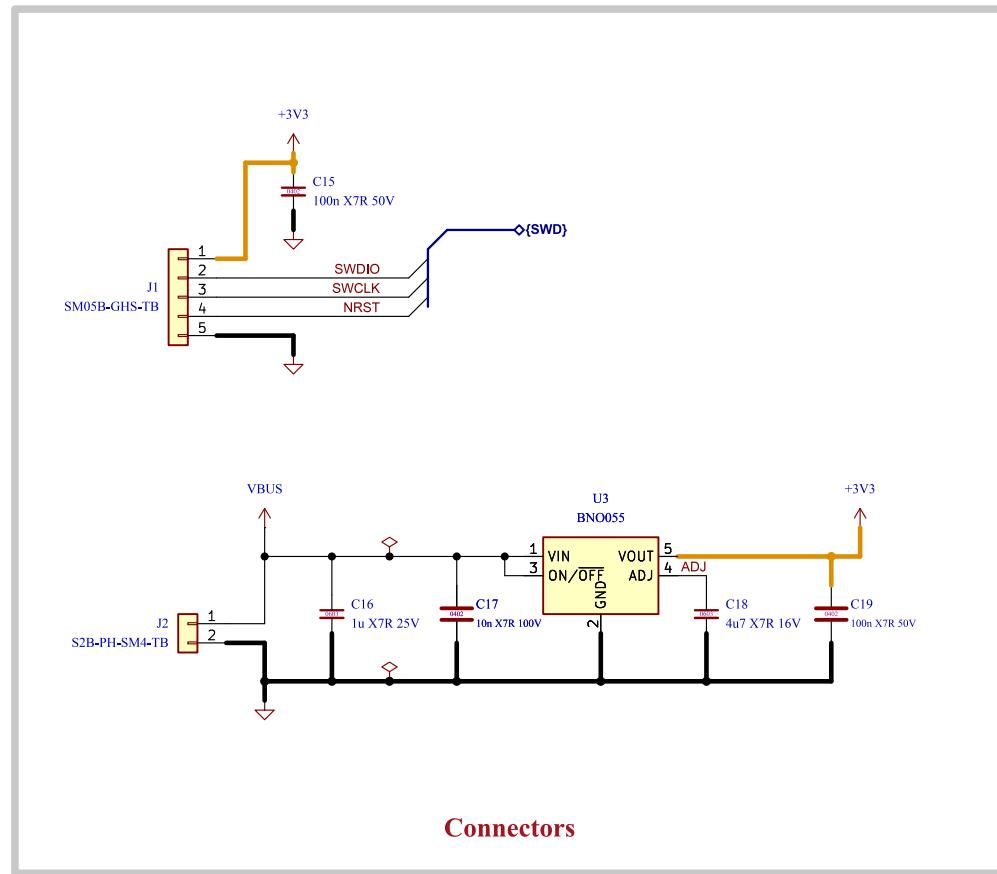


C

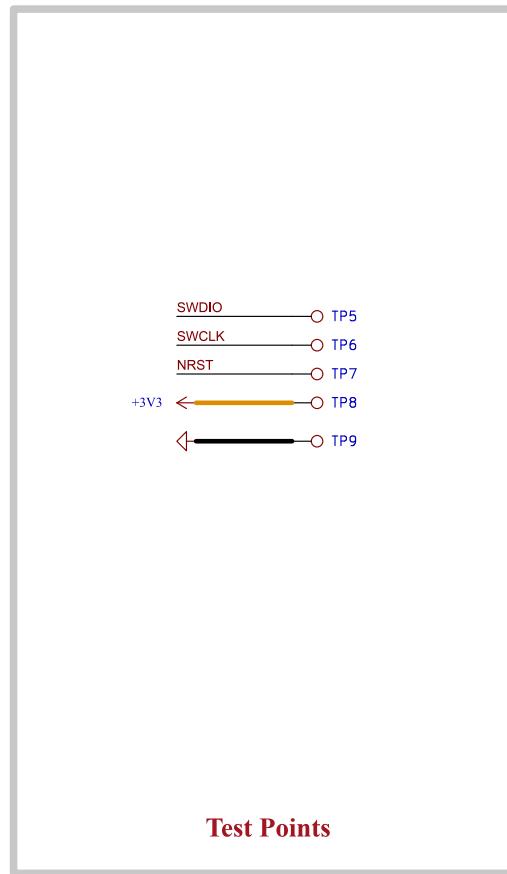
		<b>Comments:</b> Company: Company Name	<b>Variant:</b> CHECKED	<b>Git Hash:</b> 77e9e33
		<b>Board Name:</b> <b>Board Name</b>		<b>Project Name:</b> <b>Project Name</b>
		<b>Sheet Title:</b> IMU	<b>File Name:</b> IMU.kicad_sch	<b>Designer:</b> Author
		<b>Sheet Path:</b> /Project Architecture/IMU/		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

# [6] Connectors

A



A



B

C

D

C

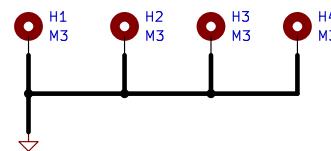
D

B

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: 1.0.0+ (Unreleased)
	Sheet Path: /Project Architecture/Holes, Fiducials/			Reviewer: <b>A4</b> Sheet: <b>7</b> of <b>8</b>

# [8] Revision History

A	<p><b>Version 1.0.0 - 2025-03-02</b></p> <hr/> <p>Added</p> <ul style="list-style-type: none"><li>- First version of the PCB and schematic.</li></ul>	<p><b>Version 1.0.1 not found.</b></p> <p>Version 1.0.1 not found.</p>				A
B						B
C						C
D		<p>Comments:</p> <p>Company: Company Name</p> <p>Board Name: <b>Board Name</b></p> <p>Sheet Title: Revision History</p> <p>File Name: Revision History.kicad_sch</p> <p>Designer: Author</p> <p>Date: 2025-01-12</p> <p>Revision: 1.0.0+ (Unreleased)</p> <p>Sheet Path: /Revision History/</p>	<p>Variant: CHECKED</p> <p>Git Hash: 77e9e33</p> <p>Project Name: <b>Project Name</b></p> <p>Reviewer:</p> <p>Size: <b>A4</b></p> <p>Sheet: <b>8</b> of <b>8</b></p>			D