

Lab 1 Simple Electronic System

Group Name: KFC

Group Member: Renwei Chen, Shuo Feng and Xujie Kan

Student ID:11911736

Date: September 26, 2022

The main objective of this lab is to understand the basic components of a typical electronic system. To this end, we will build a simple electronic system shown in Fig. 1.

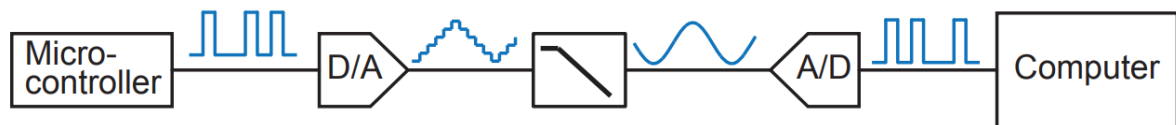


Fig. 1: The system in Lab 1.

The core of this report is in the final part, i.e. "**5 Procedures**".

1 Objectives

Understand the voltage regulators, micro-controllers, ADC, DAC, serial programming interface, and the use of the printed circuit board.

2 Deliverables

In this part, a table with all the items to be submitted are listed.

3 Prelabs

In this part, the introduction covers all the models listed below:

Micro-controllers, voltage regulators, Digital to Analog Converter (DAC) and Analog to Digital Converter (ADC), serial interface, printed circuit board design, PCB assembly technology and so on.

4 Equipment and Supplies

- 1 × bench power supply.
- 1 × bench oscilloscope.
- 1 × bench signal generator.
- 1 × bench multimeter.
- 1 × Texas Instruments breadboard.
- 1 × box of breadboard jumper wires.
- 1 × Teensy 4.0 development board.
- 1 × LM317 IC.

- 1 × 8-DIP MCP4921 DAC IC.
- 1 × LT1009 precision reference IC.
- 1 × 14-DIP TL974IN quad Op-Amp IC.
- misc resistors. Provided by the TA.
- misc capacitors. Provided by the TA.
- 1 × 2k potentiometer
- 1 × 20k potentiometer.
- 1 × small screwdriver.
- 1 × Analog Discovery.

5 Procedures

Our group firstly used the bread board to build the system. This is a easy way to check for the mistakes and to varyify whether the system can operate correctly. The overall system set up on the bread board is shown in Fig. 2.

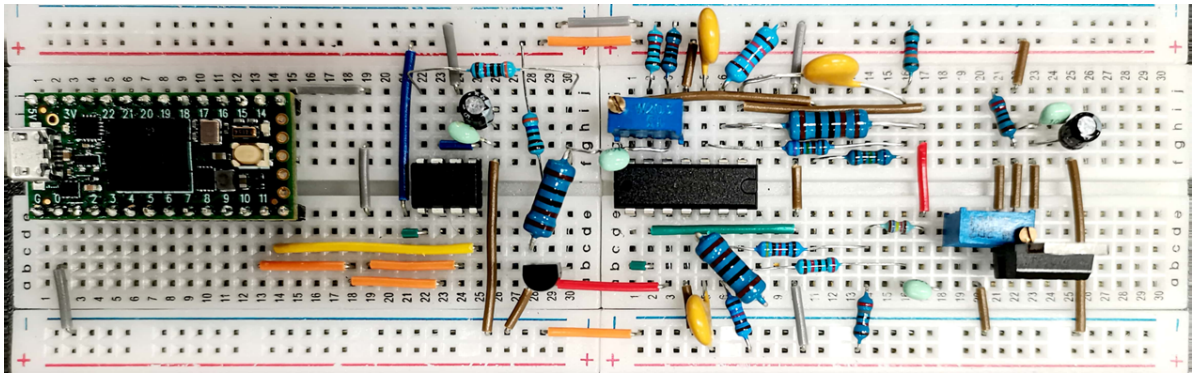


Fig. 2: The overall system set up on the bread board.

5.1 Power supply/voltage regulator

In this system, we choose LM317 as the IC regulator. The schematic of the voltage regulator circuit using LM317 is shown in Fig. 3.

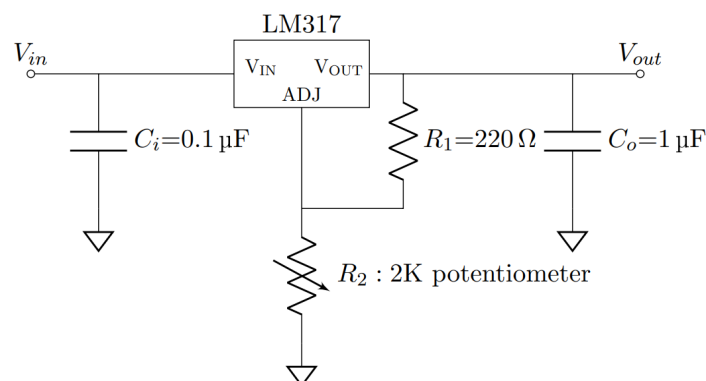


Fig. 3: The schematic of the voltage regulator circuit using LM317.

In our breadboard, as shown in Fig. 4, the voltage regulator is in the right part.

