

Board 3: Final Report

1. POR

a. Purpose

The purpose of this board is to expand upon our Golden Arduino board, with improved design features and added components such as color changing LEDs and a heartbeat sensor. It will include design features to reduce noise such as improved routing, using 2 extra PCB layers to make it a 4-layer board, and decoupling capacitors around the power rails and switching I/Os. To create the heartbeat sensor we will use a photodiode placed between two LEDs, which will measure the light reflected off the blood in the finger that is placed over the heartbeat sensor. Additionally, we will include individually addressable LEDs that are controlled via I2C.

b. Schedule

- i. Wk 1, Tues 11/11, 9 am: POR, BOM, and schematic due
- ii. Wk 1, Fri 11/15, 9 am: PCB layout started
- iii. Wk 2, Tues 11/19, 9 am: layout done and release to fab
- iv. Wk 5, Tues 12/3, 9 am: assembly, bring up, testing, final report due

c. Power Budget

The input power will be the same as what is required for powering an Arduino Uno. The recommended range of input voltage is 7 to 12 volts. If supplied with less than 7 V, the 5 V output pin may supply less than five volts and the board may become unstable. If using more than 12 V, the voltage regulator could overheat and damage the board. The Atmega 328 will be utilizing 5 V instead of 3.3 V to have the fastest clock speed.

d. Risk Management

- i. Receiving faulty components is a potential risk that can be mitigated by ordering extras.
- ii. Miss-orienting the IC chips like the Atmega 328 could result in incorrect connections to the pins. We will check for the correct orientation before soldering it in..
- iii. The mechanical stresses placed on the output pins could break connections. We will use through-hole mounts to ensure this doesn't happen.
- iv. When soldering there is the potential for cold solder joints. We can mitigate this by inspecting each connection after the joint is formed.
- v. Identifying where parts go on the board could become difficult. The inclusion of silk screen identifications will make this easier.

e. Layout Plan

To reduce noise, decoupling capacitors will be added near power rails. Also, traces will be routed with mutual loop inductance in mind. This means that routing will be done in a way that avoids long return paths and large aggressor loops that induce current in victim loops contained within them. We will use ground planes for the middle two layers to reduce ground bounce noise.

