DESIGN OF AN AUTONOMOUS GREENHOUSE

Group 6

March 21, 2021

Contents

List of Figures

List of Tables

1 Control Interface

1.1 Control PCB

The entire greenhouse system is controlled by a custom designed PCB based on the ATmega32u4 microcontoller from Microchip.

1.1.1 Controller Design

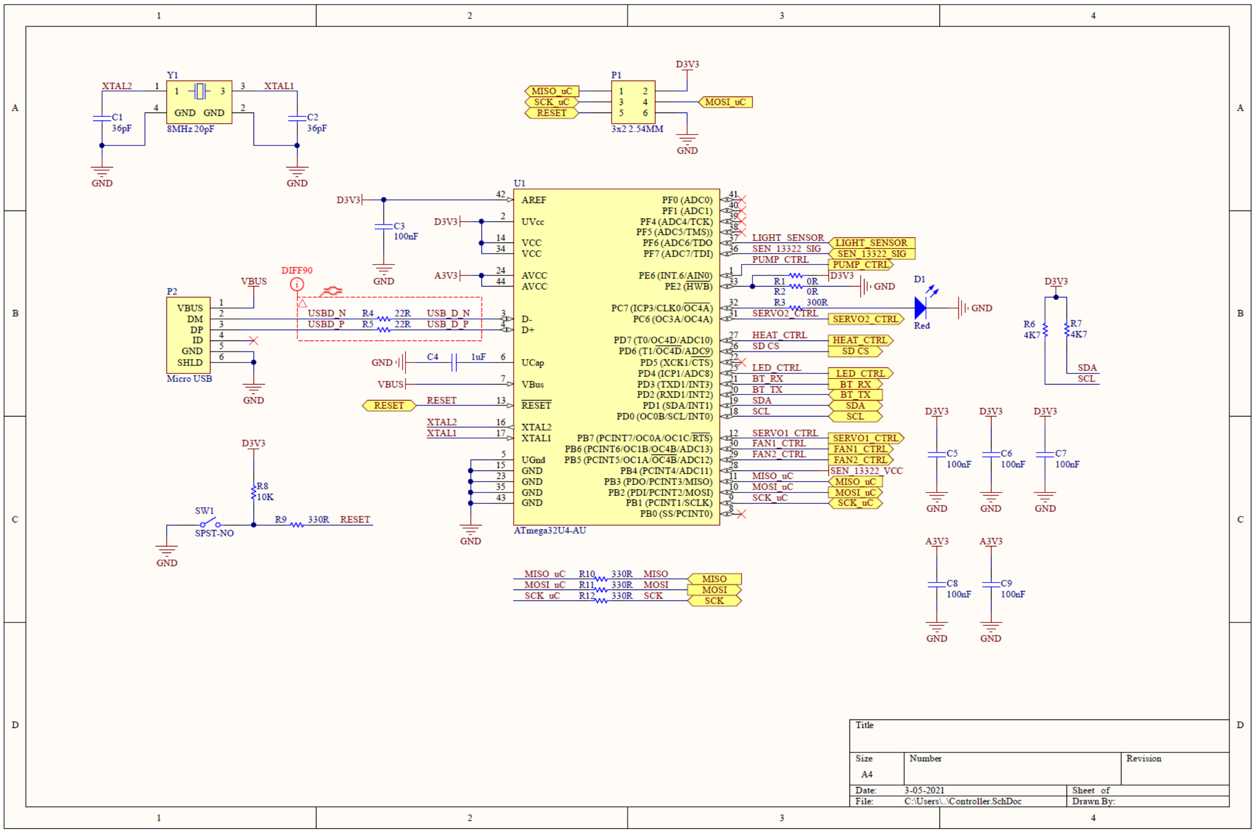

Figure 1: Controller

U1 represents the ATmega32u4 microcontroller. The ATmega32u4 is an 8-bit AVR RISC based microcontroller with 32KB of flash memory, 2.5KB of SRAM and 1KB EEPROM. This controller was selected due to its ease of programming (Arduino IDE support), package (QFP-44) and native USB support.

P1 represents the ISP programmer header which is used to burn the bootloader and set the fuse bits on the microcontroller to allow it to be programmed directly through USB and to set the clock frequency. This interface utilizes a modified SPI bus (RESET replaces chip select).

Y1 represents the system clock crystal. Due to selecting to power the microcontroller off 3.3V instead of 5V, the microcontroller must be run at 8MHz instead of 16MHz. This is sufficient for this application as there are no fast ADC reads being done or time specific image processing. A crystal with 20pF load capacitance was selected as this is the most common value found on the market. To calculate the load capacitors (C1 and C2), the following formula was used

$$C_{load} = \frac{C_x^2}{2C_x} + C_{stray} \tag{1}$$

$C_{load} = 20pF$ (from datasheet)
$C_{stray} = 2pF$ (safe assumption if traces are short)
Solving for C_x in (1) gives $C_x = 36pF$

As stated in AVR042 [1], if additional devices are connected to the SPI bus when the ISP lines are connected, 330R series resistors must be placed on the bus as shown below. Since the SD card uses the SPI bus to interface with the microcontroller, the series resistors were added to prevent driving the SD card from the ISP programmer.

