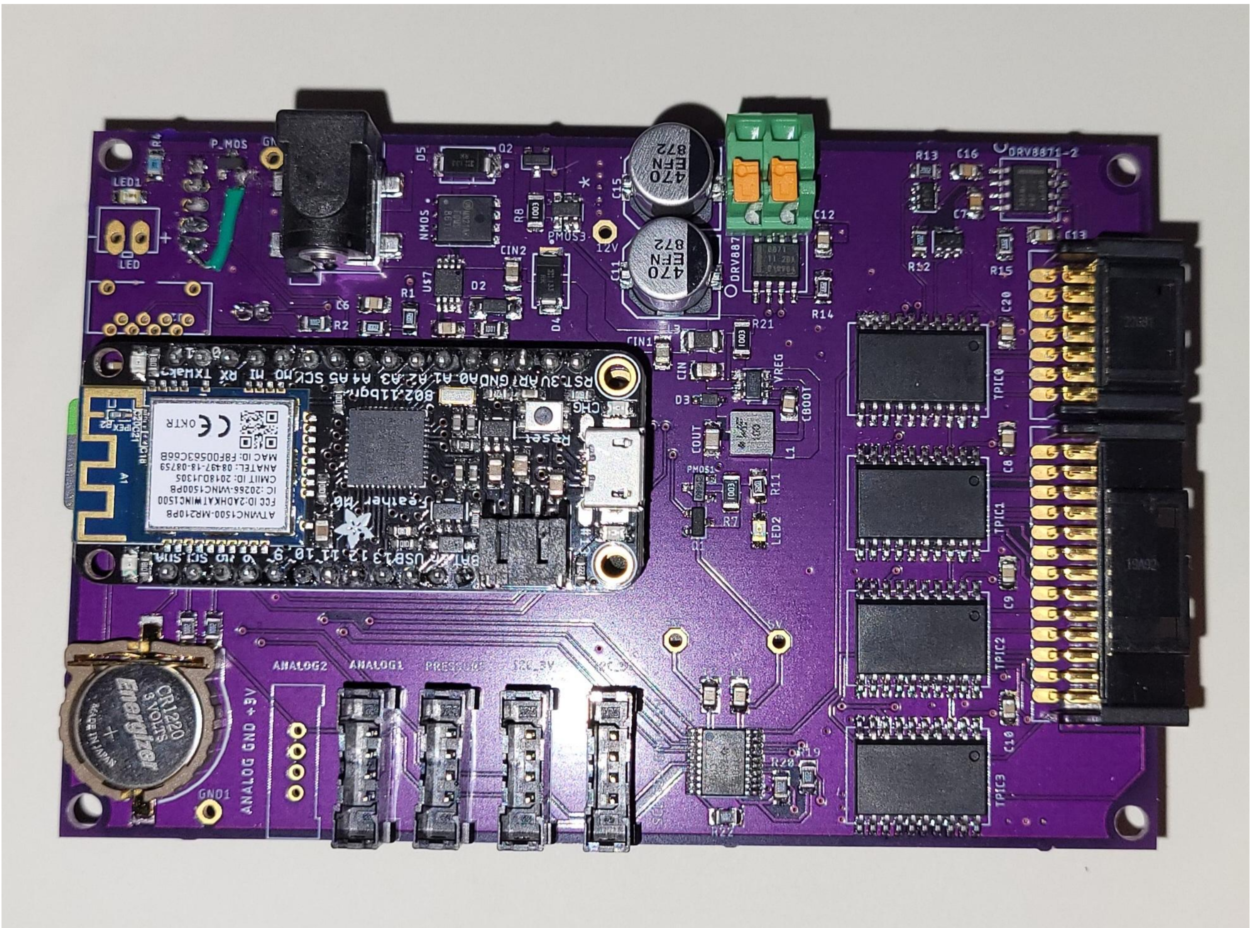
# eDNA Electronics V3

By Kai Quintin Kurisaka Roy



# Table of Contents

# Top Level Block Diagrams

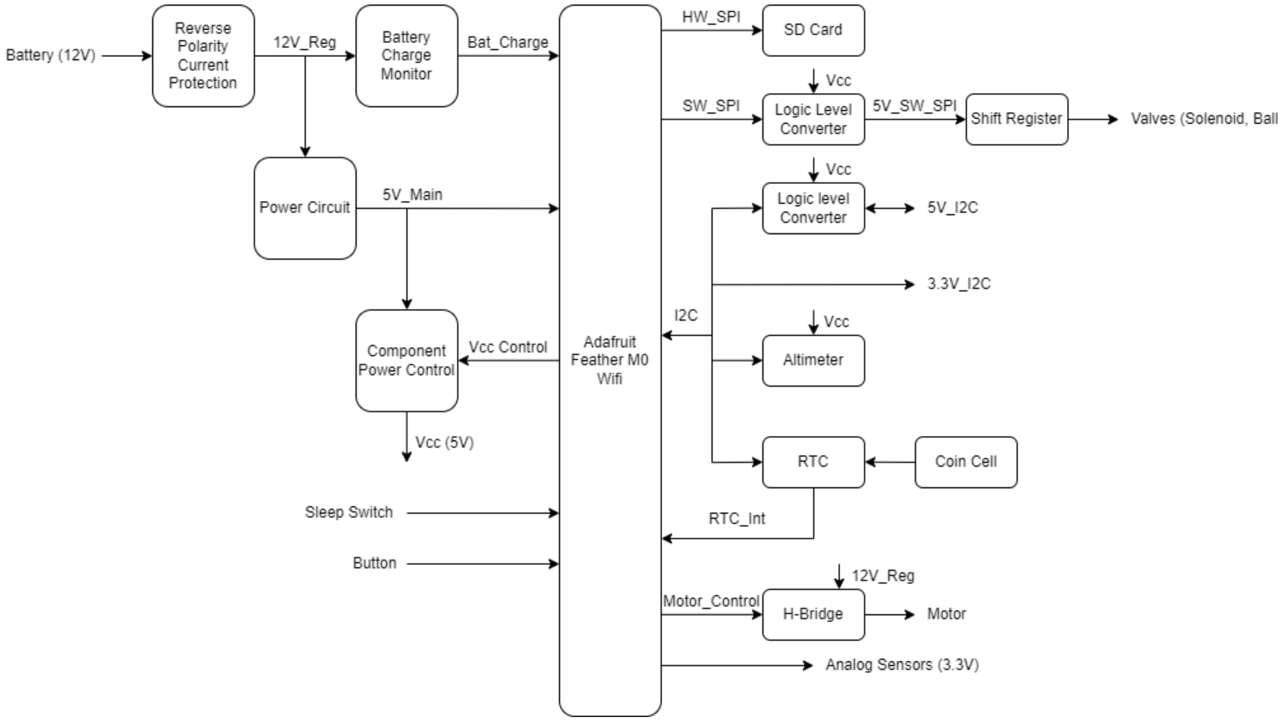



*Figure #: Top-Level Block Diagram*

# Whole Circuit Schematics and PCB

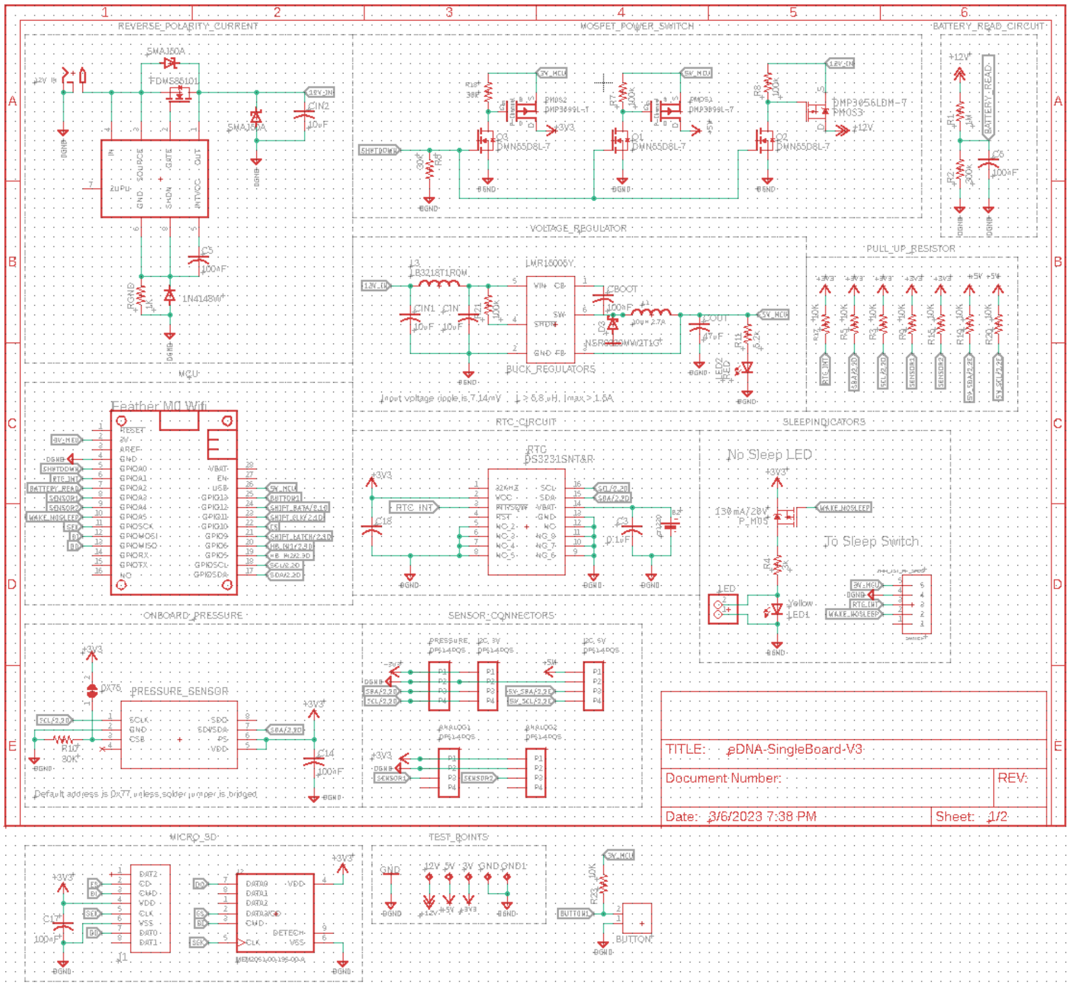



*Figure #: Sheet 1 of the eDNA Board Schematic*

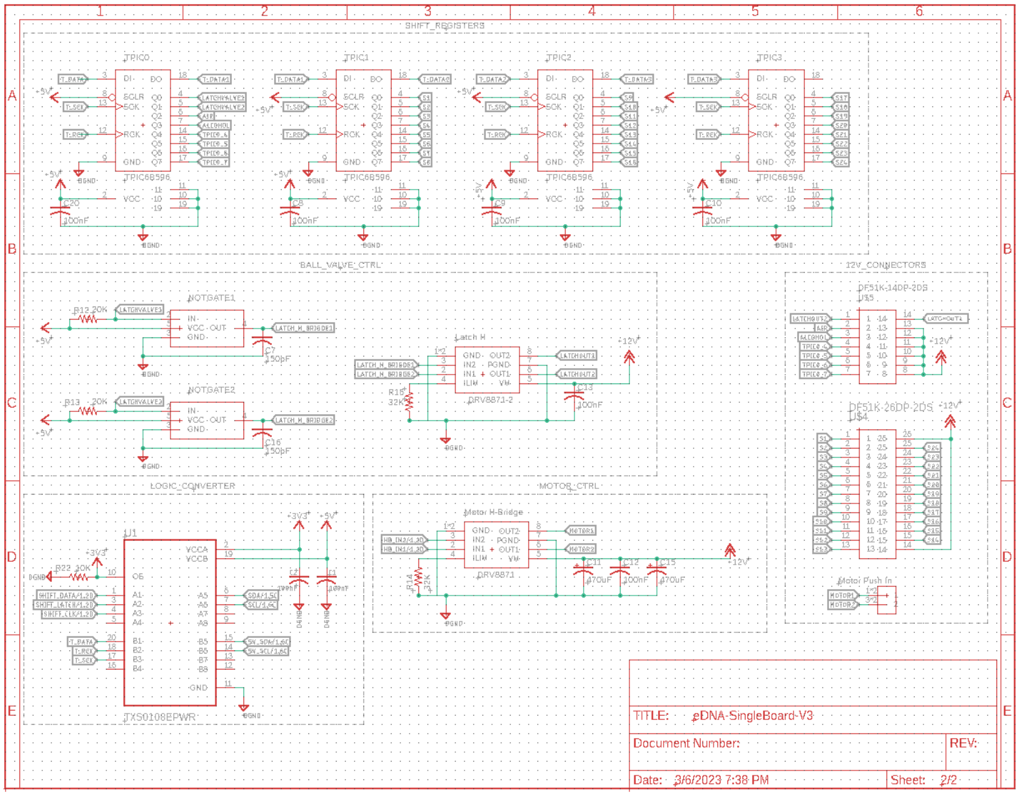



*Figure #: Sheet 2 of the eDNA Board Schematic*

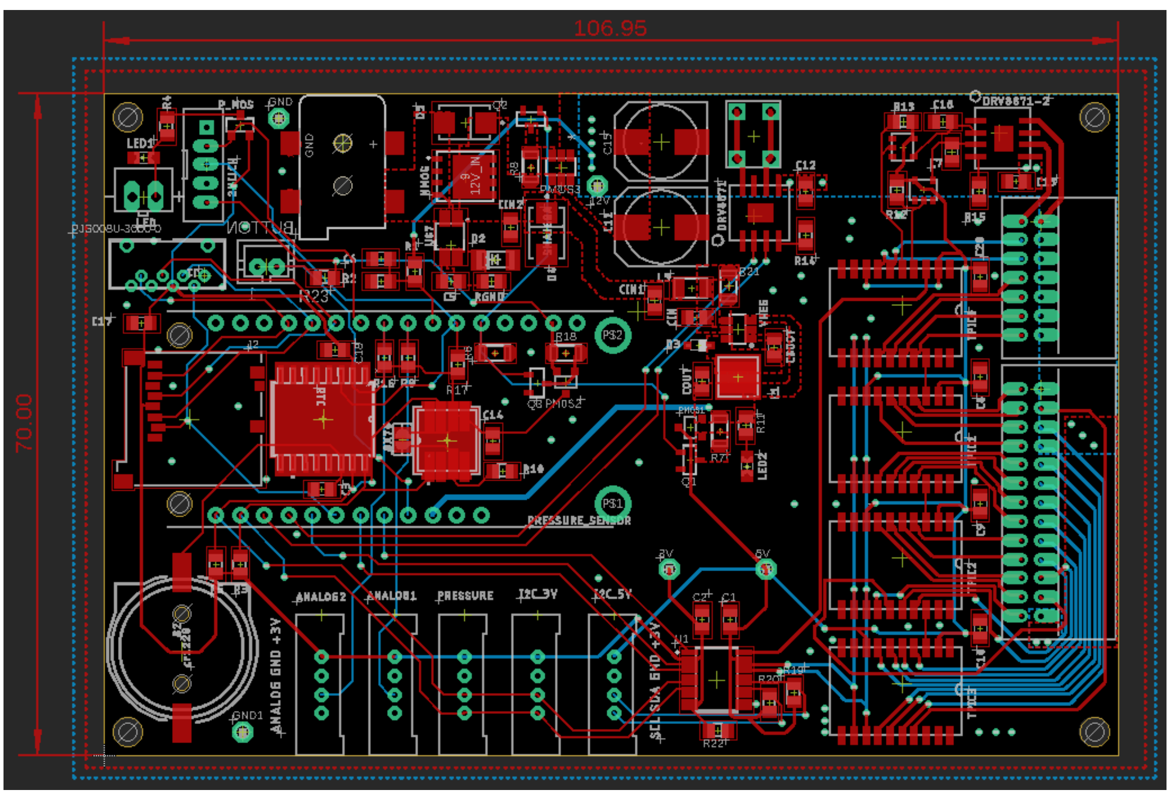



*Figure #: eDNA Board PCB Layout*

# Micocontroller - Adafruit Feather M0 WiFi

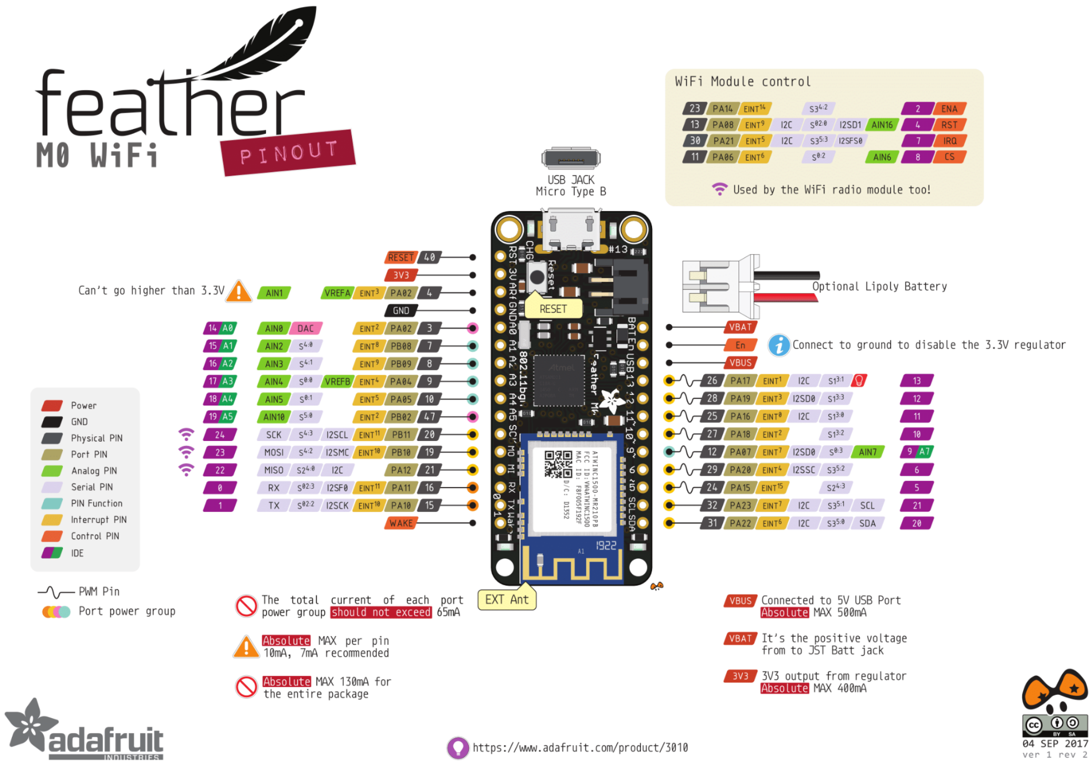
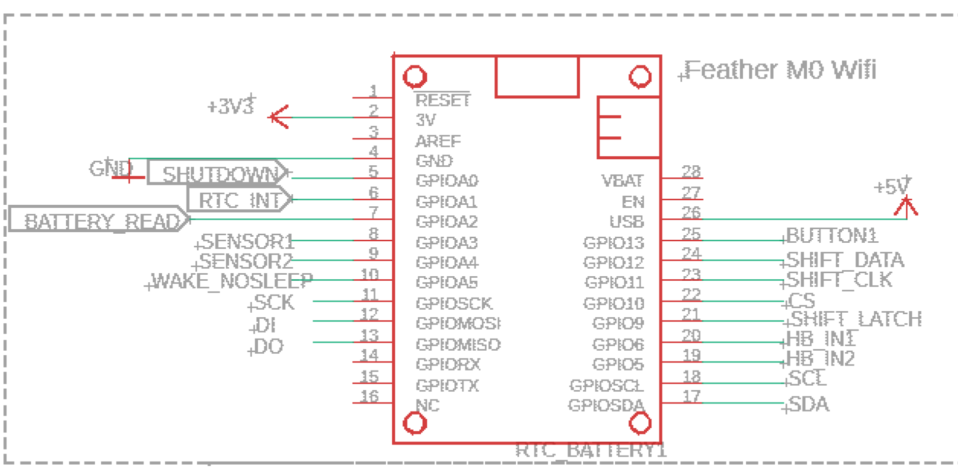



*Figure #: Pinout of the Feather M0 WiFi from Adafruit ([Link](https://www.adafruit.com/product/3010))*

