A Better Cosmic Watch Muon Detector: CosmicWatchBeta: The Amplifier

Asher Merrill
University of New Hampshire

May
2018

1 Introduction

The development of inexpensive, small-package particle detectors is important to the progress of particle physics for a number of reasons: First, by utilizing a silicon photo-multiplier, (SiPM,) the need for a large, cumbersome photo-multiplier tube is avoided; this can allow detectors to be placed in many more applications where the detector is size-limited. Second, by developing procedures and building an understanding of how to construct high-speed filters and amplifiers, the need for the expensive filters and amplifiers typically required by particle detectors can be avoided.

MIT's CosmicWatch[1] particle detector accomplishes many of the goals of this project: it is both a radical reduction in size of a traditional particle detector, and accomplishes the detection of events with relatively inexpensive off-the-shelf operational amplifiers. MIT's CosmicWatch is fundamentally limited, however: by design it only counts the number of events that occur. To serve more utility, this can be improved to find the energy of each event that occurs.

To find the energy of each event that occurs, a faster filter and amplifier must be used. A faster filter and amplifier combination would allow a capacitor to be charged with the incoming pulse from the SiPM, and subsequently alert an ADC to readout the discharge time of the capacitor. This time can then be used to calculate the total charge on the capacitor, or energy of the event.

CosmicWatchBeta works to construct a faster filter/amplifier pair to allow this charge measurement to occur.

2 Design of the Amplifier

2.1 Design Decisions & Philosophy

The design of the amplifier circuit is fundamentally identical to the original CosmicWatch with minor changes. The choice to use a nearly-identical design stemmed from desiring to conserve

[1]See `http://cosmicwatch.lns.mit.edu/` for MIT's website detailing the foundation for this project.

effort, as well as make full use of what existing resources are available. Additionally, to build off their schematic is validated because the requirements for the improved amplifier are only that it have a higher bandwidth. This could in theory be achieved with only a change the opamp used in the circuit with whatever changes would be associated with the changed opamp.

From the beginning of the project it was assumed that virtually all components attached to the amplifier would need to be mounted to a printed circuit board, (PCB,) and more than likely a surface-mount PCB, (SMT for surface-mount technology(ies) or SMD for surface-mount device(s).) This was decided to minimize trace length and the possibility and/or influence of RF noise in the frequencies that the opamp will be working.

All of the files and the associated design developments of this project can be found at the project's GitHub repository, available at `https://github.com/asher-m/CosmicWatchBeta`.

2.2 Schematic Design in Eagle

The schematic from the original CosmicWatch was duplicated in Eagle with a newly selected, faster opamp: the Maxim Integrated MAX4104 opamp. This MAX4104 is a high-bandwidth unity-gain-stable (as we would come to learn) opamp. Its bandwidth is 740 MHz, and it has a slew-rate of $400\,\mathrm{V\,\mu s^{-1}}$.

A secondary, physically-identical opamp was selected to serve as a fill in with slightly different characteristics. This opamp, the Linear Technologies LT6200 opamp, has a slightly reduced bandwidth, but is more closely identical to the original CosmicWatch in that it is rail-to-rail. Its bandwidth is 1.4 GHz and it has a slew rate of $340\,\mathrm{V\,\mu s^{-1}}$, and is designed to be stable at 10x gains.

The design was changed from the CosmicWatch in two key ways:

- Because of the opamps chosen for use, our design required a V_{-} to be added to the amplifier board.
- To allow the package to be more modular, we chose to connect the detector to the amplifier board through an SMA cable. This required the inputs of the amplifier to be impedance-matched to $50\,\Omega$ SMA cable and terminations.

To help with the second item above, we contacted SensL, the manufacturer of the sensor to be used with the scintillating material to detect events. After a number of messages, the engineer from SensL kindly directed us to `http://www.mantaro.com/resources/impedance-calculator.htm`. This tool was used to find the necessary trace width to match the required $50\,\Omega$ impedance.[2]

A schematic of the amplifier circuit can be found in Figure 1.

[2]Incidentally, numbers *must* be correctly entered into this tool for it to be valid. For example, if it is [incorrectly] used that the thickness of the material to be used for the PCBs is merely 1.4 mil, the tool will give an invalid result.

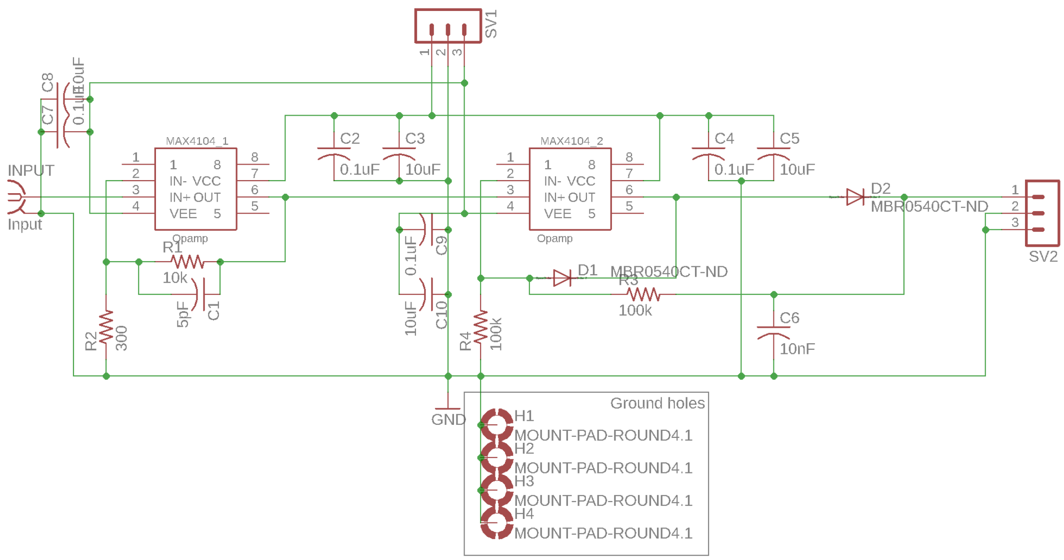

Figure 1: Amplifier Schematic

2.3 Board Design in Autodesk Eagle & Board Production

As it would ultimately be necessary to produce the PCBs to build the amplifier, the circuit boards for the amplifier were laid out in Eagle. Eagle served well for this purpose as we had previous experience with the tool, Eagle has a free license (limited to 2-layer PCBs,) and it is extremely well documented.

Renderings of the amplifier PCB can be found in Figures 2 and 3.

