

# PCB NINA PROJECT

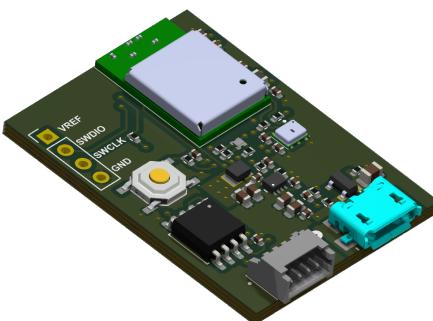
## Variant: DRAFT

2025-05-21

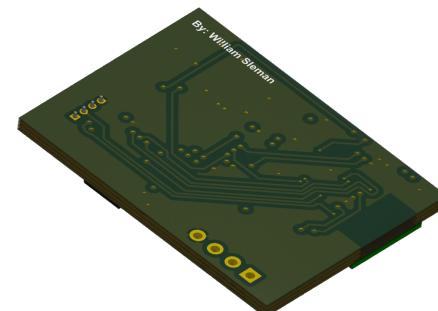
Rev 1.0

Page	Index	Page	Index	Page	Index	Page	Index
1	Cover Page	11	.....	21	.....	31	.....
2	Block Diagram	12	.....	22	.....	32	.....
3	Power Supply	13	.....	23	.....	33	.....
4	MCU	14	.....	24	.....	34	.....
5	Revision History	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



### 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.

### NOTES

Not fitted components are marked as

DRAFT - Very early stage of schematic, ignore details.

PRELIMINARY - Close to final schematic.

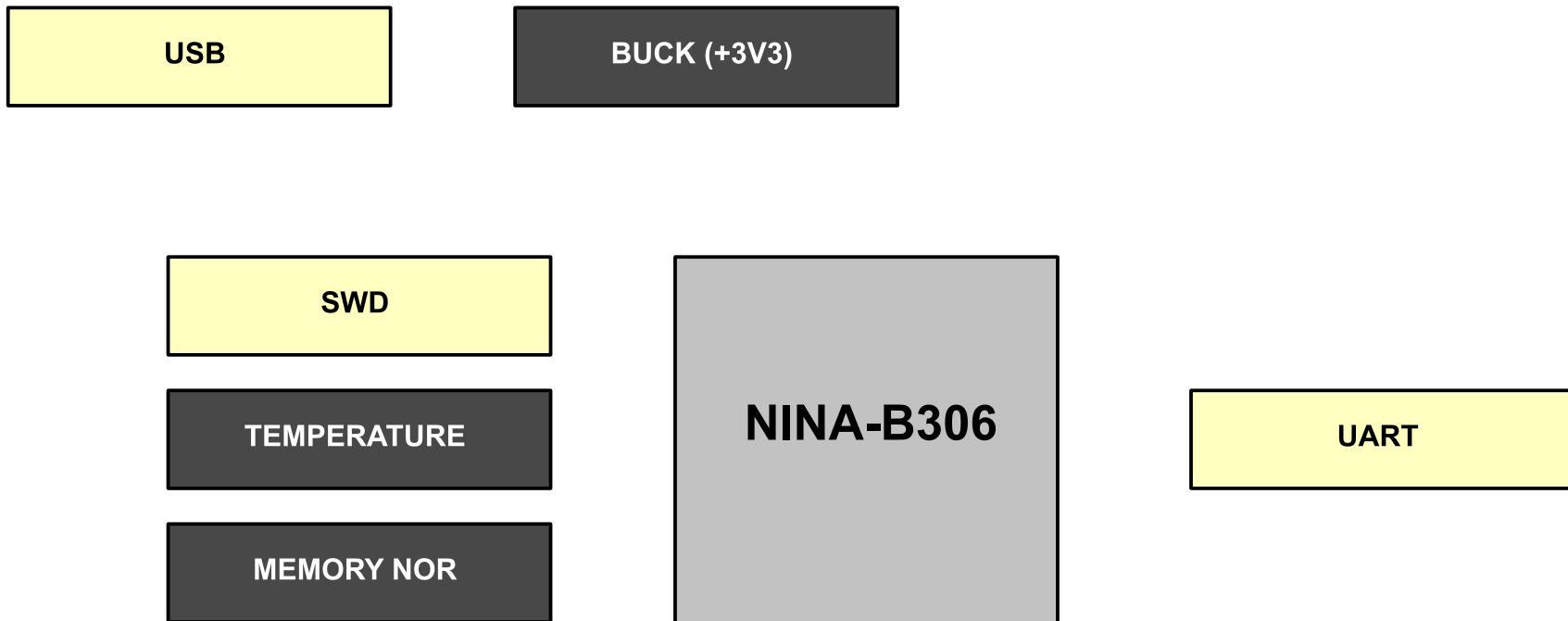
CHECKED - There shouldn't be any mistakes. Contact the engineer if you find any.