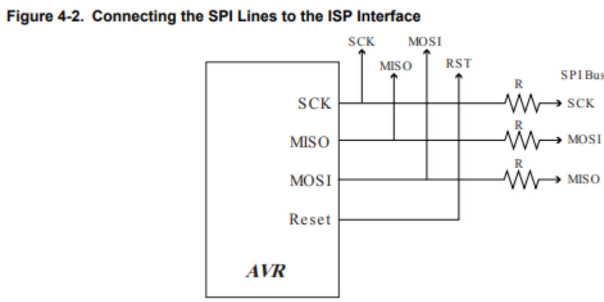

Figure 2: SPI/ISP resistor connections

In order to achieve the required differential impedance of $90\Omega \pm 10\%$ on the USB differential pairs (labeled DIFF90), the following formula is used to solve for w, the trace width

$$Z_d = \frac{174}{\sqrt{\epsilon_r + 1.41}} \ln\left(\frac{5.98 \times h}{0.8 \times w + t}\right)\left(1 - 0.48 \exp\left(-0.96\frac{d}{h}\right)\right) \tag{2}$$

Where
Z_d = target differential impedance, $90\Omega \pm 10\%$ $(81\Omega - 99\Omega)$
t = layer thickness, 1.378 mils
h = dielectric thickness, 7.874 mils
d = trace separation, 5 mils
ϵ_r = relative dielectric constant, 4.6
These values were obtained from the board stackup presented below

#	Name	Material	Type	Weight	Thickness	Dk
	Top Overlay		Overlay			
	Top Solder	Solder Resist	Solder Mask		0.5mil	3.8
1	TOP SIGNAL		Signal	1oz	1.378mil	
	Dielectric 1	7628	Prepreg		7.874mil	4.6
2	GND		Signal	1/2oz	0.689mil	
	Core	FR-4	Core		41.929mil	4.5
3	POWER		Signal	1/2oz	0.689mil	
	Dielectric 2	7628	Prepreg		7.874mil	4.6
4	BOTTOM SIGNAL		Signal	1oz	1.378mil	
	Bottom Solder	Solder Resist	Solder Mask		0.5mil	3.8
	Bottom Overlay		Overlay			

Figure 3: 4 layer PCB stackup

Using (2) it can be calculated that w $\approx$ 8.866 mil. This can also be verified using Altium Designer's impedance calculator which gives w = 8.745 mil, both answers which are well within the 10% tolerance.

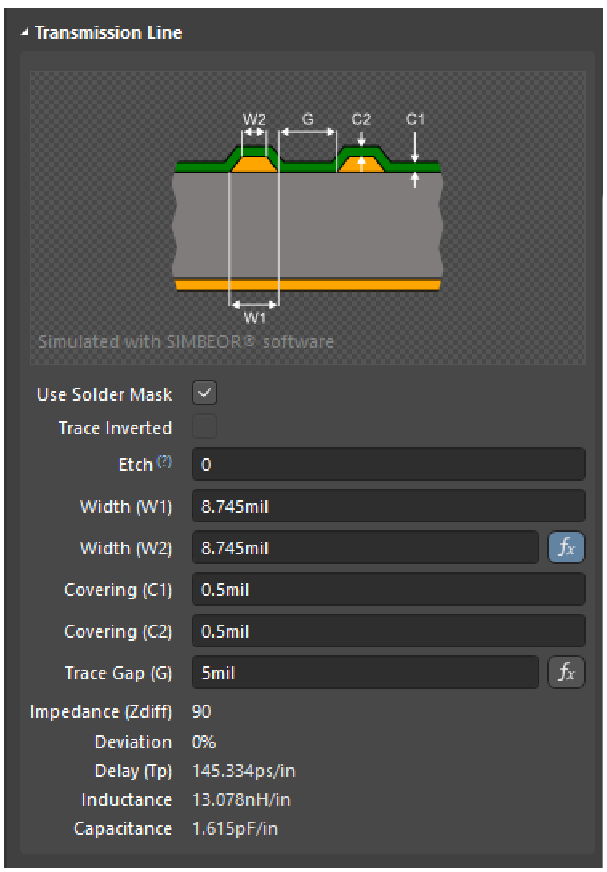

Figure 4: Differential transmission line

Assuming the following
Etch factor = 0, making W1 = W2 = W
Solder mask thickness (C1, C2) = 0.5 mils
Trace gap (G) = 5 mils
Gives a width (W) = 8.745 mils

