Last update: 2023A 20/10/22

Experiment #7 – operational amplifiers

Objectives

Understanding op amps: applications, performance, properties.

Maximum time allotted for this experiment: **10 hours**.

Recommended sources

1. Sedra Adel, S. Kenneth, C. Smith. Microelectronic Circuits, 6^{th} Ed., New York, Oxford. Chap 2, 12.
2. Millman, C., C. Halkias. Integrated Electronics Chap 15, 16.
3. “VIDEO: Experiment 1: Operational amplifiers” on our Moodle website.

Mandatory reading/viewing (files available on our Moodle website)

1. “VIDEO: Using the oscilloscope – the basics”
2. “INFOBIT - How to measure Slew Rate”

Your personal tour guide

- In your preparatory report, you will analyze the circuits used in the experiment.
- The purpose of the purple text is to give you an overview of what you’re doing – why are you analyzing this circuit? How does it relate to the other circuits? What is the purpose?

Preliminary questions and simulations

Note: in your simulations you will use the LM741 amplifier, where the V-, V+ voltage supply terminals are disconnected. Make sure to connect them +/-15V DC supply voltage when simulating, e.g. as depicted in the following figure:

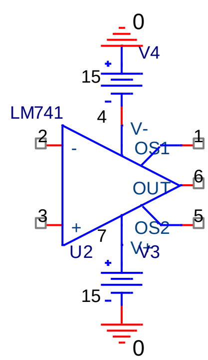


Figure 1

- Depending on your PSPICE version, you may want to use the model “uA741” in library “EVAL”, in case “LM741” doesn’t work.

Last update: 2023A 20/10/22

1. General questions

1.1. Use the Op Amps data sheets (TL071M, TL061M, LM741/LM741A), available on the course website, and summarize their properties in a table.

1.1.1. OpAmp model & manufacturer

1.1.2. Input offset voltage, V_{io}

1.1.3. Large signal voltage gain / differential voltage amplification

1.1.4. Common-mode rejection ratio (CMRR)

1.1.5. Slew-rate (SR) at unity gain

1.1.6. Bandwidth (ω_{-3dB}), unity-gain bandwidth (ω_t) or gain-bandwidth product (GBW)

2. Using the document "how to read an op-amp data sheet" and your knowledge from the "analog circuits" course, explain each of the above properties (up to two lines per item). Pay close attention to the differences between the items in subsection 1.1.6 and their usage in the three datasheets.

- In the experiment, you will have a circuit board with one of the above op-amps installed. **The goal of the experiment is to deduce which op-amp is installed** from measurements of the above properties.

- In this preliminary report, we will follow some of the sections and measurements you will perform in the lab. There, you will wholly rely on your answers here.

- Note: all your simulations incorporate a LM741 op-amp. Since you do not know which op-amp you will end up with, treat your results here qualitatively.

- In some of the following sections you will be required to choose numerical values for hand-calculations or simulations. If you have a lab partner, the last three digits of your and your partner's ID number will hence be referred to as "ABC" and "DEF", respectively. If you are attending the lab alone, use your last six digits, i.e. "ABCDEF".

3. Consider the following types of voltage definitions: Vamplitude, Vmax, Vmean, Vpp, VRMS.

3.1. For a given input signal, $v_{IN} = (DEF/100)\sin(\omega t)$, calculate each of the above.

3.2. Repeat the previous section with $v_{IN} = 10 + (DEF/100)\sin(\omega t)$.

3.3. Repeat the previous section with $v_{IN} = -10 + (DEF/100)\sin(\omega t)$. Note the sign of your result for VRMS.

4. Input offset voltage

- Refer to the experimental procedure, section 2.

Last update: 2023A 20/10/22

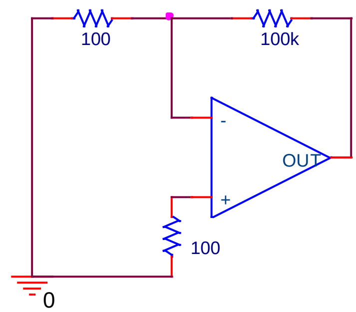


Figure 2

4.1. If we assume an op-amp is ideal in the sense of input offset voltage, $V_{io} = 0$, then its output voltage would be zero. Observe the op-amp circuit in Figure 2. Assume that V_{out} is equal to ABC millivolts, recall the op-amp lecture from the course "Analog Circuits", and redraw the circuit with an implementation of the non-ideal op-amp, where $V_{io} \neq 0$.

4.2. According to the previous section, calculate V_{io}. In the lab, you are required to deduce V_{io}; However, assuming the electrical noise level in the lab is roughly 50mV, is it even measureable? Explain here how will you find V_{io} (see experiment section 2).

- Once you've realized there is a DC offset in your circuit's output, it is only natural to try and compensate for it, setting it to zero. This can't be done using the method you've learned in class – in reality you never connect an additional source "$-V_{io}$" to the inputs!

- Instead, recall that the first stage in an op-amp is a differential amplifier. The asymmetry present in this stage is that which usually causes the output DC offset. Let's try to fix it by introducing an opposite asymmetry.

Consider the following circuit, representing the first stage in an op-amp (for the BJTs, use model Q2N2222):

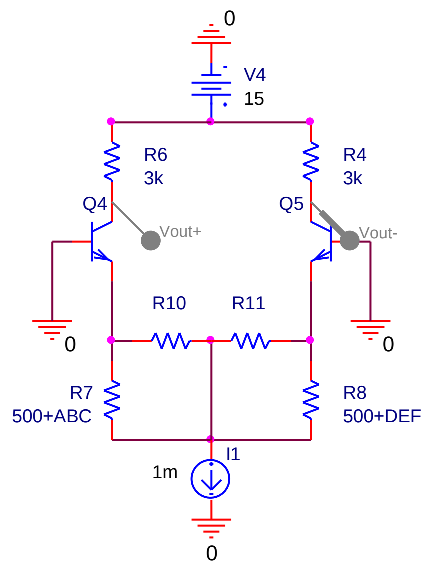


Figure 3

4.3. Set $R_7 = 500 + ABC$ and $R_8 = 500 + DEF$. At this point, R_{10}, R_{11} are DISCONNECTED from the circuit. Run a PSPICE bias point simulation and write down the DC values Vout+ and Vout- (no print needed). The difference between R_7, R_8 should produce an offset between the outputs ΔV_{out} – it emulates any kind of non-ideality that may result from asymmetry in the circuit.

4.3.1. Suppose you now add a potentiometer $R_{10} + R_{11} = 10k\Omega$ as depicted above. What is the ratio, $0 < R_{10} : R_{11} \leq 1$ required to set the output to zero? (you may use a computational software to solve this mathematical problem; there's no need for elaborate hand-calculations. Note though, that a simple solution is possible using symmetry considerations!).

4.3.2. Simulate the circuit with your choice for R_{10}, R_{11}, and make sure the difference between the outputs has been compensated (no print needed).

4.3.3. Suppose you couldn't set the potentiometer exactly, and change the ratio $R_{10} : R_{11}$ by 10%. Simulate again, and attach a print. Write down the output difference, $\Delta V_{out}^{10\%}(10k\Omega)$.

In the lab, it is not certain that you will have a 10kΩ potentiometer.

5. CMRR

Refer to the experimental procedure, section 3.

5.1. Define the figure of merit "CMRR" and write down two different expressions – in decibels and without units. Show the calculations for differential mode gain A_{dm} given "CMRR"=60+AB dB and $A_{cm} = DEF/1000[V/V]$.

In the lab, you will build two circuits in order to measure CMRR; these are depicted in Figure 4 and Figure 5.

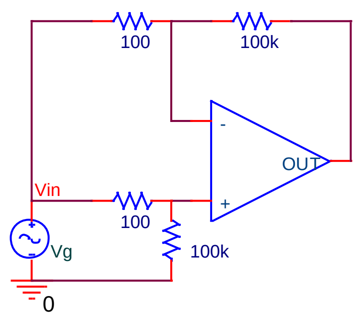


Figure 4

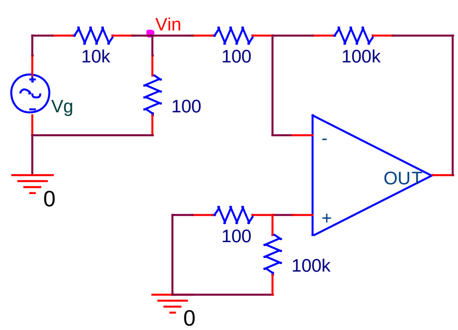


Figure 5

5.2. Calculate the transfer functions for both circuits, $A_4 = \frac{v_{out}}{v_{in}}$ and $A_5 = \frac{v_{out}}{v_{in}}$, assuming the op-amps are ideal. Which circuit will you use for measuring A_{cm} and which for A_{dm}? Write down the CMRR using A_4 and A_5.

5.3. In the lab, the op-amps will not be ideal. Explain: in which range will you choose your input amplitude for the circuit in Figure 4, 2-4VRMS or 50m-300mVRMS?

5.4. Repeat the previous section for the circuit in Figure 5. Why are the $100\Omega, 10k\Omega$ resistors connected between v_g and v_{in} absolutely required for an accurate gain measurement?

6. Slew-rate and Full-power bandwidth

- Refer to the experimental procedure, section 4.
- Both of these quantities give a measure to whether a signal passing through an op-amp will become distorted. You should review "Sedra & Smith", 6th ed., section 2.8, "Large-signal operation of op-amps" , to answer the following.
- Following the formulation in the book, an important note to remember: our op-amps may be modeled by a single-pole transfer-function.

6.1. Write down the formulas for slew-rate (SR) and full-power bandwidth. Define the relationship between them and explain the term "full-power bandwidth" and its usage.

6.2. What is the difference between the effect of SR and the effect of a finite (rather than ideal, infinite) bandwidth, on the output signal of an op-amp? Give a short explanation on the electronic origin of each.

6.3. Given the op-amp in Figure 6, with $SR = [ABC/100] V/\mu s$ and bandwidth $BW = [1 + DEF/100] MHz$,

6.3.1. Given $v_g = a \cdot \sin(2\pi f t),\ f = 1kHz$ What is the maximal input amplitude $a_{\max}$ for which the op amp's output could still follow its own SR? (Note: here you assume SR is the only limiting effect)

6.3.2. Calculate the full-power bandwidth for DC supply voltages of $\pm 2V$, $\pm 5V$. Your calculation results might not be feasible in the lab, but theoretically acceptable.

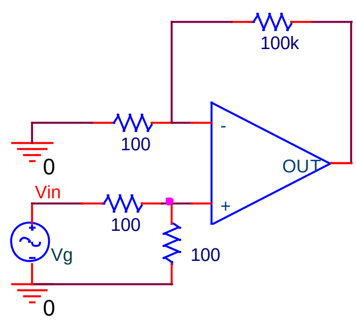


Figure 6

- You will use the circuit in Figure 6 to measure slew-rate. In the lab, you will choose a square-wave as an input signal. Here, you will now explain how to measure slew-rate using a step-function.

6.4. Suppose that a capacitor is connected to Figure 6's output. Your input is defined as an ideal step function: $V_g(t<0)=0$, $V_g(t>0)=a$. Qualitatively draw the output of the circuit for different values of a where the circuit is not limited by SR, and for several values of a where it is limited by slew-rate.

6.5. In the lab, the scope's screen is 10cm wide, divided to ten 1cm squares. What scale (sec/cm) would you use for the horizontal time-axis in order to see SR comfortably?

6.6. On the basis of your answer in the previous section, explain here how you would measure SR in the lab. Hint: while in theory you can measure SR by changing frequency, it would not work in the lab, because in higher frequencies there are additional mechanisms distorting the signal.

7. Gain-bandwidth product

- Refer to the experimental procedure, section 5.
- To answer the following questions, you might want to read Sedra & Smith, 5th ed. section 2.5, or 6th ed. section 2.7, titled "Effect of finite open-loop gain and bandwidth".
- At the end of this preparatory report, there is a section that teaches you how to perform certain measurements, such as bandwidth. Make sure you are well-prepared by reading it and answering the questions there!

7.1. It has been mentioned previously that we assume our op-amps have a dominant (single) pole. Write down the general single-pole transfer function, and draw a qualitative semi-logarithmic graph of the gain versus frequency. Note that the open-loop bandwidth of op-amps is only several Hz!!

7.2. Explain the following claim: "the unity-gain bandwidth and the gain-bandwidth product are identical if and only if one assumes the single-pole model".

7.3. Assume a DC-gain of $[25+AB]dB$ and a pole at $[50+DE]kHz$. Draw these numbers on your graph and calculate the slope, the bandwidth, and the unity-gain bandwidth. What is the GBW for these parameters?