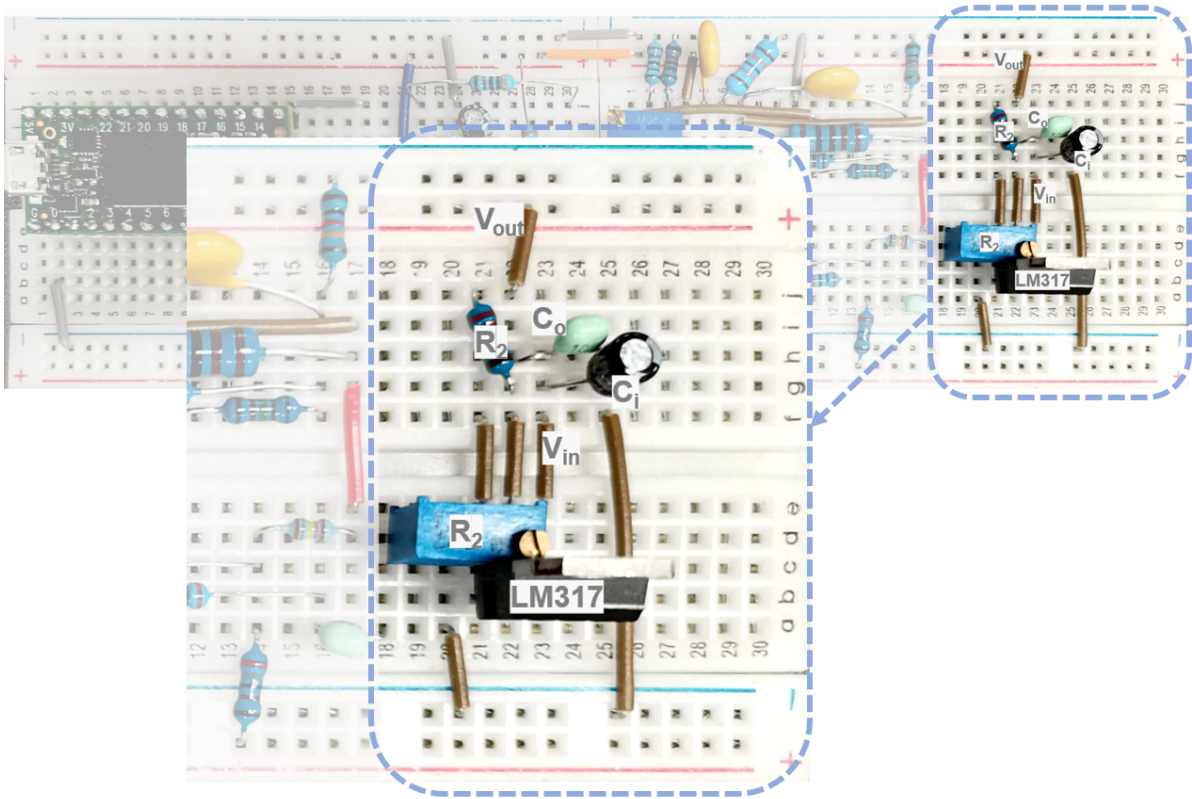


Fig. 4: Our breadboard layout of the circuit in Fig. 3.

For testing the circuit, we will first use the lab bench power supply as our input power. Set the power supply voltage to 8 V and connect it to the input of your voltage regulator circuit. Adjust the R2 potentiometer until the output voltage is 5 V. You can use the lab multimeter to measure the output voltage.

Calculate the Line regulation:

(a) Adjust the input voltage from 8 V to 5 V at 0.25 V intervals, and then from 5 V to 2 V at 0.5 V intervals, record the regulator output voltage using the multimeter;

Answer: The output voltage is shown in the following table.

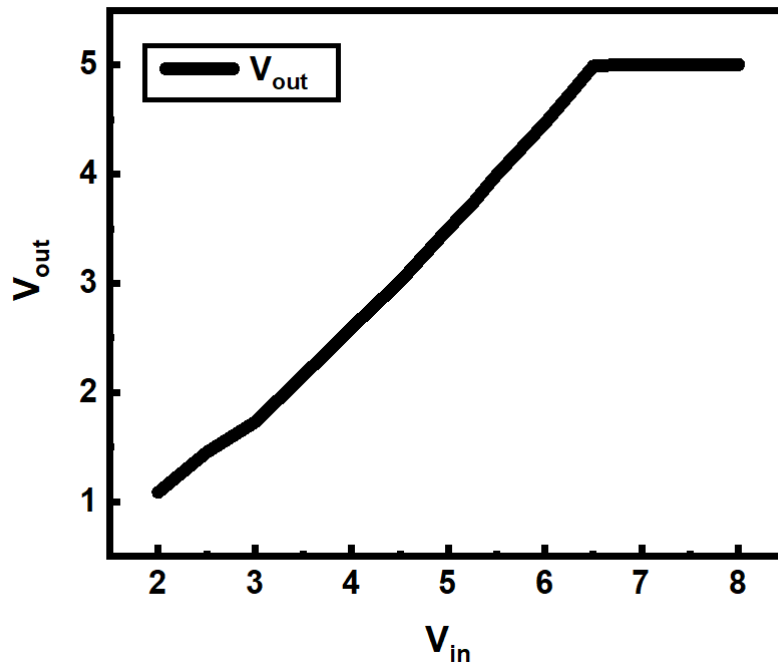
Vin/V	Vout/V
8	5.005
7.75	5.005
7.5	5.005
7.25	5.004
7	5.004
6.75	5.004
6.5	4.991
6.25	4.725
6	4.468
5.75	4.231
5.5	3.993
5.25	3.73
5	3.501
4.5	3.015
4	2.593
3.5	2.155
3	1.728
2.5	1.452
2	1.086

(b) Plot your results. In what input voltage range does the LM317 provide good line regulation?

Answer:

A good line regulation range is when the input voltage in 6.5V ~ 8V.

The plot is shown below:



5.2 Precision voltage reference

Build circuit shown in Fig. 10 on your breadboard. Here, we use one of the op-amps in the TI TL974IN quad op-amp IC as a voltage follower (buffer) to provide sufficient current drive capability. A pinout diagram of TL974IN is shown in Fig. 5.

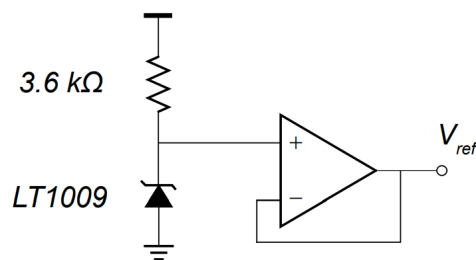


Fig. 5: Schematic of the LT1009 precision voltage reference circuit.

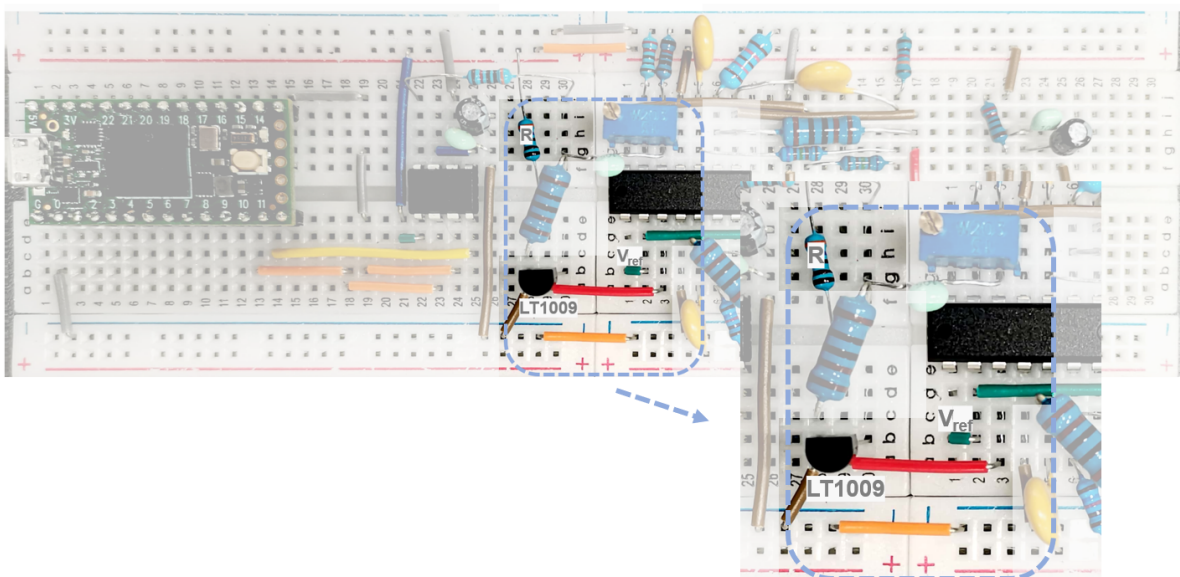


Fig. 6: Our breadboard layout of the circuit in Fig. 5.

