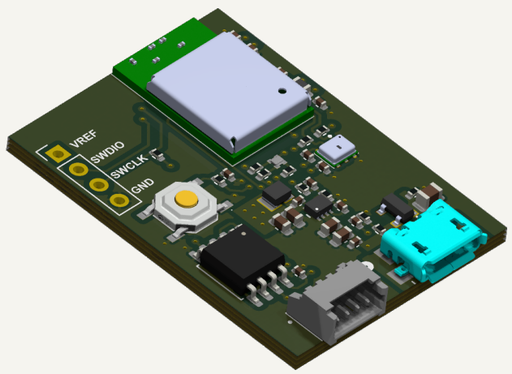
PCB NINA PROJECT

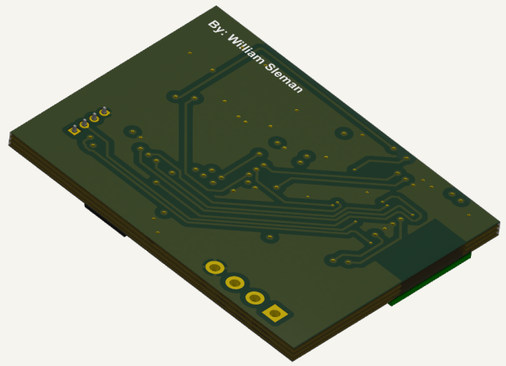
Variant: CHECKED

2026-02-13
Rev 1.0

TOP VIEW



BOTTOM VIEW



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.

CHECKED 06-JAN-2026

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.