1.1.2 Peripheral Design

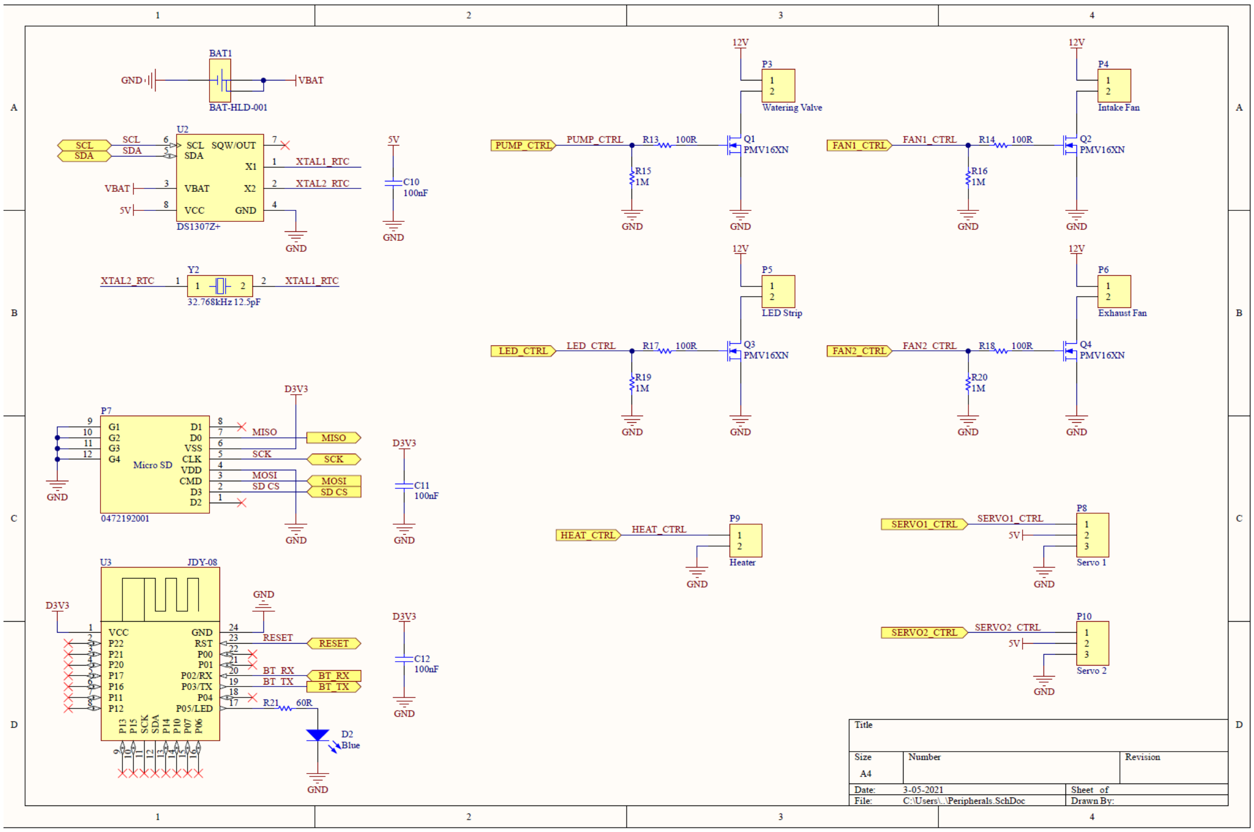

Figure 5: Peripherals

Each microcontroller pin can only sink up to 40mA, so in order to be able to drive the LED strip (P5), fans (P4 and P6) and valve (P3), a MOSFET driver is required. The MOSFET that was selected was the PMV16XN.

I_D (max)	8.6A
V_{GS}	±12V
V_{DS}	20V

Table 1: PMV16XN specs [2]

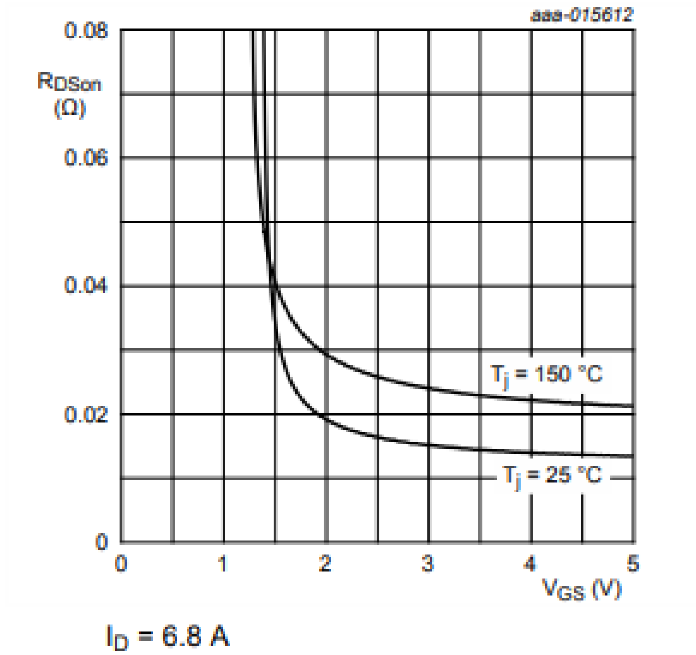

Figure 6: R_{DS} vs V_{GS} [2]

The PMV16XN was selected due to it having a V_{GS} of ±12 V, which means that the MOSFET can be controlled with 3.3V and having a V_{DS} of 20V means a 12V load can be driven with no problem. Looking at Figure 6 above, it can be seen that at a V_{GS} of 3.3V, $R_{DS_{on}} \approx 15m\Omega$. This means at $I_D = 8.6$A, the MOSFET will produce 1.11W of heat, no heatsink required. A 1MΩ pulldown resistor was added so that the output would not be floating in the even that the microcontroller pins are in high impedance mode.

The DS1307 real time clock (U2) is used to keep accurate time for data logging even when the PCB is not powered (from external CR2032 battery). The DS1307 supply must be 5V, however 3.3V falls in the logic high range for the device. This means that the DS1307 can be interfaced by a 3.3V microcontroller while still being powered by 5V (this is important as in this application the ATmega32u4 is running at 3.3V). The DS1307 also draws 1.5mA when active. This is the worst case, however it can be assumed that it constantly draws 1.5mA for power draw calculations.

I^{2}C Address	0x68
V_{CC}	4.5 to 5.5V
Logic 1 Input (V_{IH})	2.2 to (VDD + 0.3)V
Logic 0 Input (V_{IL})	-0.3 to 0.8V
Standby Current	200μA (when SDA, SCL = 5V)
Active Current	1.5mA

Table 2: DS1307 real time clock specs [3]

The JDY-08 Bluetooth module (U3) is used to send data wirelessly though Bluetooth to the Raspberry Pi. For interfacing, this particular module can use software UART, i.e. any digital pins can be used and not just the RX/TX pins (RX is connected to PB5 and TX is connected to PB6). The blue LED (D2) on P05 lights up solid when a Bluetooth connection is established.

The portable car heater (connected to P9) that is used to heat up the greenhouse is controlled by a solid state relay, which is able to drive a 25A, 12V load directly from the microcontroller pin. In testing, it was measured that the input current to the solid state relay was 2mA at 3.1V.

The servos (connected to P8 and P10) are used to control the intake and exhaust fan flaps. The angle of rotation on the servo is controlled by a PWM wave from the microcontroller which is then decoded by the servo's control circuitry.