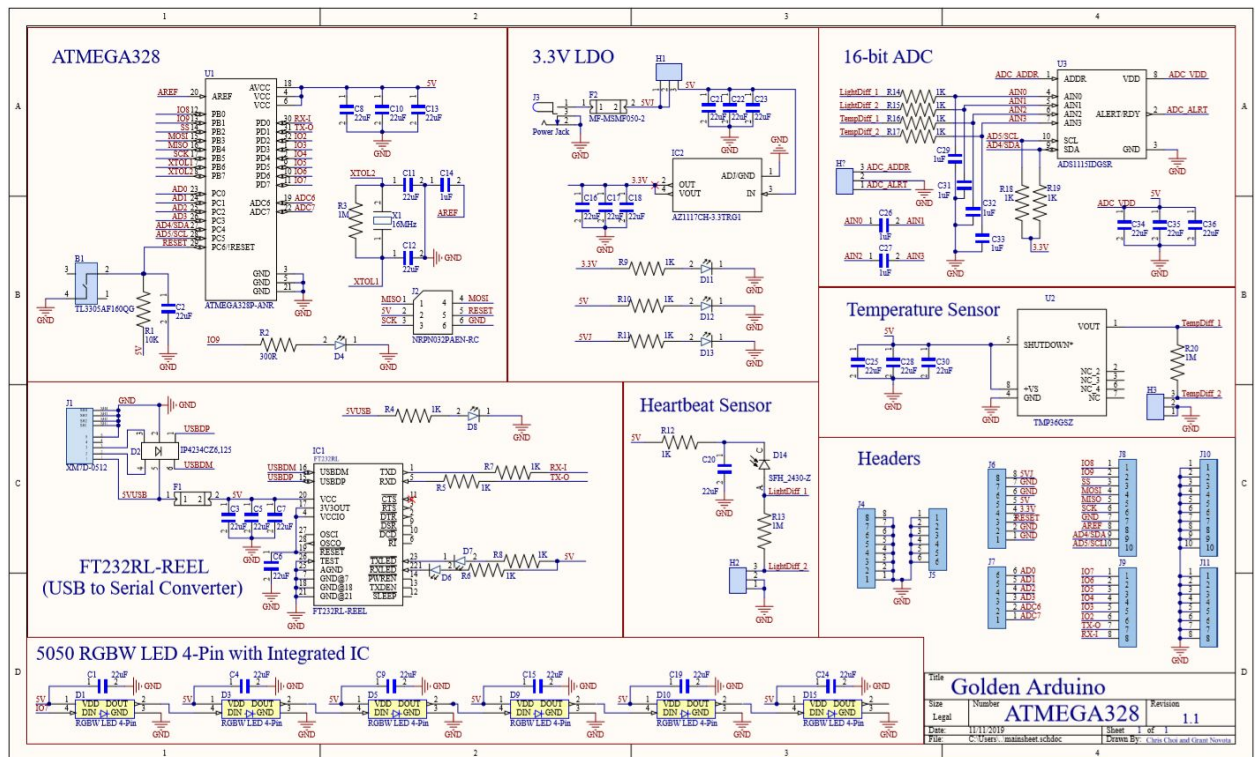
f. Bring-Up Plan

Switches will be placed to isolate the board's subcircuits and allow us to assemble and test pieces of the board one at a time and ensure that each piece is working correctly before moving on and assembling the rest of the board.