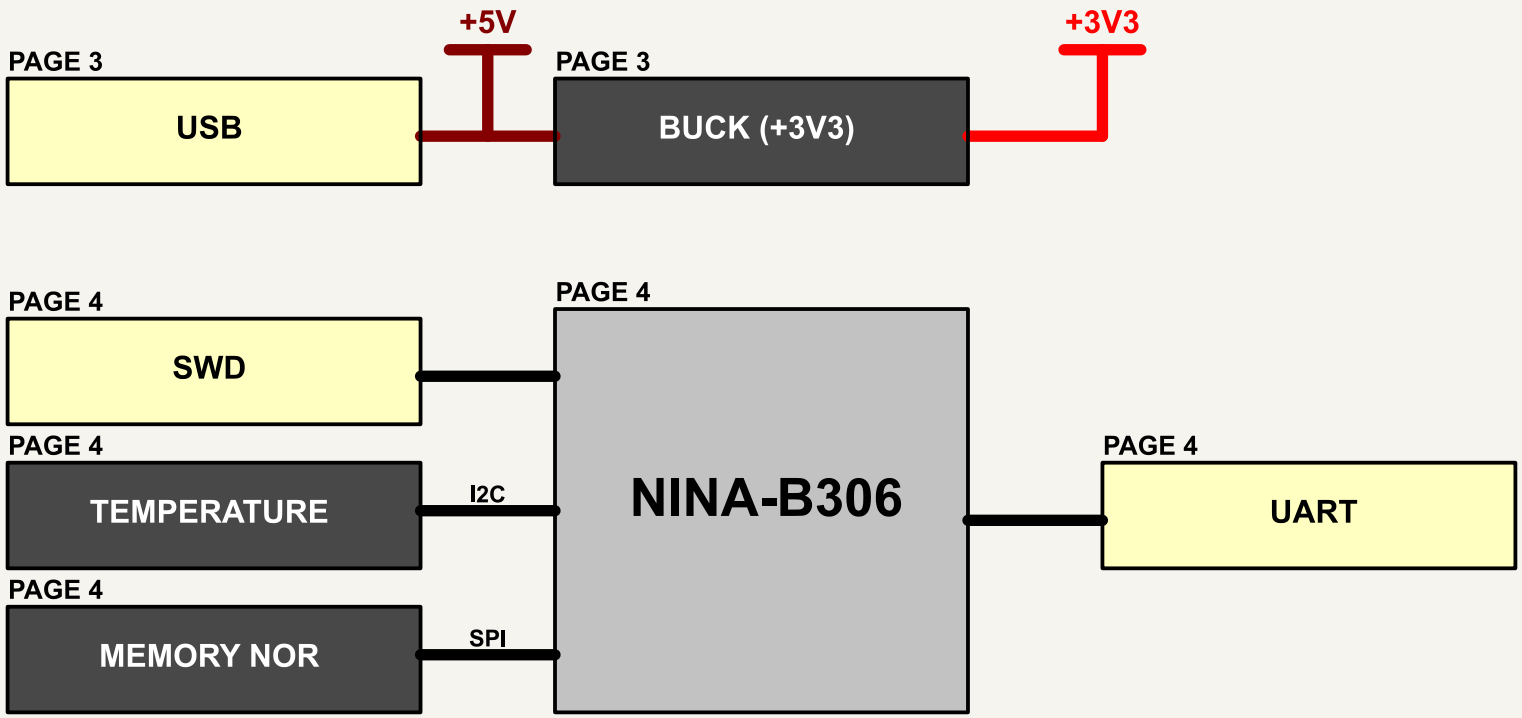
Author: William Sleman

Sheet: /
File: HW.kicad_sch

Title: PCB NRF52 PROJECT

Size: A3 | Date: 2025-05-16 | Rev: 1.0
KiCad E.D.A. 9.0.0 | Id: 1/5

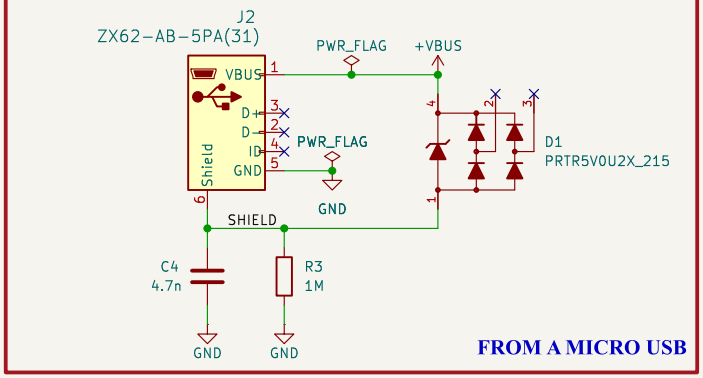
[02] - Block Diagram



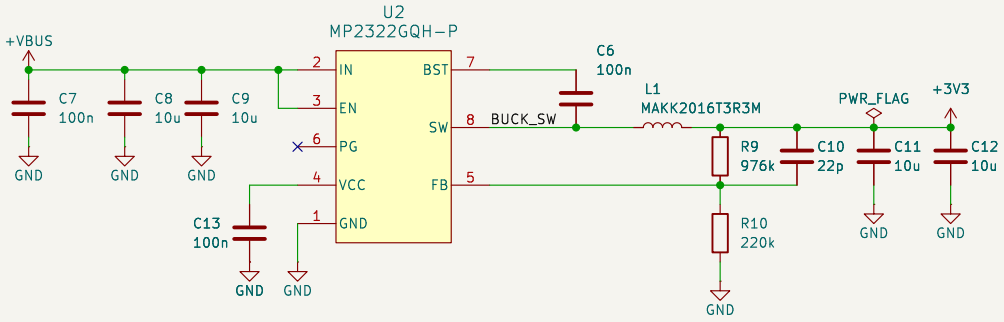
Author: William Sleman
Sheet: /[02] - Block Diagram/
File: sheet2.kicad_sch
Title: PCB NRF52 PROJECT
Size: A4 | Date: 2025-05-16 | Rev: 1.0
KiCad E.D.A. 9.0.0 | Id: 2/5