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

DRAFT 16-MAY-2025

Author: William Sleman	Sheet: /	Date: 2025-05-16	Rev: 1.0
File: PCB_TCC.kicad_sch			
Title: PCB NRF52 PROJECT			
Size: A3			
KiCad E.D.A. 9.0.2			
			Id: 1/5

# [02] - Block Diagram



Author: William Sleman

Sheet: /[02] – Block Diagram/

File: sheet2\_sch.kicad\_sch

Title: PCB NRF52 PROJECT

Size: A4 Date: 2025-05-16

KiCad E.D.A. 9.0.2

Rev: 1.0

Id: 2/5

# [03] - Power Supply

A

A

B

B

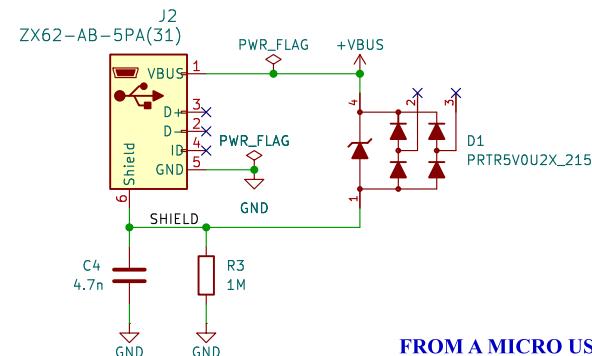
C

C

D

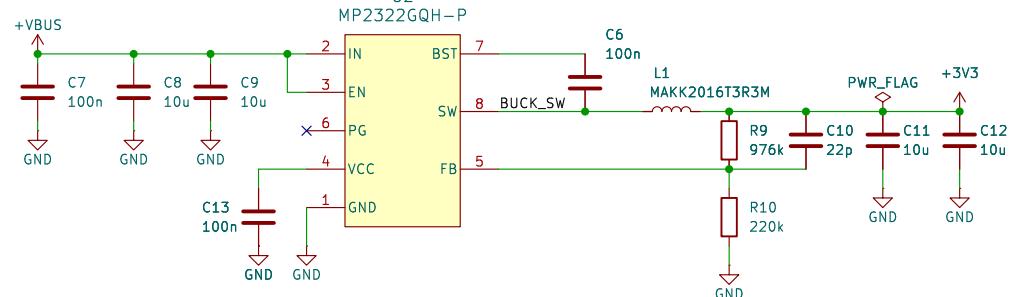
D

## CONNECTOR



FROM A MICRO USB

## BUCK (+3V3)



Author: William Sleman

Sheet: /[03] - Power Supply/

File: sheet3\_sch.kicad\_sch

Title: PCB NRF52 PROJECT

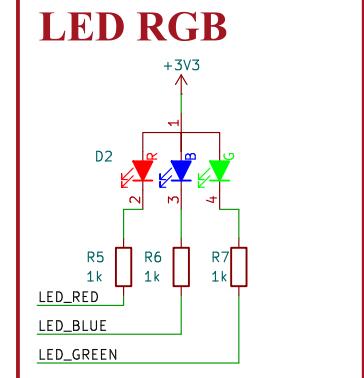