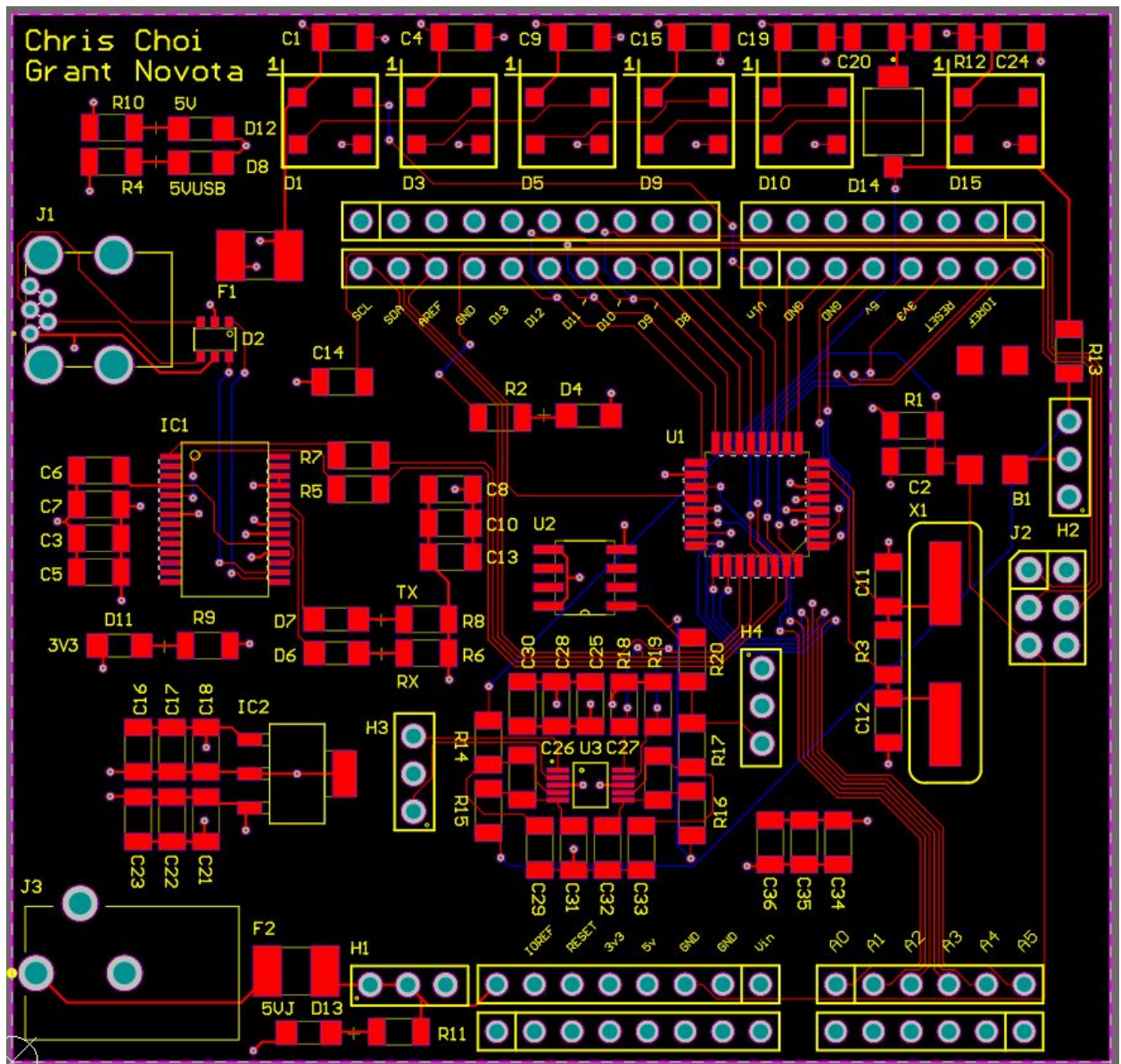
2. BOM

Component	Digikey Catalog	Count	Cost
16-bit ADC	296-38849-1-ND	1	\$5.52
Temperature Sensor	TMP36GSZ-ND	1	\$1.48
Photodiode	475-2579-1-ND	1	\$1.86
Programmable LED	1830-1209-1-ND	11	\$6.70

