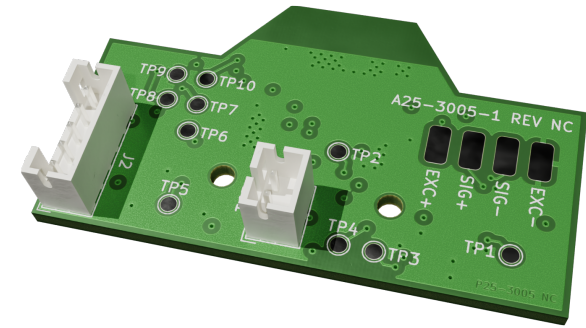
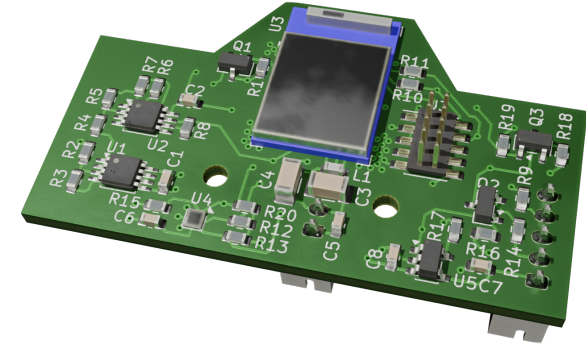


| REVISION HISTORY | | |
|------------------|-------------|------|
| REV | DESCRIPTION | ZONE |
| | | |

TABLE OF CONTENTS

| | | | |
|----------|---------------------------|----------|---------------------------|
| Sheet 1 | Cover Page | Sheet 11 | Error: XML File not found |
| Sheet 2 | Error: XML File not found | Sheet 12 | Error: XML File not found |
| Sheet 3 | Error: XML File not found | Sheet 13 | Error: XML File not found |
| Sheet 4 | Error: XML File not found | Sheet 14 | Error: XML File not found |
| Sheet 5 | Error: XML File not found | Sheet 15 | Error: XML File not found |
| Sheet 6 | Error: XML File not found | Sheet 16 | Error: XML File not found |
| Sheet 7 | Error: XML File not found | Sheet 17 | Error: XML File not found |
| Sheet 8 | Error: XML File not found | Sheet 18 | Error: XML File not found |
| Sheet 9 | Error: XML File not found | Sheet 19 | Error: XML File not found |
| Sheet 10 | Error: XML File not found | Sheet 20 | Error: XML File not found |

PCBA RENDERS



SCHEMATIC NOTES

Unless otherwise specified, components Values are in ohms, μ F, and μ H.

DESIGN NOTES

| | | | | |
|--|---|---|--|---|
| DESIGN NOTE: Example text for informational design notes. | DESIGN NOTE: Example text for cautionary design notes. | DESIGN NOTE: Example text for debug notes. | LAYOUT NOTE: Example text for critical layout guidelines. | DESIGN NOTE: Example text for critical design notes. |
|--|---|---|--|---|

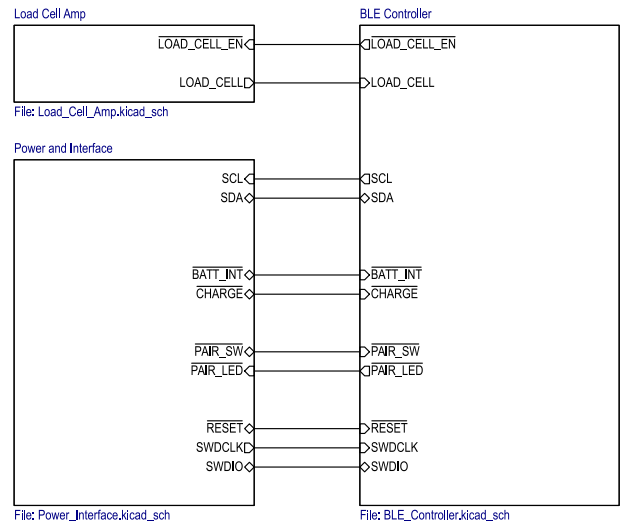


Drawing Title
Schematic, Assembly Name

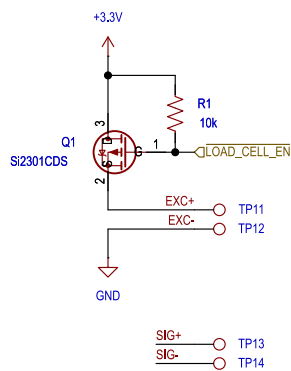
| | | | | | | | |
|----------|----------|-----------|------------------|--------------------|----------|--------------|----------------------|
| Designer | JPK | Size | A4 | Drawing Number | S99-9000 | Revision | NO_TAG+ (Unreleased) |
| Used On | A99-9000 | File Name | Test_5.kicad_sch | Release State | DRAFT | Release Date | 2026-03-06 |
| Used On | A99-9000 | Scale | 2:1 | Git Hash | fc4a488 | Sheet | 1 of 7 |
| | | | | KICad E.D.A. 9.0.7 | | | |

