

Sheet: External Input

Estimated BOM cost: \$0.03

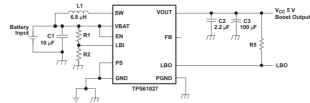


Alt Power Input

File: ExternalInput.sch

Sheet: Boost Converters

Estimated BOM cost: \$0.60



List of Components:
U1 = TI9811027
L1 = 10µH
C1, C2 = 10µF
D1 = 1N4148

Figure 24. Power Supply Solution for Maximum Output Power Operating From a Single Alkaline Cell

Power Conversion

File: Boost.sch

Sheet: USB Input

Estimated BOM cost: \$1.00
(\$0.02 without serial to usb)



USB Input
(Power/serial)

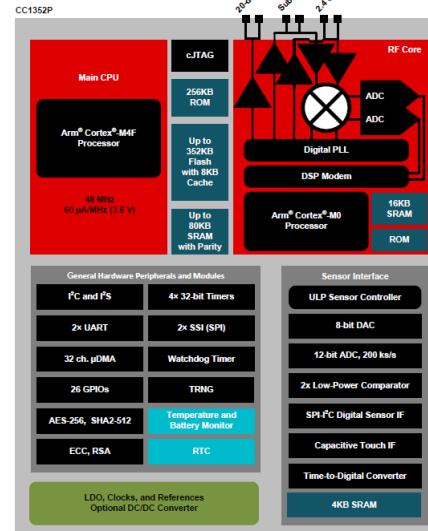
File: USB_Input.sch

Sheet: Sensors

File: Sensors.sch

Sheet: RF

Estimated BOM cost: \$1.27

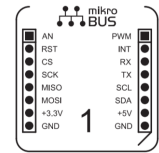
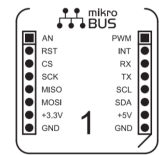