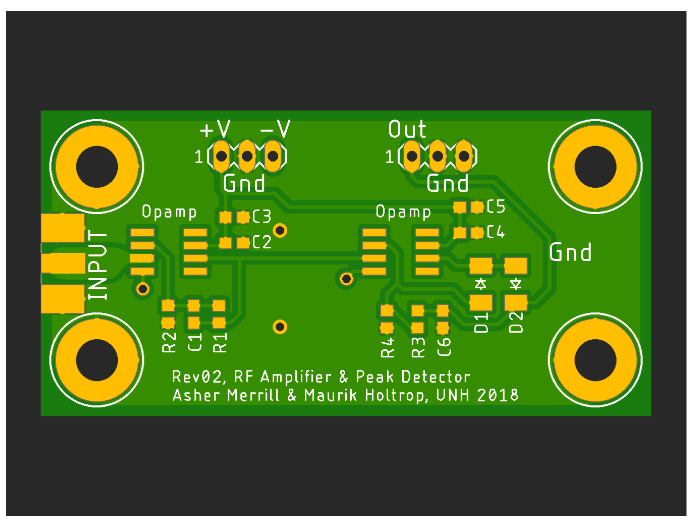

Figure 2: Amplifier PCB top

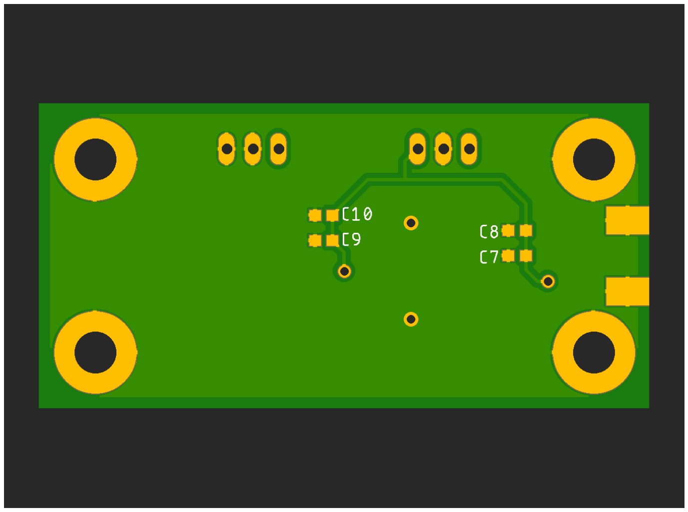

Figure 3: Amplifier PCB bottom

To have the PCBs prototyped, a colleague from the Mechanical Engineering department was asked to mill the PCBs on a CNC from double-sided copper-clad boards, (more or less identical to FR4.)

2.4 Detector Board Design, Layout & Production

The CosmicWatchBeta project originally involved the development of boards to host the SiPM. Unlike the amplifier board, this could be *exactly* identical to the board used in the original CosmicWatch.

Ultimately, slight changes to the design were included to allow the amplifier to be connected via an SMA cable to allow each piece to be more modular.

For the schematic and board designs, please refer to Appendix: *Detector Schematic and Board*, Figures 19, 20 and 21.

These too were kindly produced by our connection to the ME department. Unfortunately, they remain, for the mean time, unpopulated and not used.

3 SMT Soldering in a Toaster Oven

3.1 Idea

As discussed in 2.1, the electronics being used in the project virtually required the use the SMT. While it *is* possible to use a hand-soldering iron to work with SMT devices, this technique requires extreme care and often results in less-than desirable connections between components and the PCB. To avoid the need to hand-populate our PCBs, which were already very dense to minimize trace length and thus RF interference, we looked to experiment with various amateur reflow oven projects or ideas.

With that in mind, we purchased a toaster oven.

3.2 Parts

Throughout this project 0603 parts were used for the SMT soldering. These were large enough to see, but small enough that they required a reflow oven. (Some hobbyists may disagree and say that they can hand-solder these devices, and some SMT rework was necessary when working with the MAX4104 and LT6200. This process was rather difficult, however, and a reflow oven is encouraged.)

3.3 Notes & Procedures

It was found that solder paste could be relatively easily applied to the PCBs using a syringe of solder paste with a flexible plastic tip.[3] Solder paste could also be removed by simply wetting a paper towel with isopropyl alcohol and wiping off the paste, (this is helpful if a sleeve is accidentally caught in the paste and it is smeared.) Additionally, we used 91% isopropyl, though any percentage of alcohol (or perhaps even water) is likely to work fine. Ultimately it seemed as though the alcohol served more to wipe the residue of the paste off the board than it acted as a solvent.

During all reflow processes we merely watched the oven to visually confirm when the solder paste turned from its dull grey to a bright silver. At this point we would turn off the toaster oven and let the boards cool.[4]

It was eventually found that low-temperature solder paste worked best. This finding was because of the uneven and rapid heating of the toaster oven, (violating virtually any imaginable soldering profile[5],) the low-temperature solder paste reflowed long before the temperature of the oven could damage the PCB substrate or any of the components on the board. This allowed the oven to stay comfortably in-spec for all the devices used and avoided over heating of the PCB substrate.

In all of the procedures below a thermocouple was placed in the oven. Ultimately it was found that the thermocouple served best to be placed on the PCB itself with high-temperature tape or careful positioning when closing the oven door. This placement allowed the temperature of the boards to be more closely controlled by a human operator.

Additionally, in all of the following procedures the oven was heated as: Turning the temperature up to 450°F, and setting the oven to 'Keep On'.

We experimented with the following procedures:

1. With *normal temperature* solder paste: Using only the metal tray included with the toaster over, the PCBs were placed directly on this tray and the tray was positioned in the lower of the two positions in the oven.

[3] *The green plastic tip*, if using the same paste as we had. This tip has a relatively small hole at the end of a 1 in. extension from the syringe.

[4] Typically we allowed the boards to cool as long as our patience would allow. For most of our processes, this would mean prop the door of the toaster oven, wait about 2 minutes, and then grab the board with some tweezers.

[5] Despite how this sounds, this is extremely unlikely to cause an issue. Most devices are rated relatively low compared to what they can actually handle. An exception is the SensL detector used in this project: this should be soldered as close to specification as possible.

    - This allowed the solder paste to reflow relatively quickly and minimize the total heating time of the oven. With this method it was also observed that simple plastic components, (pin headers, etc.,) melted due to the uneven and rapid heating of the oven. Additionally, the PCB substrate was visually browned from oxidation from the high temperature of the oven. These problems were likely exacerbated by the placement of the tray directly above the lower heating element; in this way the tray served to catch all the IR radiation and heat up the components and PCB beyond the reflow temperature of the solder paste.

2. With *normal temperature* solder paste: a piece of aluminum foil was placed under the PCBs, which were placed on the oven tray in the lower position. This was also tried with 4 layers of folded aluminum foil.

    - This improved the browning the PCBs, but did not stop the headers from melting. Additionally, it was observed that the printed text on the tops of components was ablating in the high temperature; this issue was likely also caused by unwanted oxidation at the high temperatures of the oven.

      No immediate differences between 4 and 1 layer of foil were observed. In the end the primary difference between this and the previous method or setup seem to be the air gap between the tray and the PCBs. This served to insulate and slow the heating of the PCBs by some extent, allowing their temperature to be more closely controlled.

3. With *low temperature* solder paste: the second procedure above was duplicated with the use of low-temperature solder paste.

    - The use of low-temperature solder paste allowed the entire reflow process to be shorted by some amount. This decrease in time apparently resulted in the boards not browning as much as they had in the first procedure. No other differences were observed.

4. With *low temperature* solder paste: A final procedure with additional shielding for the PCBs was developed. See Section 3.4 for more details.

The above references to *low temperature* solder paste refer to our use of solder paste that was labeled as reflowing at 160 °C. Despite this labeling, however, we measured this paste to melt at 230 °C. Regardless of this labeling inaccuracy or error in our measurement, this temperature was still considerably colder than *normal* temperature solder paste.

3.4 Final Procedure & Recommendations

3.4.1 Pasting

Like the procedure for pasting detailed in 3.3, it was found that boards can be most easily pasted using a small, flexible plastic tip with a syringe.

Later it was found that low temperature solder paste worked best; this should be used whenever available.

3.4.2 Placing

Placing is most easily accomplished after pasting the board. Simply select the components for the board, and cut one type of component at a time from their respective tapes. Open the tape with a tweezers, and dump the components on the bench. Then grab and place the components under a magnifying glass.

For complex components like the opamp, be careful to place them in the correct position.