3. Schematic

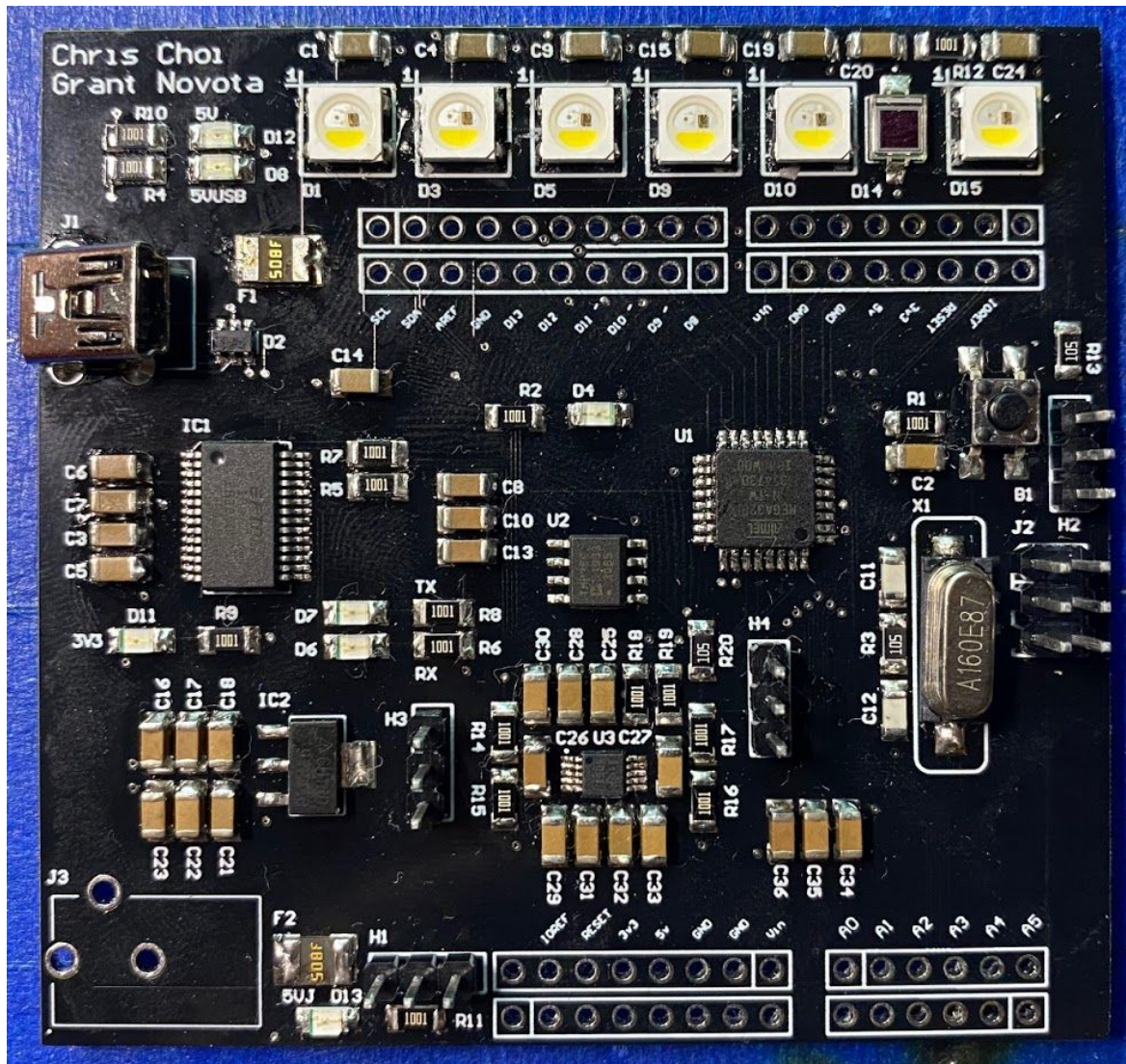


4. Layout (Power layer Pour and Ground Layer Pour Hidden to see traces)

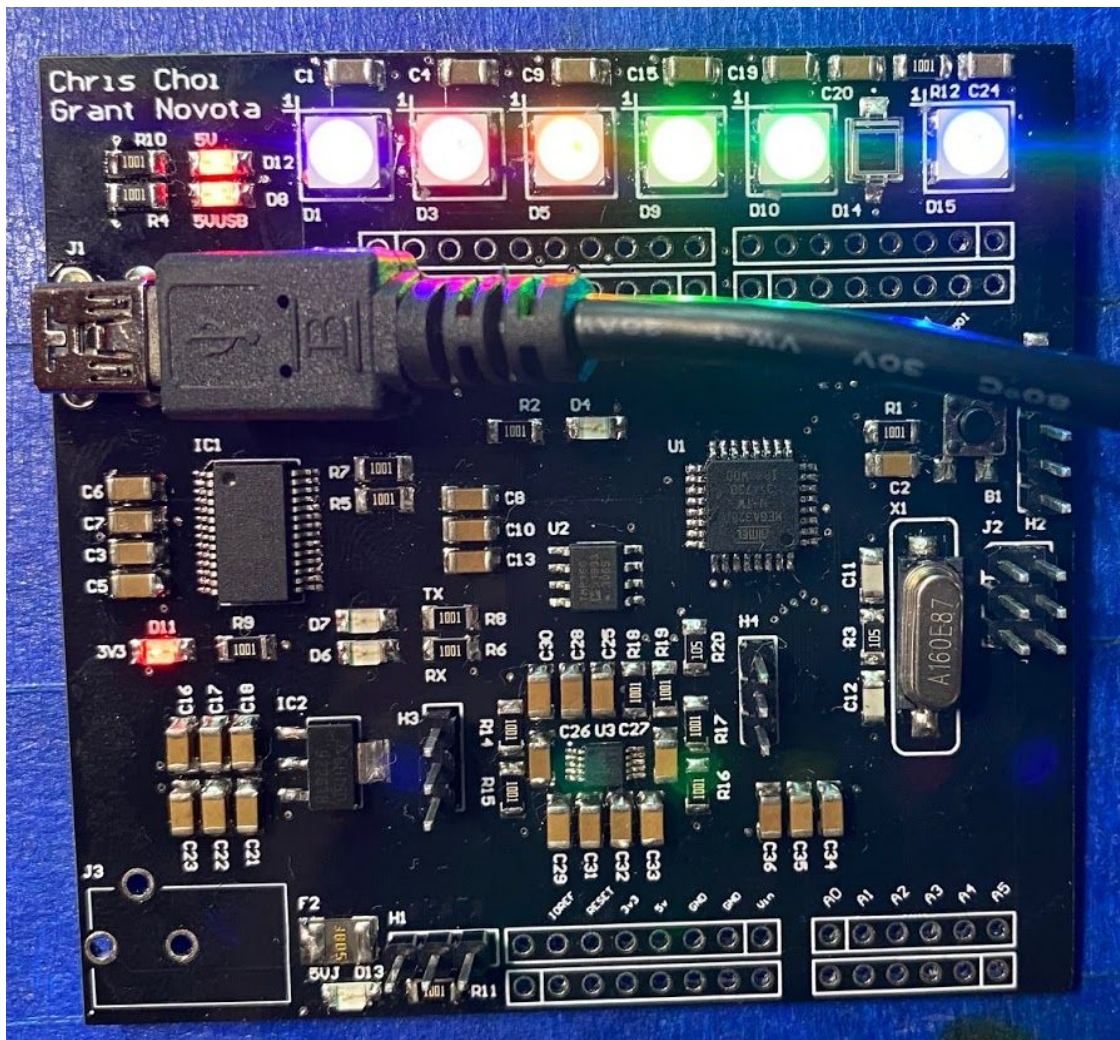


5. Assembly

The assembly of the board was primarily done at a lab bench at the Choi residence utilizing the proper (never enough) amount of flux and a microscope for chip soldering. Pictures were not taken in each step, but the photo below shows the final bring up of the board. The header pins were not added as the primary function of this board is to add LEDs and the ADC.



6. Bring up and Test