RF and Processor

File: RF.sch

Sheet: MikroBus Click

Estimated BOM cost: \$0.15



MikroBus Headers

File: MikroBusClick.sch

Sheet: Sheet50299501

Estimated BOM cost: \$0.03



Indicators

File: Indicators.sch



Sheet: User Input/Output

Estimated BOM cost: \$0.05



User I/O

File: UserInputOutput.sch

Sheet: Programming

Estimated BOM cost: \$0.00



Programming

File: Programming.sch

Schematic originally designed by Analog Life, LLC

Beagleboard.org

Sheet: //

File: BeagleConnect Freedom.sch

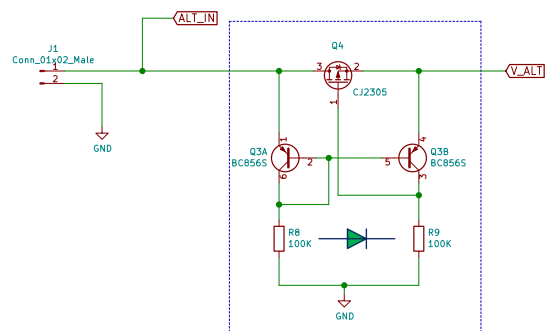
File: BeagleConnect Freedom

Size: C

Kicad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1

Rev: C2

Id: 1/10



Schematic originally designed by Analog Life, LLC

Beagleboard.org

Sheet: /External Input/

Files: ExternalInput.sch

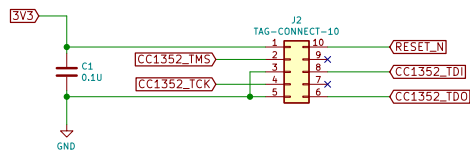
Title: BeagleConnect Freedom

Size: B | Date: 2020-11-03

KiCad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1

Rev: C2

Id: 2/10