3.4.3 Shielding

Shielding was improved as follows:

- A tent of foil, about 3 in. high in the middle, was created by folding foil over the outermost bars on the rack provided with the toaster oven. This served to avoid direct IR heating of the PCB. This was created such that the rack was ‘concave’ up, or the rack was highest in the middle.
- The tray was now used *not* to hold the PCBs, but as a shield from direct IR heating from below.
- The rack and tray were placed together in the lower position in the over. PCBs were then placed on the rack. This setup altogether avoided direct IR heating of the PCBs, and avoided any of the previously seen issues with this toaster oven reflow process.

3.4.4 Measuring the Temperature

The thermocouple was placed directly on the PCB. This allowed the PCB temperature to be most accurately read.

3.4.5 Heating

Heating was accomplished as: Turning the temperature up to 450°F, and setting the oven to ‘Keep On’.

3.4.6 Timing

The reflow process was timed by simply looking at the solder paste, waiting for it to flow from a pale grey to silver, and turning off the oven.

The boards were then allowed to cool, and then process was done. In almost all processes there was no need to clean up flux; this can be accomplished by using some high-concentration isopropyl alcohol and wiping off the boards.

4 Amplifier Testing and Design Refinements

4.1 LT1226

In preparation for work with the SMT devices, prototyping was done with a Linear Technologies LT1226 opamp. This opamp has similar characteristics to both of the other SMT devices, but serves well for prototyping because it is through-hole and can be used on a breadboard.

A schematic of the setup can be found in Figure 4.

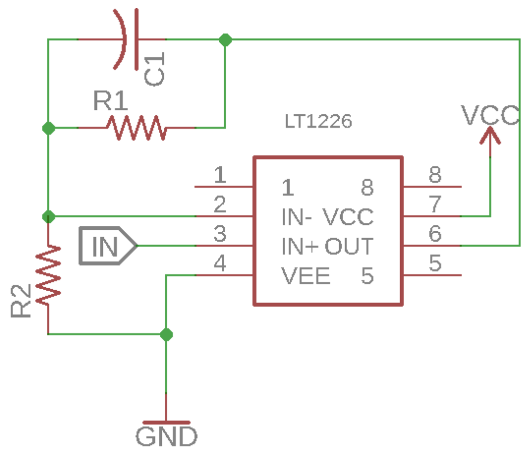

Figure 4: LT1226 Amplifier Schematic
Note that various values of R_1, R_2, and C_1 were used when testing.

The first objective in testing and setting up this amplifier was to stop any oscillations that may have been occurring when the input to the circuit was grounded. However, regardless of the values chosen for R_1, R_2, and C_1, the opamp still oscillated.

Eventually, after consulting a very helpful document from Analog Devices[6] on how to stop an opamp from oscillating, we found we had to take the following steps to quell the oscillation we found when we observed the output from the opamp with a grounded input:

- The frequency at which an opamp becomes unstable is inversely proportional to $R_1 \parallel R_2$, in the diagram above:

  $$f_{unstable} \propto \frac{1}{R_1 \parallel R_2} \tag{1}$$

  To address this, we can simply decrease the values of R_1 and R_2.[7] Eventually, due to the current limitations of the devices being used, it was decided to use R_1 and R_2 to be around $300\,\Omega$ and $30\,\Omega$, respectively, to give us a gain of around 11, depending on the exact resistors

[6] See `http://www.analog.com/media/en/technical-documentation/application-notes/an148fa.pdf` for this article. Written by an engineer with a lot of experience on the subject, it is likely to help if you're experiencing any issues working with opamps.

[7] I cannot do justice to the information provided by this article. For more information refer to footnote 6.

used. These values are radically smaller than those being used previously, (10 kΩ and 300 Ω, note the change to gain,) so some improvement was more than likely.

- The following equation must be true for the opamp to be stable:

$$R_1 * C_1 \approx R_2 * C_{parasitic} \tag{2}$$

With this in mind, and while using the Mantaro PCB layout tool, (see Appendix: B,) it was found the PCB likely had approximately 2 pF of parasitic capacitance. This together with the approximately 3 pF of input capacitance, we had to match

$$300\,\Omega * C_1 \approx 30\,\Omega * 5\,\text{pF} \;=\; 150\,\Omega\,\text{pF} \tag{3}$$

From this it was determined that because the additional capacitance required in the feedback line was so small, (indeed, essentially negligible,) the feedback capacitor could be safely ignored.

Despite all these changes, however, the most significant improvement came when it was changed how the breadboard hosting the LT1226 was connected to the Analog Discovery's (AD) function generator and oscilloscopes. As the amplifier continued to oscillate when using unshielded leads from the AD to the amplifier, despite having implemented the changes, some additional change needed to occur. This came in a breakout board for the AD's leads to BNC cables. Using shielded coax to connect the scopes and function generator of the AD to the amplifier, it was now observed that the amplifier was stable.

No data aside from how to stabilize the other amplifiers was recorded from this through-hole device. For information on how these amplifiers behave, please refer to their sections.

4.2 MAX4104

The MAX4104 amplifier was developed partially in parallel with the LT1226 breadboard amplifier. Because of this, some of the findings from the LT1226 were not immediately implemented on the circuit.

Additionally, at this point in the project it became clear that with the limited time left in the semester only some parts of the project would come to completion. It was decided to only build the amplifier section, (the left side,) of the amplifier board and get this piece to a working state.

4.2.1 Setup

The MAX4104 amplifier was connected to the Analog Discovery's power supplies, supplying 5 V and −5 V to V_{cc} and V_{ee} respectively, via the headers on the board. Additionally, more pins were soldered as:

- 2 pins soldered to the amplifier input to allow the AD's function generator to feed the amplifier and the AD to read what the amplifier was receiving via one of its scopes.
- 1 pin soldered to the output of the first opamp on the amplifier board. This would allow the second scope of the AD to read the amplifier's output.

4.2.2 Testing Stability

To test the stability of the opamp, prior to the use of BNC cables to allow less noise on the inputs and scope, leads from the AD were connected as detailed in section 4.2.1.

With this setup, no function was created on the AD's function generator, (ei., it was set to ground.) The output from the amplifier was then read on the AD's scope. No plot of the noise was recorded, but the AD correctly reported the amplifier was oscillating. Changes to the circuit were required.

Ultimately, changes to the circuit according to section 4.1; changes were made such that the feedback loop in the amplifier was identical to that detailed in the aforementioned section, (gain of 11, no feedback capacitor.) Additionally, the BNC breakout board for the AD was used to connect the AD to the amplifier, instead of merely using the leads from the AD.

Having now stabilized the MAX4104 amplifier, the output was recorded as shown in Figure 5.