5/5 Points for the assembly of the board, burning the bootloader, and successfully manipulating the LEDs

7. Documentation

Scoring the schematic and the layout: **10/10**

All of the guidance the standards set by Professor Bogatin for the design of the board have been followed hence leading to a fully functional arduino board.

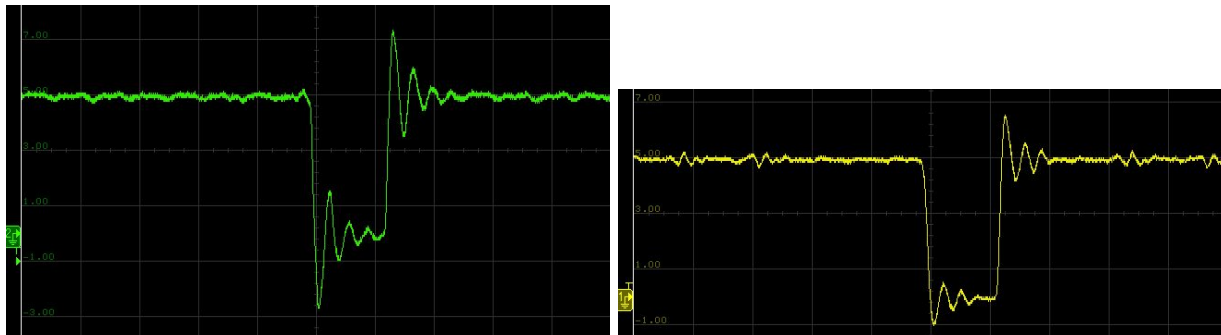
Throughout the bringup of the board, each chip was checked through a microscope to ensure the pins were not shorted together. The crystal was given care during soldering to ensure minimal thermal damage.