The entire board is built around the Feather M0 Wifi (A M0 with an ATWINC1500 Module). The Feather M0 was chosen as it is the microcontroller of choice for the OPEnS Lab. The use of low profile headers reduces the height of the fully assembled electronics board.

## Pin Usage in our Circuit:



- Pins 12, 11, and 9 are used to control the 4 Shift Registers that control the Valves.
    - Pin 12 is the Data pin,
    - Pin 11 is the Clock(CLK) pin,
    - Pin 9 is the Latch Pin.
- Pins A1 and A5 connect to the sleep Circuit. Pin A1 is connected to the RTC Interrupt (RTC_INT on the schematic).
    - Pin A5 is used by the Feather in order to determine if it is in wake mode or sleep mode.
    - Pins A3 and A4 are used for the 2 Analog Sensors
- Pins SCK, MO, MI, and 10 are used to control the Micro SD Card Reader.
    - Pin 10 is the SD Card Readers Chip Select (CS) Pin.
    - The other pins are the typical SPI Communication Pins (SLCK, MOSI, MISO).
- Pins 5 and 6 are used to control the motor via the H-Bridge (DRV8871).
    - Using two pins allows the feather to run the motor in both directions.
    - The H-Bridge allows us to power the motor with 12V while controlling it with the Feather.
- Pin A2 is the Battery Read Pin
- Pin A0 is the Shutdown Pin
- Pin 13 is marked as a button pin but is not connected to anything on the Current PCB Design.
- The 3V and USB pins are connected to 3.3V and 5V lines respectively.
- Pins SCL and SDA are used to control any $I^2C$ device.
    - Current $I^2C$ devices include the RTC and the In-Line Pressure Sensor.
- Pins RST, ARf, RX, TX, Wake, Bat, En are not used.