5.3 Function generator

In this lab, we will try to emulate a DDS using a micro-controller and a DAC IC. We are interested in generating a triangle wave which will be useful in later labs. Since DAC outputs are discrete in nature, we are really just approximating the triangle wave with a stair-case waveform. Obviously, the higher the resolution of the DAC, the better this approximate is.

It is very important to keep the circuit (ICs, resistors, capacitors, wires, etc) organized!

Signals on oscilloscope

Requirement: In your report, include a screen capture of both the triangle (at VOUT) and sync (at SYNC) output signals on the oscilloscope. Record their amplitudes and periods.

Modify your code to **set the triangle wave amplitude to 2.5 V**. Hint: change the gain setting of the MCP4921.

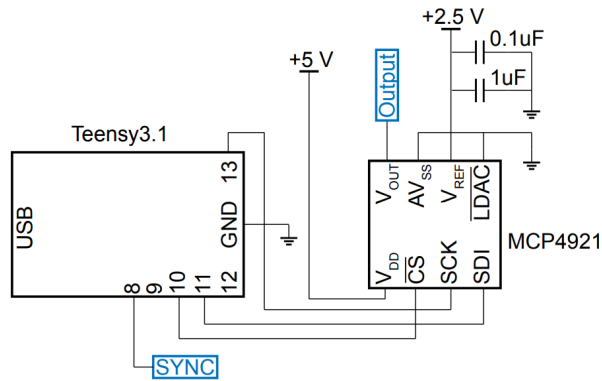


Fig. 7: Schematic of the LT1009 precision voltage reference circuit.

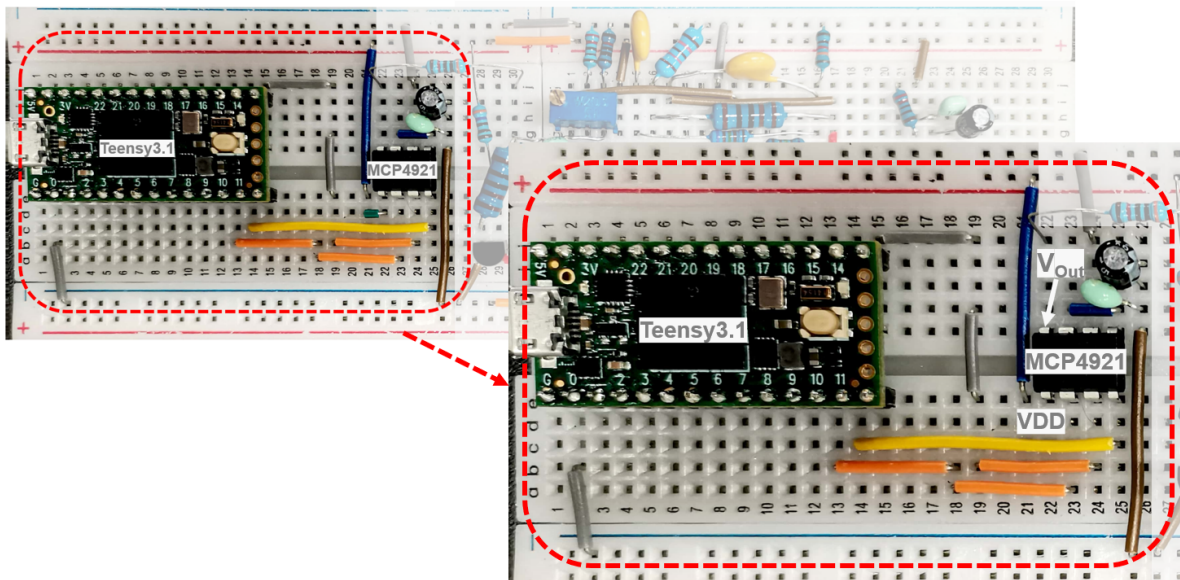


Fig. 8: Our breadboard layout of the circuit in Fig. 7.

Answer:

The screen capture of both the triangle (at VOUT) and sync (at SYNC) output signals on the oscilloscope is shown below (Fig. 9).

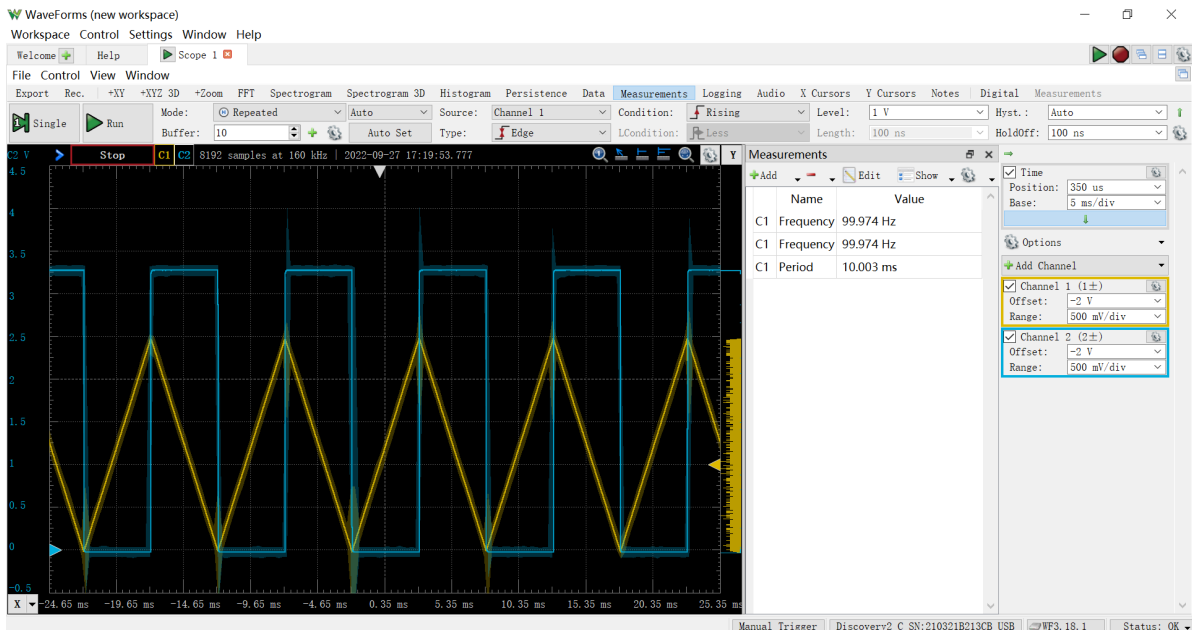


Fig. 9: screen capture of both the triangle (at VOUT) and sync (at SYNC) output signals.