1.1.3 Sensor Design

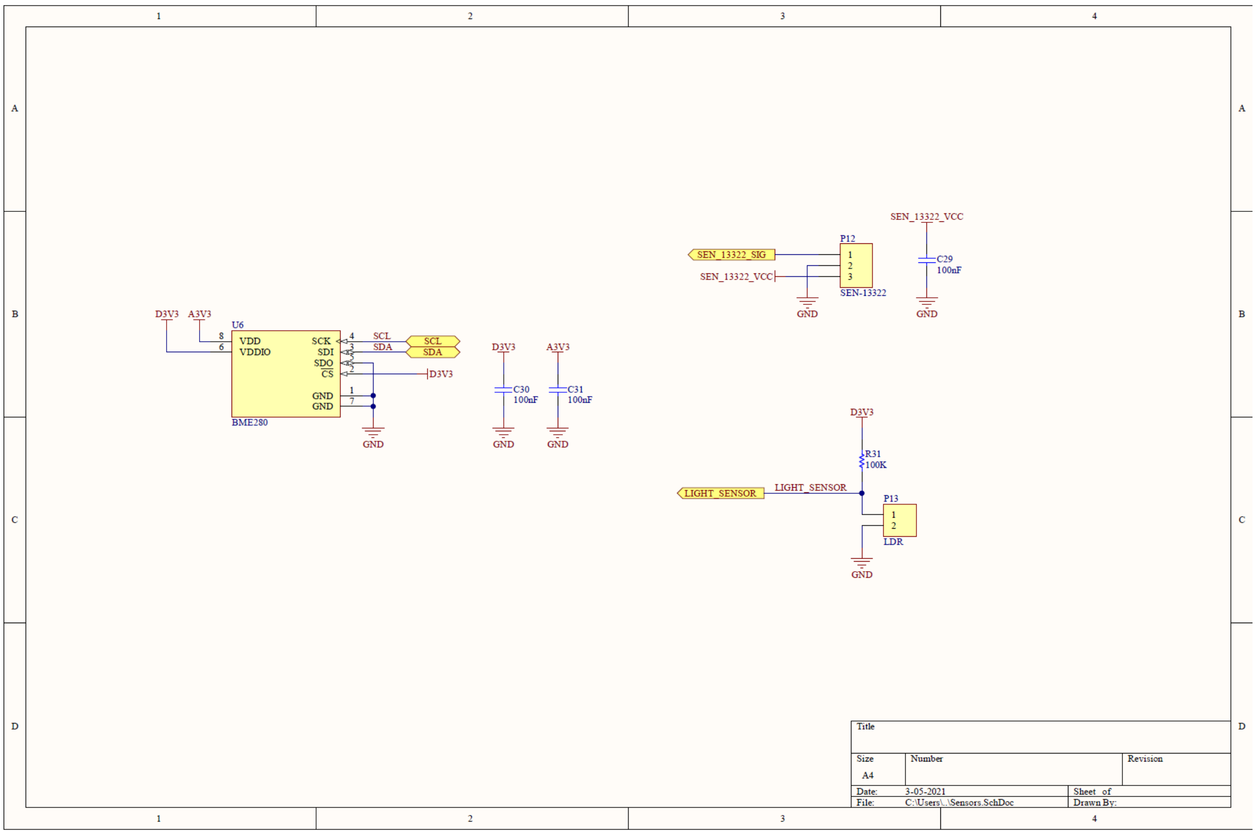

Figure 7: Sensors

The BME280 (U6) can be run at 1.8V or 3.3V (for this application it is run at 3.3V). This sensor requires the analog supply voltage ripple (V_{DD}) to be under 50mVpp. The BME280 also requires I^2C pullup resistors on both the SDA and SCL lines. The value was selected as 4K7 as this meets the requirements for I^2C pullups for both the BME280 and DS1307.

I^2C Address	0x76 or 0x77
V_{DD} (A3V3)	1.71 to 3.6V (50mVpp ripple max)
V_{DDIO} (D3V3)	1.2 to 3.6V
Standby Current	0.5 μA
Humidity Measurement Current	340μA
Temperature Measurement Current	350μA
Pressure Measurement Current	714μA

Table 3: BME280 temperature/humidity sensor specs [4]

The SEN-13322 soil moisture sensor (connected to P12) has 3 pins for interfacing: VCC, GND and SIG. The sensor's SIG pin connects to any microcontroller ADC pin. It is recommended by the manufacturer to only power the device when reading, and since the current draw of the device is only 0.33mA, it can be safely driven from a microcontroller pin.

V_{CC}	3.3 to 5.0V
Active Current	0.33mA ($3.3V/10k\Omega = 0.33mA$)

Table 4: SEN-13322 soil moisture sensor specs

The GL5516 LDR (connected to P13) is used to measure the light level. Since it is difficult to calibrate a light dependent resistor without a luxmeter, the ADC value is simply read at pitch black and direct sunlight and a linear equation is fit to the two data points. This is sufficient enough to detect when the sun is out or not (take measurement at sunset/sunrise and set as threshold). To calculate the ADC values that are read by the microcontroller, consider the following voltage divider circuit

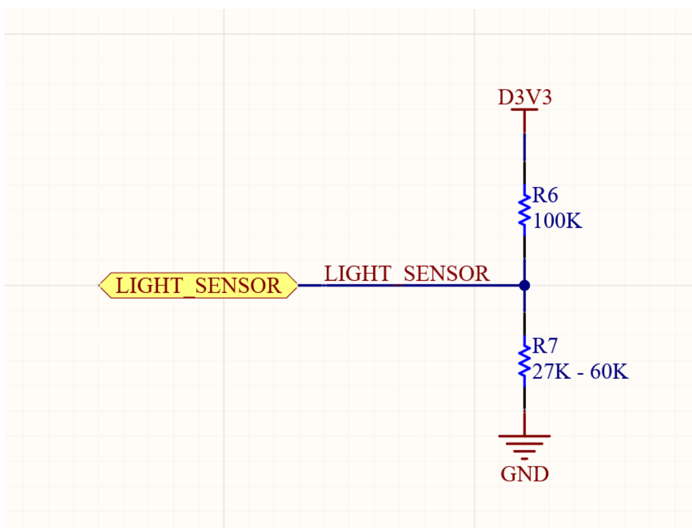

Figure 8: LDR voltage divider

Using the following formula

$$V_{OUT} = V_S \left[\frac{R_7}{R_6 + R_7} \right] \tag{3}$$

And plugging in $V_S = 3.3V$, R6 = 100K and R7 = 27K to 60K gives a voltage range of 0.702 to 1.238 V. That voltage range is then read as a 10 bit value (0-1023) and internally converted back to a voltage for easier programming using the following formula

$$V_{actual} = \frac{ADCReading * 3.3V}{1024} \tag{4}$$

V_{CC}	150V MAX
Resistance range	27k to 60kΩ
Current draw (Ohm's Law, 100kΩ in series with LDR)	0.02 to 0.03mA

Table 5: GL5516 light dependant resistor specs [5]