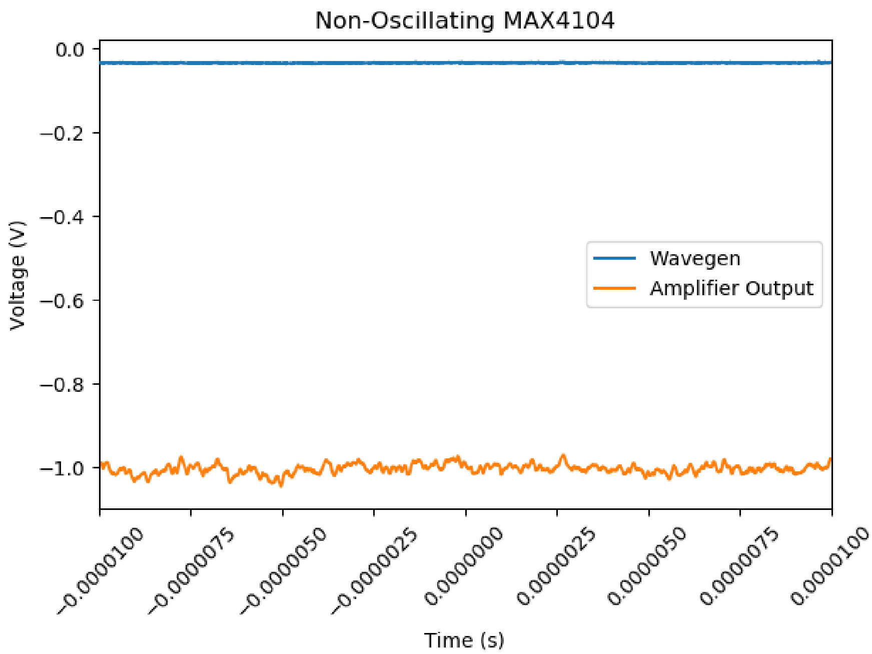

Figure 5: Non-oscillating MAX4104 Amplifier
Note that the function generator (Wavegen) is at a small offset below ground. This is likely caused by some voltage drop across R_2, or the second capacitor in the feedback loop.

When operating the amplifier using the function generator, we found the results detailed in Figures 6 and 7. More plots are available in Appendix: *MAX4104 Amplifier: More Plots*.

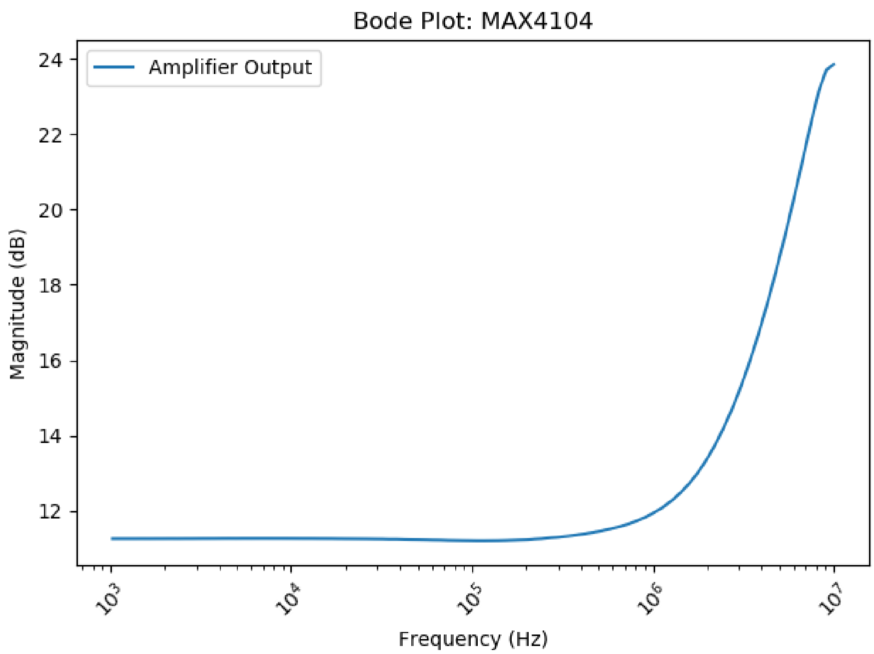

Figure 6: Driven MAX4104 Amplifier: Bode Plot
Note that the gain of the amplifier apparently increases as our frequency increases. This behavior apparently also slows down around the very highest frequency tested, 10 MHz.

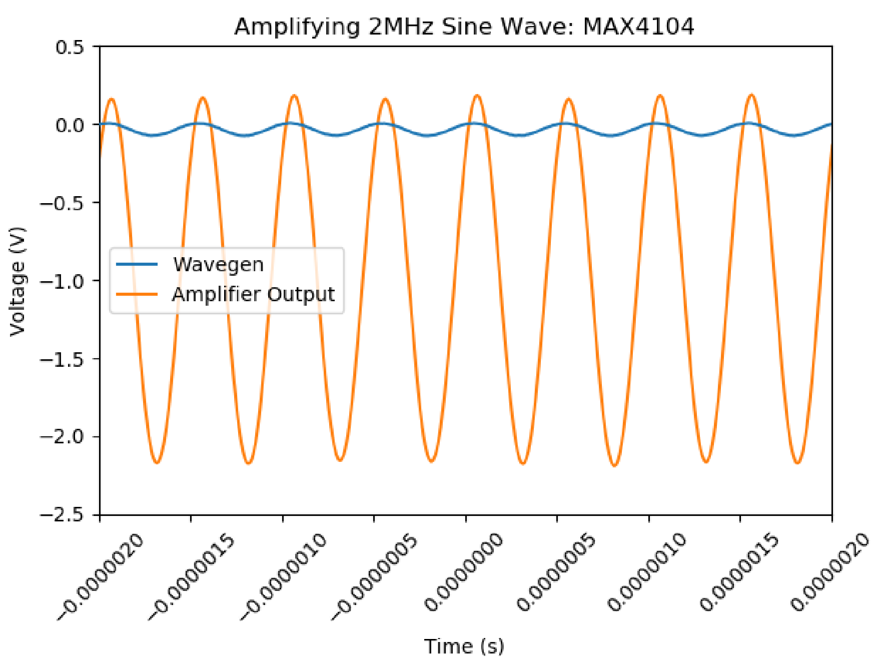

Figure 7: Driven MAX4104 Amplifier: Sine Wave

An interesting feature of this amplifier is that its offset voltage is so large. Indeed, it is seen from Figure 7 that the offset voltage is apparently close to $-1\,\mathrm{V}$.

Because this amplifier circuit would eventually be used to feed a peak detector, which requires its input to be [relatively] high to trigger, such a large negative offset would not suit the needs for the projects. Because of this, it was decided to use the LT6200.

4.3 LT6200

4.3.1 Setup

The setup of the LT6200 is identical to the MAX4104. See section 4.2.1.

4.3.2 Testing Stability

Because this amplifier was constructed after the learning experience of working with the LT1226, the LT6200 amplifier was built identically to the revised MAX4104. Despite including the revisions from the MAX4104, however, the LT6200 still exhibited some oscillations. See Figure 8.

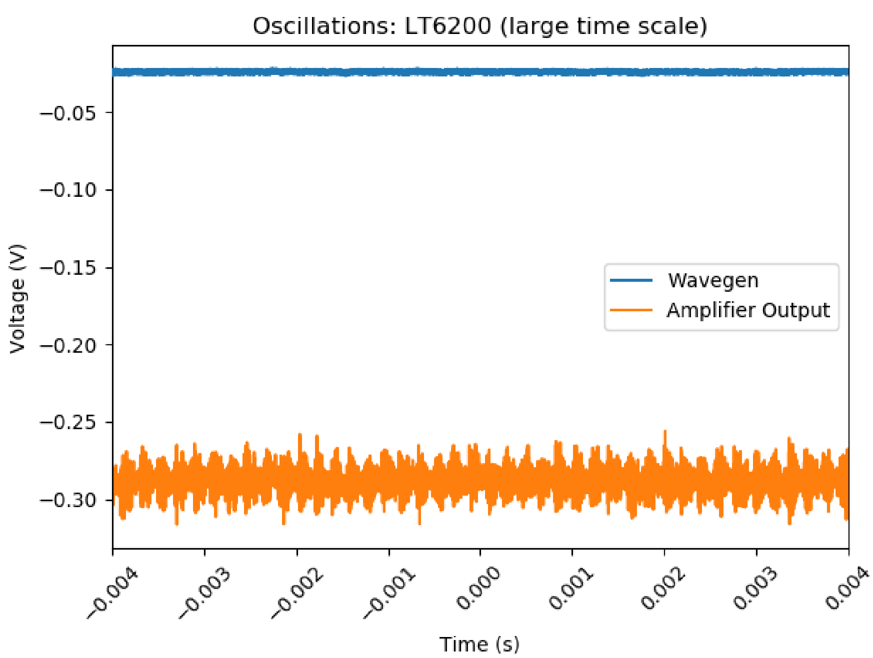

Figure 8: Oscillating LT6200 Amplifier, large time scale

Upon inspection, these oscillations were unlike those seen with the MAX4104. A smaller time-scale of the oscillations are seen in Figure 9.