8. Documentation of Switching Noise and Noise Comparison with Commercial Uno including Quiet High and Low Noise

Below are two images where the switching noises from pins 9-13 are compared. The green scope shots represent the commercial board, the yellow scope shots represent our in house board, and the blue signals represent trigger signals to get a clean, consistent scopeshot.

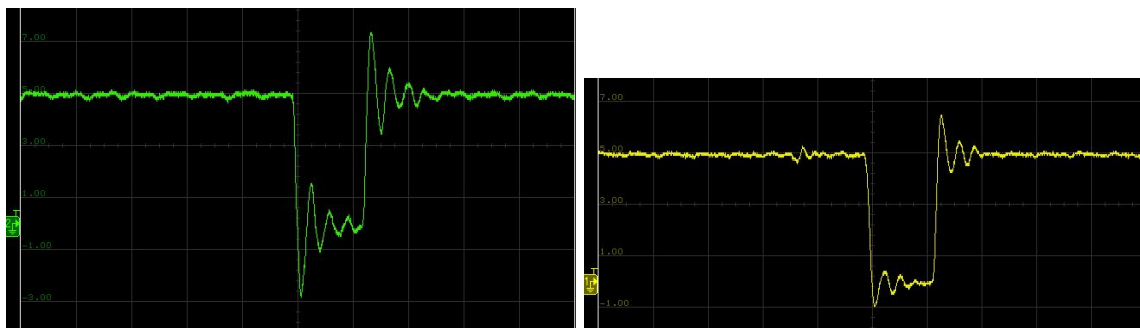
Pin 9

It is hard to see the labeling on the scope shots, but the commercial board (green) has a peak to peak switching noise of about four volts. Our in House board has a switching noise of approximately two volts. This is a 50% reduction in switching noise on pin 9.



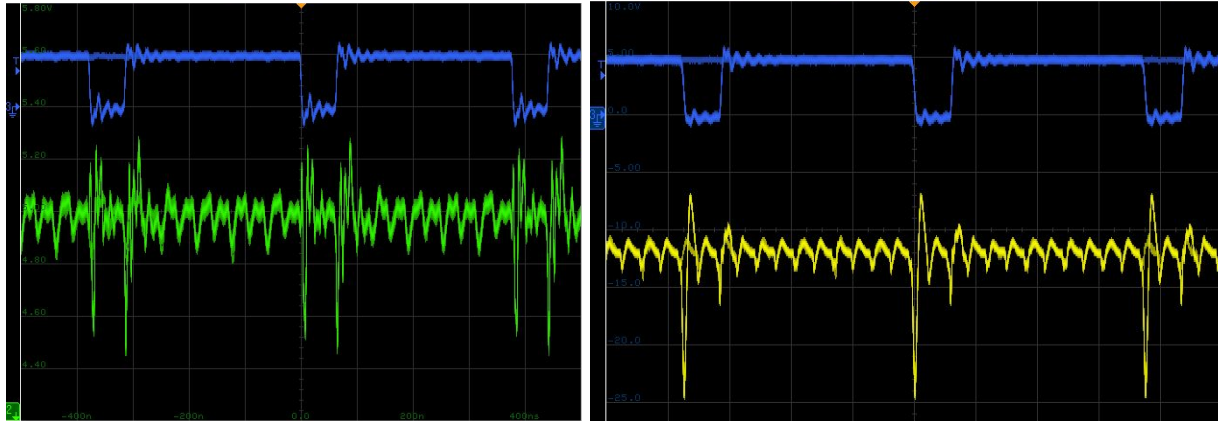
Pin 10

On pin 10, the switching noise on the commercial board is greater than four volts. It is sitting at approximately 4.2 volts peak to peak. On our in house board, the switching noise is just above two volts peak to peak. This is about a 50% reduction in switching noise.