[03] - Power Supply

CONNECTOR

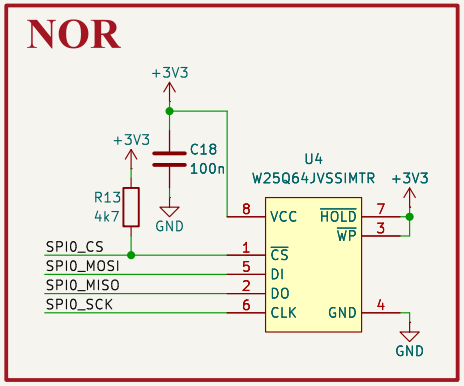


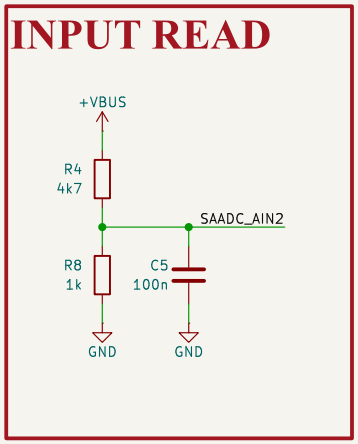
BUCK (+3V3)

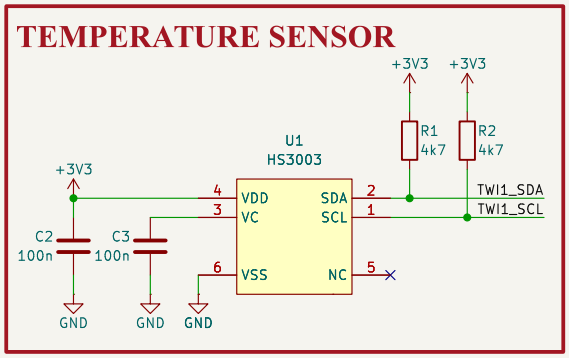


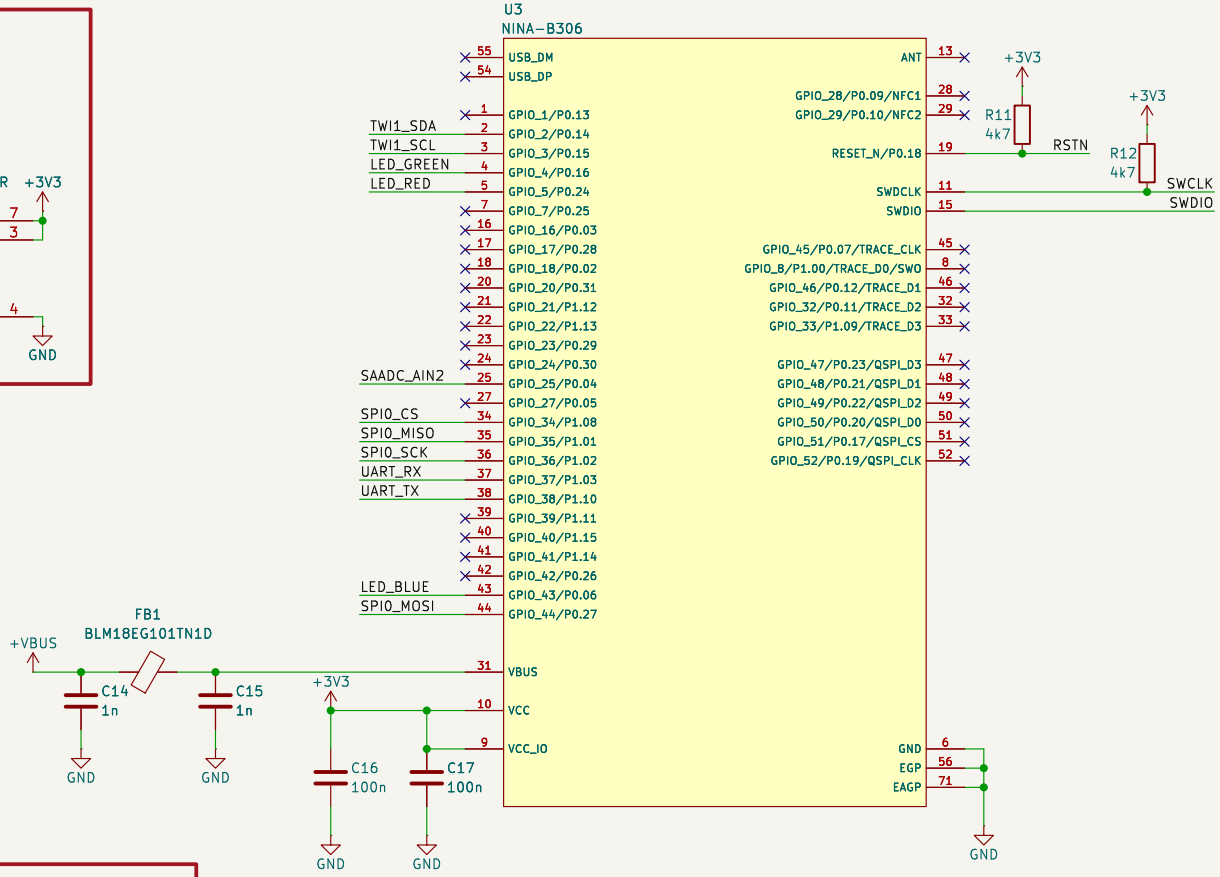
Author: William Sleman		
Sheet: /[03] - Power Supply/		
File: sheet3.kicad_sch		
Title: PCB NRF52 PROJECT		
Size: A4	Date: 2025-05-16	Rev: 1.0
KiCad E.D.A. 9.0.0		Id: 3/5