1.1.4 Power System Design

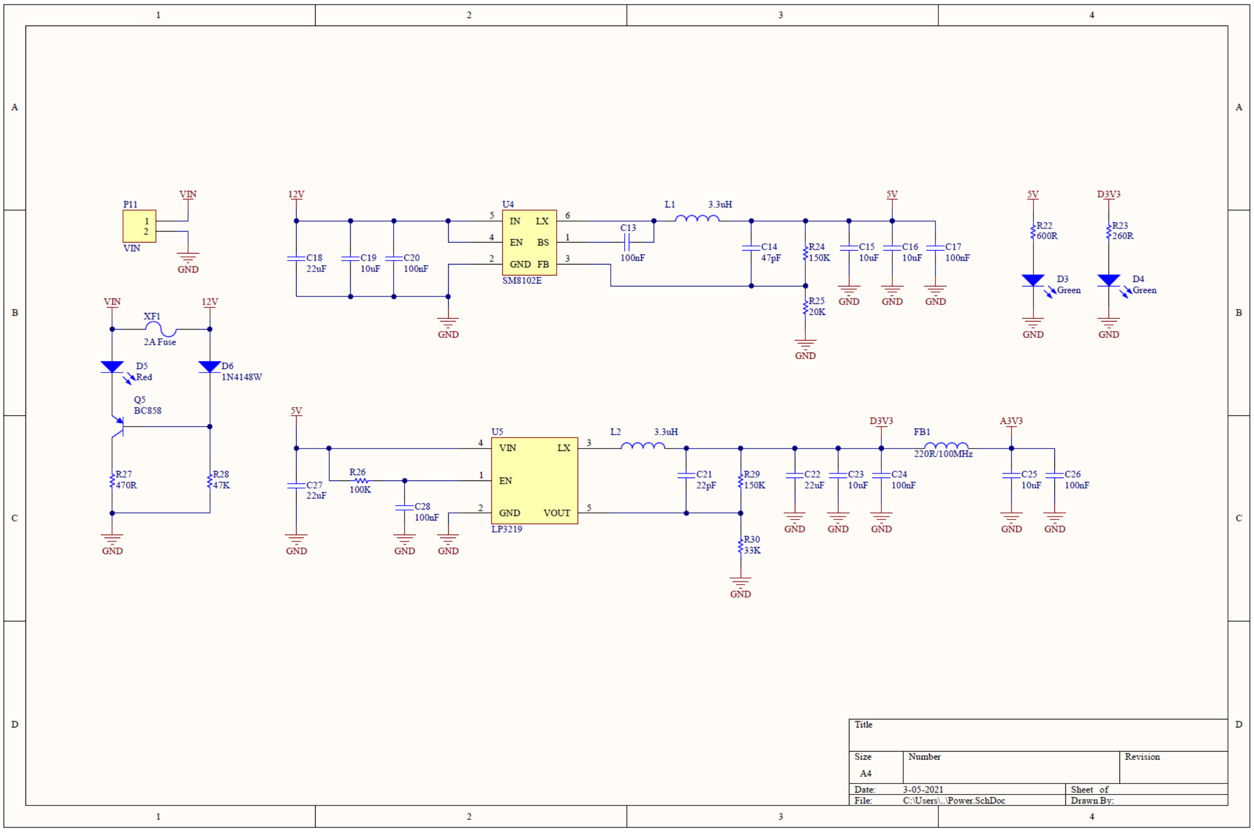

Figure 9: Power system

The board receives 12V (VIN) from the car battery, which is then fed into two step down regulators (12V-5V, 5V-3.3V).

SM8102		LP3219	
Input Voltage Range	4.2V-18V	Input Voltage Range	2.5V-5.5V
Max Output Current	2A	Max Output Current	2A
Switching Frequency	500kHz	Switching Frequency	1.5MHz

Table 6: SM8102 [6] and LP3219 [7] specs

These regulators were selected due to them meeting the power requirements (input voltage and output current) and their high efficiency (due to them being switching power supplies as opposed to low dropout regulators). The bypass/decoupling capacitors (C18, C19, C20, ...) placed on the input and output of each regulator stabilize the voltages and reduce ripple. As every electrical engineer knows, the current through a capacitor is given by

$$i(t) = C\frac{dV}{dt} \tag{5}$$

Since an instantaneous change in voltage would cause $\frac{dV}{dt}$ to be very large, in order for (5) to hold, $i(t_x)$ would have to be equally as large, which is not possible in practice. This property allows capacitors to be used to smooth power supplies and to supply steady voltage levels to digital circuits. The reason for different values in parallel is that the impedance of a capacitor changes as the frequency changes. The impedance of a "real" capacitor can be represented as

$$Z_C = X_C + X_L + ESR \tag{6}$$

Where ESR is the equivalent series resistance, which come from the imperfections in the capacitor. Since the reactance of a capacitor and inductor are given by