# Reverse Current Protection (RCP) Circuit

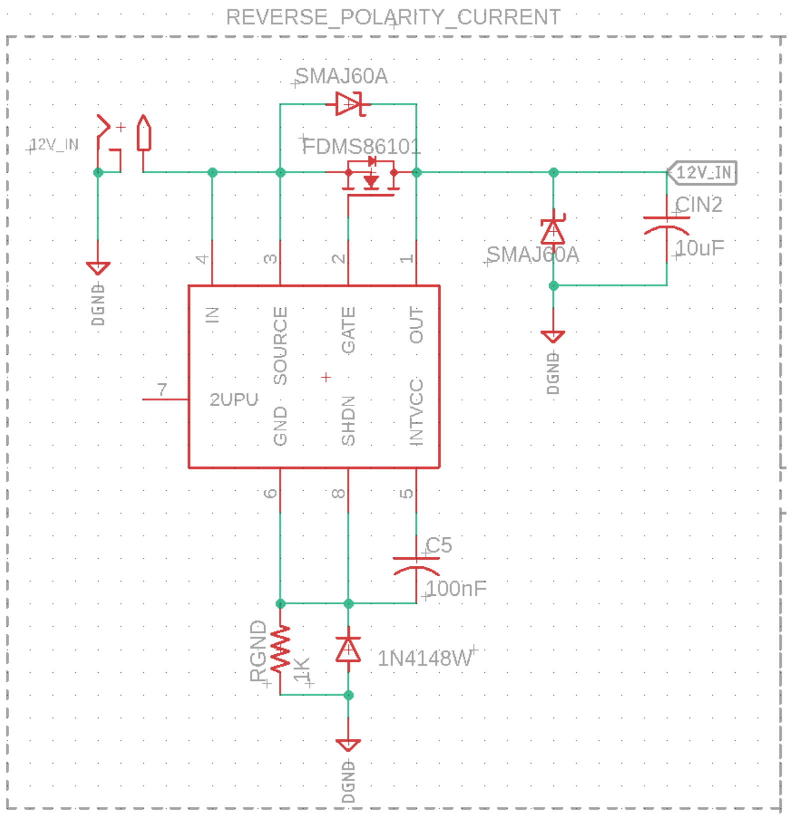



*Figure #: Schematic of the Reverse Current Protection Circuit*

This circuit is implemented to protect the battery in case something causes the current to flow towards the power source. While our battery of choice has its own protection circuits, when those circuits are triggered, the battery stops providing power and needs to be reset with the charger. By having this circuit in place, we can prevent any reverse current and prevent the battery from shutting off unexpectedly.

The integrated circuit used for this circuit is the LTC4372 Ideal Diode Controller. The supporting circuit was taken from page 18 of the LTC4372 Datasheet, specifically the left half of Figure 15, “Micropower 12V Surge Stopper with Ideal Diode.”

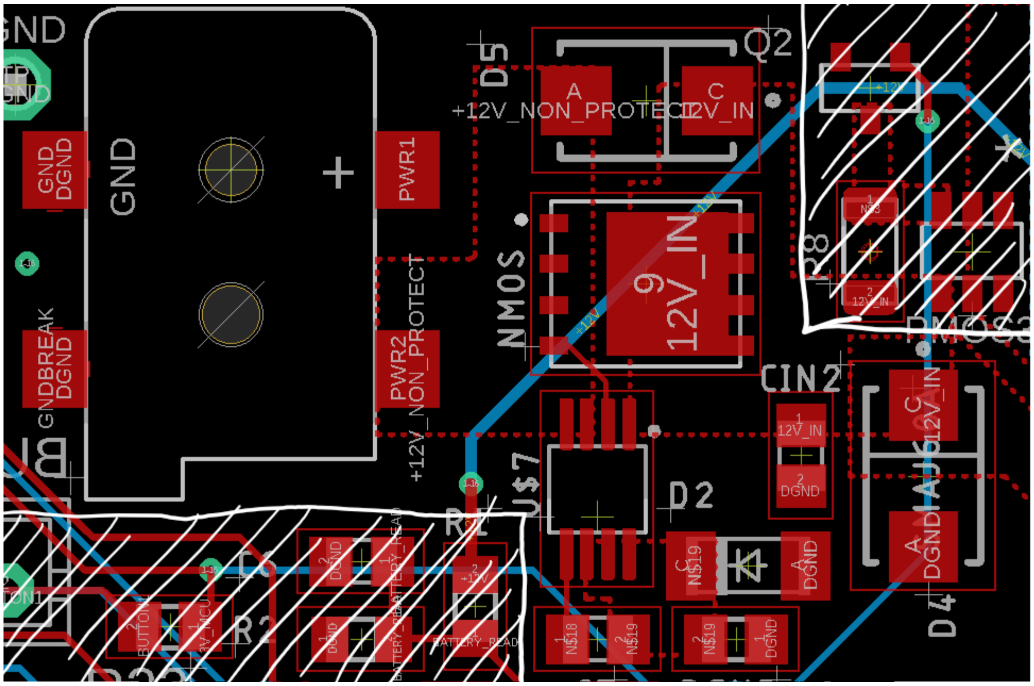



*Figure #: PCB Layout of the Reverse Current Protection Circuit*

The components were layed out so that the small copper plans could be used to connect the different components together. This is allows for both a high amount of current to flow through the plane, as well as assist with heat dissipation of the components. There is a "12V_Non_Protect" plane that connects the 12V input connector with the Protection IC and the supporting components. The output of the IC is the "12V_IN" plane which later connects to both the Mosfet Power Control Circuit and the 5V Buck Converter Circuit.

The 0805 package was used for most of the 2 terminal devices (capacitors, resistors, etc.) to reduce the overall size of the layout while maintaining the ability for the board to be assembled by hand. In many cases, any non-0805 component is chosen because of a lack of a 0805 version of that device. *Components that face higher currents and voltages typically need to be larger to handle the higher load.*

The marked out area in the top right corner of the image contains some of the components for the Mosfet Power Control circuit, specifically the components needed for the 12V/High Power Rail. This circuit will be discussed later in this report.