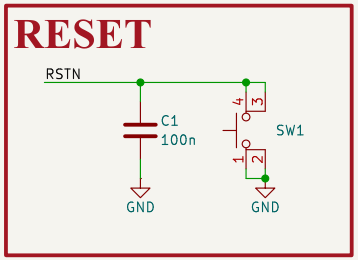
[04] - MCU

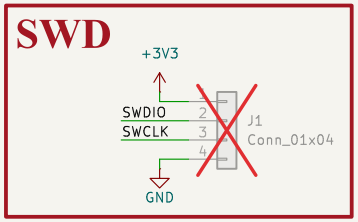


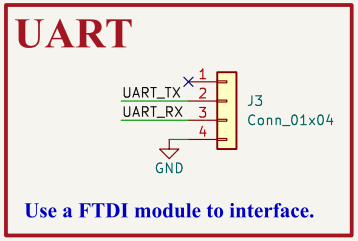


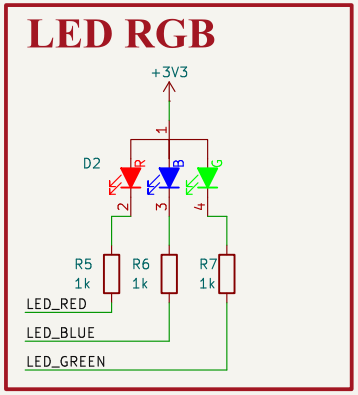












Author: William Sleman

Sheet: /[04] - MCU/
File: sheet4.kicad_sch

Title: PCB NRF52 PROJECT

Size: A4	Date: 2025-05-16	**Rev: 1.0**
KiCad E.D.A. 9.0.0		Id: 4/5

[05] - Revision History

DD.MM.YYYY - xxx Revision
Variant: xxx

-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-

DD.MM.YYYY - xxx Revision
Variant: xxx

-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-

DD.MM.YYYY - xxx Revision
Variant: xxx

-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-

DD.MM.YYYY - xxx Revision
Variant: xxx

-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-
-

Author: William Sleman

Sheet: /[05] - Revision History/
File: sheet5.kicad_sch

Title: PCB NRF52 PROJECT

Size: A4	Date: 2025-05-16	**Rev: 1.0**
KiCad E.D.A. 9.0.0		Id: 5/5