7.4. For the parameters in the previous section, find the gain at frequency $10*[50+DE]kHz$ and the frequency where the gain is $0.1*[25+AB]dB$.

Observe the following circuits:

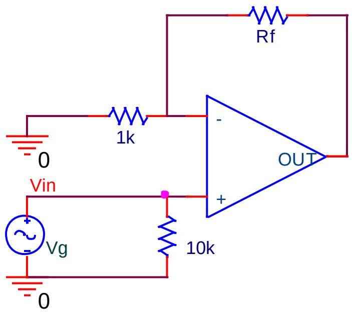


Figure 7

$$\frac{V_{out}(s)}{V_{in}(s)} = \frac{1 + R_2 / R_1}{1 + \frac{1}{A_{OL}}\left(1 + \frac{R_2}{R_1}\right) + \frac{s}{\omega_{3dB}}}$$

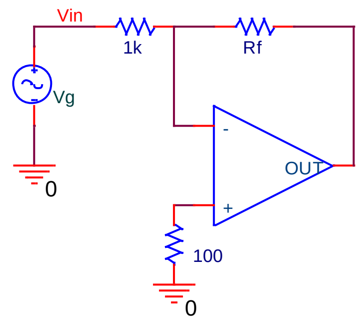


Figure 8

$$\frac{V_{out}(s)}{V_{in}(s)} = \frac{- R_2 / R_1}{1 + \frac{1}{A_{OL}}\left(1 + \frac{R_2}{R_1}\right) + \frac{s}{\omega_{3dB}}}$$

Assuming the op-amps have a single pole and a finite open-loop gain, $A_{OL}[V/V]$, the frequency-dependent transfer functions are written above, where the 3dB frequency is given by $\omega_{3dB} = \omega_t / (1 + R_2 / R_1)$. Referring to the figures above, $R_2 = R_f$ and $R_1 = 1k\Omega$.

7.5. Use the expressions above, and write down the gain at DC, multiplied by the bandwidth, symbolically, for both circuits. This is in fact the Gain-BandWidth (GBW) product.

7.6. Now assume A_{OL} is very large, and find the GBW for each circuit.

7.7. What condition must be met so that the two GBW you found will be approximately equal?

8. Open-loop gain

- Refer to the experimental procedure, section 6.
- The open-loop gain A_{OL} of op-amps is usually in the order of ~100,000x (i.e. ~100dB). This means that if you connect to the inputs a voltage difference of 50mV, which is roughly the electrical noise level in our lab, then the output would be 5,000V!
- Considering that the voltage supplies are +/-15V, the above, of course, will not happen.
- Conclusion: you cannot connect a small-enough voltage to the inputs in order to directly measure the open-loop gain. So how will we do it??

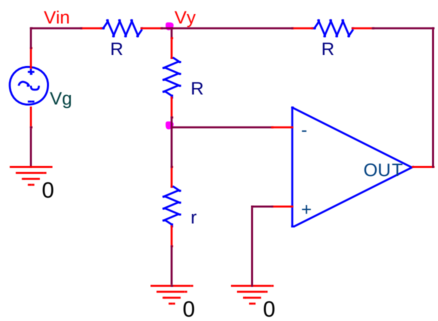


Figure 9

$$A_{OL} = \frac{1}{k}\left(\frac{V_{in}}{V_y} - 3\right),\ k = \frac{r}{r+R}$$

8.1. Calculate the ratios $\frac{V_y}{V_g}$ and $\frac{V_o}{V_g}$ for an ideal Op-Amp.

8.2. For a Non-ideal Op-Amp as given in figure 9, which voltage points could you measure in the lab to find A_{OL} and why? Describe the input signal (shape, frequency and amplitude) required for this measurement and the expected measured signals.

✓ Hint: in the lab, the op-amp is not ideal in many respects, which is why you cannot use your *approximate* derivation of vout/vin for finding out AOL.

Let us assume that the signal in V_y is as small as the measurement noise, explain how the following Scope functions could help: Averaging, BW limit, Fine.

9. Op-amp as an integrator

- Refer to the experimental procedure, section 8.
- While the title says "integrator", here you will see that this claim is frequency-dependent: the circuit will not operate as an "ideal" integrator in all frequencies! So when does it work as "expected"? Let's see!

We will investigate the following circuit:

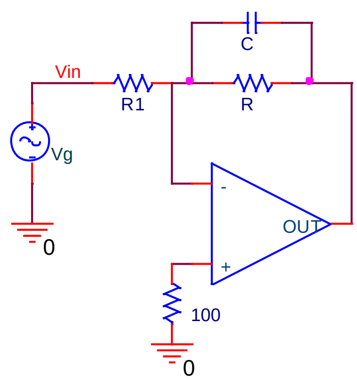


Figure 10

9.1. Assuming the op-amp is ideal, write down the frequency-dependent transfer function. How many poles and zeros does this system have? Make sure your final result is in the canonical form, i.e. $H(s) = \frac{V_{out}(s)}{V_{in}(s)} = H_0 \frac{(\omega_{z1}+s)\cdot(\omega_{z2}+s)\cdot..}{(\omega_{p1}+s)\cdot(\omega_{p2}+s)\cdot..}$. What is the expression for the dominant pole, ω_p?

9.2. Suppose that you input a signal with frequency much smaller or larger than the circuit's dominant pole. Find two approximations to the transfer-function for these cases, i.e. find $H(\omega << \omega_p)$ and $H(\omega >> \omega_p)$. Then, for a square-wave input of frequency ω, 9.2.1. Qualitatively draw the output signal for the case $H(\omega << \omega_p)$. What is the action that the circuit performs on the input?

9.2.2. Qualitatively draw the output signal for the case $H(\omega >> \omega_p)$. What is the action that the circuit performs on the input?

9.3. Given $R_1 = 10k\Omega$, $C = 10nF$ and $R = 20k\Omega$, simulate the bode plot of the circuit and find f_p (i.e. ω_p in units of Hz).

9.4. Using the results from the simulation, find the -3db frequency.

10. Op-amp as a summation circuit

- Refer to the experimental procedure, section 9.
- In this section you will design a circuit that will perform an arithmetic summation of two functions. You must design it according to your ID numbers, as explained below.

Observe the following example of a *summation circuit*:

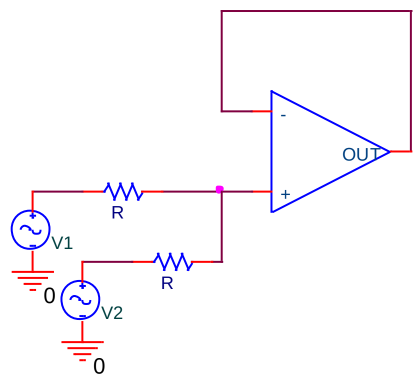


Figure 11

10.1. What is the output of this circuit, as a function of the two inputs $V_0(V_1, V_2) = ?$ What mathematical operation does this circuit perform on the inputs?

10.2. Calculate from your ID numbers: F-first number, L-last number Then, use these values to choose a function from the table below:

F-odd / L-odd	F-even / L-odd	F-odd / L-even	F-even / L-even
$-5V_1+2V_2$	$-0.5V_1-2.5V_2$	$-V_1+4V_2$	$-2V_1+2V_2$
$R_f = 5k\Omega$		$R_f = 10k\Omega$	

For your convenience, the value of R_f is chosen for you (bottom row). Also, a "master circuit" is presented below, which is the most general case that allows for any of these functions to be realized:

Last update: 2023A 20/10/22

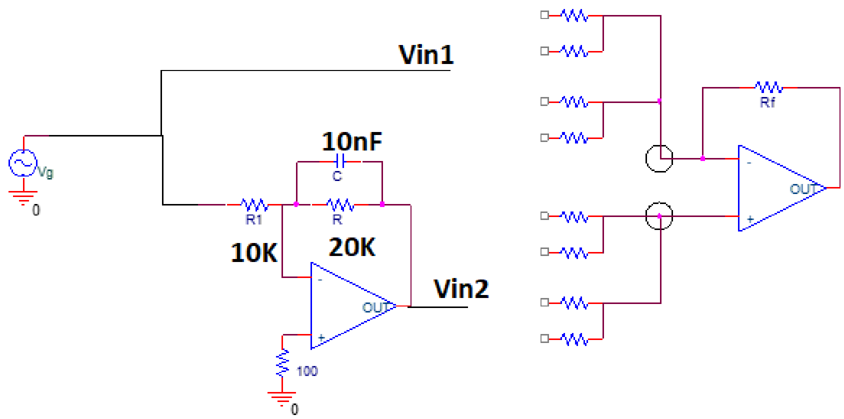


Figure 12

10.3. You may connect any free node in Figure 12 to the ground, to one of the voltage sources, or leave it as is (disconnected from the circuit). The resistors may be replaced with a wire (short-circuit) or disconnected completely from the circuit. Design the correct circuit, based on this general solution, to match the transfer-function you've been assigned. You may only use the free resistors available to you on the circuit board; see photo of the board on the website.

- ✓ Important: going "brute-force" and trying to solve the general case is **not** recommended. Instead, use your knowledge and understanding of operational amplifiers to first narrow the options down.

- ✓ Hint: you do not need more than 4 resistors in total to solve any of the functions.

10.4. Use PSPICE to simulate your circuit, choosing Vg1 = square wave. Attach simulations of Vg with Vin1, Vin2 (make sure amplitude Vin1 is equal to amplitude Vin2), and a simulation of both inputs (Vin1, Vin2) and the output, and make sure your results prove to the reader that you've succeeded in your task by providing important numbers on the simulation graphs.

- Note: all your simulations incorporate a LM741 op-amp. Since you do not know which op-amp you will end up with, treat your results here qualitatively.

The preparatory report is not finished! There are more questions to be answered below!

Self-quiz and preparation for actual measurements

You're almost finished with the preparatory report. Don't stop now..

- In the experiment you'll be "thrown in the water", so you must know exactly what's going on and what you should do at all times. By answering these questions correctly, you'll be assured that you can perform the required tasks in the lab.

11. 3dB frequency measurement

In Electronics Lab 1, you learned the "SWEEP" method for measuring the 3dB frequency. We do not endorse this method in our lab, which is why you will perform it only once, in this experiment.

As a quick reminder about the "SWEEP" method, use the horizontal cursors to measure the peak-to-peak value ΔV of the curve away from any poles or zeros (i.e. where the response curve is flat), then calculate: $V_{-3dB} = \Delta V / \left(2\sqrt{2}\right)$. Set the top horizontal cursor to read the calculated V_{-3dB}, then use the vertical cursors to measure the time difference Δt from the beginning of the sweep to the intersection with V_{-3dB}. The 3dB frequency is then $f_{-3dB} = \Delta t \times \Delta f_{sweep} / \Delta t_{sweep}$. Review the relevant material from Electronics Lab 1 for further insight.

Now, answer the following questions:

11.1. What horizontal scale must you use to perform this measurement?
11.2. Assuming the sweep looks like an LPF, what is the relationship between the peak-to-peak voltage in low frequencies and the amplitude voltage in the 3dB frequency?

- We will now explain the method you will use in all experiments in our lab.

11.3. First, we must learn how the circuit behaves when changing the frequency. Does it resemble a LPF, BPF or HPF? Perhaps it's not even working correctly?
11.4. For the sake of this explanation, assume it is an LPF. In this case, we know that the 3dB frequency has an amplitude which is lower by 3dB, i.e. by a factor of $1/\sqrt{2}$, from the (flat) amplitude in low frequencies. "Low" depends on the circuit's frequency response, of course, and as a rule of thumb is in a frequency range $f << f_{3dB}$.
11.5. By constantly measuring the output and input voltages, and dividing them: v_o / v_i, and changing the input signal's frequency from low to high, you are "traversing" the circuit's frequency response curve, i.e. the LPF; therefore, at some point the division v_o / v_i will grow smaller, until it reaches your calculated $1/\sqrt{2}$ -factor value. This is the 3dB frequency.

- We will now explain the actual measurements in the lab.

11.6. First, you must know what you expect to measure. Is it LPF? BPF? Or HPF? Here we will assume it has the shape of a LPF.
11.7. Perform an AC sweep measurement on the output, v_o, and input, v_i. Make sure you show both on-screen. Change the frequency range so you can see both the "flat" low frequency range and well past the 3dB frequency.