| | | | | | | |
|---|---|---|---|---|---|---|
| | 1 | 2 | 3 | 4 | 5 | 6 |
| A | | | | | | |
| B | | | | | | |
| C | | | | | | |
| D | | | | | | |
| | 1 | 2 | 3 | 4 | 5 | 6 |

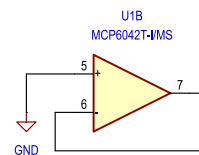
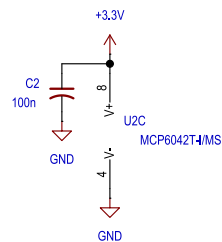
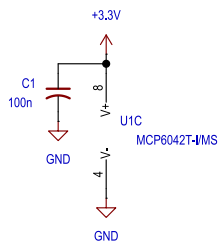
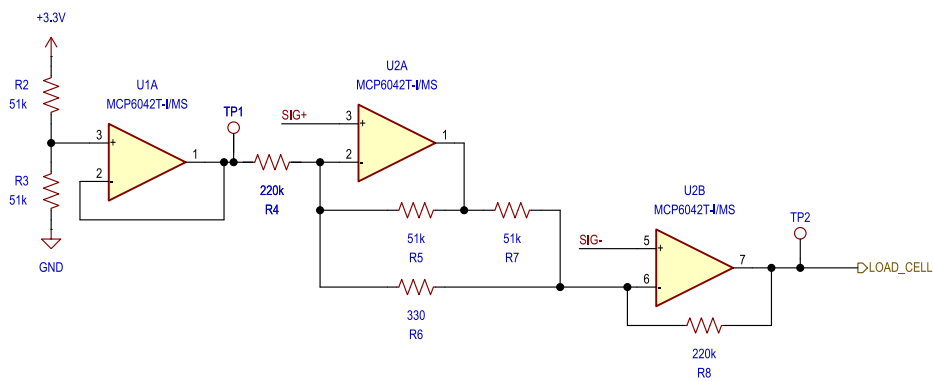
| | | | | | |
|---------------|--------------------|--|----------------------|--------------------------|--|
| Sheet Title | | | | Block Diagram | |
| Drawing Title | | | | Schematic, Assembly Name | |
| Size | Drawing Number | | Revision | | |
| A4 | S99-9000 | | NO_TAG+ (Unreleased) | | |
| File Name | | | | Block_Diagram.kicad_sch | |
| Release State | Release Date | | Git Hash | | |
| DRAFT | | | fc4a488 | | |
| Scale | KICad E.D.A. 9.0.7 | | Sheet | | |
| 2:1 | | | 2 of 7 | | |