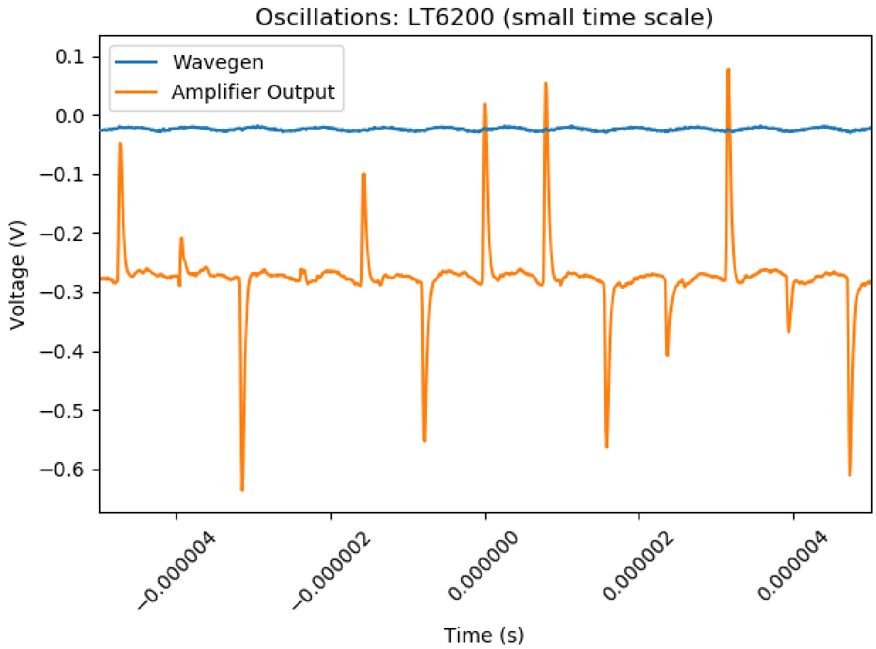

Figure 9: Oscillating LT6200 Amplifier, small time scale

Note that there seems to be a positive or negative spike every 1 µs, or the period of oscillations seems to be 2 µs.

These were suppressed by the addition of a 10 pF feedback capacitor. See Figure 10.

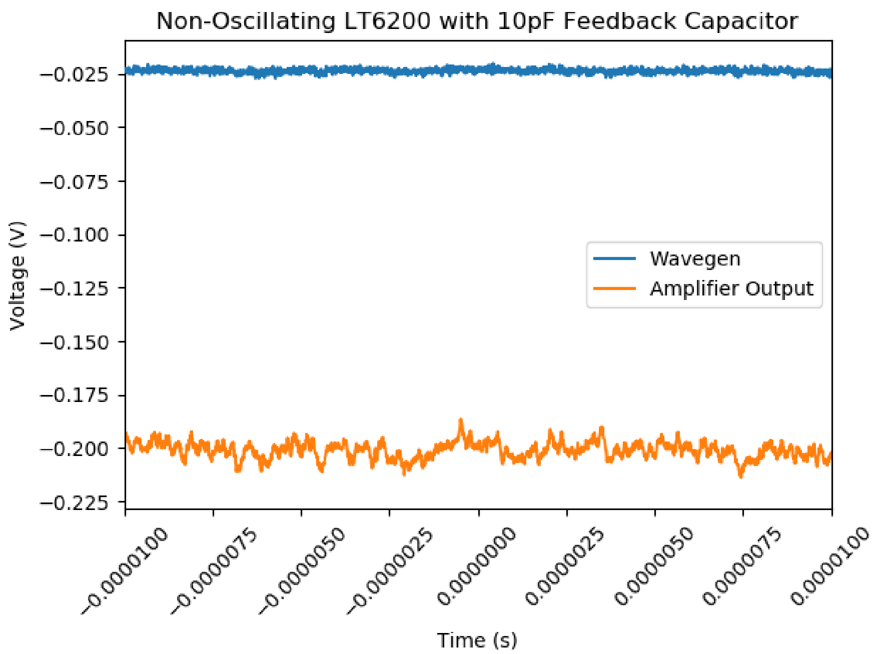

Figure 10: Non-oscillating LT6200 Amplifier with 10 pF Feedback Capacitor
Notice that the output of the amplifier still looks a bit noisey; this is gone in the remainder of testing.

The plots below were made of the response of the amplifier. Additional plots can be found in Appendix: *LT6200 Amplifier: More Plots*.