Schematic originally designed by Analog Life, LLC

Beagleboard.org

Sheet: /Programming/

Files: Programming.sch

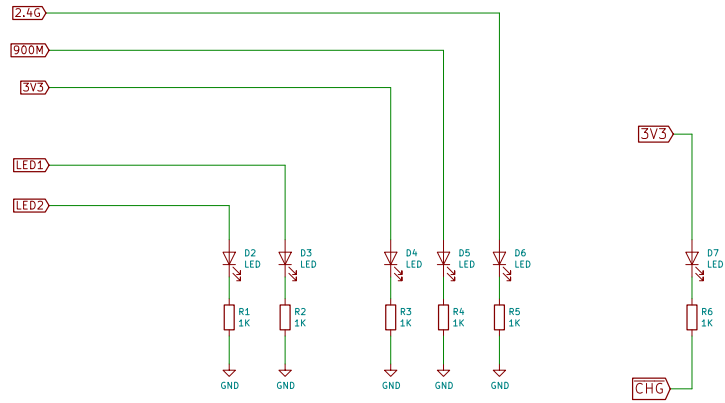
Title: BeagleConnect Freedom

Size: B | Date: 2020-11-03

KiCad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1

Rev: C2

Id: 3/10



Schematic originally designed by Analog Life, LLC

[Beagleboard.org](https://beagleboard.org)

Sheet: /Sheet5D099F01/

Files: indicators.sch

Title: BeagleConnect Freedom

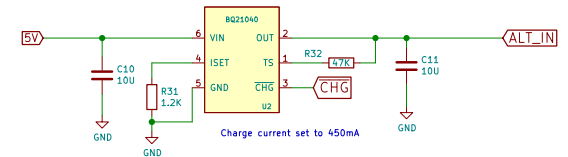
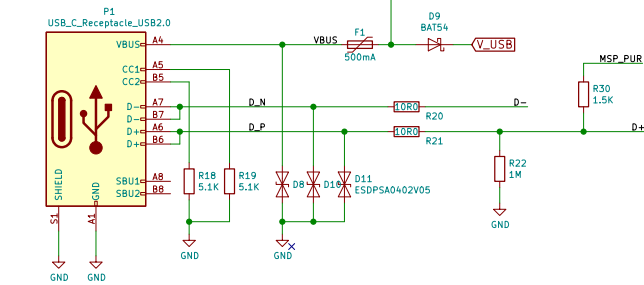
Size: B | Date: 2020-11-03

KiCad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1

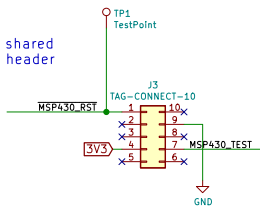
Rev: C2

Id: 4/10

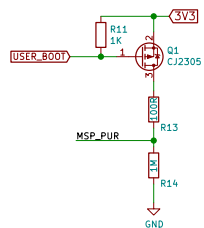
USB C input



Decoupling cap shared with other prog header

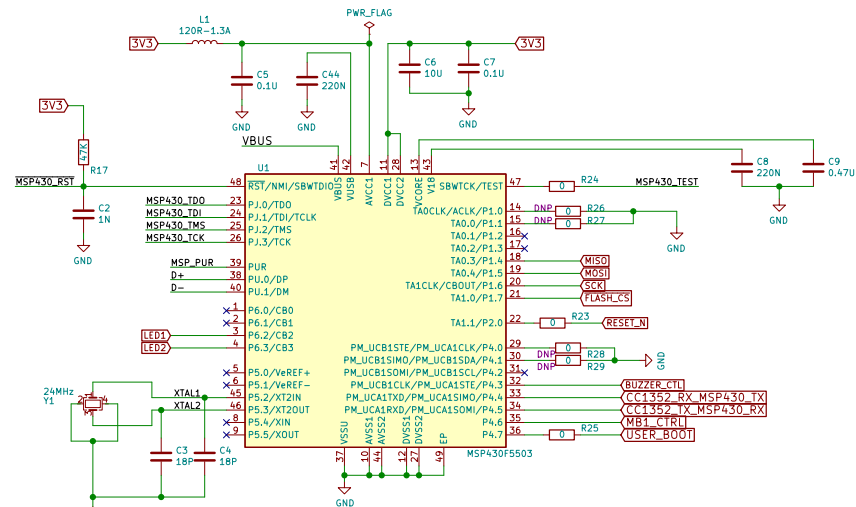


Verify all connections with Erik Larson



When USER_BOOT button is pressed during USB power up, it pulls MSP_PUR high. This puts the MSP430 into programming mode.