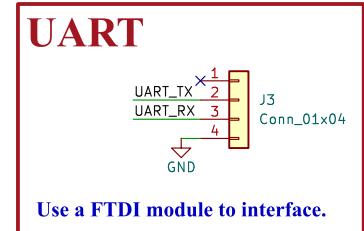
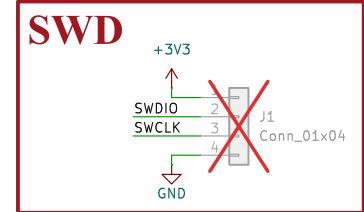
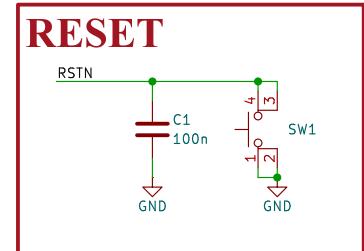
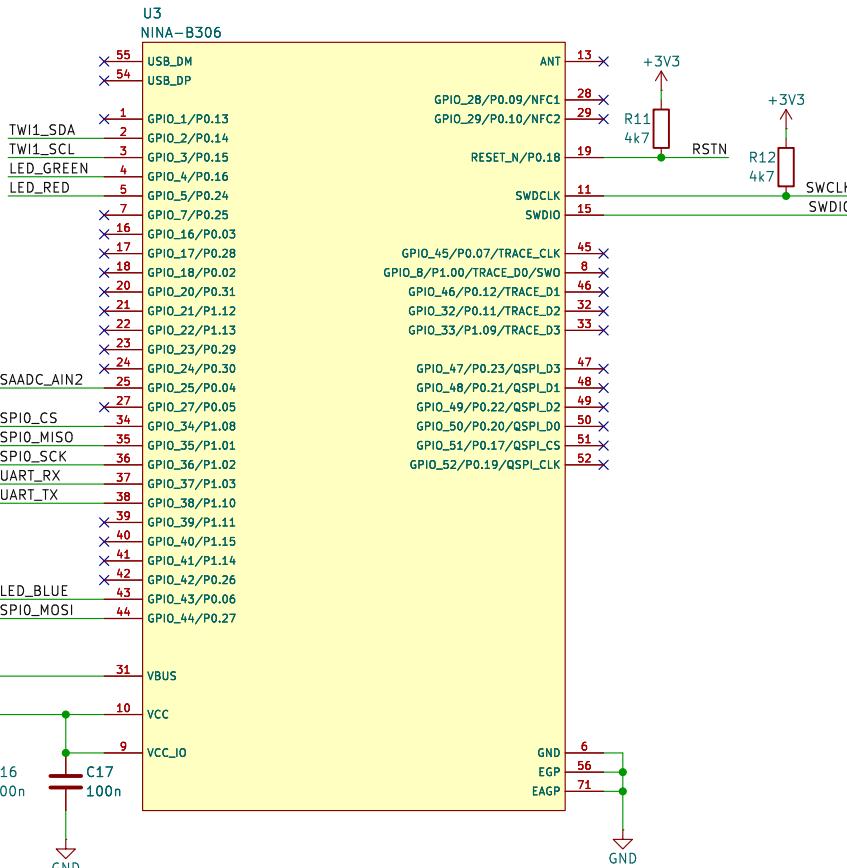
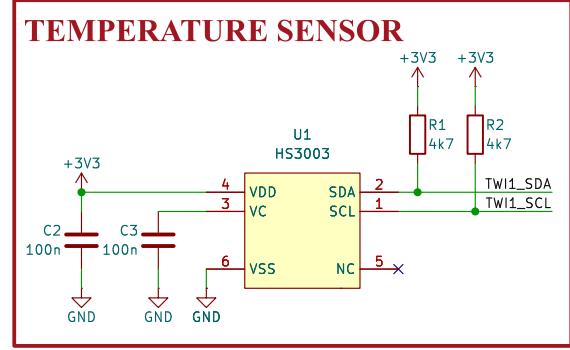
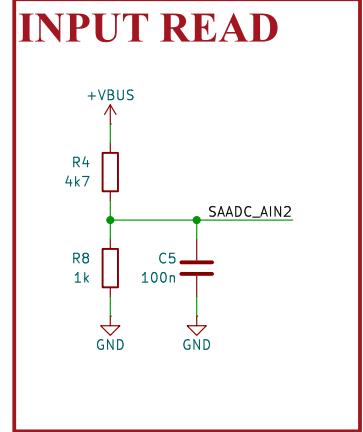
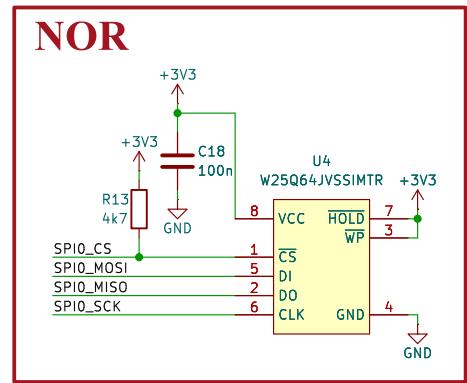
Size: A4 Date: 2025-05-16

KiCad E.D.A. 9.0.2

Rev: 1.0

Id: 3/5

# [04] - MCU



Author: William Sleman

Sheet: /[04] - MCU/  
File: sheet4\_sch.kicad\_sch

Title: PCB NRF52 PROJECT

Size: A4 Date: 2025-05-16  
KiCad E.D.A. 9.0.2

Rev: 1.0  
Id: 4/5

## [05] - Revision History

DD.MM.YYYY - xxx Revision  
Variant: xxx

Author: William Sleman

Sheet: /[05] – Revision History/  
File: sheet4.kicad\_sch

Title: PCB NRF52 PROJECT

Size: A4

Rev: 1.0

REV. 1.0