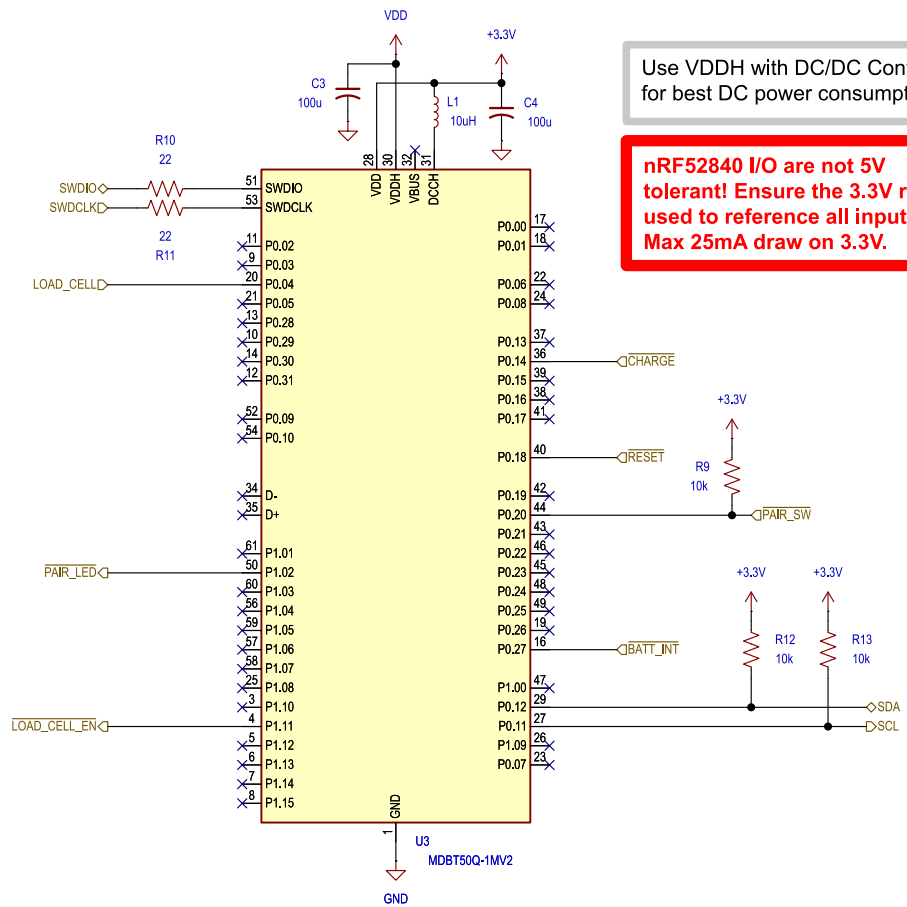
| | | | |
|--|-------------------------|-------------------------------|------------------|
| Sheet Title Project Architecture | | | |
| Drawing Title Schematic, Assembly Name | | | |
| Size A4 | Drawing Number S99-9000 | Revision NO_TAG+ (Unreleased) | |
| File Name Architecture.kicad_sch | | | |
| Release State DRAFT | Release Date | | Git Hash fc4a488 |
| Scale 2:1 | KICad E.D.A. 9.0.7 | | Sheet 3 of 7 |



Place four, 1.5mm x 3mm solder pads for Load Cell connections (signals EXC+, EXC-, SIG+, and SIG-). Mark with silkscreen as "E+", "E-", "S+", "S-".



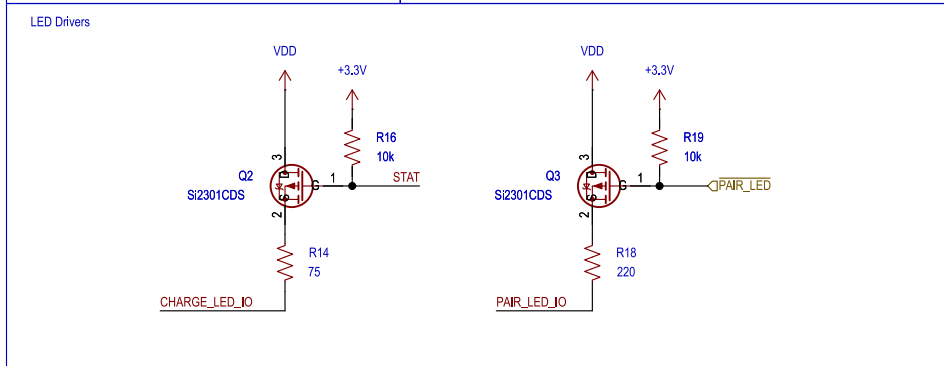
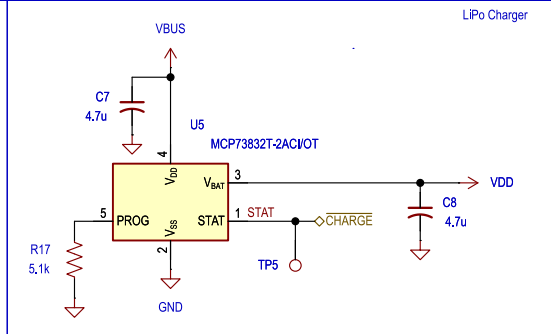
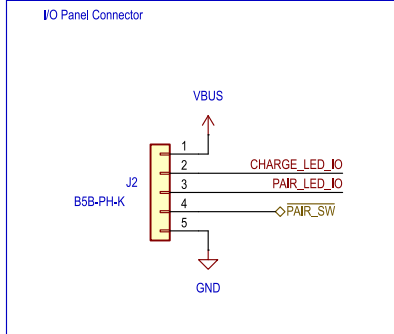
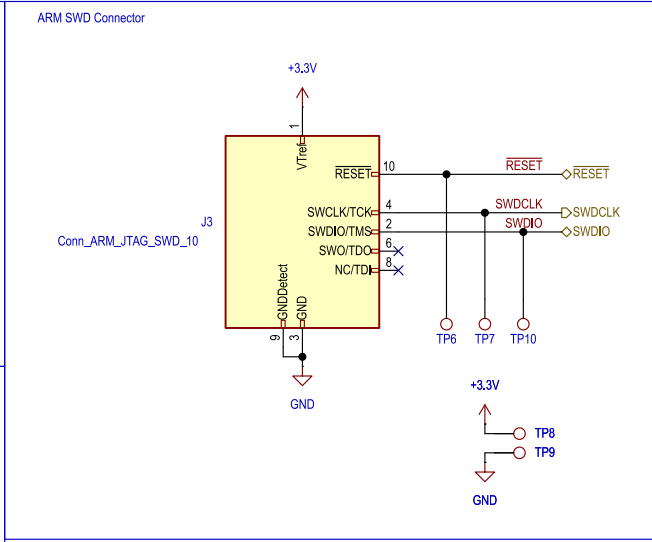
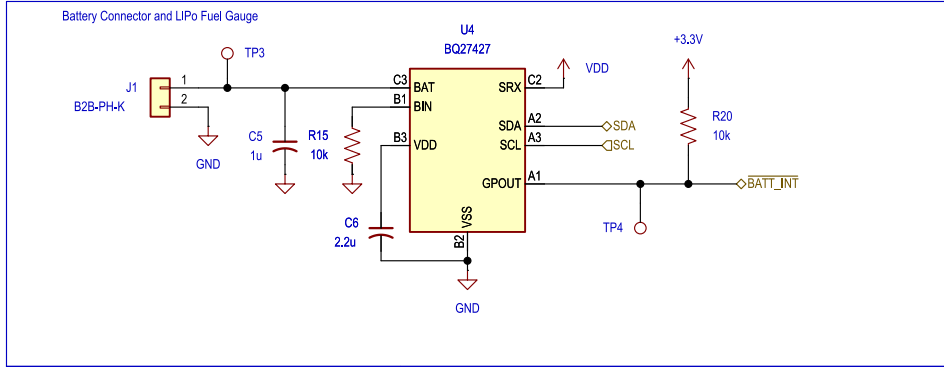
| | | |
|--|-------------------------|-------------------------------|
| Sheet Title Load Cell Amp | | |
| Drawing Title Schematic, Assembly Name | | |
| Size A4 | Drawing Number S99-9000 | Revision NO_TAG+ (Unreleased) |
| File Name Load_Cell_Amp.kicad_sch | | |
| Release State DRAFT | Release Date | Git Hash fc4a488 |
| Scale 2:1 | KICad E.D.A. 9.0.7 | Sheet 4 of 7 |