The USER_BOOT button is also used as an input to the CC1352R for it's programming, but is active low



May want to shrink center pad or connect manually

On rev B prototype, the soldermask peeled back and many of the pins on the edge of the QFN would short to the center pad after attempted repair to the part (reflow, component replacement)

Schematic originally designed by Analog Life, LLC

Beagleboard.org

Sheet: /USB Input/

Files: USB_Input.sch

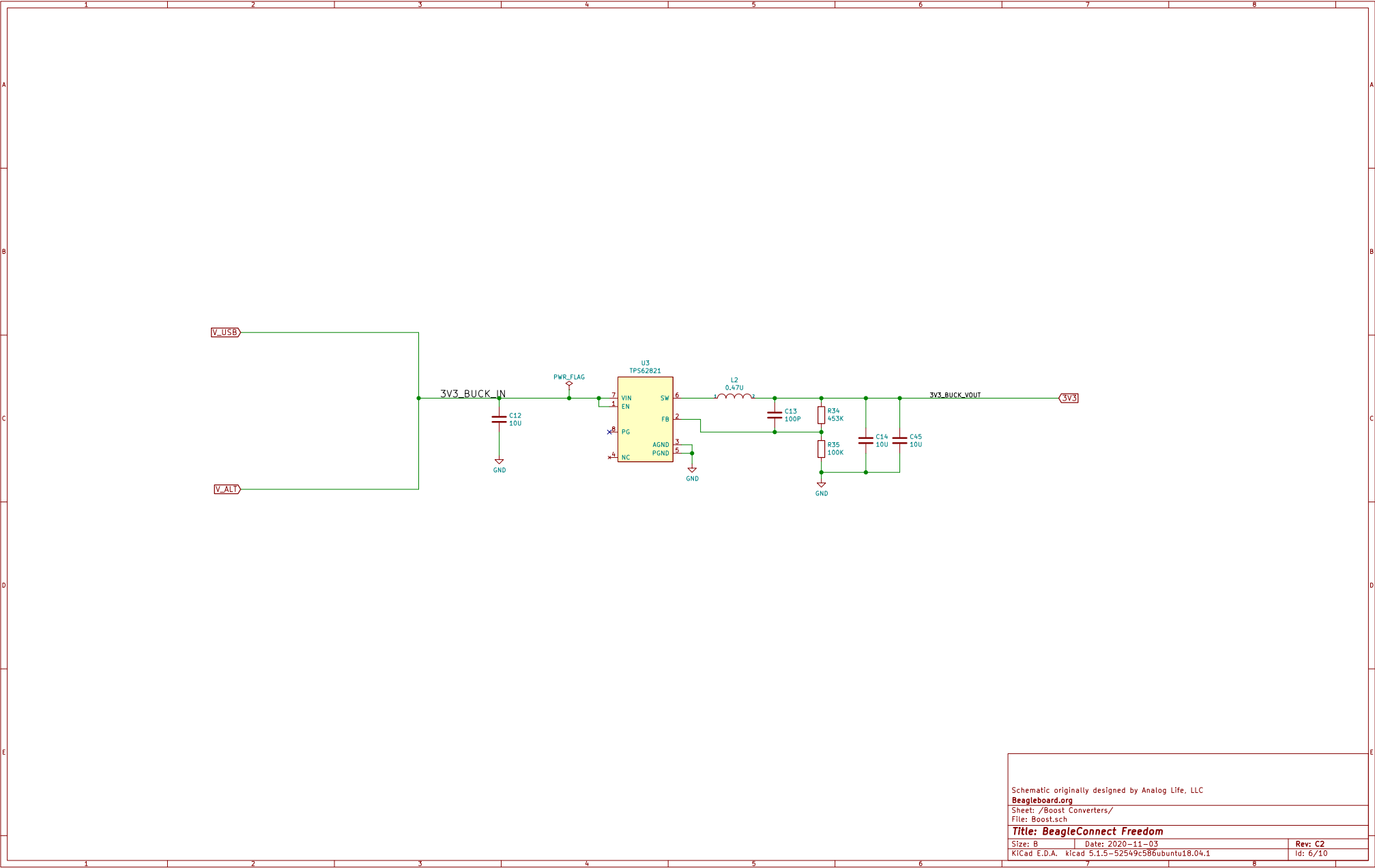
Title: BeagleConnect Freedom

Size: B Date: 2020-11-03

KiCad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1

Rev: C2

Id: 5/10



Schematic originally designed by Analog Life, LLC

Beagleboard.org

Sheet: /Boost Converters/
Files: Boost.sch

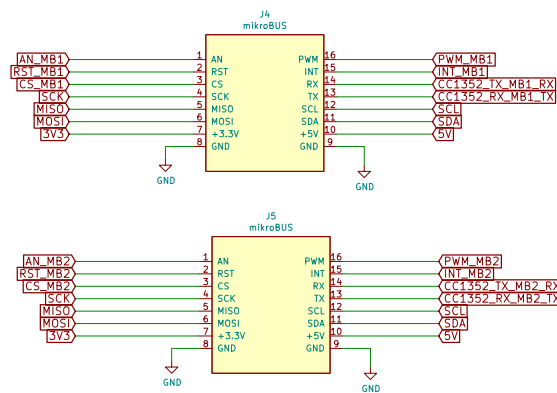
Title: BeagleConnect Freedom

Size: B | Date: 2020-11-03 | Rev: C2

KiCad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1 | Id: 6/10

*Footprint is component footprint plus 1 mm per side.

Part	Manufacturer	Part Number	Quantity	Total Price (\$)	Attribute	Total Footprint (mm²)	Top View	Edit
Cin	TDK	C1608X7S1A475K085AC	1	0.06	Cap = 4.7 µF Total Dissipatd Cap = 1.5 µF VDC = 16 V ESR = 3.73 mΩ Package = 0603	4.68		SELECT ALTERNATE PART
L1	Murata	DPE201610E-W47M-P2	1	0.15	L = 470 nH DCR = 36 mΩ IDC = 3.6 A	7.8		SELECT ALTERNATE PART
U1	Texas Instruments	TPS62821DLDR	1	0.47	Cap = 22 µF	7.5		