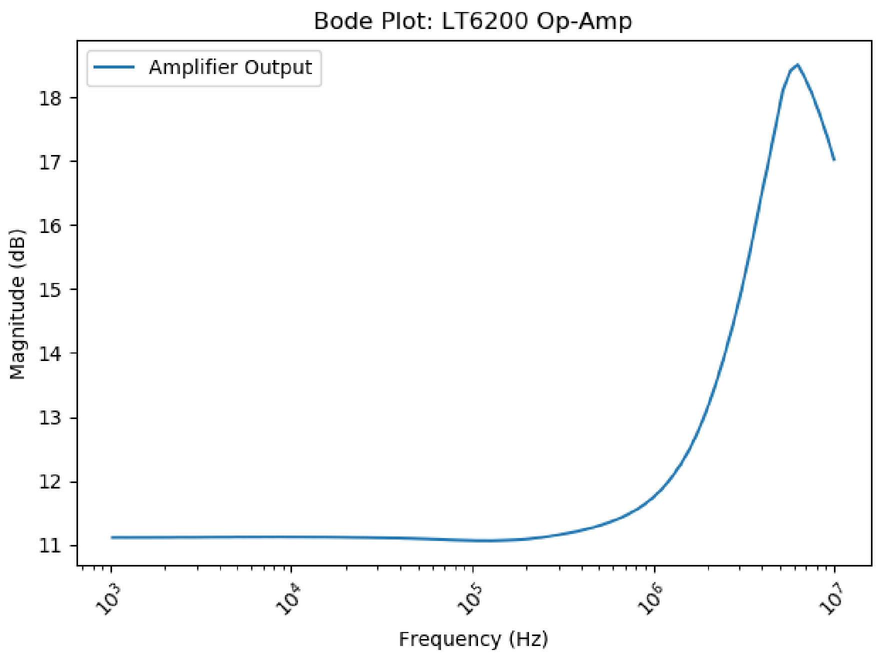

Figure 11: Driven LT6200 Amplifier: Bode Plot

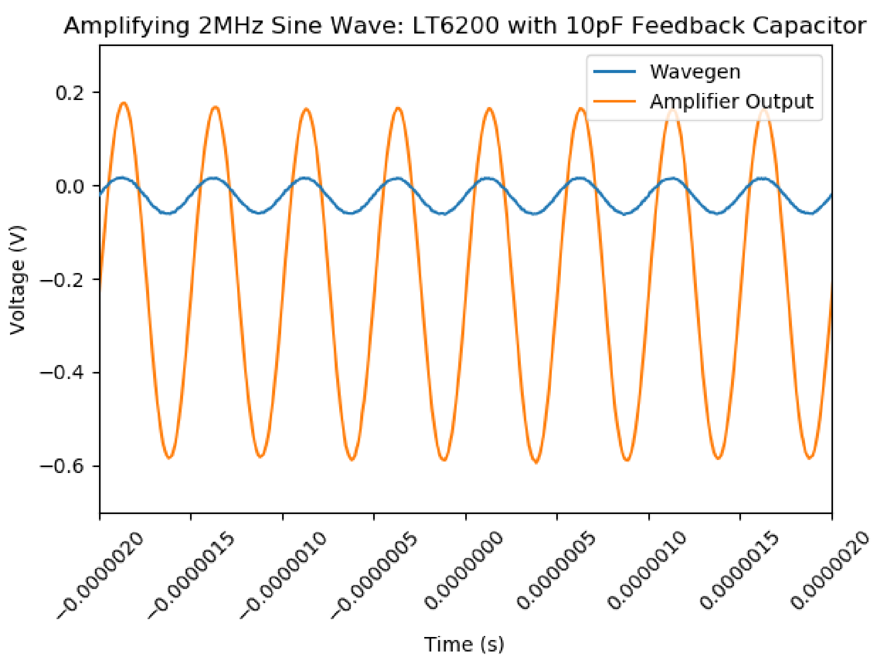

Figure 12: Driven LT6200 Amplifier: Sine Wave

Notice that the offset voltage of the LT6200 amplifier is considerably smaller than the MAX4104 amplifier, (approximately $-0.2\,\mathrm{V}$ vs. approx. $-1\,\mathrm{V}$. This was the smaller negative offset we were looking for when we constructed the amplifier, though further investigation is required to determine if this will ultimately work as an amplifier for the sensor.

4.3.3 Testing Stability: Repeated Inability to Reproduce Stability

In subsequent testing of the LT6200 amplifier, on some occasions we were unable to reproduce the stability of the amplifier that allowed us to collect data for the plots in the previous sections.

No immediate cause for this instability is obvious, as in all previous cases the instability of the amplifier was mitigated and removed by the use of BNC cables to probe and feed the amplifier, and by satisfying the equations discussed in Section 4.1.

5 Conclusion

The use of an off-the-shelf opamp in a particle detector amplifier at fast enough speeds to find interaction energy should be possible. Nonetheless, to bring the knowledge to this point would require further investigation.

To produce an amplifier circuit that is capable of working at the speeds required for energy integration or measurement, the recommendation is to contact a manufacturer and obtain a recommendation on their part for the circuit setup and design. Alternatively, the Electrical Engineering department at UNH may be able to provide some recommendations or advice for the construction of these high speed devices. These resources would serve well to further the project; ultimately this project was limited in our experience with opamps and high-frequency devices.

Apart from that, the project still served to teach a number of useful and important skills, including:

- SMT rework
  - This project required a lot of soldering and unsoldering of devices to test how changes to the feedback loop of an amplifier would affect its output. I had never tried SMT rework, but I found it to be not too hard if one has patience and a magnifying glass.
- PCB layout and design
  - I too had never tried laying out a PCB or what considerations become evident when trying to route your own traces. This project gave me *much* more experience with Eagle than I previously had.
- RF signals and associated familiarity
  - As a subset of not having too much prior experience with Eagle, I didn't have any experience working with high frequencies. This project required me to familiarize myself with some best-practices, as well as consider how to implement these on the PCB itself.
- Noise considerations in circuit

- We had talked all semester about how to minimize noise in a circuit, but it didn't evident just how large an issue noise can be until using the leads (not the coax) from the AD was the source of the oscillations of both the MAX4104 and the LT1226.

  Of course, the addition of the coax solved these problems.

Appendix

A Special Thanks

This project would not have been possible without the support from both the [above-nameless] engineer and technician from the University's Mechanical Engineering department. These two individuals proved to be invaluable in their support, both in recommendations for the project and design of the amplifier itself. These people are:

- Maurik Holtrop
  University of New Hampshire
  Professor, Physics
- Jim Abare
  University of New Hampshire
  Mechanical Engineering Technician
- Aidan Browne
  SensL
  Electrical Engineer

B Important Links

There are numerous important links that could be helpful for students wishing to continue this project in the future. Look at:

- CosmicWatchBeta GitHub: `https://github.com/asher-m/CosmicWatchBeta`
- CosmicWatch Original Project: `http://cosmicwatch.lns.mit.edu/`
- PCB Layout Tool from Mantaro: `http://www.mantaro.com/resources/impedance-calculator.htm`
- Analog Devices, 'Does Your Op Amp Oscillate?': `http://www.analog.com/media/en/technical-documentation/application-notes/an148fa.pdf`

C Various Plots

C.1 MAX4104 Amplifier: More Plots

This is a continuation of Section 4.2.2. Also see Figures 6 and 7.