$$X_C = \frac{1}{j\omega C} \tag{7}$$

and

$$X_L = j\omega L \tag{8}$$

Substituting (7) and (8) into (6) it can be shown that

$$Z_C = \frac{1}{j\omega C} + j\omega L + ESR \tag{9}$$

If this function is graphed, something similar to the following is obtained

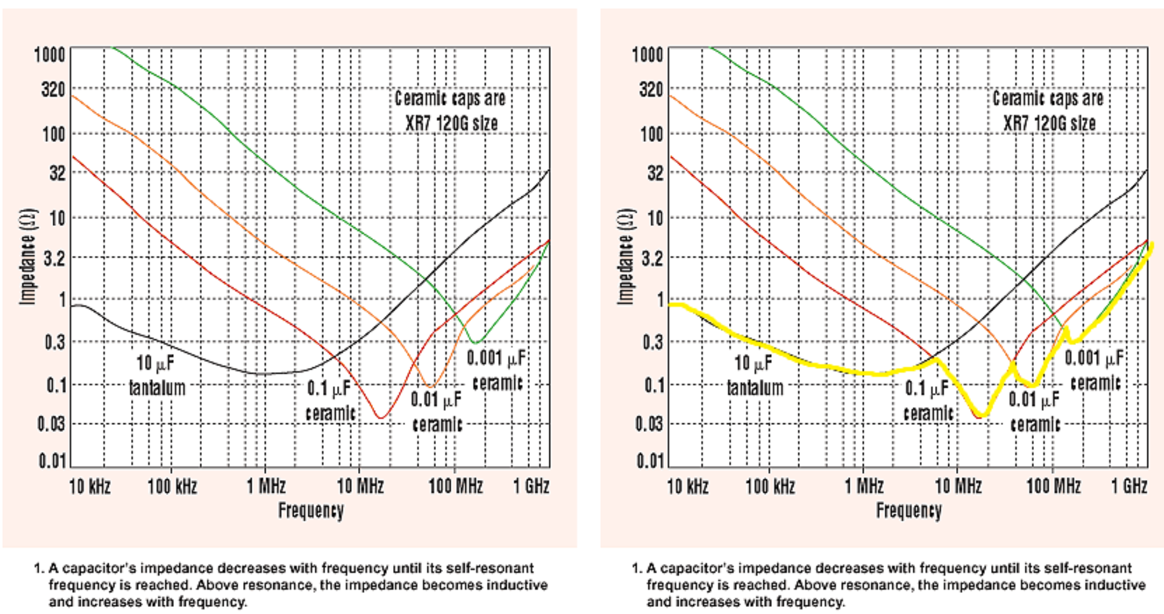

Figure 10: Capacitor impedance curve [8]

As ω increases, X_c decreases and X_L increases. When $X_c = X_L$ (resonant frequency) the total impedance is the lowest. When multiple capacitors are placed in parallel, the curve becomes more like the yellow curve on the right. This gets more complicated when you factor in the composition (ceramic, tantalum, electrolytic) and size (0402, 0603, 0805) as these all play a role in equation (9), specifically the ESR component. The values of the capacitors in the design were selected for ideal performance by the manufacturer of the regulator chips.

To set the output voltage of U4 and U5, the resistors R24, R25, R29 and R30 are selected to set a specific voltage at the FB/VOUT pins. The required resistors for both regulators are calculated using the following formula

$$V_{OUT} = 0.6(1 + \frac{R_H}{R_L}) \tag{10}$$

This formula was used to calculate the following values

SM8102

R_H	150KΩ
R_L	20KΩ
V_{OUT}	5.1V

LP3219

R_H	150KΩ
R_L	33KΩ
V_{OUT}	3.32V

Table 7: Voltage regulator resistors

The closest resistors that are available were selected that give a bit of overshoot on the output, as the output voltage will lower when loaded. Each voltage was measured with a multimeter at the appropriate output capacitor for 5V and 3.3V. The theoretical values were obtained from (10).

Theoretical Value	**Measured Value**	**% Error**
5.100V	5.107V	0.137
3.320V	3.385V	1.958

Table 8: Voltage regulator output measurements

The ferrite bead (FB1) separates the digital 3.3V supply from the analog 3.3V supply. While not completely necessary, adding a bead in series reduces the noise on the more sensitive analog supply as it acts as a second order filter along with the bypass capacitors. In order for the supply to function properly, the output voltage ripple must be measured on an oscilloscope and fall under 3% of the supply voltage, according to Microchip [9]. This is a general rule of thumb, however the BME280 datasheet specifies the 3.3V supplies are required to be under 50mVpp ripple due to the sensitive analog components. To measure the analog and digital 3.3V supplies (A3V3 and D3V3), a ground spring was connected to an oscilloscope probe to lower the ground loop size, ie. two ground points on the same circuit have different voltage potentials.

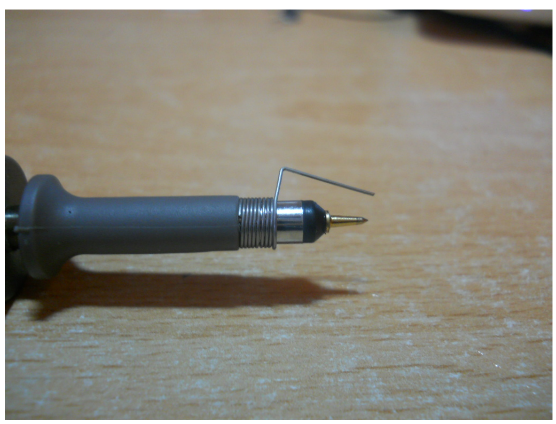

Figure 11: Oscilloscope ground spring

The probe was then used to measure the ripple on the output capacitors (C23 for D3V3, C25 for A3V3). The following waveforms were observed.