- ✓ Don't tempt yourself with the notion of sweeping across all frequencies between 1Hz and 30MHz. Like you've seen in "Electronics 1" course, experiment #9, when the frequency is high enough, the elements in the circuit cannot be regarded as "lumped" anymore, and we must use the distributed element model (transmission-line model), meaning the sweep results <u>are not reliable</u>. **As a rule in our lab, never go above a 4MHz input frequency.**

- ✓ On the other hand, don't choose the high-frequency to always be 4MHz. You will lose much resolution if the 3dB frequency is 1kHz, for example.

- ✓ Also, don't choose the lowest frequency to be 1Hz. A 1Hz sine completes one period in 1sec. If the sweep time is 1sec, starting a sweep from 1Hz means the sweep would be severely under-sampled!

11.8. Choose a sweep time of 1sec: in this interval, the waveform generator will output a whole range of frequencies; some circuits don't respond well to a change too fast. In such a case you will have odd fluctuations in the sweep curve, and will have to choose a longer sweep time (you could always choose 10sec to be on the safe side).

11.9. Once you have deduced the circuit is working as expected, stop the AC sweep and measure the RMS voltage in low frequencies of both input $v_i|_{low}$ and output $v_o|_{low}$ signals. This frequency should be one where the input and output voltages are independent of frequency (i.e., the "flat" frequency range).

11.10. Use the frequency knob on the waveform generator and raise the frequency while constantly monitoring the input and output RMS measurements.

- ✓ While making a measurement in the scope (and using the scope built-in measure tool), pay attention to the signal on your screen. It must be seen with good resolution (not too big for the screen nor too small), and the screen must contain several signal cycles. Ignoring this may cause <u>false or unreliable</u> results.

11.11. Once the output is equal to $v_o|_{low}/\sqrt{2}$, you have *approximately* reached the 3dB frequency. Write it down (in Hz).

- ✓ Why approximately? This depends whether v_i was constant in frequency or not:

11.12. Before you finish, remember: the 3dB frequency originally means the **transfer function's gain** diminishes by 3dB, i.e. you should look for the frequency where: $\left.\frac{v_o}{v_i}\right|_{(f=f_{3dB})} = \frac{1}{\sqrt{2}}\left.\frac{v_o}{v_i}\right|_{low}$. However, **in most of our applications v_i is independent of frequency** for the whole range of interest. If it only changes slightly, an approximation of the 3dB frequency is usually accepted. Otherwise, you should use a table measurement to find a more accurate measurement of the 3dB frequency.

11.13. The concept of a transfer function implies linearity, which means we are always talking about **sine waves**. If your output signal is distorted, then there's no use in looking for the 3dB frequency. You should then try to fix this distortion, for example, by reducing the input voltage as needed, so the circuit operates linearly again.

- 💡 Let's see if you got it!

11.14. You've performed the first step in measuring the 3dB frequency of a circuit in the lab, and you find that the AC sweep resembles a HPF. Explain all the steps for measuring the 3dB frequency in the lab, in this case. What should you be careful with?

12. <u>Table measurement (frequency dependence)</u>

A sweep measurement gives a qualitative assessment of the circuit's dependence on frequency. To gain a quantitative perspective, one must (unfortunately) take a series of manual measurements of signal amplitudes or phase, and draw a graph. You will be supplied with a table and an embedded Excel sheet; follow the experiment, section 3.5.

12.1. Normally, before filling the table, you would perform an AC sweep measurement. This is done for two reasons:
- To make sure the circuit is operating, and operating as you expect it to.
- Even if you simulated the circuit, this doesn't mean the bandwidth or gain would be the same. By sweeping first, you can iteratively choose the sweep range and decide on the table's frequency range.

12.2. Once you've decided on an appropriate frequency range, one that would represent the transfer function adequately, choose 9 frequencies along this range, with at least 5 around the 3dB frequency.

12.3. Notice that when you change frequencies, you should change the scope's scale to accommodate at least 5 cycles with good resolution and scale on screen.

12.4. When done, take one additional measurement at the 3dB frequency itself.

12.5. Most usually, you'll be asked for a <u>gain</u> measurement. This means you have to measure the input and output of the transfer function (it's not always the input and output of the circuit!), using a sine wave with specified frequency.
- The input will not always remain constant, so you must always measure it as well.
- It is usually preferable to measure RMS rather than VPP, because electrical noise might create unwanted "peaks" that would give false results.

✓ Remember! You are assuming the system is <u>linear</u>, which for this measurement means both input and output are <u>undistorted sine waves</u>.

12.6. If the output is nonlinearly distorted, try reducing the input voltage. If that doesn't work, and you can't otherwise fix this, the system is no longer linear and the results might not be reliable.

12.7. If there's a V/V column for the gain, make sure you include the sign (if it's definitive). Otherwise, there will be a dB column: calculate using 20*log(gain).

12.8. If you are also asked to measure the phase, you may use the scope's phase measurement but take care to add 180 degrees when the phase flips sign. Also, make sure you are measure the phase correctly (the phase of Vo/Vi rather than Vi/Vo).

Experimental procedure

אור שאול		8	5

Last update: 2023A 20/10/22

אריאל רנה			
Your names	**Student IDs**	**Board number**	**Booth number**

Reminder:

a) **All the answers for the practical and theoretical questions shall be written down in this document, during lab hours only.**
b) **Save this file as "report1_XXX_YYY" when XXX, YYY are your students ID's. Please save this document right now as to avoid any future inconvenience.**
c) **At the end of this lab you must upload this file under the right assignment to the course site and click "hand in assignment":**
   - **If you have yet to complete this experiment in its entirety, please upload the file under the "Progress report" assignment.**
   - **If you had completed this experiment and answered all of the questions, please upload the file under the "Post lab (Final)" assignment.**

d) **You cannot "fix" sections that have been answered without a special approval from your lab instructors.**
e) **You should not attempt to upload or send the assignments from your home or after/before lab hours.**

1. Before we begin..

1.1. As a reminder, the circuit board has one of three operational amplifiers installed: TL071, TL061 or LM741. They have similar traits but different behavior. During this experiment you will collect enough data in order to deduce which op-amp you are working with.

1.2. Pay attention to the board number you have. This number ("YY") is written down on the bottom right side of the board, in the following pattern "SN: XXX-YY". Write it down now at the top of this page, in the appropriate place. When you get to the next lab session, make sure you receive the same board.

1.3. In this document you will find tips colored in cyan. These pearls of wisdom will come in handy in this experiment, and those to follow. Remember them!

Last update: 2023A 20/10/22

1.4. Safety notice

The board contains many metallic parts, some of which with relatively **high voltage** and the ability to conduct **high current**. This means you may very easily burn things or in some extreme cases even get electrocuted.

NEVER "THROW" WIRES AROUND ON THE BOARD, OR LEAVE A WIRE UNATTACHED!

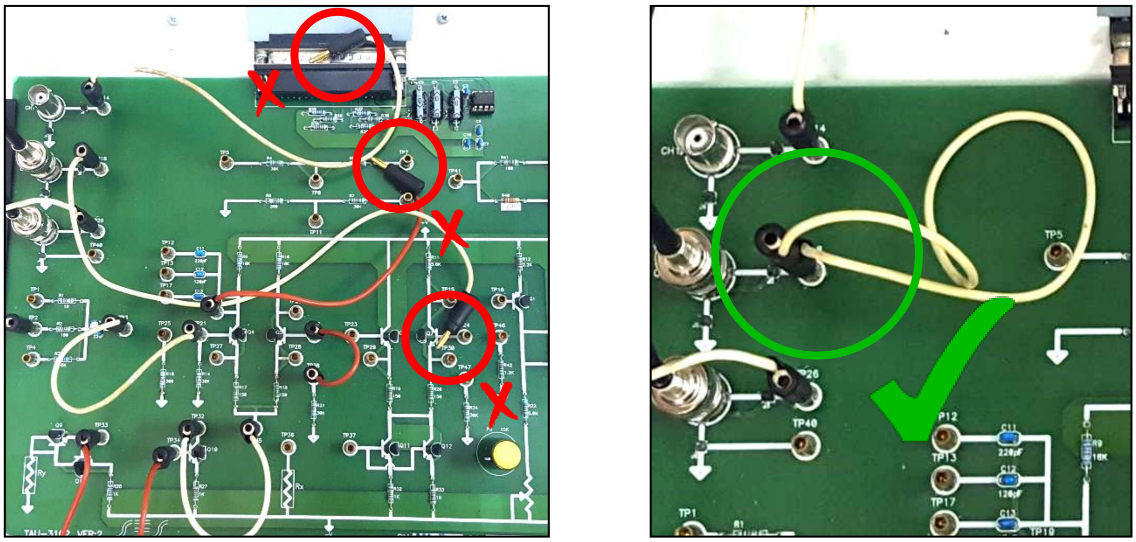


Figure 13

2. Measuring the input offset voltage

- Refer to the preliminary report, section 4.

✓ Wherever we quote a section number, such as "section 4" above, we've included a "hyperlink" for quick navigation. Try pointing your mouse on the section number ("4") and then CTRL+LEFT CLICK.

2.1. Build the required circuit, and measure V_{out} using the multimeter. What coupling did you use? What is V_{io}?

✓ Don't forget to write appropriate units next to every measurement or at the top of a table column, in case there none are mentioned.

Coupling = DC
V_{out} =-0.12v
V_{io} =0.12v

2.2. From now on, and for the rest of this experiment, we'd like to work with an offset-free op-amp. Connect a potentiometer (P1) to the appropriate op-amp terminals, then use it to set the output to zero (within the accepted electrical-noise level, as mentioned in the preliminary report). Use the multimeter to measure the output, and write down the final, measured value:

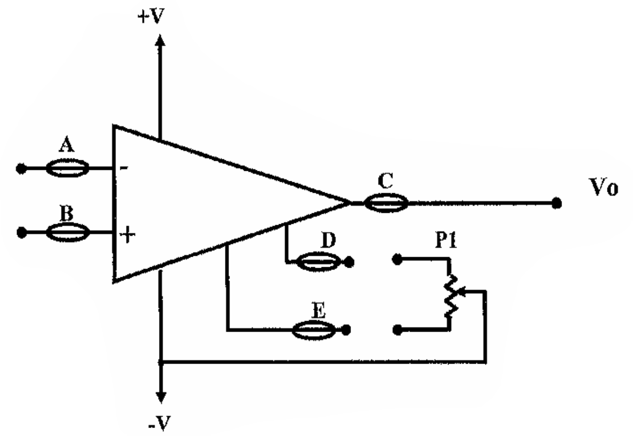


Figure 14

Coupling = DC
V_{out0} = 25.5mv

✓ Do not disconnect the potentiometer until you finish today's session, and remember to reconnect and calibrate it again next time!

3. Measuring CMRR

Refer to the preliminary report, section5.

3.1. Build the circuit in Figure 5. Set Vg to a 100Hz-sine wave and choose the input according to the preliminary report.

✓ In order to save a print from the scope to your report, you'll need to use the "Agilent" software. There is a shortcut for it on your desktop.

- Click Instruments->MSO-X2012A or Instruments->Agilent 2000/3000
- Make sure that the "Greyscale" and "Invert" checkboxes are not ticked.
- Click the "Get Data" Icon that lies on your toolbar – This will copy your scope's screen.
- Using CTRL+C or Edit->Copy save the image to your pasteboard. Change your active window into this document, and paste the picture.