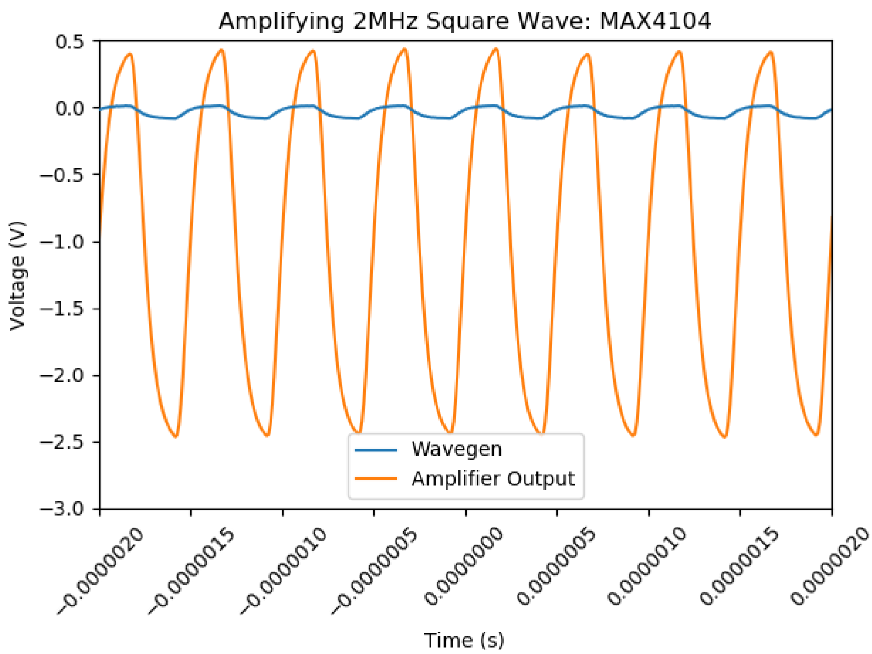

Figure 13: Driven MAX4104 Amplifier: Square Wave

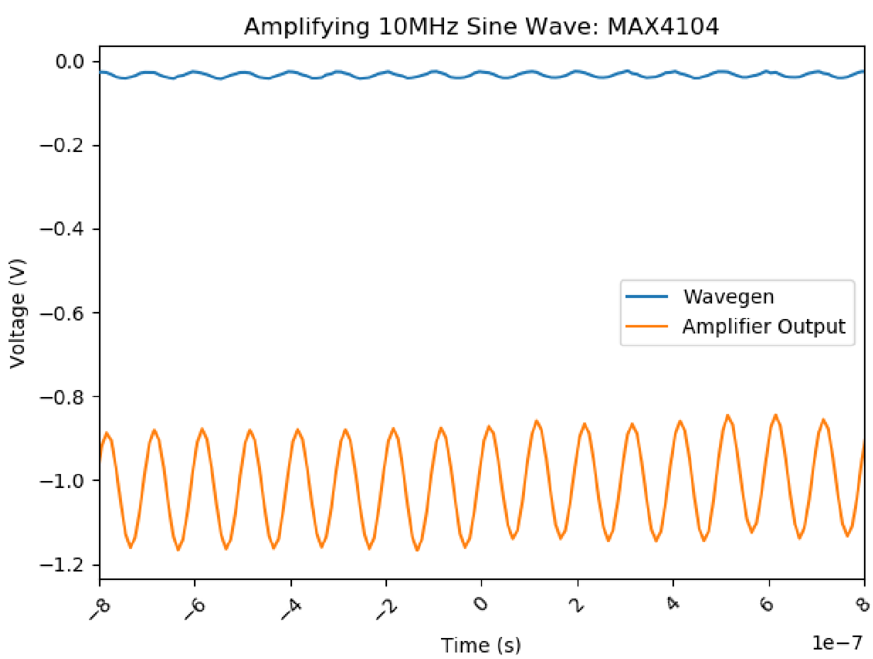

Figure 14: Driven MAX4104 Amplifier: Sine Wave

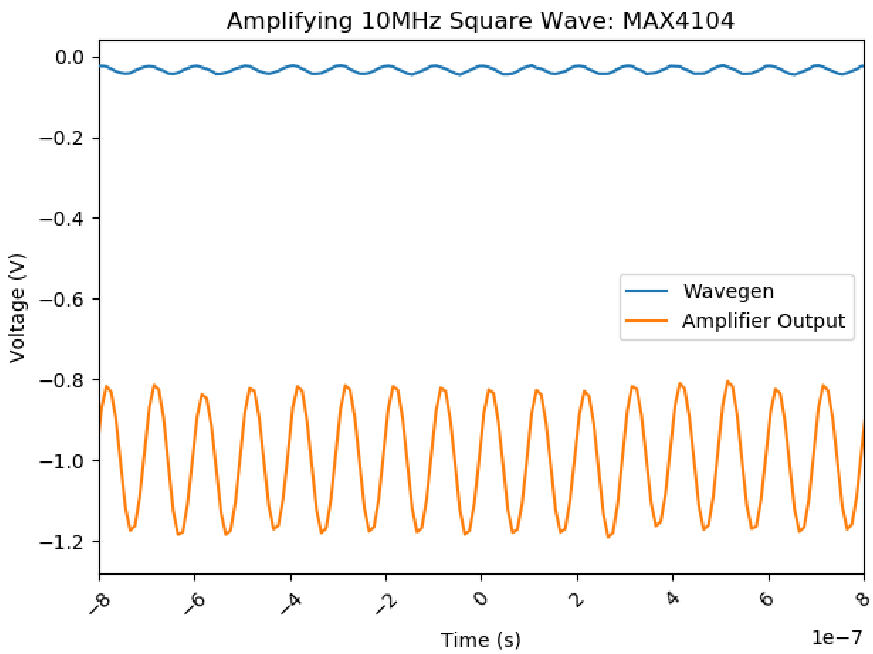

Figure 15: Driven MAX4104 Amplifier: Square Wave

Notice that in both Figures 14 and 15 the apparent gain of the amplifier falls. This must occur because the opamp is approaching unity gain.

Also notice that in the same figures the function generator is distinctly far from the square-wave output it is meant to be producing. This may be because of some input capacitance on the amplifier or simply that the AD cannot achieve clean functions at these speeds. Additionally, it should also be noted that this 10 MHz *is* the fastest that the AD is rated for; perhaps this degradation in signal quality is simply that the AD cannot reach these speeds.

C.2 LT6200 Amplifier: More Plots

This is a continuation of Section 4.3.2. Also see Figures 11 and 12.