# Voltage Regulator (VReg) Circuit

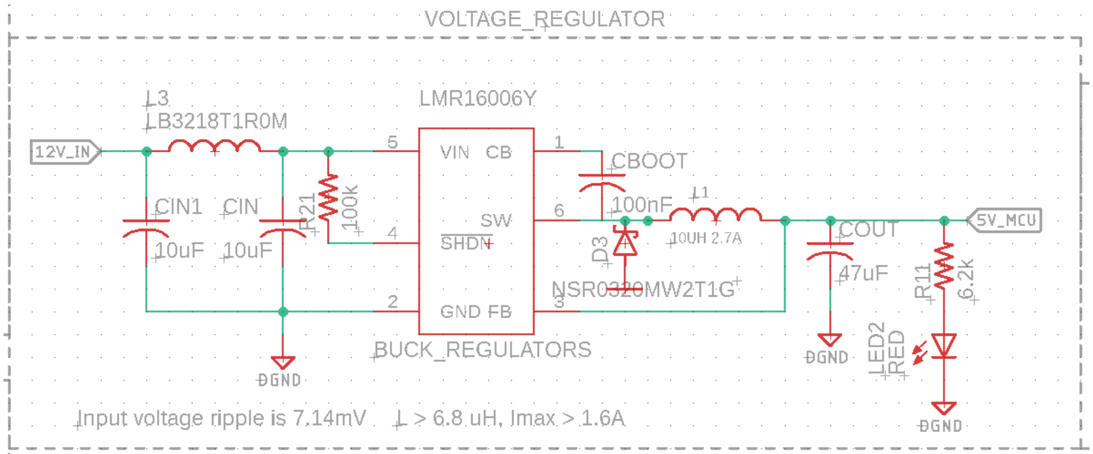



*Figure #: Schematic of the Voltage Regulator Circuit*

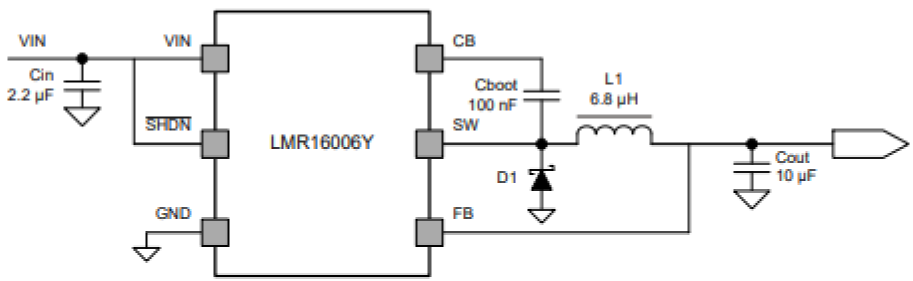


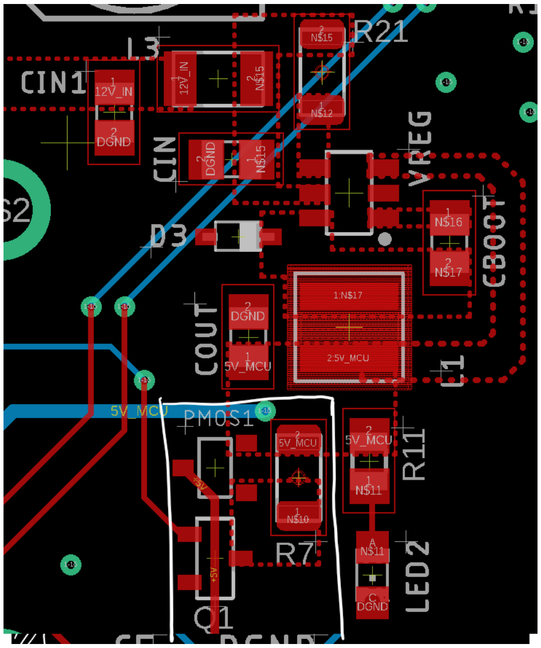

*Figure #: Manufacture recommended schematic for the LMR16006YQ5. Page X of the datasheet.*

Since a 12V battery is used to power all of the electromechanical devices in the eDNA sampler, this input voltage needs to be reduced to 5V and 3.3V for all of the logic components used in the board. Since the Feather M0 has a 5V-to-3.3V regulator built-in to the board, we only need to handle the 12V to 5V conversion.

The LMR16006YQ5 was chosen as it is a Buck Regulator. The buck regulator aspect of the IC is useful for any fluctuations in voltage caused by the electromechanical devices of the eDNA system as the IC uses a feedback pin to maintain the stable 5V output.

An additional LC filter was added to the input of the LMR16006YQ5 to further reduce the noise that comes out of the IC's output. A resistor and LED combo was also added as an indicator of the Voltage Regulator functioning. Other deviations from the recommended layout include:

- The $C_{IN1}$ capacitor changed from 2.2uF to 10uF.
- The $L_1$ inductor changed from 6.8uH to 10uH.
- The $C_{out}$ capacitor changed from 10uF to 47uF.



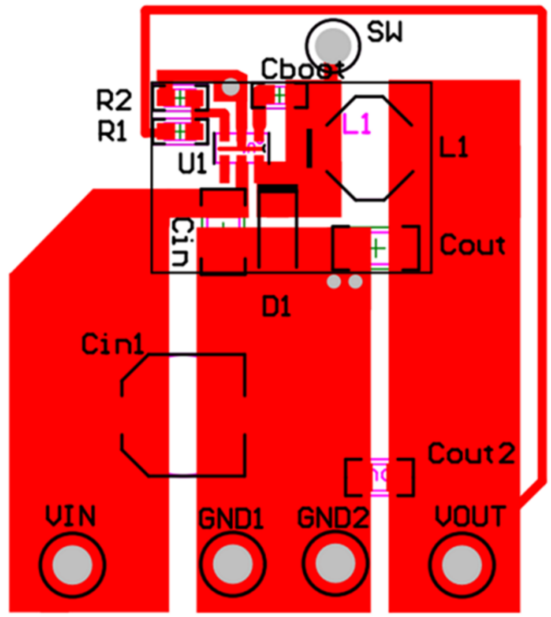




*Figure # (Left): PCB Layout of the Voltage Regulator Circuit*
*Figure # (Right): Manufacture recommended layout for the LMR16006Y IC.*

The PCB Layout was designed to follow a similar layout to the one recommended by the manufacturer with some modifications to accommodate the additional components. The Circuit was also placed as close to the RCP Circuit to reduce the overall size of the "12V_IN" copper plane/trace.

The components outlined in white are for the Mosfet Power Control Circuit which will be covered later in the report. Specifically these are the mosfets used to control the 5V rail for all of the peripheral devices.

# MOSFET Power Control Circuit

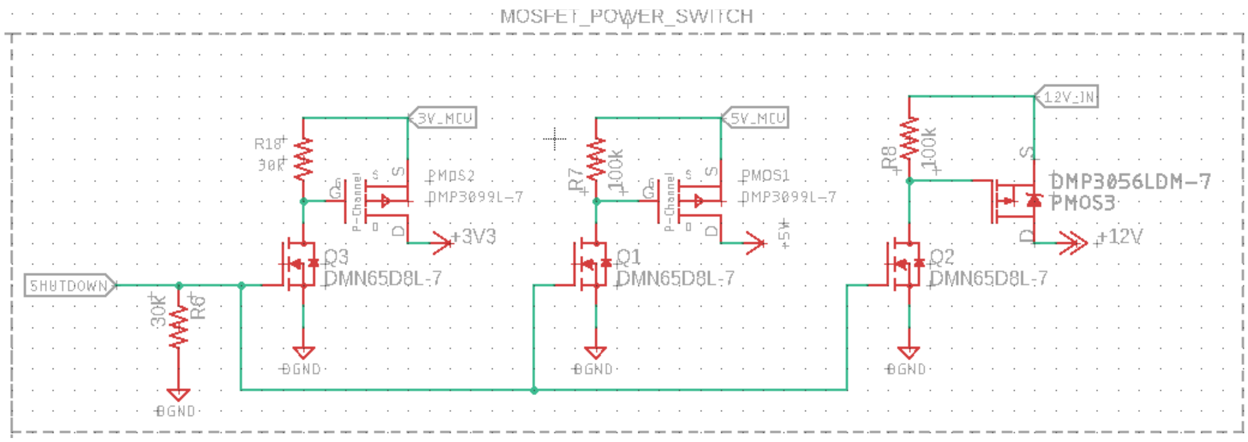
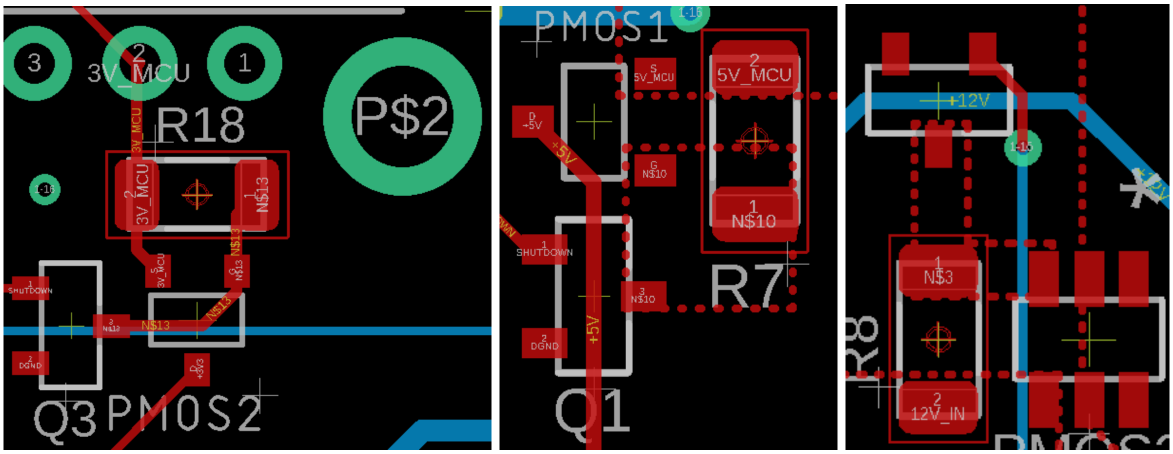