3.2. Measure the 3dB frequency according to the preliminary report, section 11, using the "SWEEP" method, then answer the questions in the table below. This is the only time you will use this method. <u>You do *not* need to add a print to the report at this stage.</u>

Last update: 2023A 20/10/22

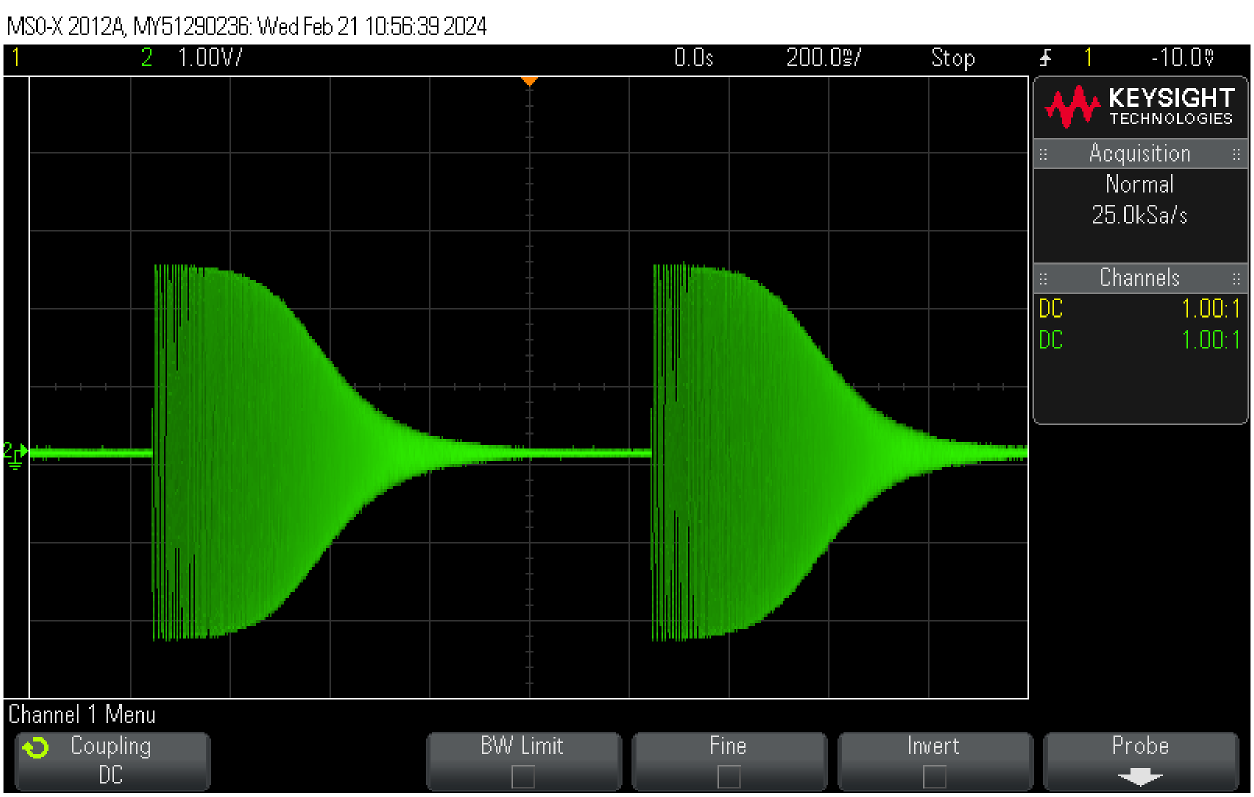
What is the 3dB frequency, according to the “SWEEP” measurement?
1512Hz
Is the 3dB frequency you measured here equal to the 3dB frequency of the transfer function vout/vg, or vout/vin, or both? Why? (Review prelab section 11 to answer this correctly)
אף אחד מהם, כנראה טעות במדידה

3.3. Now we will use our lab’s method. As a first step, attach a print of the AC sweep, with VPP measurements for the input (vin) and output (vout) signals (total of 2 measurements), and write down the sweep parameters in the table below.

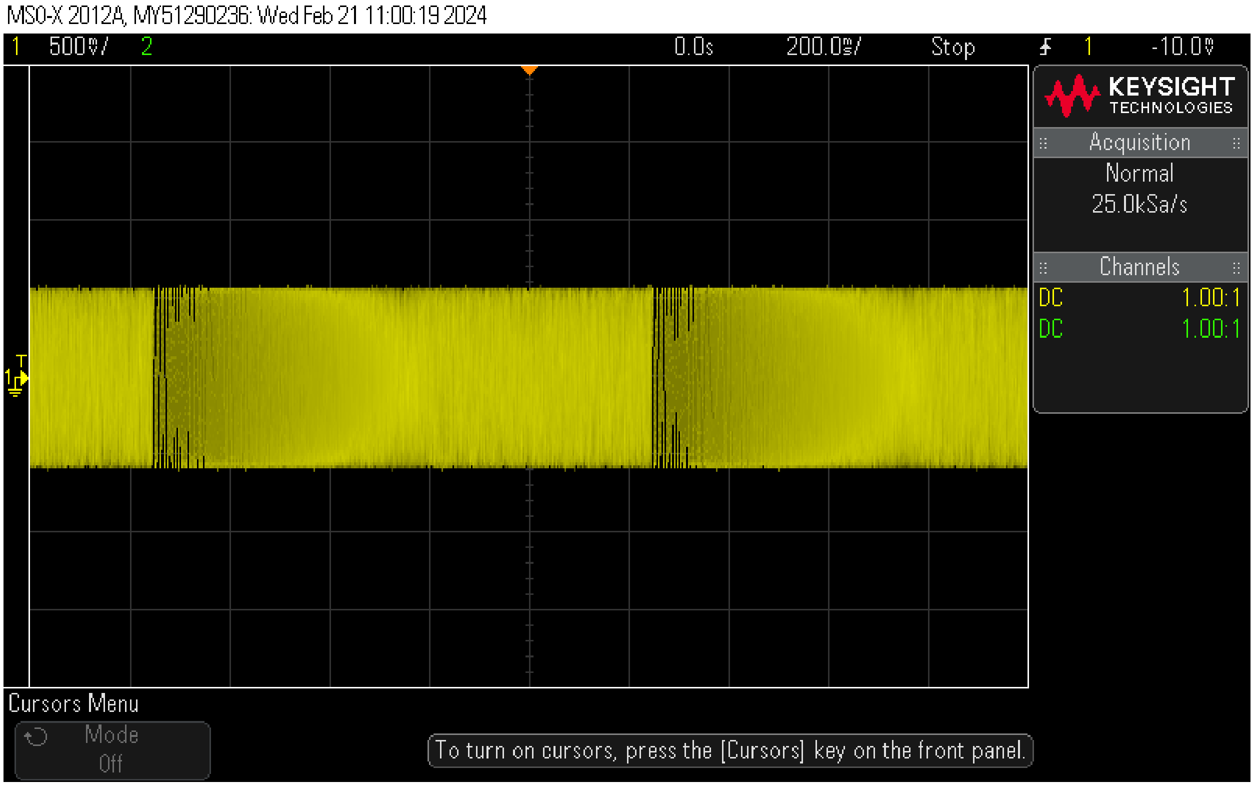
SWEEP MEASUREMENT	
Frequency range	100Hz-1Meg Hz
Sweep time	1sec
Input voltage	200mvRMS

✓ NOTE: Sweep print-outs must always be in logarithmic scale!

Print: input and output



Last update: 2023A 20/10/22



3.4. Now perform the measurement according to our lab's method (see preliminary report section 11). Fill in the table below.

What is the 3dB frequency, according to our method?
1475.63 [Hz]
Following your answers in section 3.2, we repeat the question: is the 3dB frequency you measured here equal to the 3dB frequency of the transfer function Vout/Vg, or Vout/Vin, or both? (Review prelab section 11 to answer this correctly)
התוצאות אמורות להיות זהות
Compare the two methods. Which do you think is better, and why? (up to two lines)
לדעתנו, השיטה השנייה טובה יותר בגלל שתדר הברך יותר מדויק באופן הזה ולא צריך למדוד באופן ידני "לפי העין" כמו בשיטה הראשונה.

3.5. Measure the gain of the circuit, for CMRR calculation later, according to the preliminary report, section 1212. Notice that we are asking for several columns of measurements, but you will choose the transfer function to draw in the graph. Also, use the "Auto-Scale" button to change the scales automatically as you change frequency.

✓ You may choose to use the multimeter as an additional measurement tool. Just remember not to measure the same node with both multimeter and scope!!

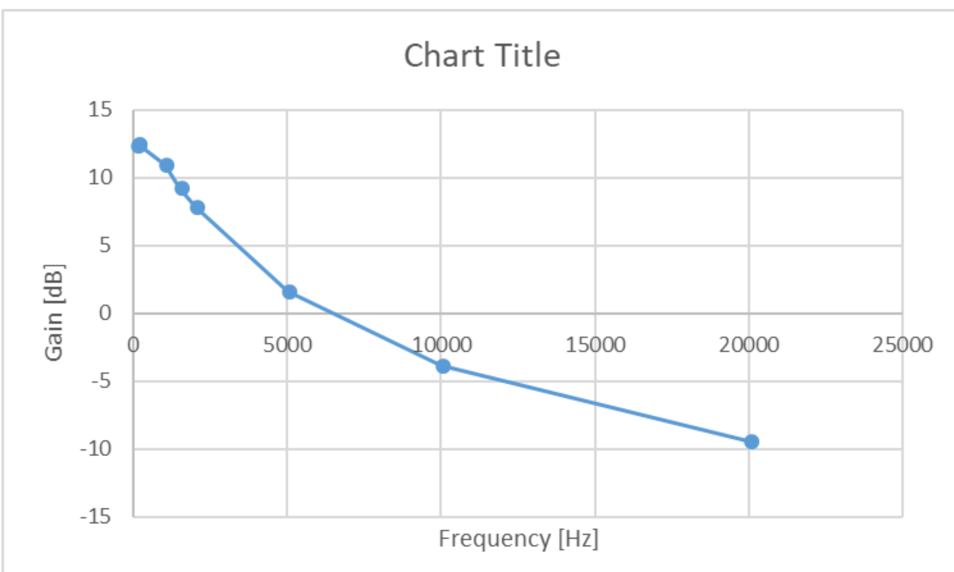
Last update: 2023A 20/10/22

- ✓ When the "Auto-Scale" button is pressed on the scope, both channels automatically reset themselves back into DC coupling. To avoid this, click "Auto-Scale" once, and on the bottom left side of the screen click the "Fast Debug" button. From now on, your coupling and other changes you have made to the scopes settings will be saved even after clicking the "Auto-Scale".

TABLE MEASUREMENT				
Transfer Function	(Answer here: which transfer function are you drawing? What is Vx and Vy in Vx/Vy=?)			
$f[Hz]$	V_g	V_{in}	V_{out}	$A[dB]$
100	200mvRMS	1.19v	4.9v	12.29
120	200mvRMS	1.19v	5v	12.46
130	200mvRMS	1.19v	5v	12.46
1000	200mvRMS	1.19v	4.2v	10.95
1500	200mvRMS	1.19v	3.46v	9.2
2000	200mvRMS	1.19v	2.93v	7.82
5000	200mvRMS	1.19v	1.43v	1.59
10000	200mvRMS	1.19v	760mv	-3.89
20000	200mvRMS	1.19v	400mv	-9.46
(3dB frequency=1475Hz)	200mvRMS	1.19v	3.5v	9.3

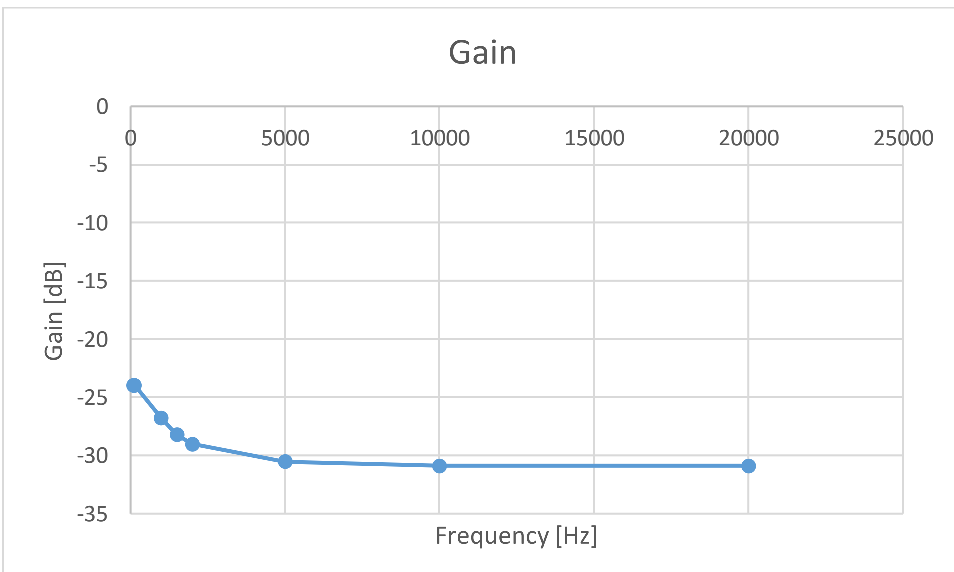
- ✓ We will supply you with sheets like these, although, they will not always have the equations, units or headlines inside. Pay attention to what's missing and fill them out in order to hand in an organized report.

- ✓ In order to edit the excel sheet, double click it and select Sheet1. Type in the data and return to Chart1 so that on leaving the excel sheet the plot would be displayed.



3.6. Connect the circuit in Figure 4, and measure its gain. **Use the same frequencies you just measured!** This will help you in CMRR calculation later. Once you're done, answer the question in the next table.