Schematic originally designed by Analog Life, LLC

Beagleboard.org

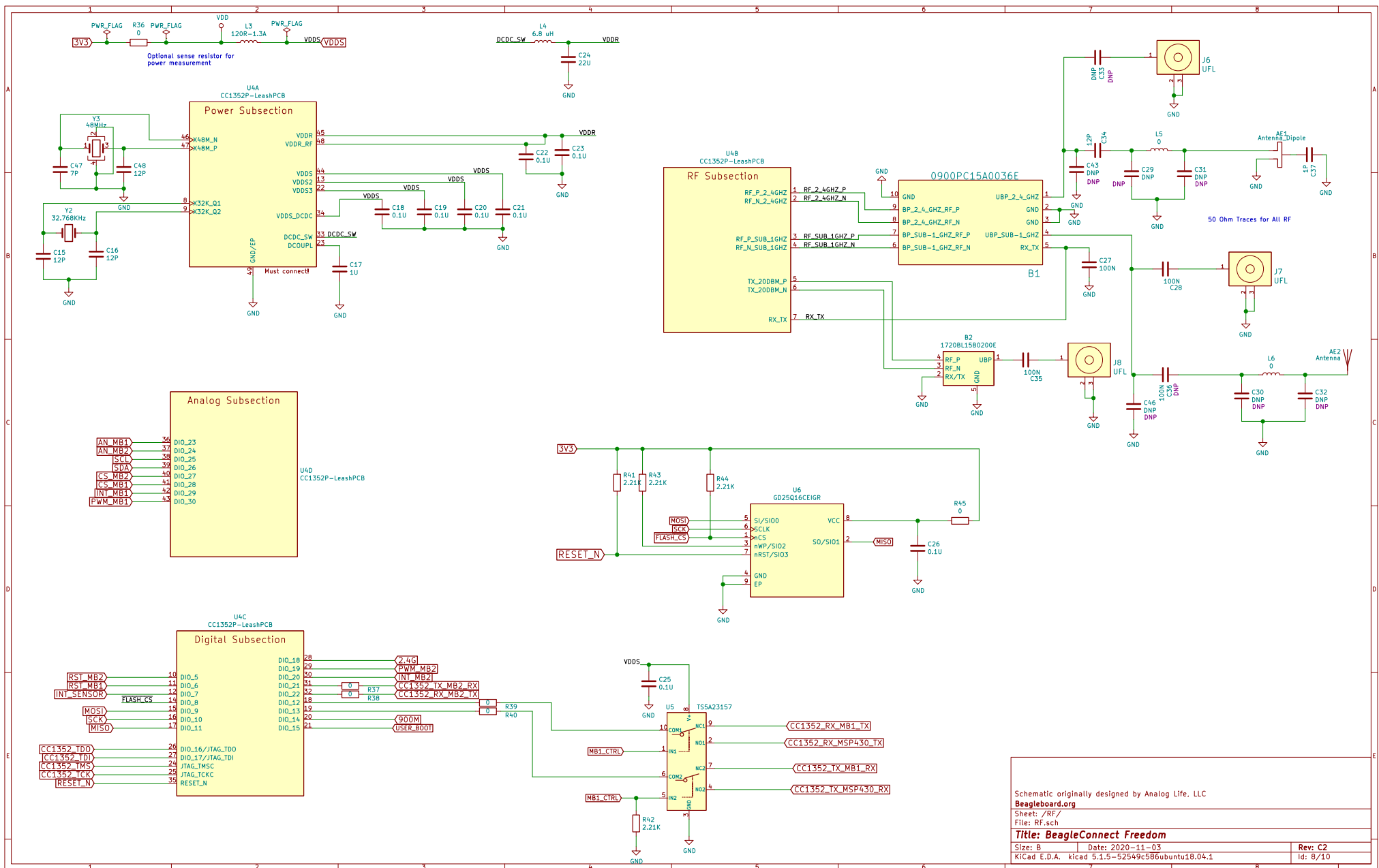
Sheet: /MikroBus Click/

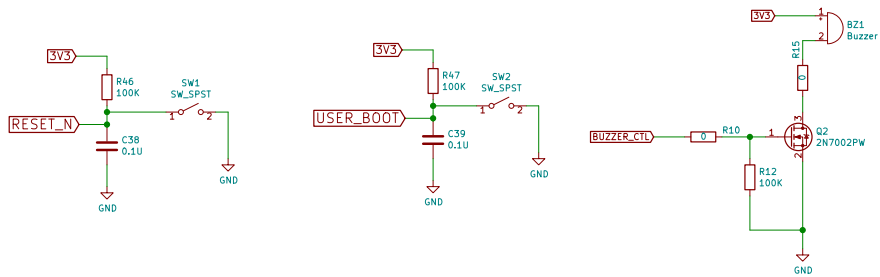
Files: MikroBusClick.sch

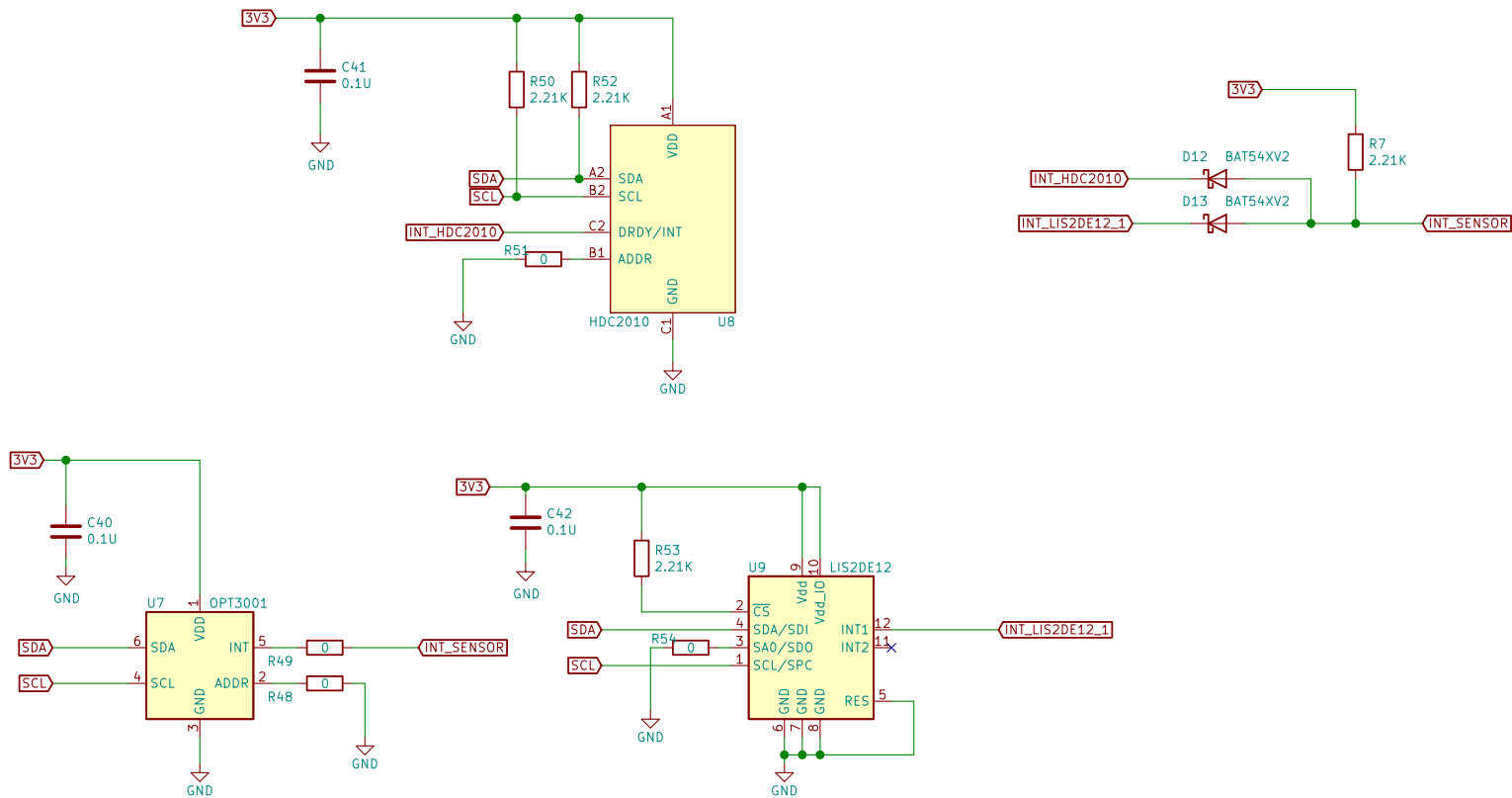
Title: BeagleConnect Freedom

Size: B | Date: 2020-11-03
KiCad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1

Rev: C2
Id: 7/10







Schematic originally designed by Analog Life, LLC

Beagleboard.org

Sheet: /Sensors/

File: Sensors.sch

Title: BeagleConnect Freedom

Size: A4

Date: 2020-11-03

Rev: C2

KiCad E.D.A. kicad 5.1.5-52549c586ubuntu18.04.1

Id: 10/10