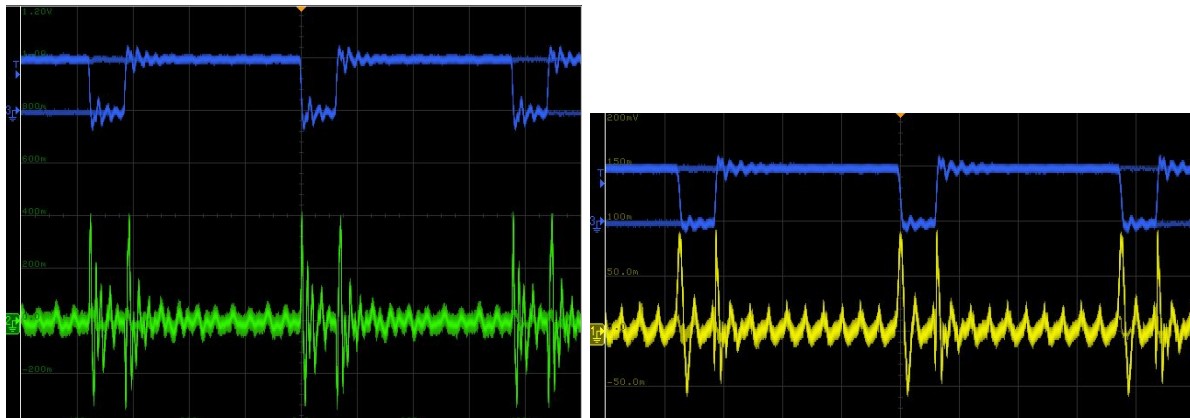
Pin 11

On pin 11 on the commercial board, the switching noise on the high-to-low is about 800 mV and the low-to-high switching noise is about 900 mV. On our in house board, the switching noise on the high-to-low is about 800 mV and our switching noise on the low-to-high is approximately 200mV. We reduced the low-to-high switching noise dramatically, but did not really affect the high-to-low switching noise.



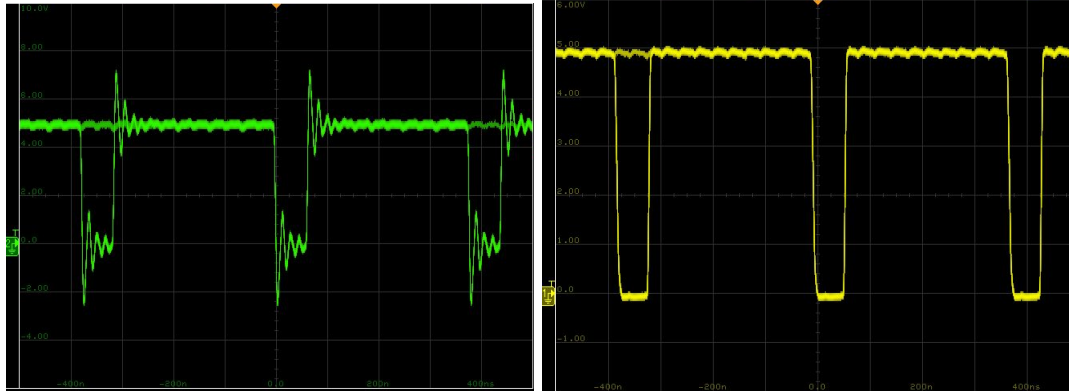
Pin 12

On pin 12, we used a switching IO to trigger the scope shots in both the commercial board and the in house board. In the commercial board, the switching noise is about 700mV peak to peak and the in house board has a peak to peak of 150mV. This is approximately a 20% decrease in the switching noise.



Pin 13

On pin 13, the commercial board shows a switching noise of four volts peak to peak and our in house board shows little to no switching noise. This is a massive difference and is cause for celebration.



9. Additional Documentation

As a supplement, the following steps to successful soldering has been detailed below:

1. Melt a small amount of solder onto one pad of a component.
2. Grab the component and place it in its correct orientation.
3. Reflow the previous solder so the solder grabs onto the component.
4. Let the solder cool. The part should now be locked into place.
5. Solder the rest of the component as it is being held into place.

Pro tip: using flux makes this process 100 times easier and also ensures a clean solder joint.