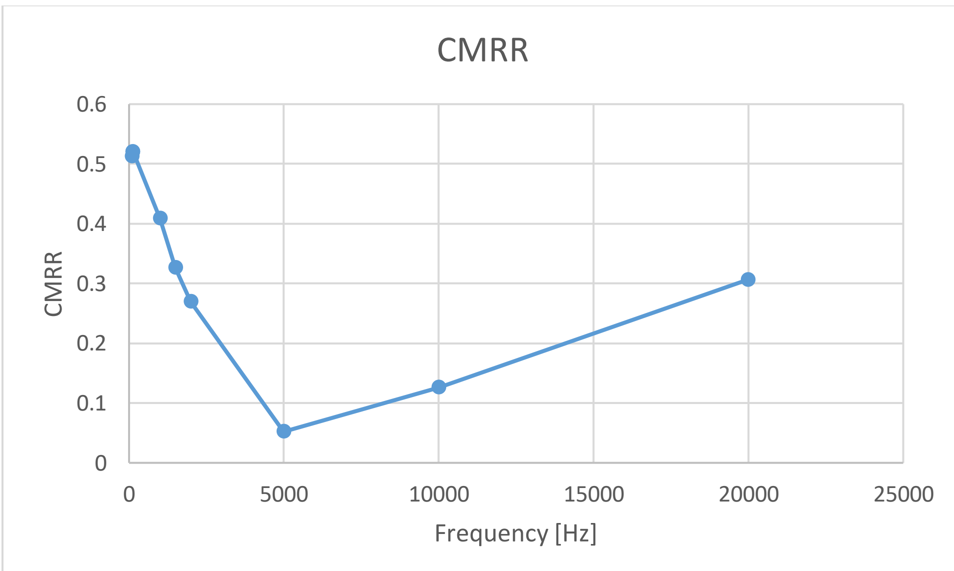
TABLE MEASUREMENT				
Transfer Function	(Which transfer function are you drawing? What is Vx and Vy in Vx/Vy=?)			
$f[Hz]$	$V_g = V_{in}$	V_{in}	V_{out}	$A[dB]$
100	17.5v		1.11v	-23.95
120	17.5v		1.11v	-23.95
130	17.5v		1.11v	-23.95
1000	17.5v		800mv	-26.79
1500	17.5v		680mv	-28.21
2000	17.5v		620mv	-29.01
5000	17.5v		520mv	-30.54
10000	17.5v		500mv	-30.88
20000	17.5v		500mv	-30.88



Observe your gain graph for this circuit. Is this what you expected? Why?
לפי הדוח המכין הגבר קומונלי במגבר שרת אידאלי הוא 0. במקרה שלנו ההגבר אינו 0 אך מאוד נמוך וניתן להסיק כי המגבר שלנו קרוב לאידאלי.

3.7. Using your previous measurements, calculate CMRR and graph it.

$f\,[Hz]$	$CMRR[dB]$
100	0.5131524
120	0.52025052
130	0.52025052
1000	0.4087346
1500	0.32612549
2000	0.26956222
5000	0.05206287
10000	0.1259715
20000	0.30634715
100	0.5131524



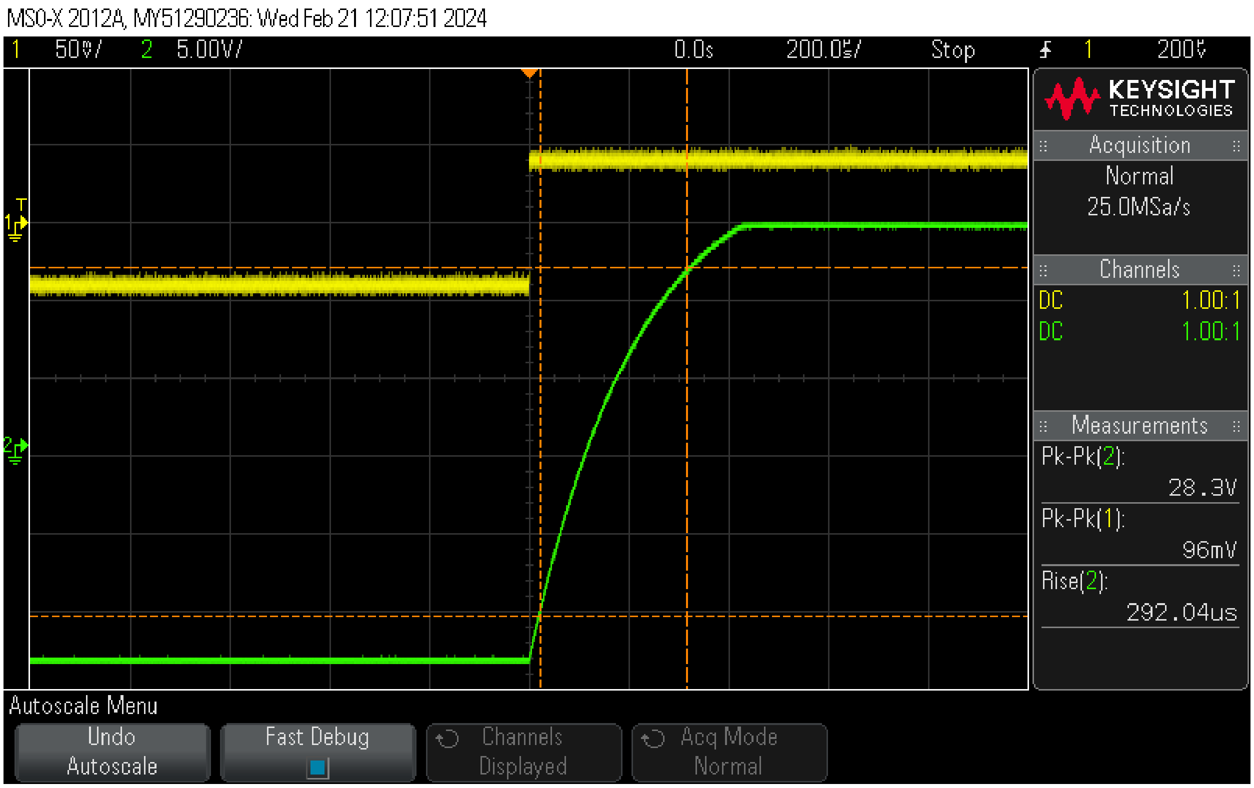
4. <u>Measuring slew-rate</u>

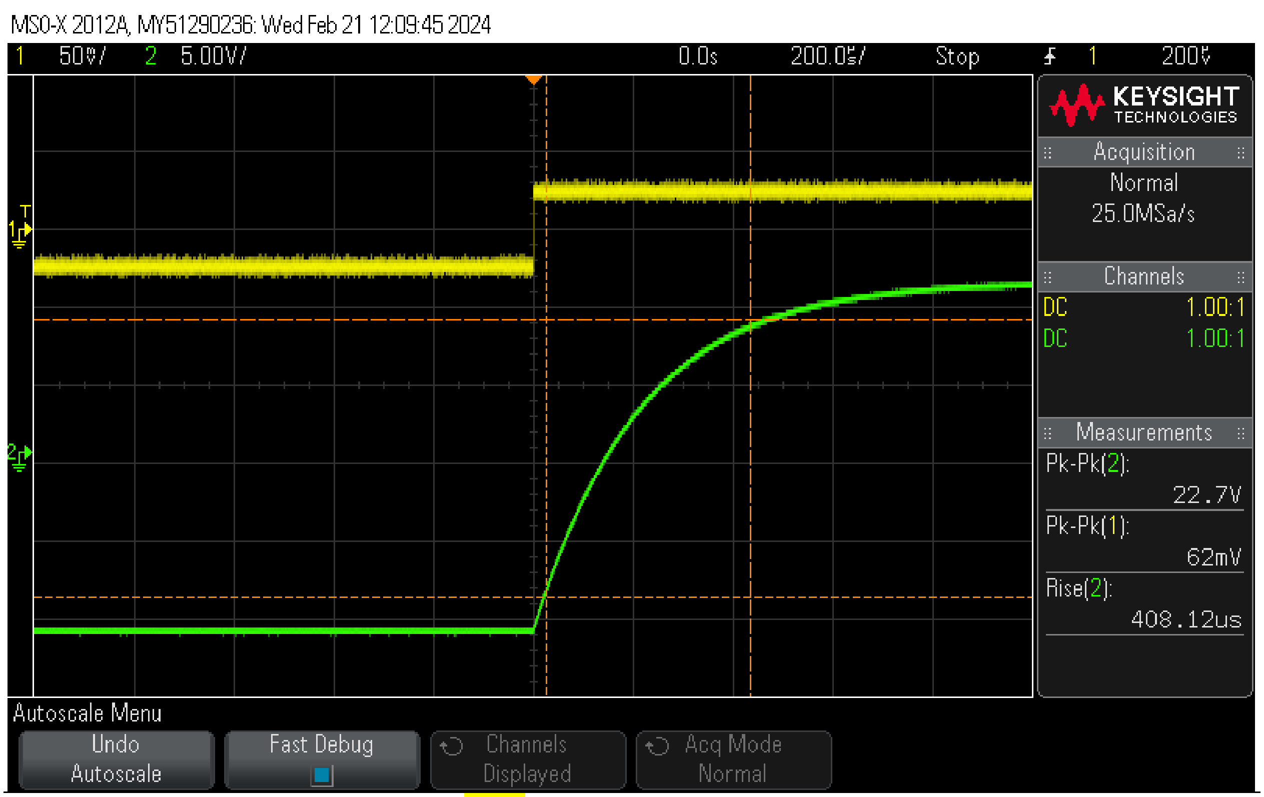
💡 <u>Refer to the preliminary report, section 6.</u>

4.1. Construct the circuit in Figure 6. Input a 100Hz-square wave with zero offset and 50% duty-cycle, and <u>attach two different prints</u> of the input and output signals, showing cases where the output *is not* SR-limited, with VPP measurements for each and a measurement of “t-rise” for the output (total of 3 measurements). Then, answer the question below.

Print: input and output, #1

Last update: 2023A 20/10/22





Print: input and output, #2

Last update: 2023A 20/10/22

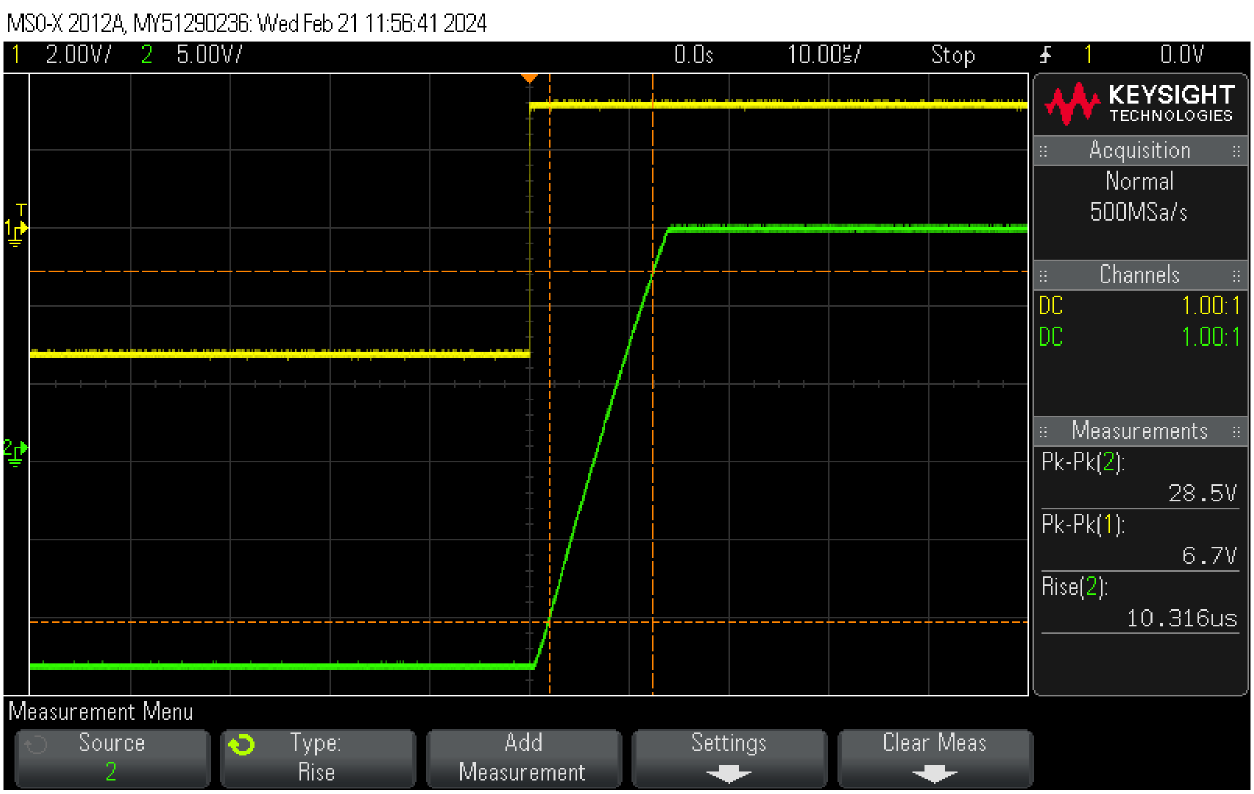
- ✓ To answer the following question, click the “Measure” button and choose t-rise using the small knob. Then, press and hold the button; after several seconds the scope will present a pop-up message with a detailed explanation on this type of measurement.