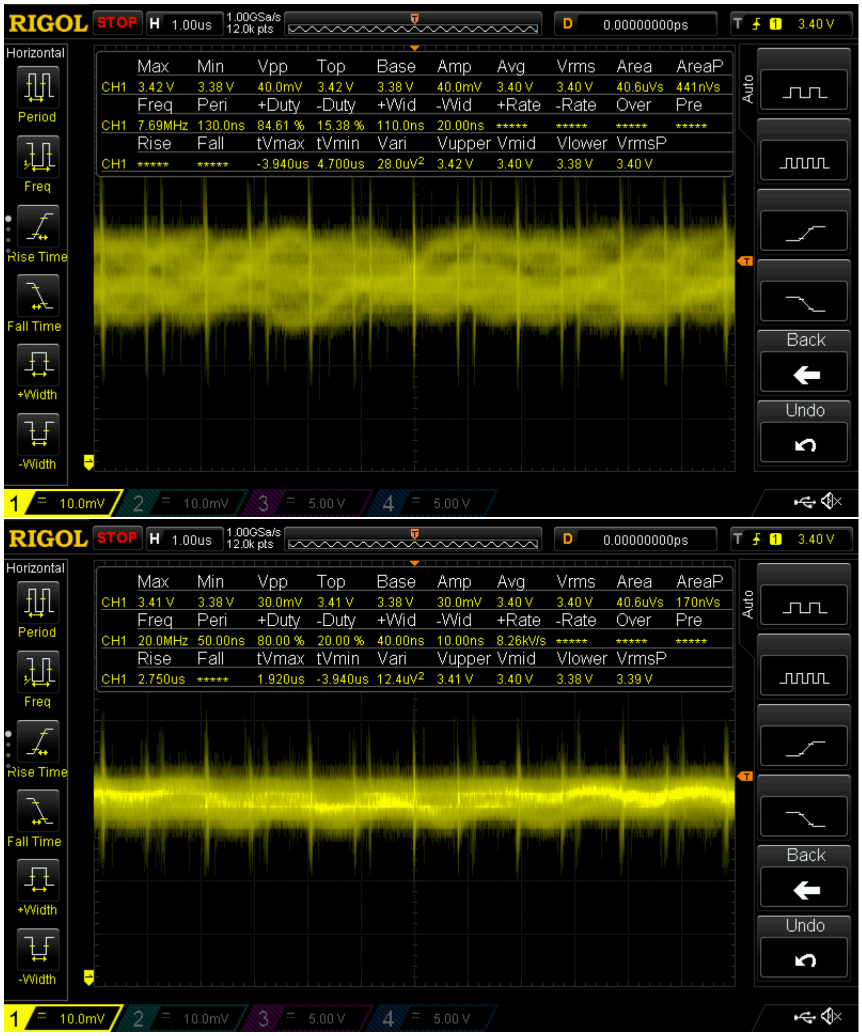

Figure 12: D3V3 ripple (top) and A3V3 ripple (bottom)

As can be seen, both waveforms are under the required 50mVpp (40mVpp for D3V3, 30mVpp for A3V3).

The power LEDs (D3, D4) require current limiting resistors. To calculate the required resistor value, the following formula is used

$$R = \frac{V_s - V_f}{I_f} \tag{11}$$

Where
V_s = supply voltage (5V or 3.3V)
V_f = forward voltage (2V for green LED)
I_f = desired forward current (5mA)
Solving for R in each case gives $R_{5V} = 600R$ and $R_{3.3V} = 260R$

The fuse indicator LED (D5) for the 2A ATO fuse is located between VIN and 12V. When the fuse is functional, the red LED will not light up and the circuit will operate as intended. When the fuse is blown, the red LED will light up. This can be seen in the following simulation results

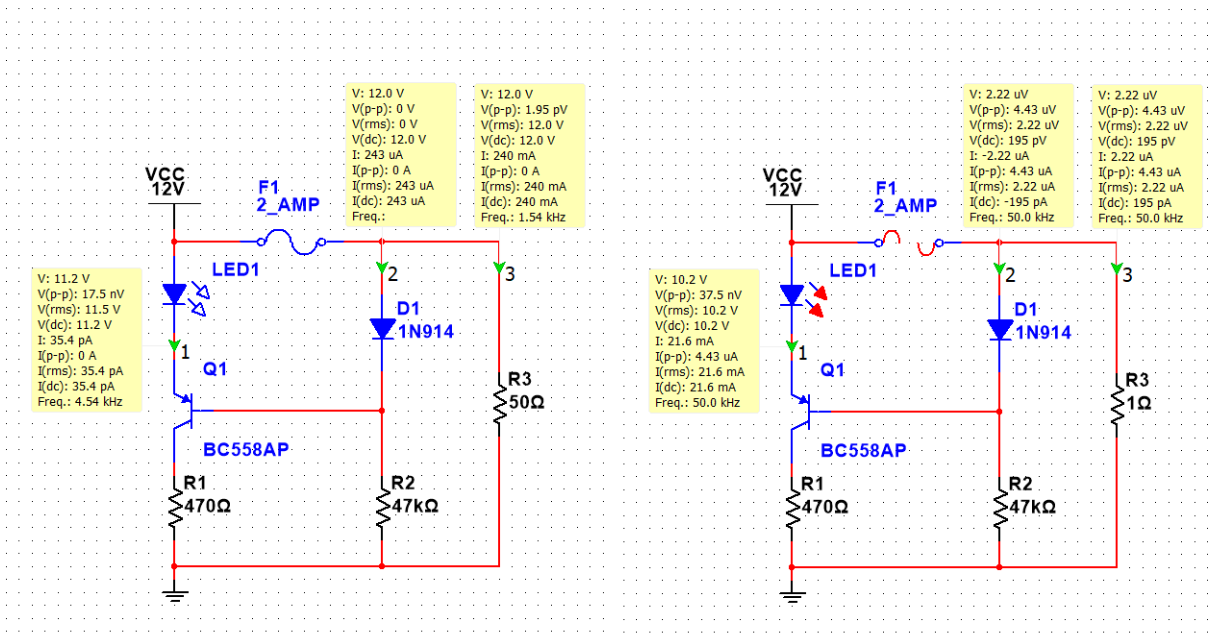

Figure 13: Fuse status indicator simulation

When a 50 ohm load is connected, a current of 240mA flows through the fuse. The current though the BJT and diode are extremely small (35.4pA and 243μA) and can be considered negligible. When a 1 ohm load is connected, a current of 12A flows through the fuse. This is more than enough to blow the fuse as seen in the right image. When the fuse is blown, the red LED illuminates. The active components used in the simulation are through hole components, however the following substitutions were made to allow all components to be SMD (substitutions have the same specs).