Question 7

How can you modify the code to change the amplitude and period of the output waveform? What is the fastest triangle wave you could generate? What do you think is the limitation to going even faster?

Answer:

For the amplitude control:

```
if (outputvalue == 0 || outputvalue == 4096){
    incr = -incr;
    digitalWrite(SYNC, !digitalRead(SYNC));
}
```

we can adjust the **outputValue** boundary, in the above block, the value is 4096.

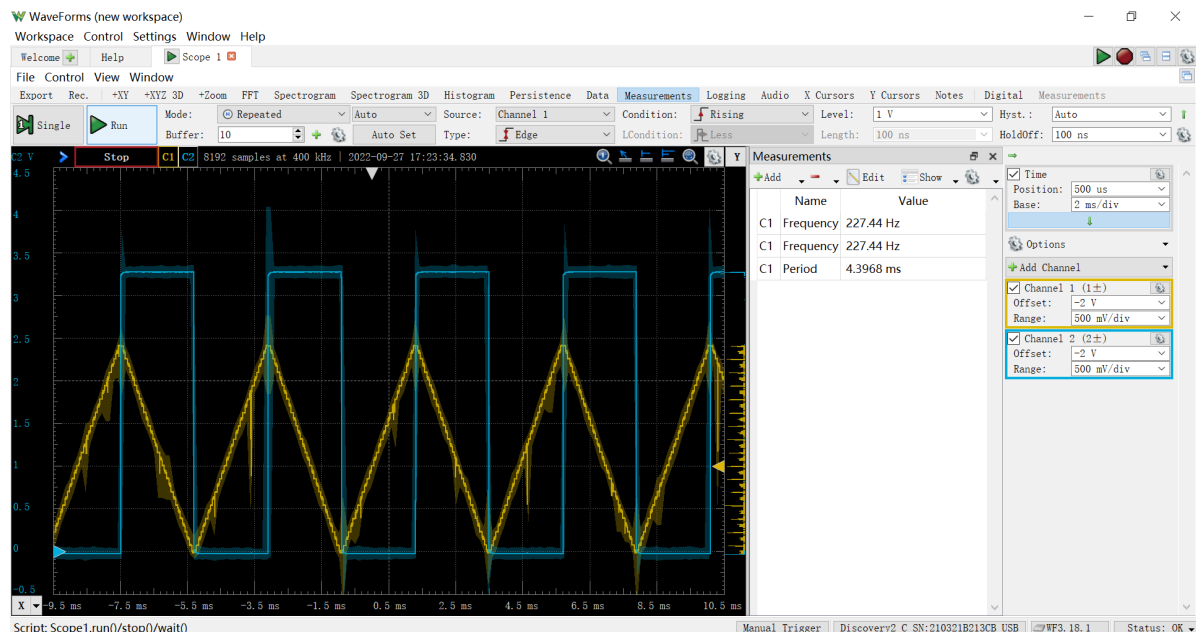
For the period control:

```
void setup()
{
    // Set pins for output
    pinMode(SYNC, OUTPUT); // SYNC pin
    digitalWrite(SYNC, LOW); // sync pulse low
    pinMode(slaveSelectPin, OUTPUT); // Slave-select (SS) pin
    SPI.begin(); // Activate the SPI bus
    SPI.beginTransaction(SPISettings(20000000, MSBFIRST, SPI_MODE0)); // Set up
    the SPI transaction; this is not very elegant as there is never a close
    transaction action.
}
```

① we can adjust the **speed**, i.e., the first place in **SPISettings()**. In the block above, the value is 20000000.

② Or we can increase the increment value, here in our code, is **incr**.

By adjusting the period, we find that the fastest triangle wave is about 227Hz, as shown in the following figure captured by the oscilloscope.



The DAC output frequency remains almost constant (reaches saturation) when the frequency is further increased. So we think that the limitation of the frequency is due to insufficient bandwidth of DAC.

Question 8

Modify the code to generate a sinusoidal wave. What is the highest frequency sinusoidal wave you can generate? The following link may give you some hint. <http://interface.khm.de/index.php/lab/experiments/arduino-dds-sinewave-generator/>

Answer:

our modified code:

```

/*
Triangle wave and sync pulse generator to control a (0-5V input range) VCO for
FMCW radar.
The MPC4921 DAC is used to generate a triangle wave with a period of 40ms.
PWM of the Arduino UNO is use to simultaneously generate the sync pulse,
used for signal processing.

*/

#include <SPI.h> // Include the SPI library

word outputValue = 0; // A word is a 16-bit number
double incr = 0.01;
double x = 0;

const int slaveSelectPin = 10; //set the slave select (chip select) pin number
const int SYNC = 8; //set the SYNC output pin number

void setup()
{
    // Set pins for output
    pinMode(SYNC, OUTPUT); // SYNC pin
    digitalWrite(SYNC, LOW); // sync pulse low
    pinMode(slaveSelectPin, OUTPUT); // Slave-select (SS) pin
    SPI.begin(); // Activate the SPI bus
}

