

1

2

3

4

5

6

A

B

C

D

TABLE OF CONTENTS

Sheet 1 Cover Page

Sheet 2 Block Diagram

Sheet 3 Project Architecture

Sheet 4 Load Cell Amp

Sheet 5 BLE Controller

Sheet 6 Power and Interface

Sheet 7 Parts List

Sheet 8 .....

Sheet 9 .....

Sheet 10 .....

Sheet 11 .....

Sheet 12 .....

Sheet 13 .....

Sheet 14 .....

Sheet 15 .....

Sheet 16 .....

Sheet 17 .....

Sheet 18 .....

Sheet 19 .....

Sheet 20 .....

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.

REVISION HISTORY

REV

DESCRIPTION

ZONE

PCBA RENDERS

Replace this with "kiboImageBlender3dAngledTop" to invoke the Blender 3D Rendering output.

Replace this with "kiboImageBlender3dAngledBottom" to invoke the Blender 3D Rendering output.

The Git Edit

Drawing Title

Schematic, Assembly Name

Size

A4

Drawing Number

S99-9000

Revision

NO\_TAG+ (Unreleased)

Used On

File Name

Test\_5.kicad\_sch

Used On

Release State

WORKING

Release Date

2026-03-09

Git Hash

e859a78

Used On

A99-9000

Scale

2:1

KiCad E.D.A. 9.0.7

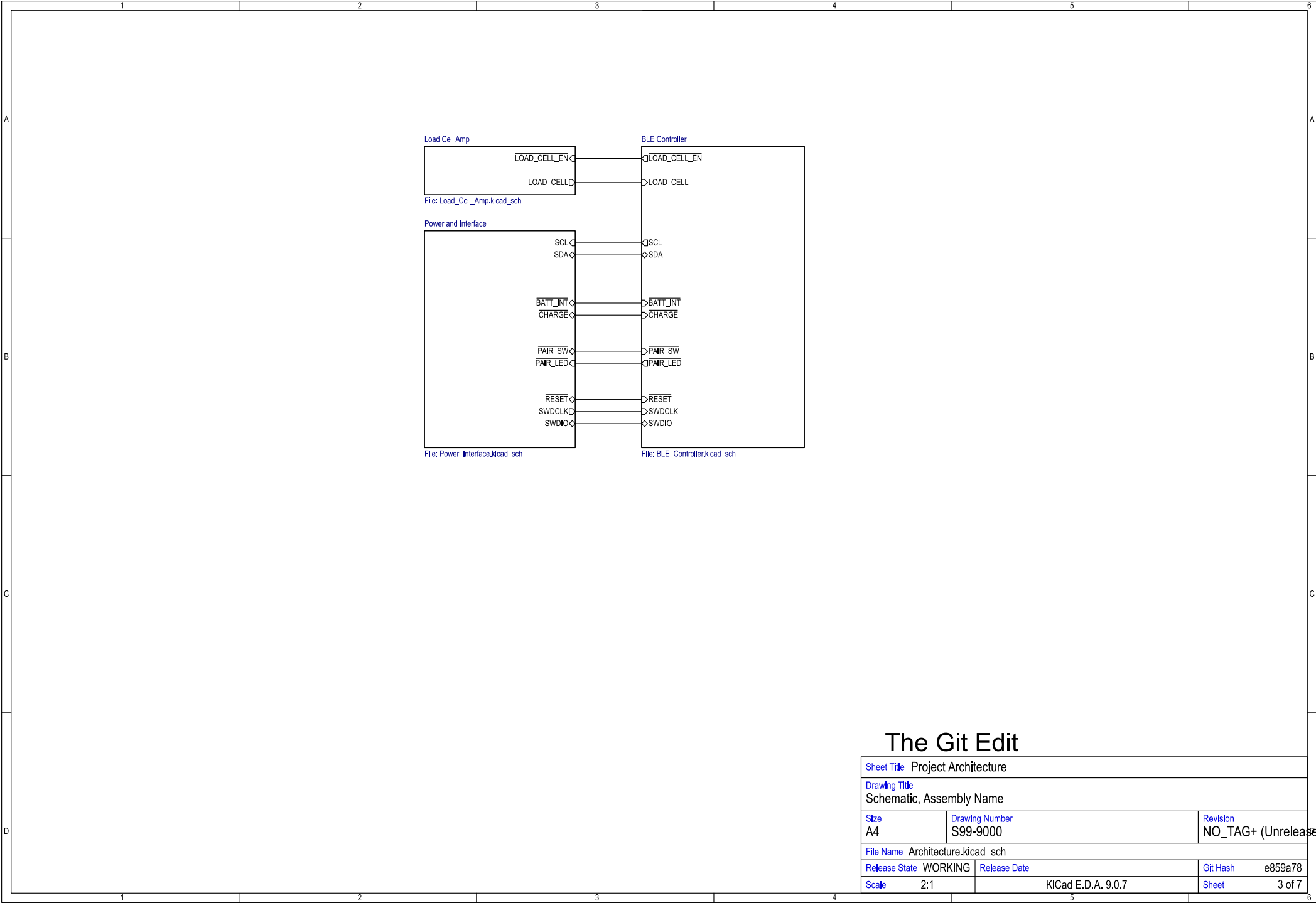
Sheet

1 of 7

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

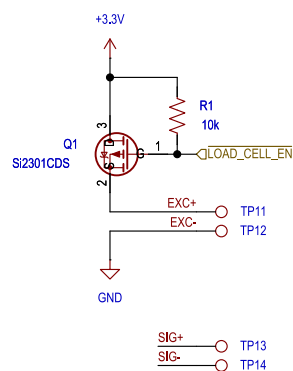
# The Git Edit

Sheet Title Block Diagram		
Drawing Title Schematic, Assembly Name		
Size A4	Drawing Number S99-9000	Revision NO_TAG+ (Unreleased)
File Name Block_Diagram.kicad_sch		
Release State WORKING	Release Date	Git Hash e859a78
Scale 2:1	KiCad E.D.A. 9.0.7	Sheet 2 of 7

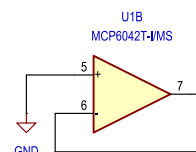
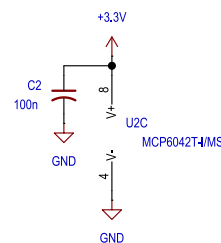
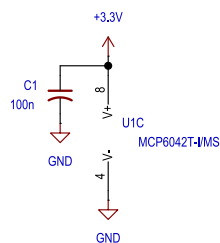
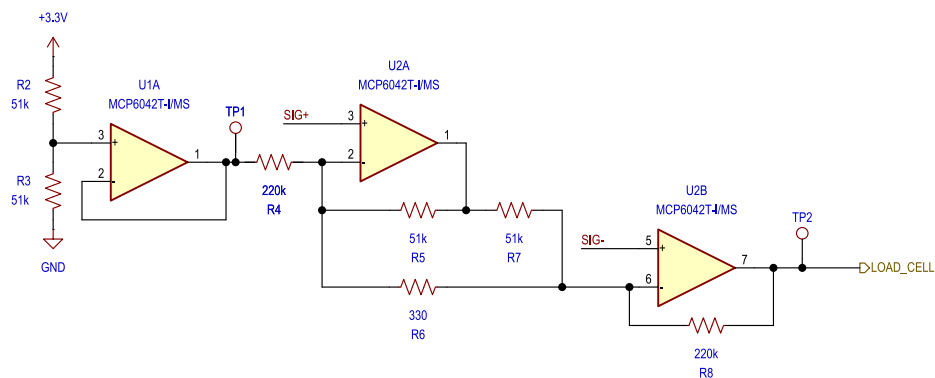


# The Git Edit

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 WORKING	Release Date	Git Hash e859a78
Scale 2:1	KiCad E.D.A. 9.0.7	Sheet 3 of 7