TH Component	**SMD Equivalent**	**SMD Package**
1N914	1N4148W	SOD-123
BC558	BC858	SOT-23

Table 9: Surface mount equivalent of through hole components used in simulation

To calculate the power trace thicknesses, the current draw of every peripheral was determined from the datasheets and are presented below

Peripheral	**Voltage**	**Current Draw**
Valve	12V	320mA
PC fans	12V	2x300mA = 600mA
LED strip	12V	850mA
Servos	5V	2x700mA = 1400mA
DS1307	5V	1.5mA
Microcontroller, sensors, BT module, etc	3.3V	<200mA

Table 10: Peripheral current draws

Using some approximations, it can be estimated that the 12V bus will draw around 3A (320mA + 600mA + 850mA + ∼500mA (for all 5V and 3.3V peripherals) ≈ 3A, allowing for some margin). The largest current draw on the 5V bus are the servos, which have a stall current of around 700mA each, depending on the model. As an example, the trace width for 3A will be calculated. Using IPC-2221, the trace cross sectional area must first be calculated

$$A = \left[\frac{I}{(k * (T_r)^b)}\right]^{\frac{1}{c}} \tag{12}$$

Where
A = cross sectional area [mils^2]
I = current [A]
T_r = max temperature rise [°C]
k = 0.024 (internal), 0.048 (external)
b = 0.44
c = 0.725
k, b, c are constants obtained from curve fitting the IPC-2221 curves [10]

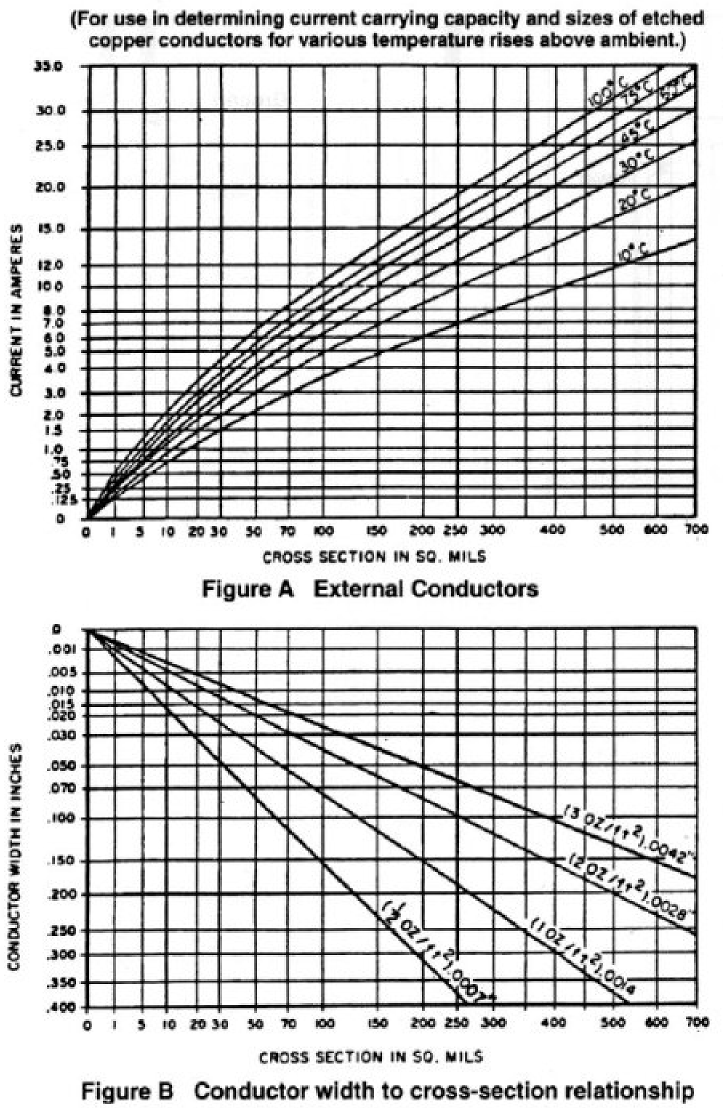

Figure 14: IPC-2221 curves [10]

With the values of T_r = 10°C and I = 3A, it can be determined that A = 192.93 mils^2 for internal traces and A = 74.16 mils^2 for external traces are required. To calculate the width, the following formula is used

$$W = \frac{A}{T * d} \tag{13}$$

Where
W = trace width [mils]
A = cross sectional area [mils^2]
T = thickness [oz]
d = 1.378 mils/oz

A = 192.93 mils^2 and T = 0.5 oz results in W = 280 mils (internal traces)
A = 74.16 mils^2 and T = 1 oz results in W = 53.8 mils (external traces)

As it can be seen, 280 mils is very thick for a trace, so for the purposes of this PCB design a polygon pour was used for the high current traces. Using (12) and (13), it can be calculated that for a trace to be able to carry 500mA it must be at least 10 mils wide. This was used as the standard trace width for most signals. The final PCB can be seen below:

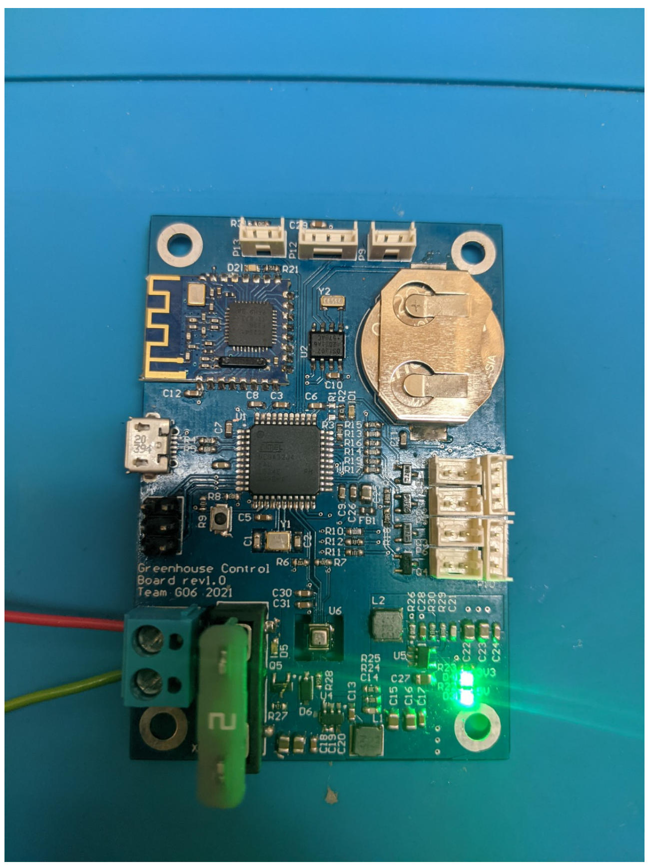

Figure 15: Control PCB