```



```

    SPI.beginTransaction(SPISettings(100000000, MSBFIRST, SPI_MODE0)); // Set
up the SPI transaction; this is not very elegant as there is never a close
transaction action.
    //SPI.beginTransaction(SPISettings(20000000, MSBFIRST, SPI_MODE0)); // Set
up the SPI transaction; this is not very elegant as there is never a close
transaction action.
}

void loop()
{
    x = x + incr;

    outputValue = 2048+ floor(2048*sin(x));

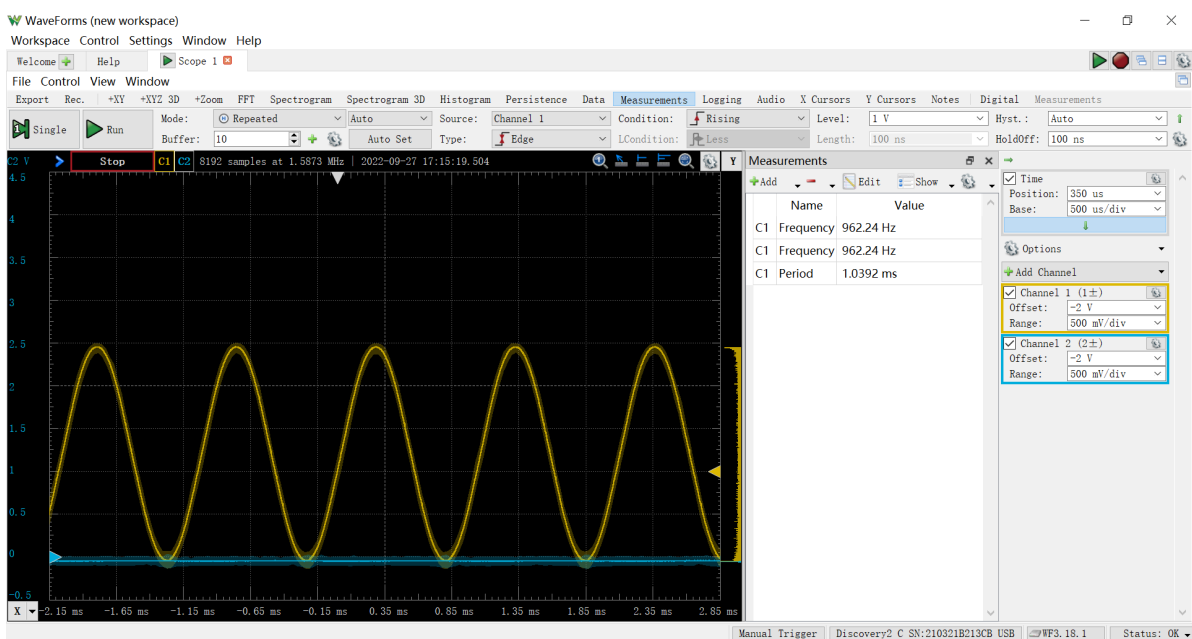
    if (x == 6.28){
        x = 0;
    }

    byte HighByte =highByte(outputValue); // Take the upper byte
    HighByte = 0b00001111 & HighByte; // Shift in the four upper bits (12
bit total)
    HighByte = 0b00110000 | HighByte; // Keep the Gain at 1 and the
Shutdown(active low) pin off
    //HighByte = 0b00010000 | HighByte; // Keep the Gain at 1 and the
Shutdown(active low) pin off
    byte LowByte = lowByte(outputValue); // shift in the 8 lower bits

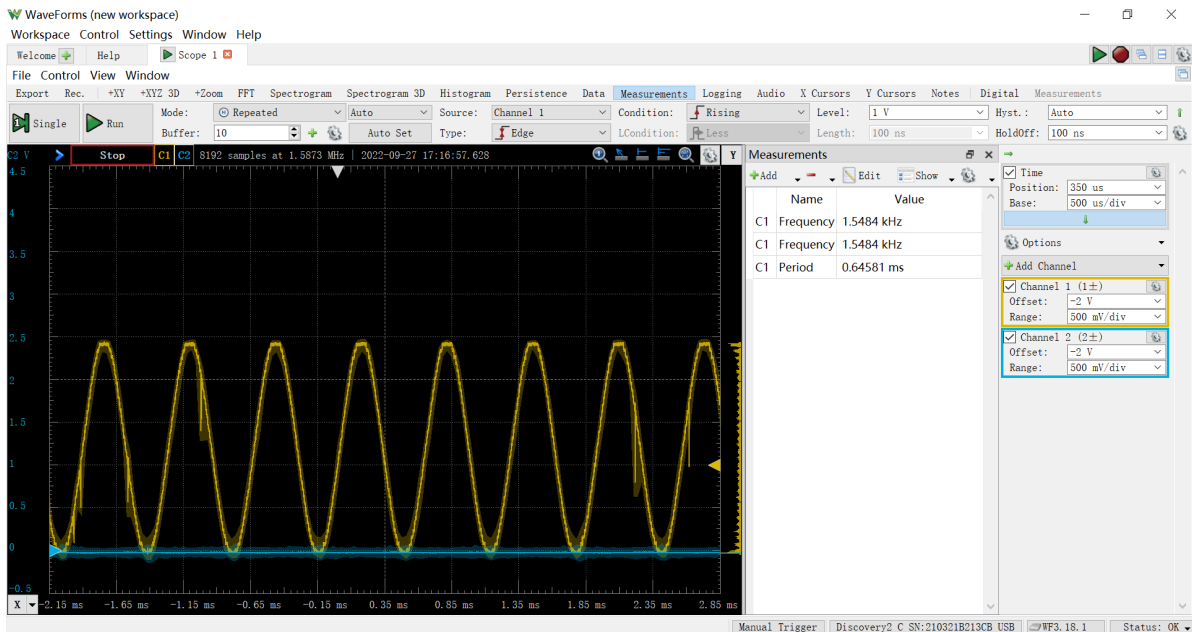
    digitalWrite(slaveSelectPin, LOW);
    SPI.transfer(HighByte); // Send the upper byte
    SPI.transfer(LowByte); // Send the lower byte
    digitalWrite(slaveSelectPin, HIGH); // Turn off the SPI transmission
}

```

The screen shot from the oscilloscope of the sinusoidal wave is shown below:



By adjusting the period (speed), the highest sinusoidal wave frequency is 1.5484 kHz.



5.4 Active low-pass filter

In this part of the lab, we will implement an active low pass filter (LPF) with an adjustable gain. In the final radar system, the active LPF will be responsible for filtering out spurious signals and amplifying the desired received baseband radar signals. We will use the TI TL9741N quad Op-Amp for both the gain stage and the active LPF. Fig. 10 shows the schematic of the active LPF circuit. The filter has a cut-off frequency of 15 kHz and was designed with the TI WEBENCH filter design tool.

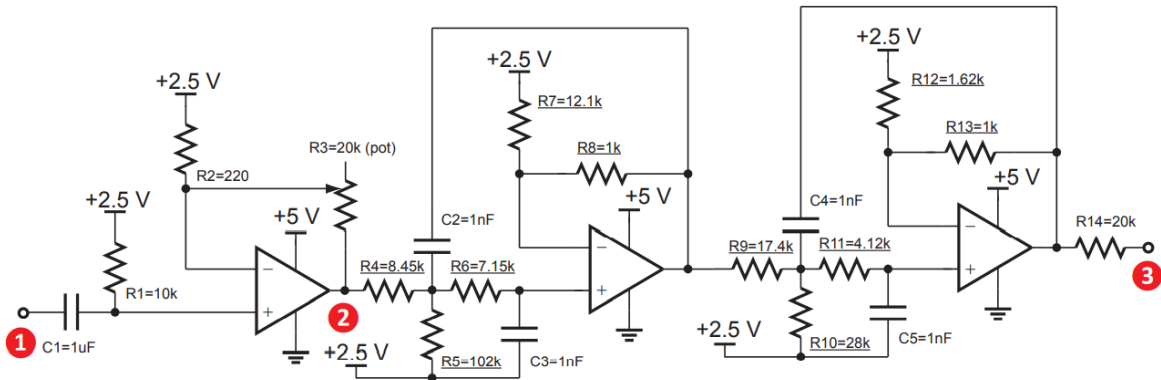


Fig. 10 Schematic of the active low-pass filter. Underlined resistors values are 1% tolerance.

5.4.1 Gain stage

Question 1

Identify the gain stage (between point 1 and point 2 Fig. 10). What is the expression of its gain? Notice that the +2.5V supply does not affect this result, rather it simply sets the bias of the overall amplifier for maximum output swing because a single 5V supply (battery) is used vs a dual-rail (both positive and negative) supply.