*Figure #: Schematic of the MOSFET Power Control Circuit*

The MOSFET Power Control (MPC) Circuit is designed to replace the old D-Flip-Flop (DFF) based power management/saving circuit. The DFF based circuit was designed to shut down the voltage regulator cutting 5V power. This meant that the only things being powered in the circuit were the DFF circuit, Voltage Regulator (even though it was not providing power) and the RTC circuit, which had its own battery. This meant that the Microcontroller was completely powered off which made the system unable to wake up from an interrupt, at least normally. The RTC interrupt had to be connected to the DFF Circuit to turn the Voltage Regulator back on, thus powering the Microcontroller.

The MPC Circuit is based off of the one used in the Hypnos Board, which is designed to use the Microcontroller's sleep system and simply cutoff power to any peripheral devices; anything that does not used to wake up the feather. The only modification made was to the 3.3V Rail MOSFET circuit. In the Hypnos design, this was controlled separately from the 5V and HighPower MOSFET circuits. Since the eDNA circuit only has one available pin for Power Control, the 5V circuit was replicated for the 3.3V Rail and connected to the same control pin.




*Figures #, #, #: PCB Layouts for the MOSFET Power Control Circuit*

For the layout of the MPC Circuits, the circuit was separated into the 3.3V, 5V, and High Power (12V) sets of components. Each set was placed as close to their respective power supplies as possible. The 12V MPC was placed next to the RCP Circuit, the 5V MPC was placed next to the VReg Circuit, and the 3V MPC was placed right next to the 3V Pin/Head of the Microcontroller.

# Valve Control Circuit:

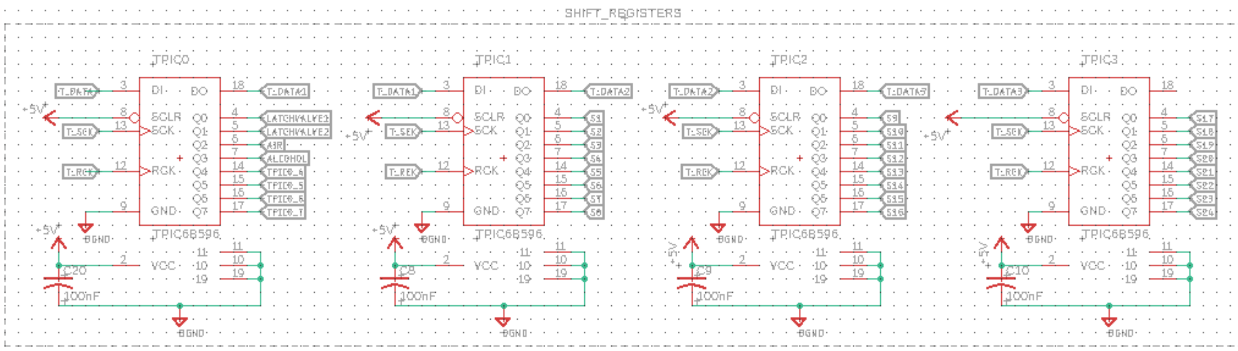



*Figure #: Schematic for the Valve Control/Shift Register Circuit*

The Valve Control Circuitry controls the activation of the 24 sampling valves as well as most of the central components. The motor is the only output device that is not controlled by this circuitry. There are four total Shift Registers (TPIC), one to control the central components and three to control the sampling valves.

The Valve Control Circuitry is controlled by the Feather via three signals: Data, Latch, and Clock. These signals go through the logic level converter to convert the 3.3V signal from the Feather into a 5V signal that is used by the TPICs. The Clock and Latch signals are connected to the TPICs in parallel, while the Data signal is connected to the TPICs in series (i.e TPIC0’s Data_In is connected to the logic level shifter data output, TPIC0’s Data_Out is connected to TPIC1’s Data_In and so on). The TPICs communicate in a way very similar to SPI with a data, clock and cs pin (the latch pin). While it should be possible to connect these TPICs to the hardware SPI pins on the microcontrolller, the previous electrical had difficulty getting both the WiFi and TPICs working simultaneously with the TPICs connected to the Hardware SPI pins.

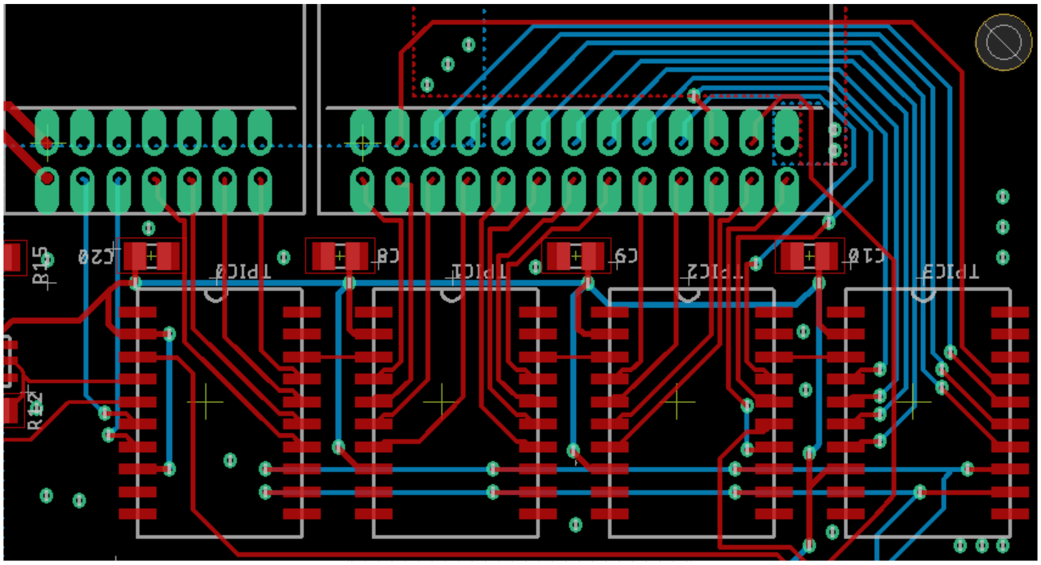


*Figure #: PCB for the Valve Control/Shift Register Circuit*

The TPICs are placed near the relevant connectors to keep the traces compact. To the left of the image is where the additional circuitry for the Ball Valve is located. I attempted to keep the circuit relatively compact but since I was keeping the overall board footprint the same size, there is a lot of extra unused space. The traces between the connector and the TPICs are slightly thicker than the rest of the circuit as these traces will have more current running through them as they serve as the drain for the 12V Valves.