Use VDDH with DC/DC Converter for best DC power consumption.

nRF52840 I/O are not 5V tolerant! Ensure the 3.3V rail is used to reference all inputs. Max 25mA draw on 3.3V.

| | | |
|--|-------------------------|-------------------------------|
| Sheet Title Circuit 2 | | |
| Drawing Title Schematic, Assembly Name | | |
| Size A4 | Drawing Number S99-9000 | Revision NO_TAG+ (Unreleased) |
| File Name BLE_Controller.kicad_sch | | |
| Release State DRAFT | Release Date | Git Hash fc4a488 |
| Scale 2:1 | KiCad E.D.A. 9.0.7 | Sheet 5 of 7 |



The LEDs on the P25-9010-2 board have the following characteristics:
 - Charge LED: 25mA If, 1.6~2.6V Vf => 75 ohm
 - Pair LED: 5mA If, 2.6~3.1V Vf => 220 ohm

| | | |
|--|-------------------------|-------------------------------|
| Sheet Title Battery Interface | | |
| Drawing Title Schematic, Assembly Name | | |
| Size A4 | Drawing Number S99-9000 | Revision NO_TAG+ (Unreleased) |
| File Name Power_Interface.kicad_sch | | |
| Release State DRAFT | Release Date | Git Hash fc4a488 |
| Scale 2:1 | KICad E.D.A. 9.0.7 | Sheet 6 of 7 |

| | | | | | | |
|---|---|---|---|---|---|---|
| | 1 | 2 | 3 | 4 | 5 | 6 |
| A | | | | | | |
| B | | | | | | |
| C | | | | | | |
| D | | | | | | |
| | 1 | 2 | 3 | 4 | 5 | 6 |

| | | | | | |
|---------------|--------------------|--|----------------------|--------------------------|--|
| Sheet Title | | | | Parts List | |
| Drawing Title | | | | Schematic, Assembly Name | |
| Size | Drawing Number | | Revision | | |
| A4 | S99-9000 | | NO_TAG+ (Unreleased) | | |
| File Name | | | | Parts_List.kicad_sch | |
| Release State | Release Date | | Git Hash | | |
| DRAFT | | | fc4a488 | | |
| Scale | KICad E.D.A. 9.0.7 | | Sheet | | |
| 2:1 | | | 7 of 7 | | |