Answer:

$$gain = \frac{V_{out}}{V_{in}} = \frac{R_2 + R_3}{R_2}$$

Question 2

Build the gain stage on your breadboard.

(a) Power the op-amp chip by using 5 V for VDD and ground for VSS.

(b) You can use the 2.5 V LT1009 reference as the bias for the input and output.

(c) The gain is controlled using a 20 k Ω potentiometer (pot). The pot can be tricky to use when you don't know where exactly the tap is. Measuring and adjusting the pot to a desired starting point before you put it on the breadboard will make your life much easier. For example, you can start with 880 Ω which will give you a voltage gain of 5 to start with.

(d) Use caution when handling the TL974 Op-Amp. As with all ICs, it is sensitive to electrostatic discharge, high voltage, and high current. Before you apply power to the circuit, always double check that

the supply voltages is set up correctly;

the bias voltage is set up correctly;

the load is not shorted or connected to too low of an impedance.

We have limited supplies of these ICs, so if you end up destroying yours, your lab progress may be delayed significantly.

Question 3

To test the circuit, generate a 1 kHz 100 mVpp sine wave signal using the lab signal generator and connect the signal to the input of the gain stage circuit. Measure the output of the circuit using the lab oscilloscope. What is the maximum peak-to-peak voltage at the output before clipping occurs? What is the gain at this point?

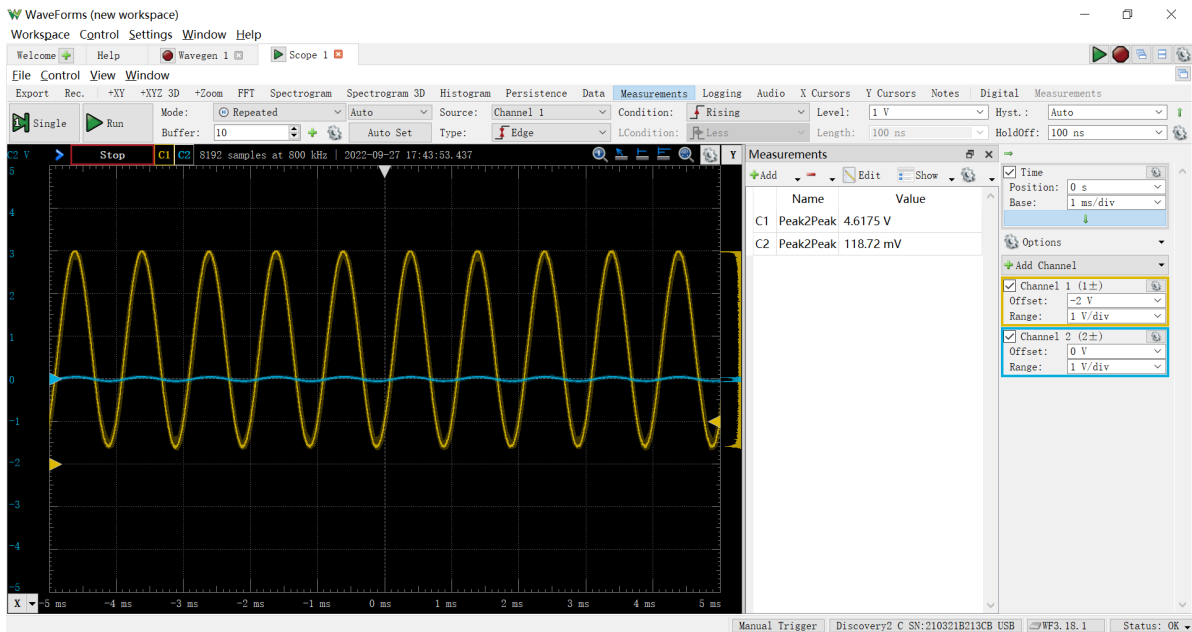
Answer:

The screen shot of the output wave as well as the input wave from the oscilloscope is shown below:

The highest peak-to-peak voltage is 4.6175 V.

The gain at this point is:

$$Gain = \frac{V_{2pp}}{V_{1pp}} = \frac{4.6175V}{119.72mV} = 38.569 = 31.7dB$$



Question 4

Adjust the potentiometer to have an output voltage swing of 3 V_{pp} for the same input as in the above step. Tabulate and plot the gain from 100 Hz to 1 MHz while keeping the pot fixed. Find the 3 dB cutoff frequency or the 3 dB bandwidth at this gain. Use at least 20 data points. What is the gain-bandwidth (GBW) product of the gain stage?

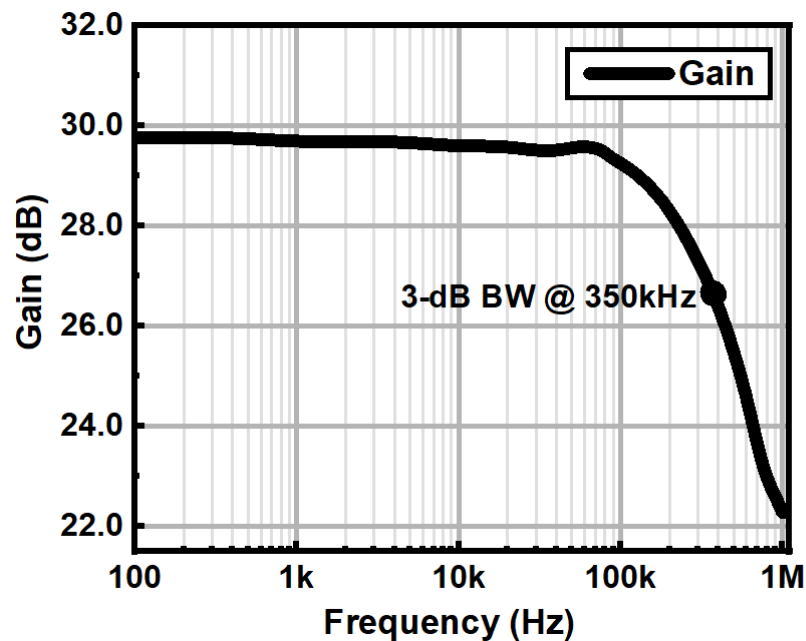
Answer:

frequency	gain (dB)
100	29.77
500	29.74
1000	29.69
5000	29.66
10000	29.6
20000	29.57
40000	29.51
80000	29.46
90000	29.34
100000	29.25
200000	28.3
300000	27.31
400000	26.4
500000	5.48
600000	24.56
700000	23.64
800000	22.98
900000	22.61
1000000	22.28

The plot is shown below, the 3-dB cutoff frequency is about 35 kHz.

Since the Gain Bandwidth is a constant for a certain system, we choose the point $f = 350 \text{ kHz}$, Gain = 26.7 dB in the following plot. Then we get the GBW:

$$GBW = 7569.45 \text{ kHz}$$



5.4.2 Active LPF

Question 1

Identify the active LPF portion of the circuit (between point 2 and point 3 in Fig. 15). The overall filter consists of 2 low pass filters. What is the order of the overall filter and how is it determined from the schematic?