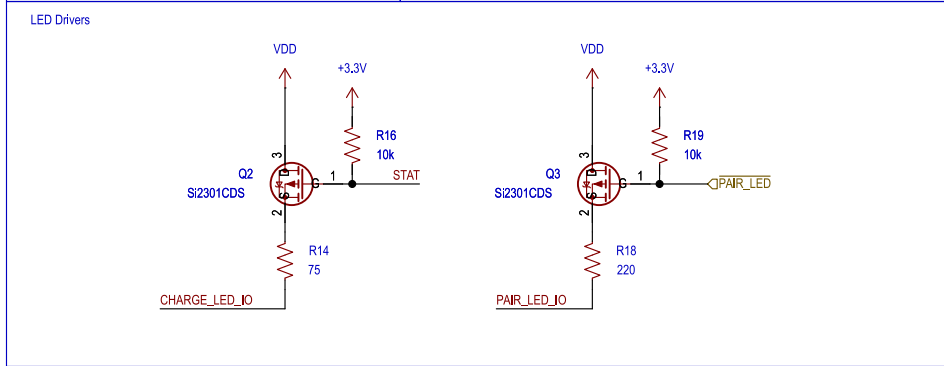
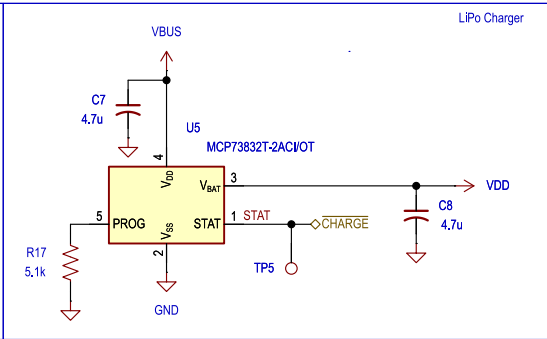
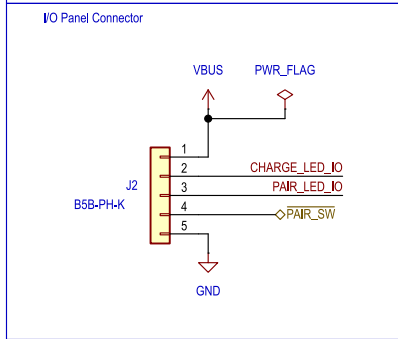
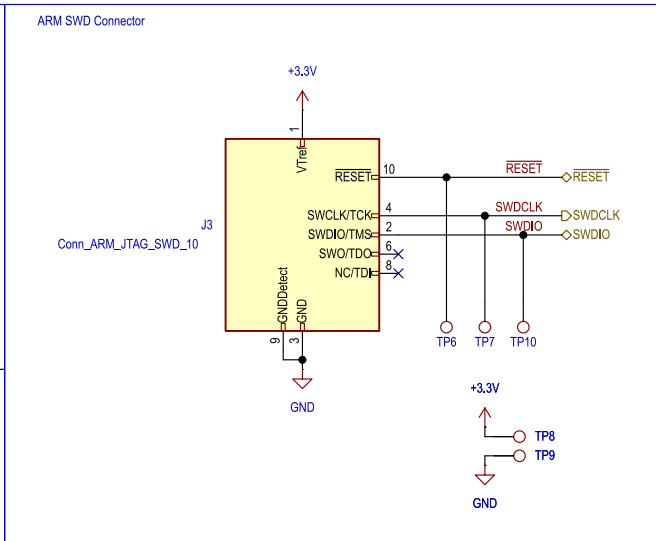
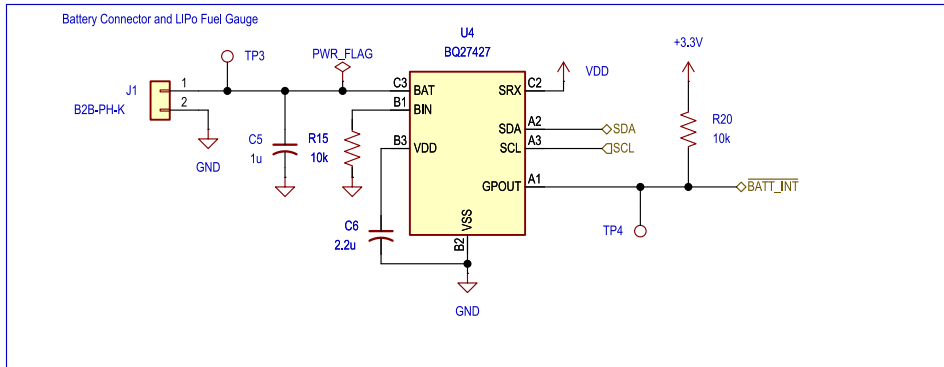
Place four, 1.5mm x 3mm solder pads for Load Cell connections (signals EXC+, EXC-, SIG+, and SIG-). Mark with silkscreen as "E+", "E-", "S+", "S-".



## The Git Edit

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 WORKING	Release Date	Git Hash e859a78
Scale 2:1	KiCad E.D.A. 9.0.7	Sheet 4 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

## The Git Edit

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 WORKING	Release Date	Git Hash e859a78
Scale 2:1	KiCad E.D.A. 9.0.7	
		Sheet 6 of 7

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

# The Git Edit

Sheet Title Parts List		
Drawing Title Schematic, Assembly Name		
Size A4	Drawing Number S99-9000	Revision NO_TAG+ (Unreleased)
File Name Parts_List.kicad_sch		
Release State WORKING	Release Date	Git Hash e859a78
Scale 2:1	KiCad E.D.A. 9.0.7	Sheet 7 of 7