# Motor/Ball Valve Control Circuit:

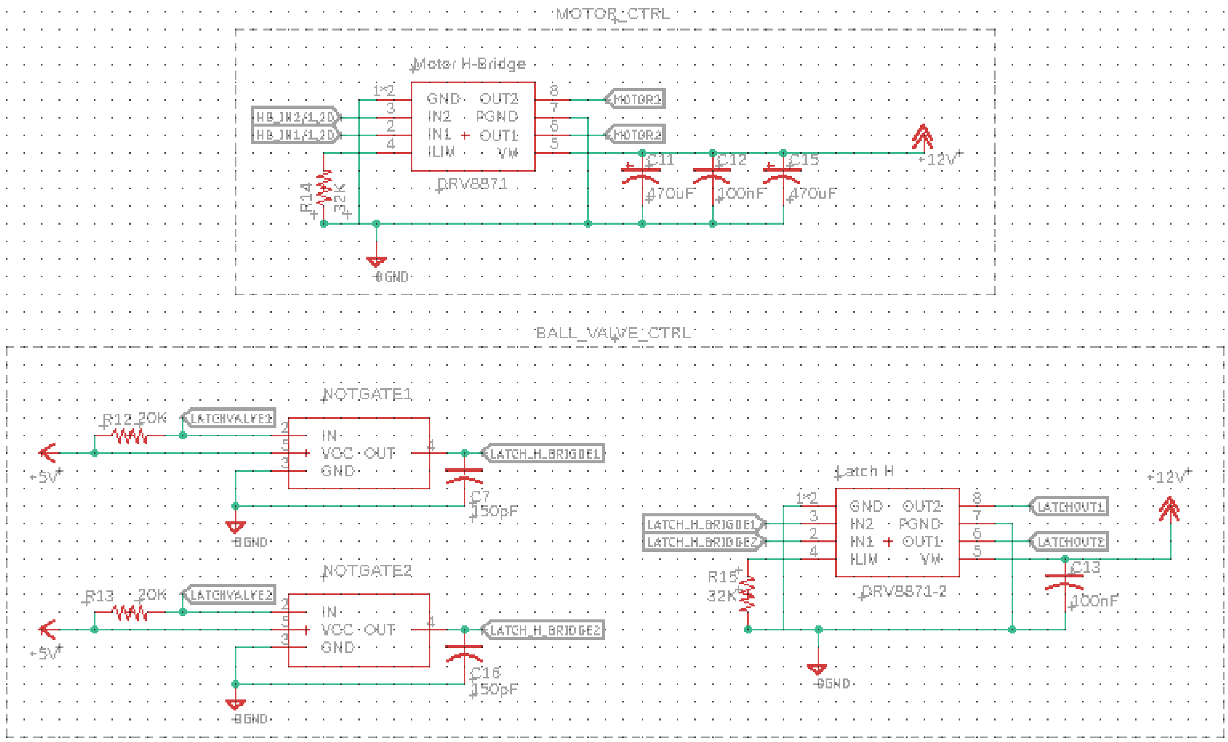



*Figure #: Schematics for the Motor and Ball Valve Circuits*

While most of the TPIC Outputs connect directly to the connectors, the two signals for the Ball Valve, LatchValve1 and LatchValve2, go through additional circuitry (right image) before connecting to the connectors. Like with the motor, the ball valve needs to be controlled by an H-Bridge. Since the TPICs pull the output lines to ground, the signal for the Ball valve needs to be inverted before entering the H-Bridge.

An H-Bridge (DRV8871) is used to control the motor in the eDNA Sampler. An H-bridge is used as it allows for the motor to be run at 4 modes of operation (Forward, Backwards, Coast, Break) at 12V. The circuit used in the eDNA Circuit is the same circuit that can be found on page 7 of the DRV8871 Datasheet. The two 470uF Capacitors function as the bulk capacitor recommended in the data sheet. The 32k ohm value for the resistor was chosen to reduce the motor current to 2A. The calculation for this resistor value can be found on page 9 of the DRV8871 Datasheet.

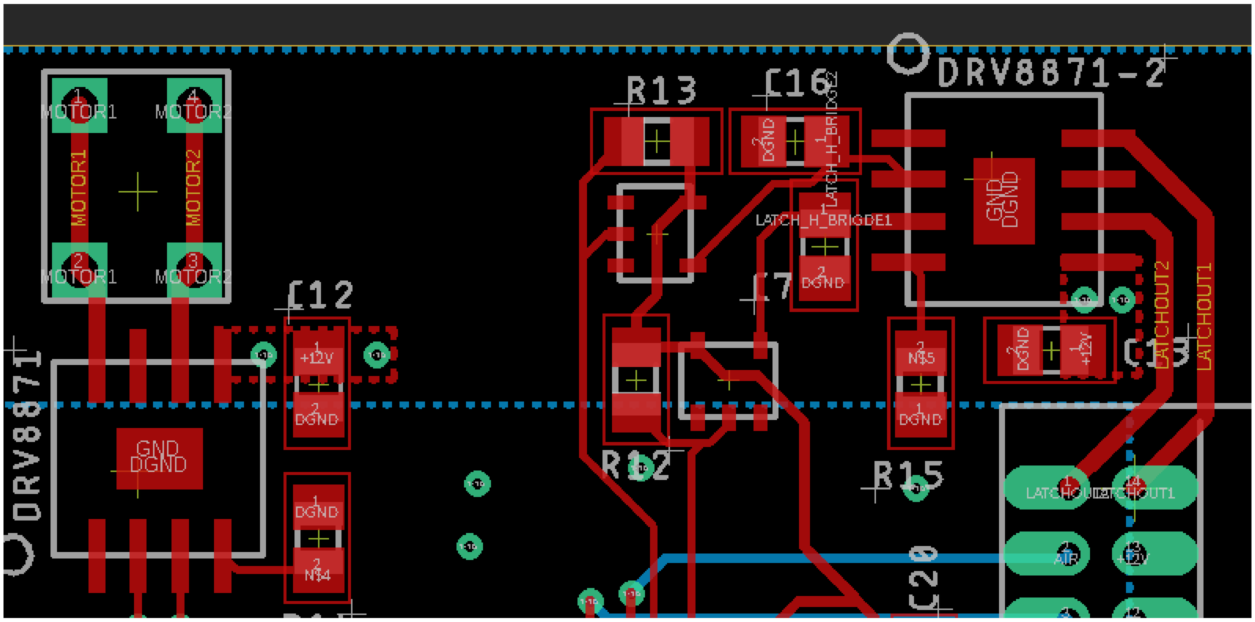
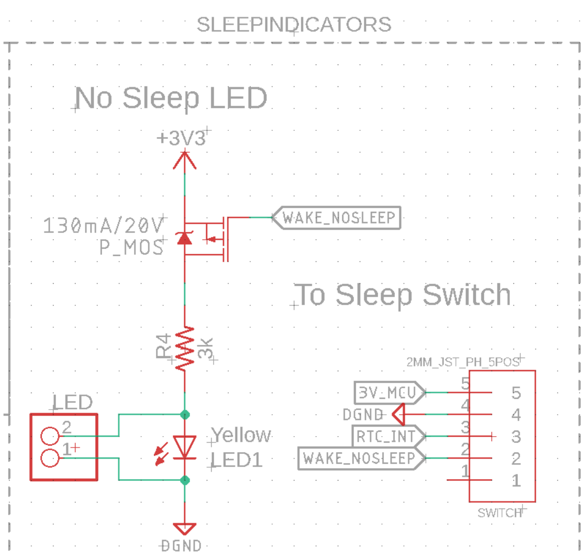



*Figure #: PCB Layout for the motor and Ball Valve Circuits*

I placed the H-bridges and additional circuitry as close to the relevant connectors as possible to reduce the length of the thinker traces for the Higher Power devices.