Answer:

The order of the overall filter is 4 order. This is because of there are four capacitors in the overall filter.

Or we can get the transfer from the overall filter, by applying property of the virtual short and virtual open for the ideal amplifiers. The transfer function is:

$$H(s) = \frac{R_5(R_7 + R_8)}{s^2 C_2 C_3 R_4 R_5 R_6 R_7 + s[(R_5 R_6 + R_4 R_6 + R_4 R_5) C_3 R_5 - R_4 R_8 C_2] + R_7(R_4 + R_5)}$$

$$\times \frac{R_{10}(R_{12} + R_{13})}{s^2 C_4 C_5 R_9 R_{10} R_{11} R_{12} + s[(R_{10} R_{11} + R_9 R_{11} + R_9 R_{10}) C_5 R_{10} - R_9 R_{10} R_{13} C_4] + R_{12}(R_{10} + R_9)}$$

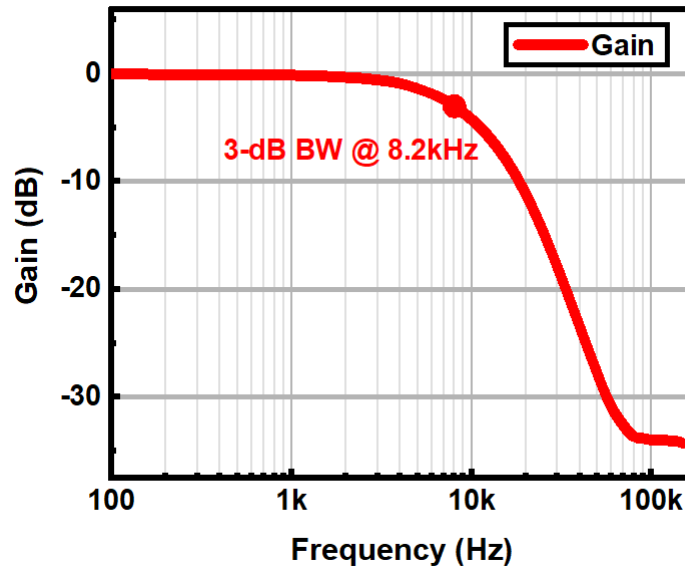
Question 2

To measure the filter response only, input a 3 Vpp signal through a series 1 μF capacitor at point 2 in Fig. 15 and measure the output through R14 (at point 3 in Fig. 15). Tabulate and plot the frequency response from 100 Hz to 150 kHz. What is the cutoff frequency of the filter?

Answer:

frequency	Gain(dB)	Gain
100	-0.027	2.99
200	-0.056	2.98
500	-0.085	2.97
800	-0.114	2.96
1000	-0.144	2.95
2000	-0.322	2.89
3000	-0.567	2.8
4000	-0.911	2.68
5000	-1.375	2.56
6000	-1.864	2.42
8000	-2.972	2.13
9000	-3.61	1.98
10000	-4.24	1.84
80000	-11.158	0.83
30000	-17.944	0.38
40000	-23.519	0.2
50000	-27.813	0.122
60000	-30.952	0.085
70000	-32.64	0.07
80000	-33.69	0.062
90000	-33.83	0.061
100000	-33.98	0.06
110000	-33.98	0.06
120000	-33.98	0.06
130000	-34.12	0.059
140000	-34.12	0.059
150000	-34.27	0.058

One can get the cutoff frequency from in below figure, which is roughly 8.2 kHz.



Question 3

How would one increase or decrease the cutoff frequency?

Answer:

By change the value of the resistors and the capacitors in the topology which form the LPF. This action actually change the poles and zeros in the system and therefore change the cutoff frequency. The larger value might result in the lower bandwidth.

5.4.3 Gain Stage + Filter

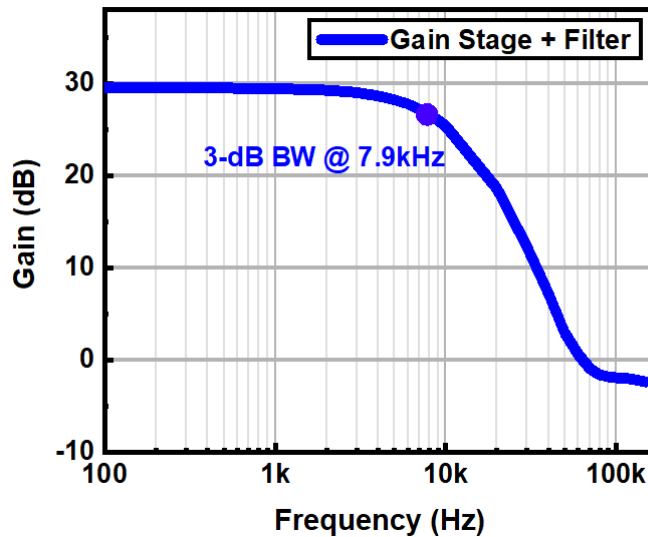
Question 1

Reconnect R4 (R14 is still disconnected) and measure the overall response of the gain stage + active LPF. Use a 100 mVpp sine wave as the input.

Answer:

frequency	Gain(dB)	Gain
100	29.57	3.01
200	29.54	3
500	29.51	2.99
800	29.48	2.98
1000	29.45	2.97
2000	29.28	2.91
3000	29.04	2.83
4000	28.66	2.71
5000	28.23	2.58
6000	27.78	2.45
8000	26.69	2.16
9000	26.02	2
10000	25.39	1.86
80000	18.59	0.85
30000	12.26	0.41
40000	7.23	0.23
50000	2.92	0.14
60000	0.588	0.107
70000	-0.915	0.09
80000	-1.618	0.083
90000	-1.83	0.081
100000	-1.938	0.08
110000	-1.938	0.08
120000	-2.047	0.079
130000	-2.158	0.078
140000	-2.27	0.077
150000	-2.38	0.076

The plot is shown below with 100 mVpp input for the Gain Stage + Filter.



Question 2

Adjust the pot for a 3 V_{pp} output at 1 kHz. Measure the frequency response from 100 Hz to 150 kHz and record the result. What is the 3 dB cutoff frequency? Compare the results to previous measurements. Plot all of the results for the report.