Explain how the scope defines and measures t-rise and t-fall.
t-rise = t(V= 90%Vpp)-t(10%VPP)
t-fall = t(V= 10%Vpp)-t(90%VPP)

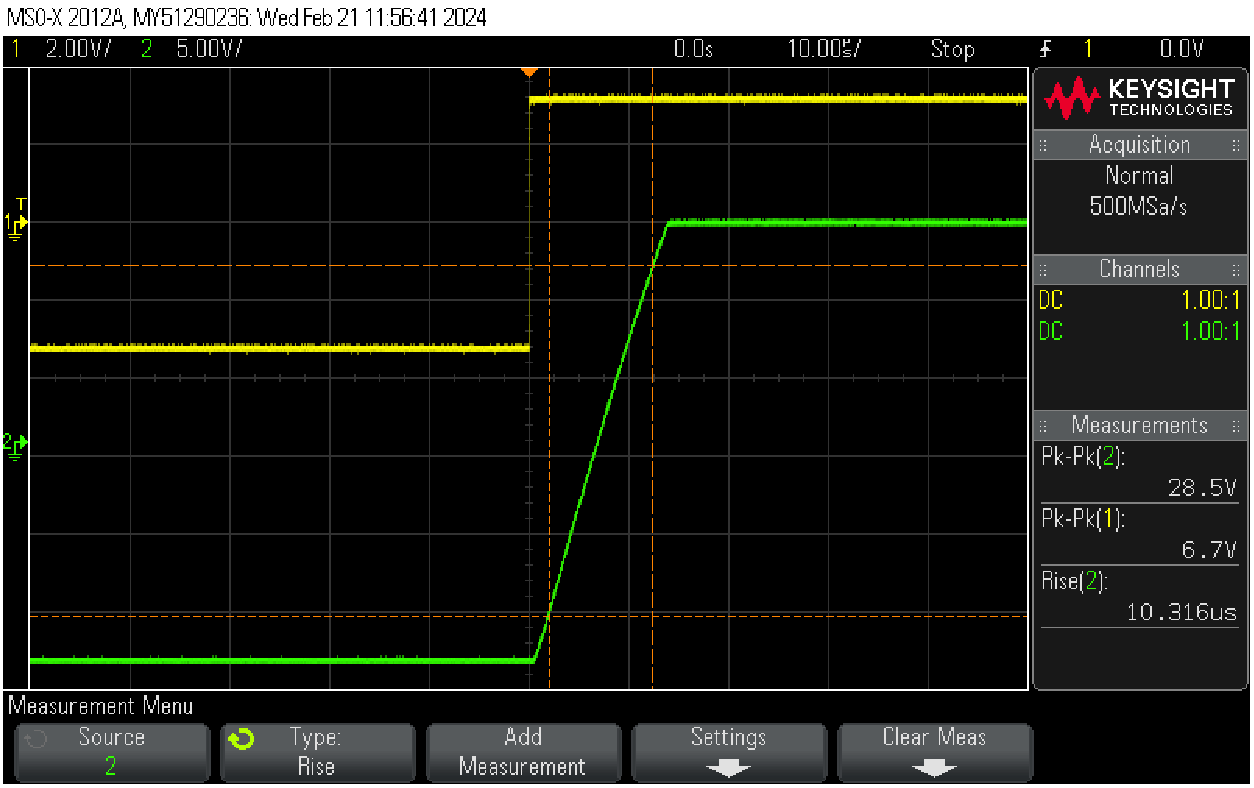
- ✓ **Try holding down the button for every function the scope has and you will get an immediate explanation on its definition!** Remember to use this option in future experiments when you find yourself befuddled with one of the scope’s functionalities.

4.2. Attach two different prints of the input and output signals, showing cases where the output *is* SR-limited, with VPP measurements for each and a measurement of “t-rise” for the outpu0t (total of 3 measurements).

Print: input and output, #1



Last update: 2023A 20/10/22



Print: input and output, #2

4.3. From your measurements, calculate SR.

$SR = 2.19$[V/usec]

5. Measuring the gain-bandwidth product

- Refer to the preliminary report, sections 77 and 11.
- NOTE: When beginning measurements, use the scope to make sure the output signal oscillates around 0VDC, since the calibration you did in section 2.2 might be off.

5.1. Write down your ID numbers here, and add them together:

ID#1	206393662
ID#2	206491920
ID#1+ID#2	412885582
Is the sum an even number?	Yes

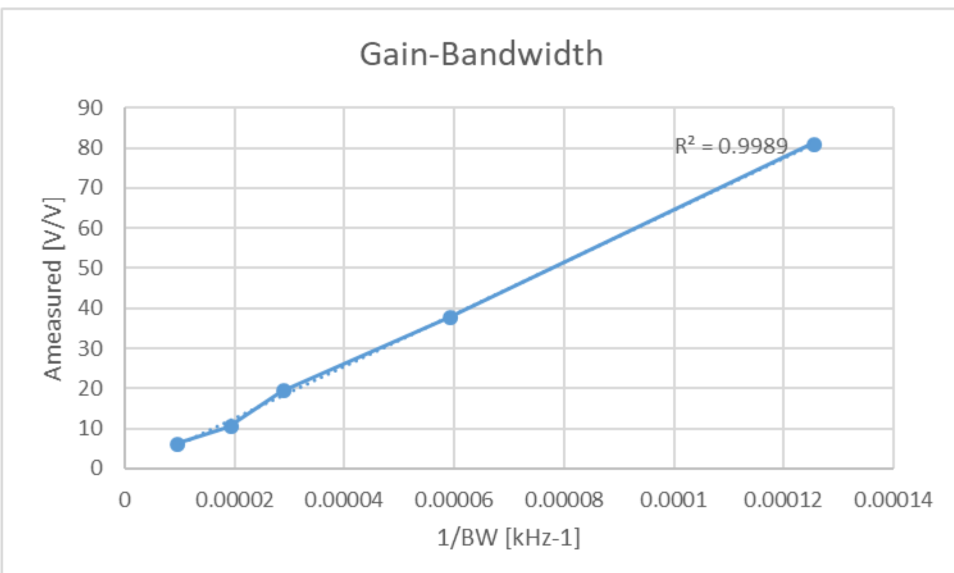
If the sum ID#1+ID#2 is an even number, construct Figure 7. Otherwise, use Figure 8. Then, fill up the following table to calculate GBW by measuring the bandwidth f_{3dB}, and measuring the correct gain in the 3dB frequency $A_{measured}^{-3dB}$.

Before you begin, calculate the theoretical DC gain A_{theory}^{DC} you should have for each R_f, and while measuring, compare the gains (A_{theory}^{DC} vs. $A_{measured}^{-3dB}$):

TABLE MEASUREMENT					
Transfer Function	(Answer here: which transfer function are you drawing? What is Vx and Vy in Vx/Vy=?)				
$R_f\,[k\Omega]$	$f_{3dB}\,[Hz]$	$A_{theory}^{DC}\,[V/V]$	V_{in}	V_{out}	$A_{measured}^{-3dB}\,[V/V]$
5	109KHz	6	2.09V	12.7V	6.0765
10	53KHz	11	2.13V	22.1V	10.37
20	35kHz	21	1.09V	21.1V	19.35
40	17KHz	41	450mV	16.9V	37.55
100	8KHz	101	350mV	28.3V	80.85

Relate your measured gain to the bandwidth (in the Excel sheet, write A in V/V and make sure the x axis is 1/BW in 1/kHz):

✓ Note: in principle, the y-axis should be $A_{measured}^{DC}[V/V]$ rather than $A_{measured}^{-3dB}[V/V]$. But we are only interested in the *slope* of the curve, this difference doesn't affect the result.



Use Excel to add a linear trend-line to the graph, and make the line's equation and R-squared value visible. What is the GBW of your amplifier? Write the result in $[kHz]$.
$GBW = ?$649729 [Hz]

6. Measuring the open-loop gain

- Refer to the preliminary report, section 8.
- NOTE: When beginning measurements, use the scope to make sure the output signal oscillates around 0VDC, since the calibration you did in section 2.2 might be off.
- ALSO: in this section, some of the boards exhibit noisy "peaks" on your sine measurements. If this happens to you, that's okay, because you measure in RMS.

6.1. Build the circuit in Figure 9 where $r = 20\Omega$ and $R = 20k\Omega$. Before you begin, check if the circuit is working as expected: take a sweep measurement of both V_y and V_{out}. Choose your frequency range from 1Hz to such frequency that V_y is seen with good resolution. Arrange the two signals so you will have room on screen for a third (see next section).

- ✓ From time to time you'll be required to present more than two signals on the scope's screen and attach them in a single print. By using "Capture Waveform" you save the screen **as an image**. This means all information regarding scale, amplitude, etc. is lost. Therefore, whenever capturing more than one signal you must write down the scale and amplitude for every signal you capture. For sweep measurements, you must add the frequency range as well, which must be equal for all signals. This is done so a comparison between your signals is viable.
- ✓ Caution! Pressing the "Auto-Scale" button will reset the scope's screen and **erase** any images you've captured!

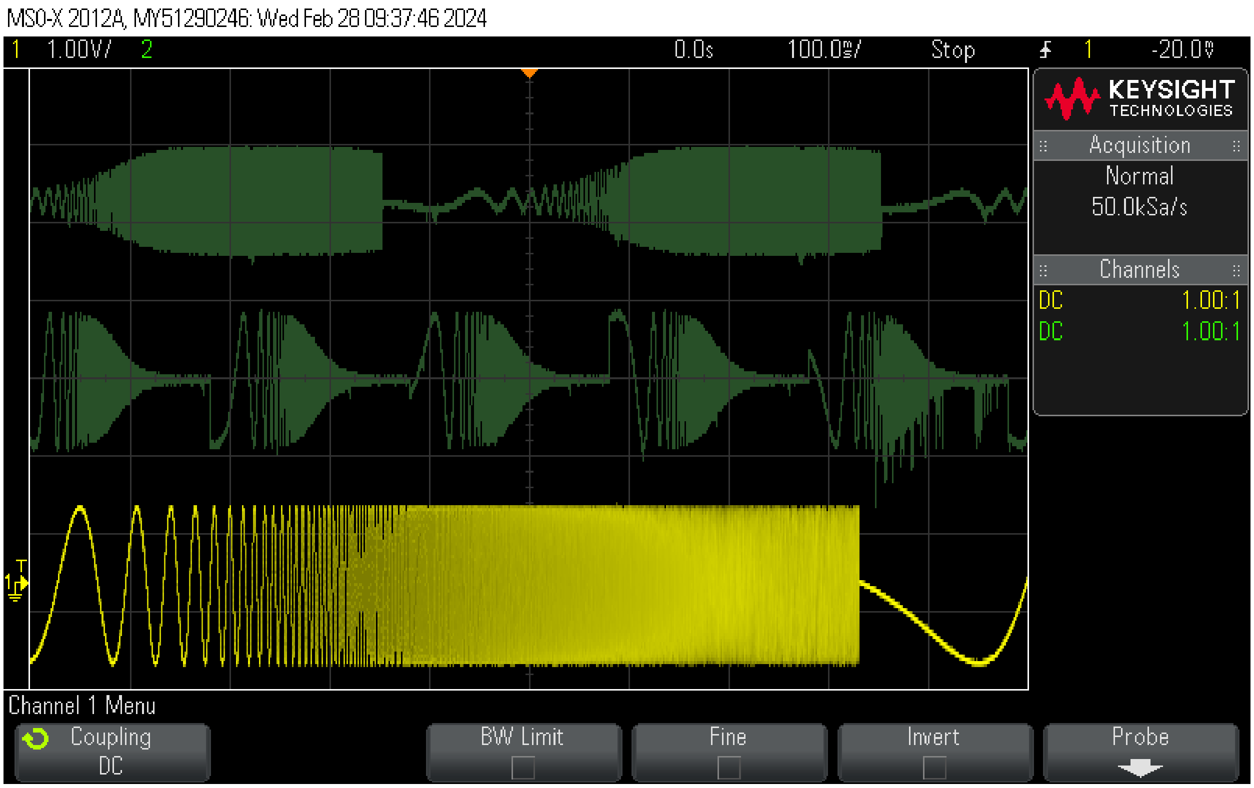
6.2. You will now add a third signal to the screen. On the scope, click "Display" and then "Capture waveform".