# Other/Miscellaneous Circuits

## Sleep Mode Control Circuit:



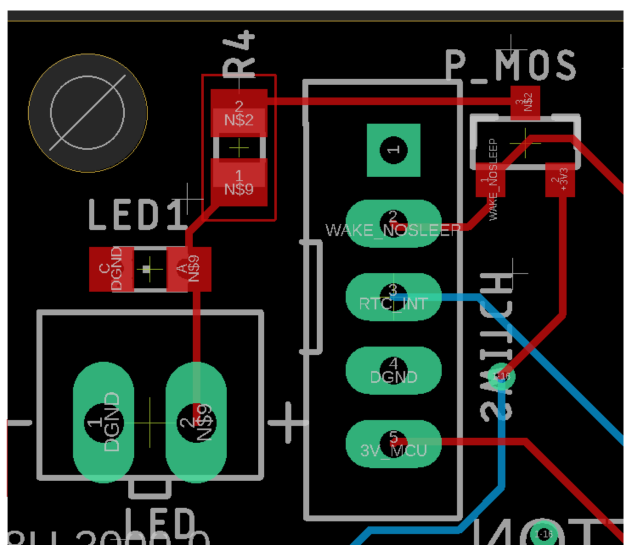
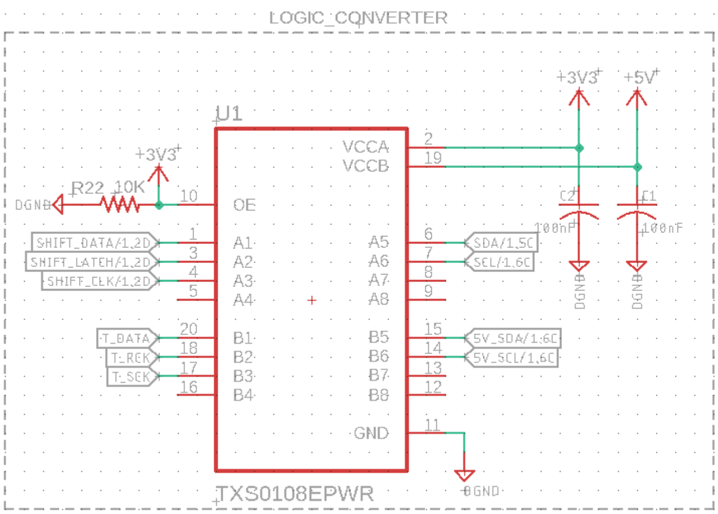




*Figure # (Left): Schematic for the Sleep Circuit*
*Figure # (Right): PCB Layout for the Sleep Circuit*

The image on the right shows two different circuits: The Switch Connector on the left and the No Sleep Indicator Circuit on the right. The Switch Connector connects to a DPDT toggle switch which is used to enable or disable sleep functionality.

The No Sleep Indicator Circuit is used to indicate whether the Sampler is in No Sleep Mode or not. When in No Sleep Mode, the Yellow LED will Light Up. There is also a connection point where an external LED can be attached if it is desired to have the sleep indicator LED outside of the electronics box.

## Logic Converter:



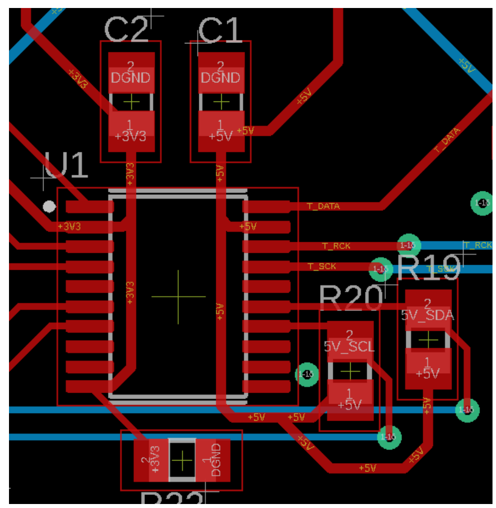




*Figure # (Left): Schematic for the Logic Converter Circuit*
*Figure # (Right): PCB Layout for the Logic Converter Circuit*

The logic level converter circuit allows for the conversion of 3.3V digital signals to 5V signals. This is important since the microcontroller functions on 3.3V logic. The logic converter IC being used is the TXS0108E, an 8-channel Logic Level Converter IC. Three of these Channels are used for the Shift Register (TPIC) circuit, while 2 others are used to allow for 5V I2C capability. Like with many other circuits on this board, this circuit simply follows the recommended layout outlined in the IC's datasheet.

## SD Card Circuit:

The SD Card is set up using the SPI Communication protocol. The 0.1uF capacitor acts as a decoupling capacitor. Outside of that the circuit set up is straightforward.

## Connectors:

Three connectors are used for any external I2C device, one of which is used by the in-line pressure sensor. This allows for 2 more I2C sensors to be added to the machine before the logic board will have to be redesigned. For the I2C protocol to function correctly, the SDA and SCL lines need to be pulled up to 5Vs. Two resistors are used for this purpose.

There are two connectors/pins allocated for analog sensors, one of which is already in use for the flow sensor. The other connector is used in the SampleNow implementation for the SampleNow Button. If more than one analog sensor needs to be added then the board will need to be redesigned. It is recommended that you try to find an I2C version for any sensor that needs to be added.

# Notes on Possible Improvements:

- Changing to a Hardware SPI instead of Software SPI for Shift Registers
  - This would allow for:
    - The use of the SPI Library in the Code
    - The freeing of 2 digital pins on the Feather.
  - Idea by Kai
- Splitting the 26-Pin Connector into two 14-Pins Connectors
  - This would allow each side (set of 12 valves) to have its own connector
  - Idea by Kai.
- Add a reset button connected to the reset pin
  - Idea by Kai
- Changing the SD Card Holder
  - Current one is a Push In, Push Out style Holder which is hard to use with the current Electronics box
    - The Space between where the SD card comes out and the wall of the box is tight
  - Change with a top latching holder or a vertical holder
    - Top Latching is not recommended as the seem to be less reliable
  - Suggested by Kai
- Embedding the micro controller into the PCB
  - More fine control of the MCU and Pins
  - No need for headers, reduces overall Board Height
  - **Pro/Con:** Depending on whether the board is designed for Pre-Assembly, could increase the size of the board as components can not be “placed under the Feather”.
  - Suggested by Kai
- Order the boards pre-assembled/machine assembled
  - Currently employees assemble the boards by hand, which limits the minimum size of components as well as keeping the board single sided
  - Buying Pre-Assembled Boards could allow the following
    - Using smaller SMD components, saving space
    - Allow for components on each side, further saving space
    - Higher consistency, i.e little human error in assembly
    - **Con:** Increased cost (double side will increase components even more
- Switch to a SPST Switch with a 3-pin connector on the board
  - RTC_INT pin is no longer needed on the switch as the sleep control circuit is different

- Switch to an external uSD Card Slot (Waterproof)
    - Allows for easy user access to the uSD Card
    - Would need a connector on the board to connect.
    - Suggested by Kai
- Change the Valve Connectors to better fit the PCB Layout OR Change the layout to better fit the connectors.
    - As it currently stands, the first Shift Register (TPIC0) is the furthest TPIC away from the logic converter, this is mostly due to the location of the power circuits which need to sit on the top right of the board (at least in this design). By switching the central valve connector with the sample valve connector(s), you switch the TPIC placement so that TPIC0 sits closer to the logic converter
    - **NOTE:** This poin may be moot as new Shift Registers will need to be found as these ones are either discontinued/non-stocked or very hard to come by.
        - Would also recommend finding smaller footprint device(s) if the board is switched to a Pre-Assembly Design.
- Move any indicator LEDs to be power off of the Peripheral Rail
    - I (Kai) forgot to switch the VReg LED to be power off of the Post-MPC Power Rails.