Answer:

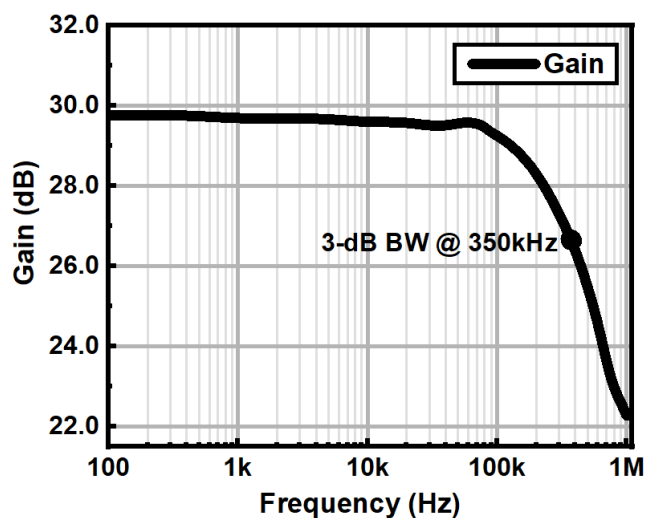
The frequency response is shown above (blue curve). The profile of the transfer function of the "Gain Stage + Active LPF" in **section 5.4.3** and transfer function of the "Active LPF" in **section 5.4.2** are quite similar.

The 3-dB cutoff frequency is 7.9 kHz.

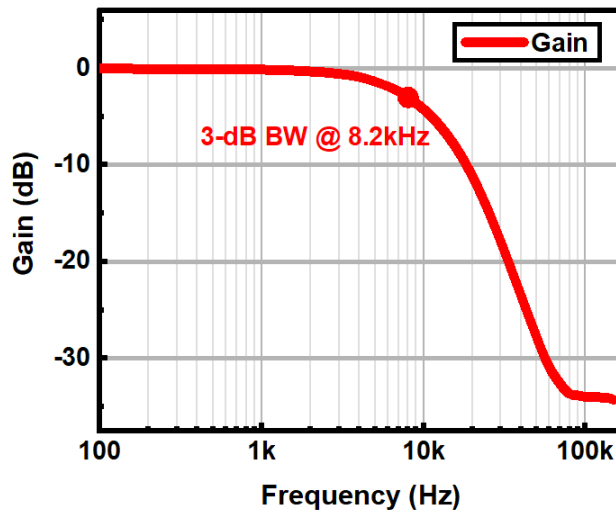
This is because that the bandwidth of the overall filter system (Gain Stage + Active LPF) is limited by the LPF. Thus, with the cascaded stages, the overall system has a bandwidth of 7.9 kHz, slightly reduced from the LPF stage BW, 8 kHz.

Here comes a summary of the transfer function:

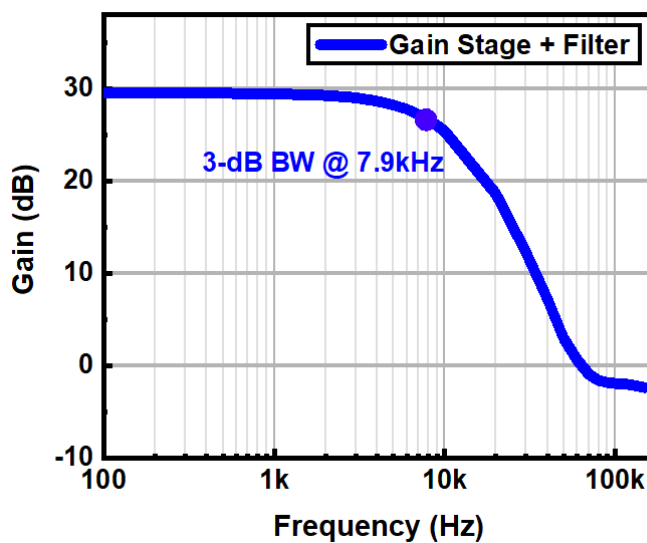
1. Gain Stage



2. Active LPF



3. Overall Filter (Gain Stage + Active LPF)



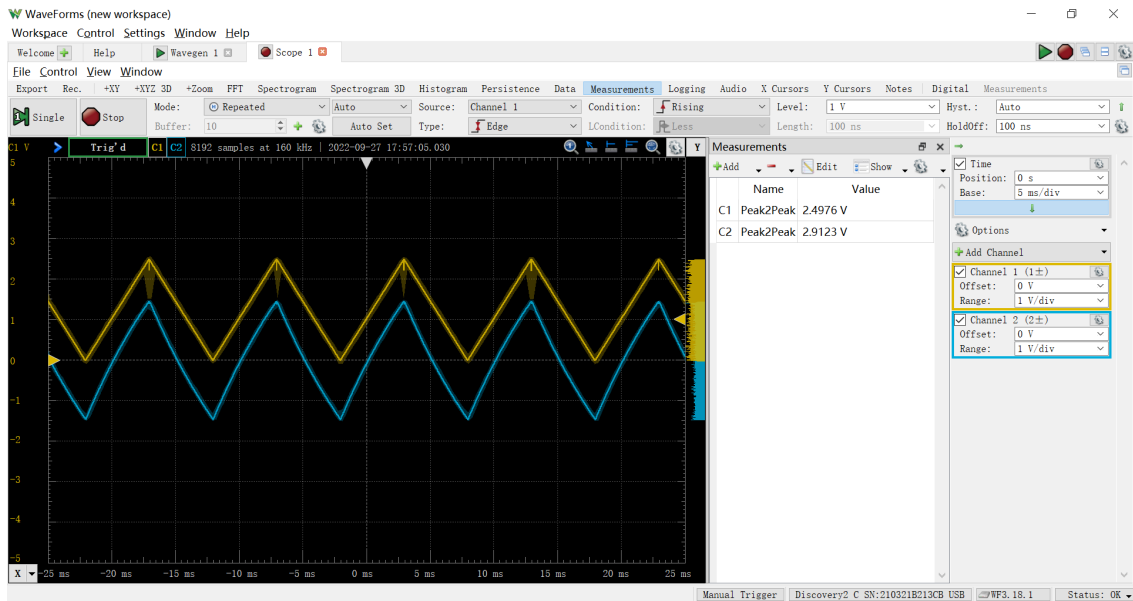
5.5 Acquiring and analyzing analog signals

The Analog Discovery is a USB-powered electronic measurement instrument. It combines the functionalities of an oscilloscope, a logic analyzer (sampling and analyzing digital signals), a function generator, and a dc power supply in a very small form factor. You should read the Analog Discovery 2's reference manual for more information: <https://digilent.com/reference/test-and-measurement/analog-discovery-2/reference-manual>.

1. Set up the system according to Fig
2. Program the Teensy with "triangle teensy audio"
3. Verify that the output of the LPF does not exceed 3 Vpp.

Answer:

The blue one shows the Output of the LPF does not exceed 3 Vpp.



4. Connect the output of the LPF to the "1+" pin of the Analog Discovery. Connect the ground reference of the LPF to the "Ground" pin. A pin out diagram of the Analog Discovery is shown in Fig. 19.
5. Connect the Analog Discovery 2 to your computer using the USB cable.
6. Open the WaveForm software. Open a "Scope" window to display the measured signal. It should look like the following.
7. Open a "Spectrum" window to display the spectrum of the measured signal.

Answer:

The blue one is the spectrum of the DAC output. And the yellow one is the overall system output.