6.3. Write down CH1 and CH2's scales, then press the scopes "2" button to disable CH2. Now, connect CH1 to the circuit's input and move the measurement so all three measurements are seen. Attach the print.

SWEEP MEASUREMENT	
Frequency range	1Hz-100KHz
Sweep time	1sec
Input voltage	1Vpp
NOTES	1V for all channels

Print: sweep with 3 different signals

Last update: 2023A 20/10/22



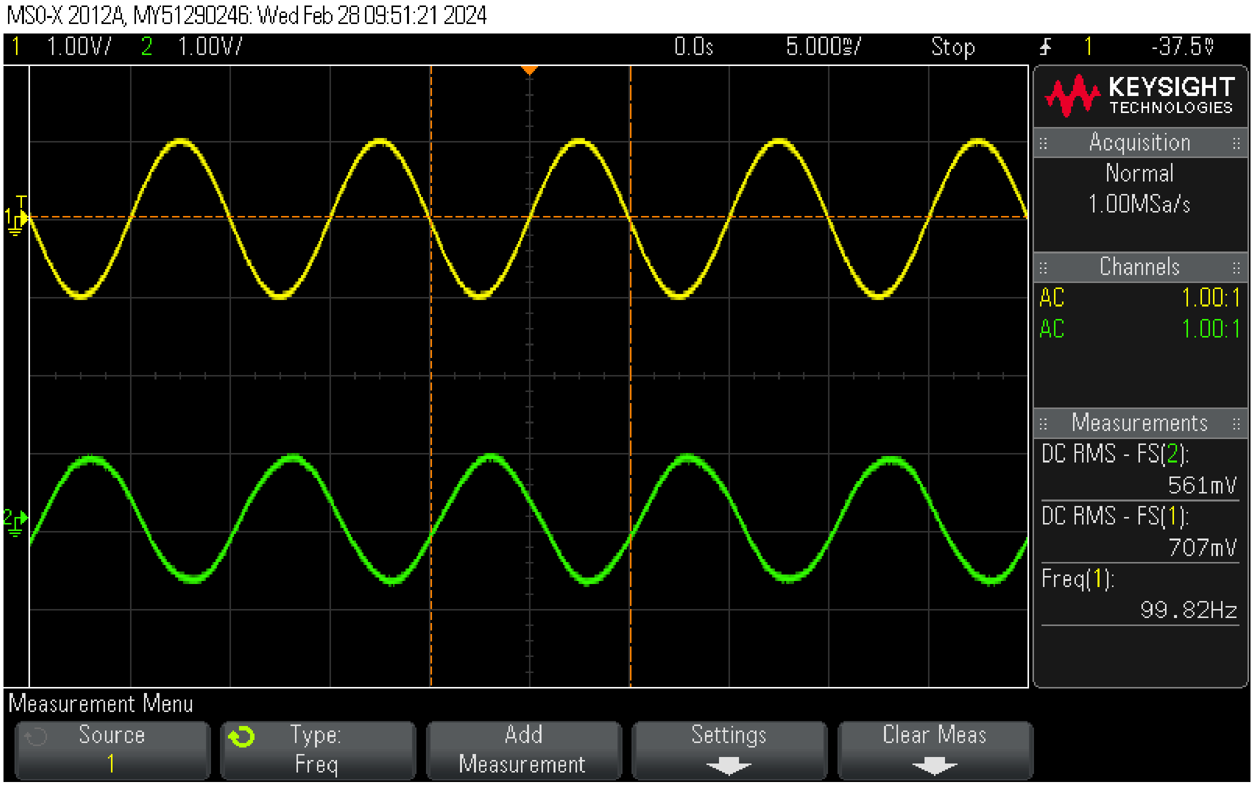
6.4. Choose an input frequency of 100Hz, and <u>attach a print</u> **showing several cycles of the input and output**, with two RMS measurements and a frequency measurement of the input (total of 3 measurements).
Make sure that on the print, the measured frequency is the correct one!

✓ Notice: For every scope measurement where you have the option of choosing between “N-Cyc” (Cycles) and “FS” (Full Screen) – Always choose FS !

✓ One of the goals of the following sections is to learn about the “AUTOSCALE” button.

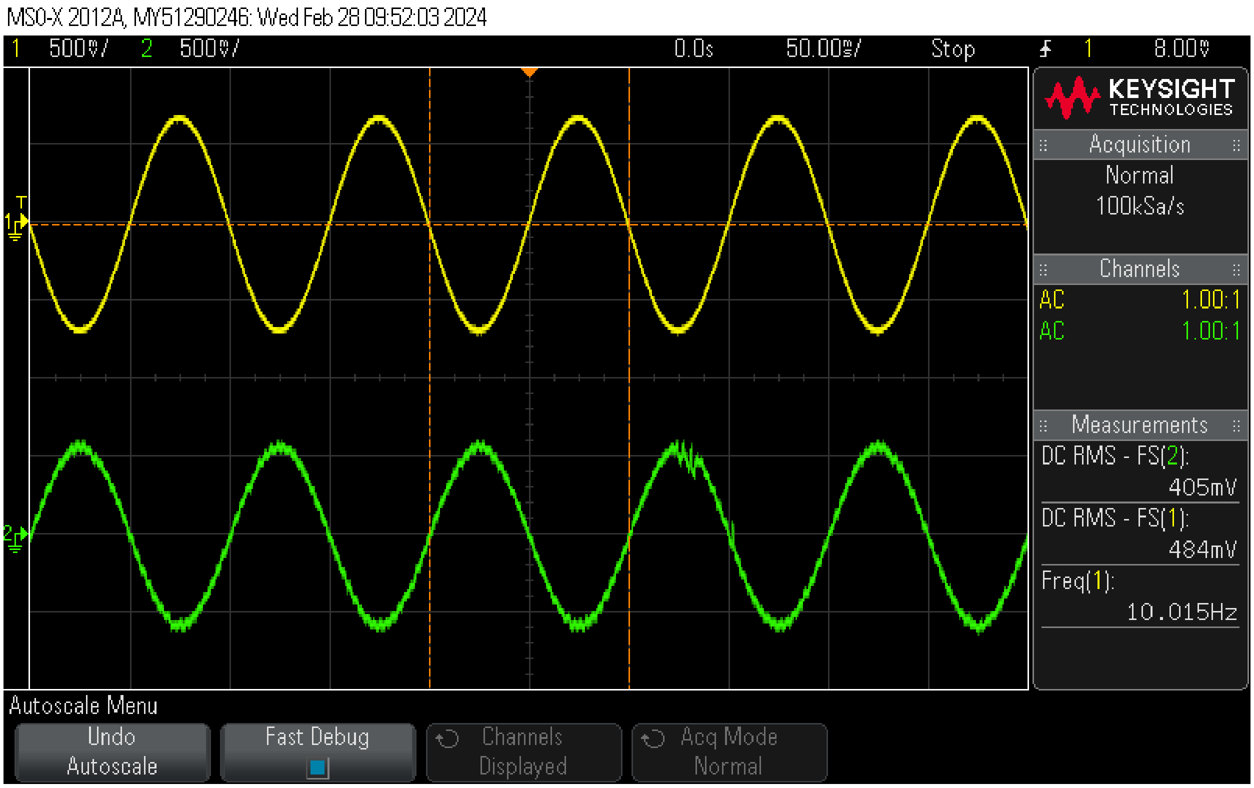
Last update: 2023A 20/10/22

Print: input and output, 100Hz*



6.5. Change the frequency to 10Hz, press “AUTOSCALE”, and make sure that on the print, the measured frequency is the correct one. Then attach a print.

Print: input and output, 10Hz

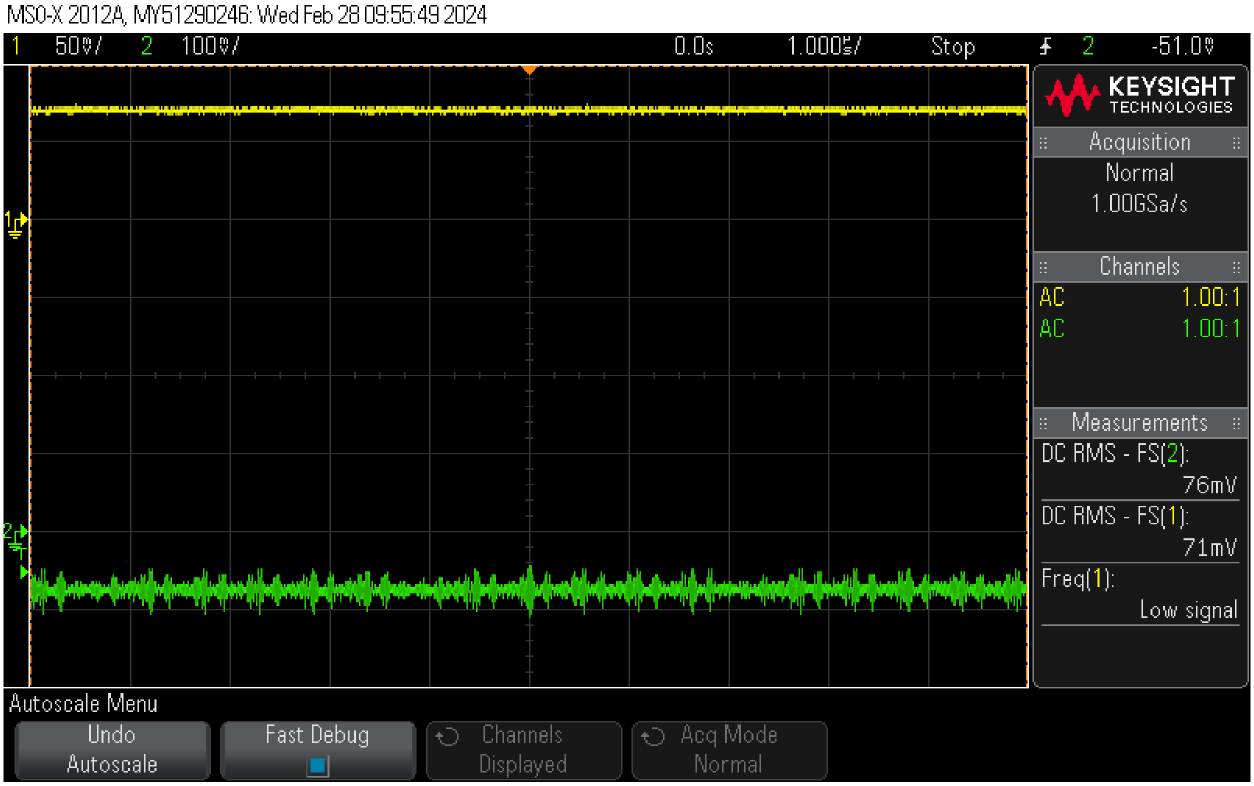


6.6. Change the frequency to 1Hz, press “AUTOSCALE”, and make sure that on the print, the measured frequency is the correct one. Then attach a print.

נשים לב שעבור תדר של 1HZ, אנחנו לא מצליחים לקבל סיגנל ברור על המסך באמצעות הAUTOSCALE. הסיבה לכך היא שפונקציית הAUTOSCALE הינה פונקציה של מעגל חשמלי, התלויה ברוחב הפס. באוסילוסקופ שלנו, רוחב הפס של הautoscale מתחיל מ10Hz ולכן בתדר נמוך יותר הפונקציה לא מצליחה למסך את הסיגנלים על המסך.

Last update: 2023A 20/10/22

Print: input and output, 1Hz



6.7. Fill the following table using the scope. Calculate the open-loop gain for each frequency, then answer the questions below.

✓ You might observe some noise on the output sine wave – similar to small "spikes" every half cycle. This is okay! Since you're using an RMS rather than VPP measurement, such noise is averaged out in the measurement.

Important: as long as the circuit operates linearly, i.e. a sine input gives a sine output without distortion, you can use a larger input amplitude. This will help you fill the following table, because a larger V_g means a larger V_y which means an easier time measuring V_y!