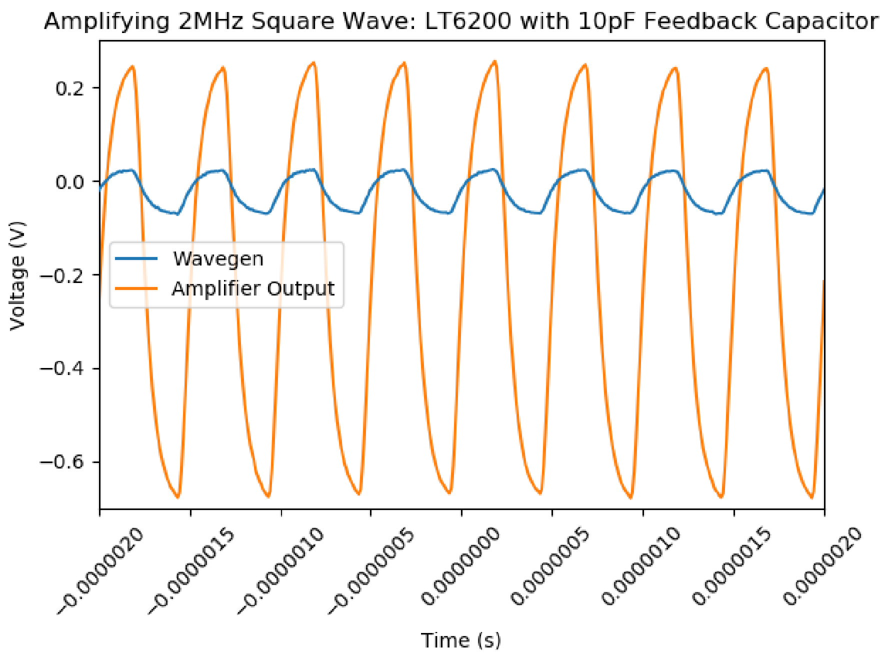

Figure 16: Driven LT6200 Amplifier: Square Wave

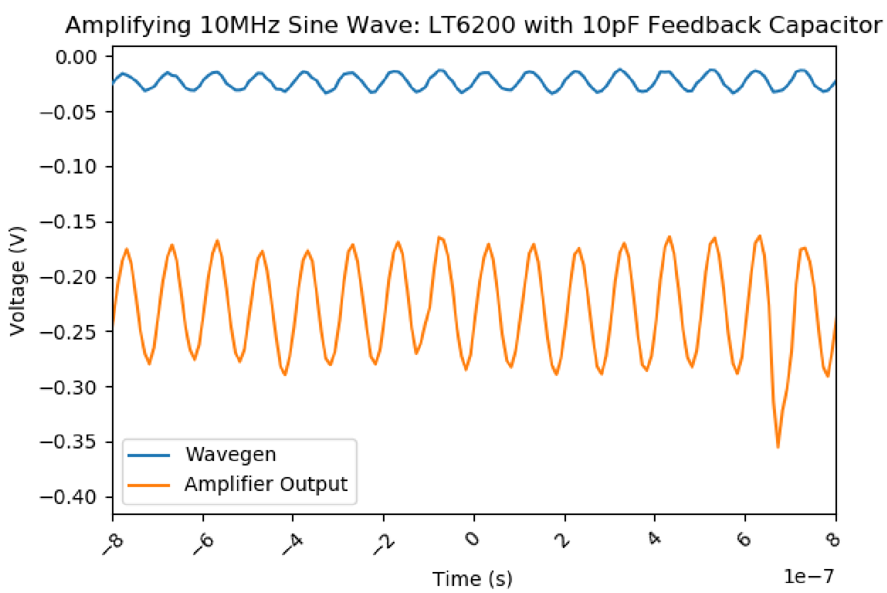

Figure 17: Driven LT6200 Amplifier: Sine Wave

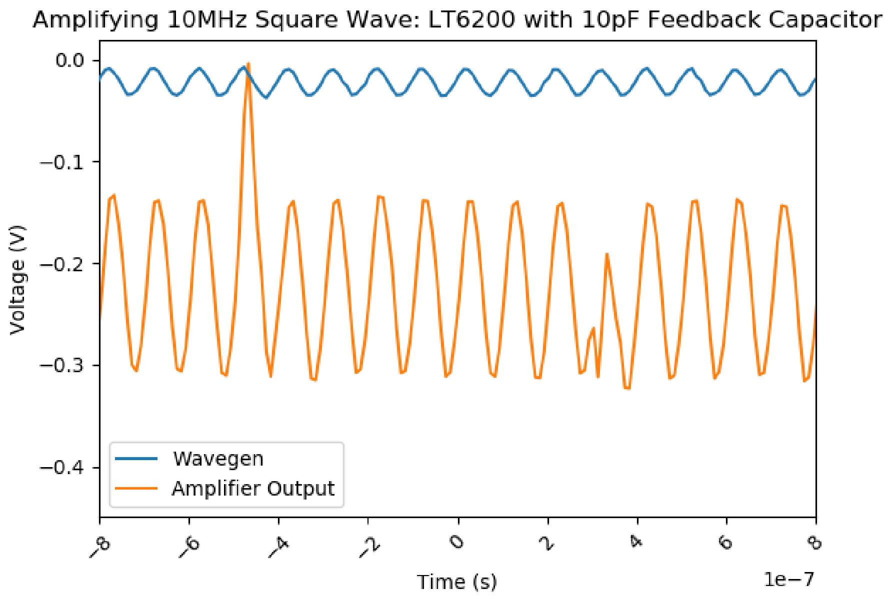

Figure 18: Driven LT6200 Amplifier: Square Wave

Notice again that the square waves in each of the above figures are not square. Again this is likely due to input capacitance of the opamp.

D Detector Schematic and Board

Below are the detector schematic and board:

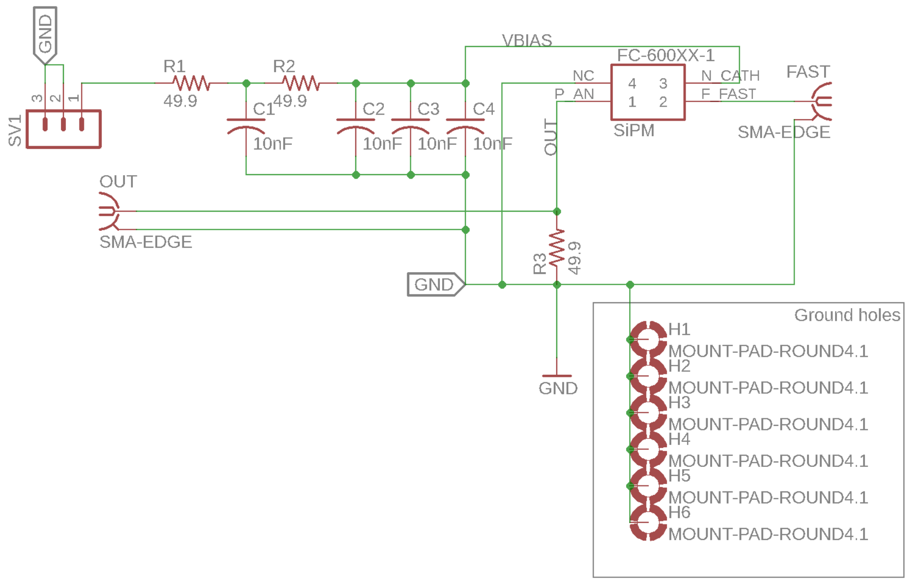

Figure 19: Detector Schematic

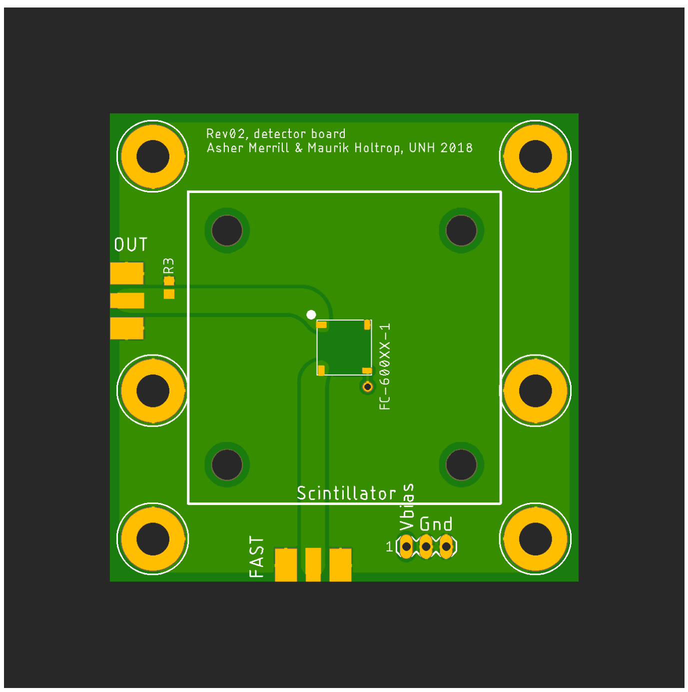

Figure 20: Detector PCB top

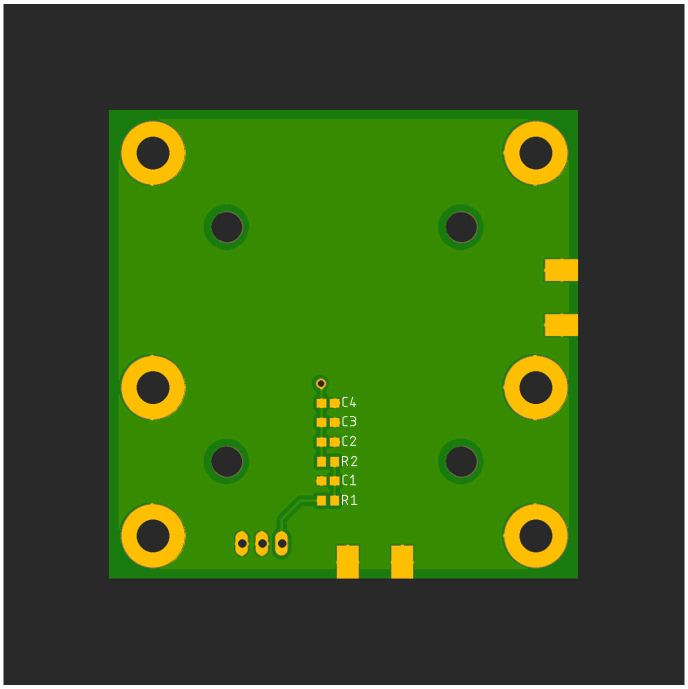

Figure 21: Detector PCB bottom