TABLE MEASUREMENT				
Transfer Function	(Answer here: which transfer function are you drawing? What is Vx and Vy in A=Vx/Vy=?)			
$f[Hz]$	$V_g = V_{in}$	V_y	V_{out}	$A[dB]$
1	25.1mV	4.31mV	15.8mV	0.632
2	115.9mV	4.59mV	128mV	1.104
8	427mV	24.8mV	358mV	0.838
9	457mV	25.7mV	383mV	0.838

10	485mV	27.7mV	404mV	0.832

What is the amplifier's open-loop gain, in DC (estimation)?
$A_{OL} =$
K=0.00099, Aol = 14796.88

What is the amplifier's open-loop bandwidth?
BW = GBW/Aol = 43.909 [Hz]

7. Conclusion

Fill the following table using your preliminary report, and your measurements so far. Calculate the GBW from the datasheet if necessary. Which op-amp do you have?

Parameter	TL071	TL061	LM741	YOUR AMP
V_{io}	3 [mV]	3 [mV]	3[mV]	0.12 [V]
Large-signal voltage gain	200 [V/mV]	6 [V/mV]	200 [V/mV]	8.3[V/mV]
CMRR	86 [db]	86 [db]	90 [db]	0.5131524 [dB]
SR	13 [V/uSec]	3.5 [V/uSec]	0.5 [V/uSec]	2.19[V/usec]
GBW	3 [MHz]	1 [MHz]	1 [MHz]	649729 [Hz]

Which amplifier do you have, and why?
לפי הערכים, המגבר שהכי קרוב למגבר שלנו הוא מגבר TL061
נבחין כי ככל הנראה היו כמה טעויות במדידות ולכן הערך מעט רחוק מהמצופה.

8. Op-amp as an integrator

Refer to the preliminary report, section 9.

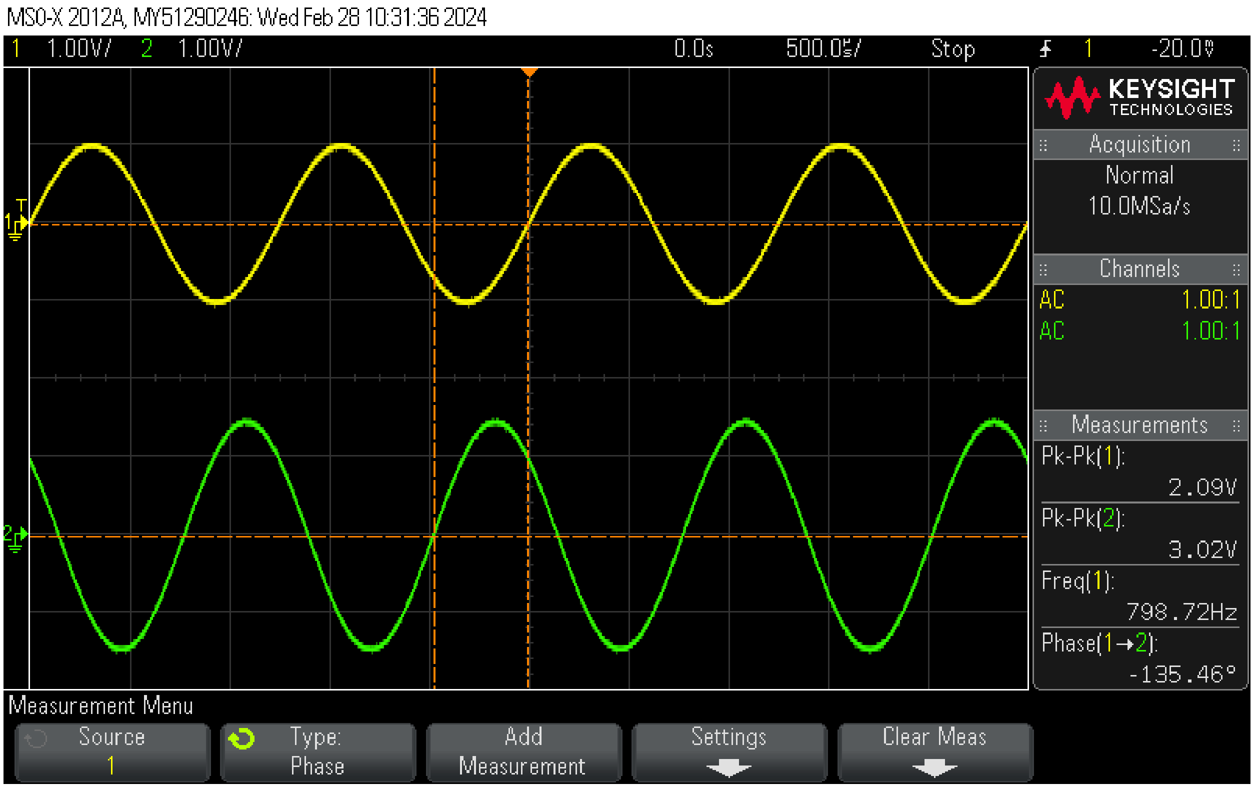
8.1. The circuit in Figure 10 is found on your circuit board, in the bottom left corner (but without the 100Ω resistor). Use a sine wave with an amplitude that would give an appreciable sinusoidal output.

8.2. Find the frequency f_p (note: the values of R_1, C and R might not have the exact values as in the preliminary report). Attach a print of the input and output with VPP measurements, input frequency, and phase (total of 4 measurements).

Last update: 2023A 20/10/22

---

*Print: input and output at f_p *

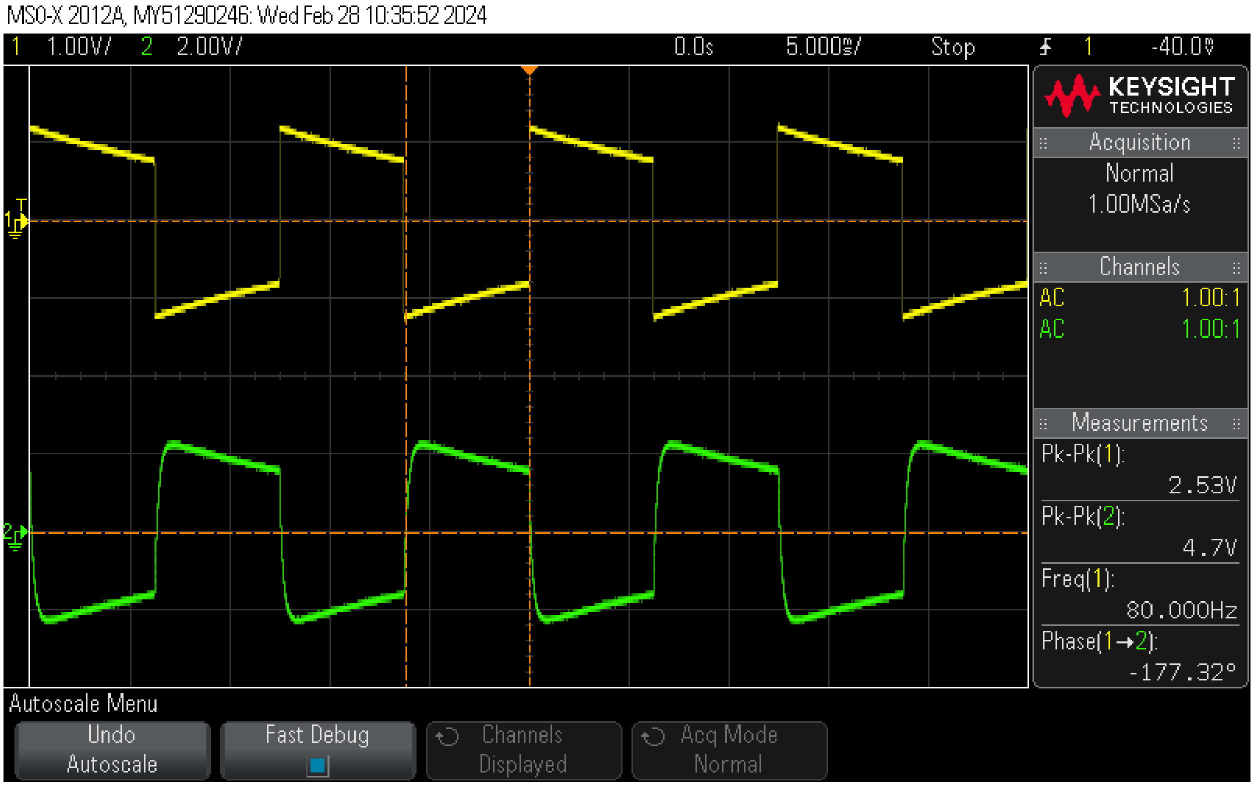


8.3. Use a square wave with 0VDC offset. Attach two prints, one at $f_l = f_p/10$ and on at $f_h = 10 f_p$, each with the same measurements as before (total of 4 measurements per print). If necessary, tune the input amplitude so the output signal is appreciable.

---

*Print: input and output at $f_l = f_p/10$ *

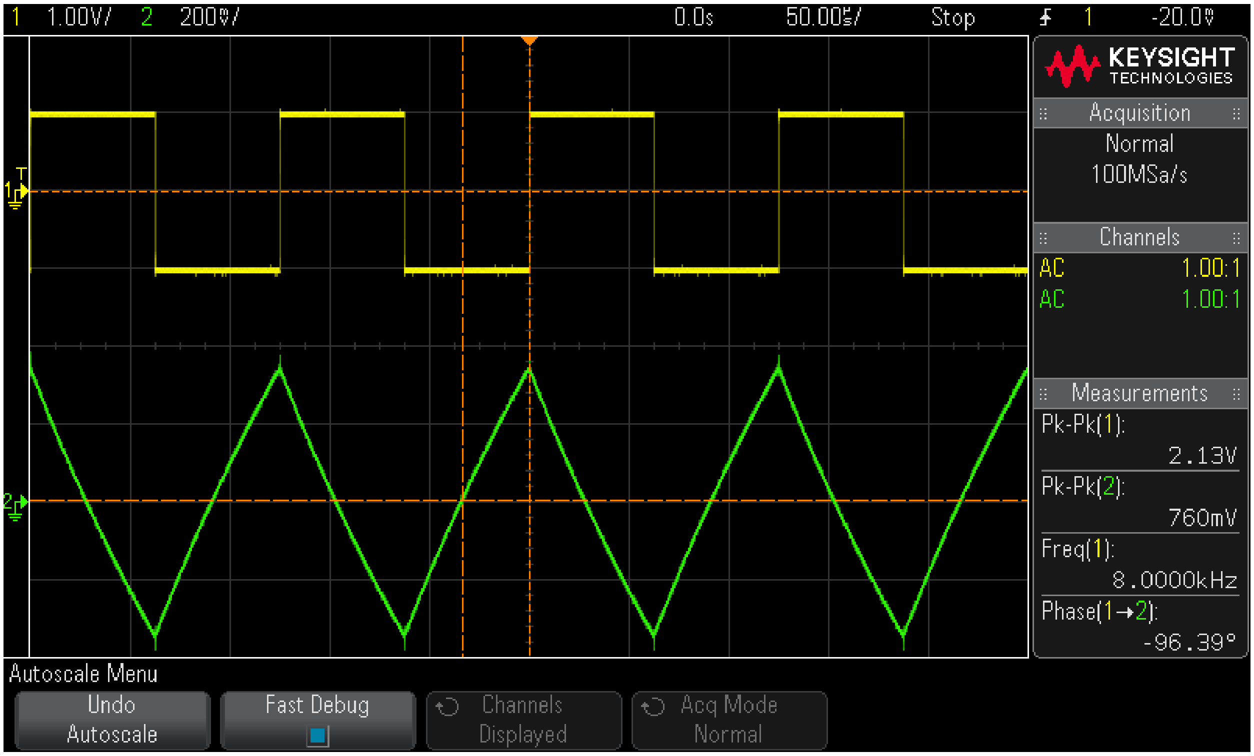
Last update: 2023A 20/10/22



Last update: 2023A 20/10/22

Print: input and output at $f_h = 10 f_p$

MSO-X 2012A, MY51290246: Wed Feb 28 10:36:28 2024



9. Op-amp as a summation circuit

- Refer to the preliminary report, section 10.
- NOTE: When beginning measurements, use the scope to make sure the output signal oscillates around 0VDC, since the calibration you did in section 2.2 might be off.

9.1. As in the previous section, use the waveform generator to output a square wave, then connect it to the integrator circuit in the bottom left corner of the board. Make sure that the integrator's output has the **same amplitude** as the input and that it is equal to 2VPP; the output will not fully convert into a triangular wave in this case, but that's okay.

9.2. What is the circuit you were requested to design in the preliminary report? Write it here:

$f(V_1, V_2) = ... -2V1 + 2V2$

9.3. Build the circuit you designed in your preliminary report, according to the instructions there. Use the waveform generator's square wave output and the integrator's triangular wave output as your inputs. Attach the following prints:

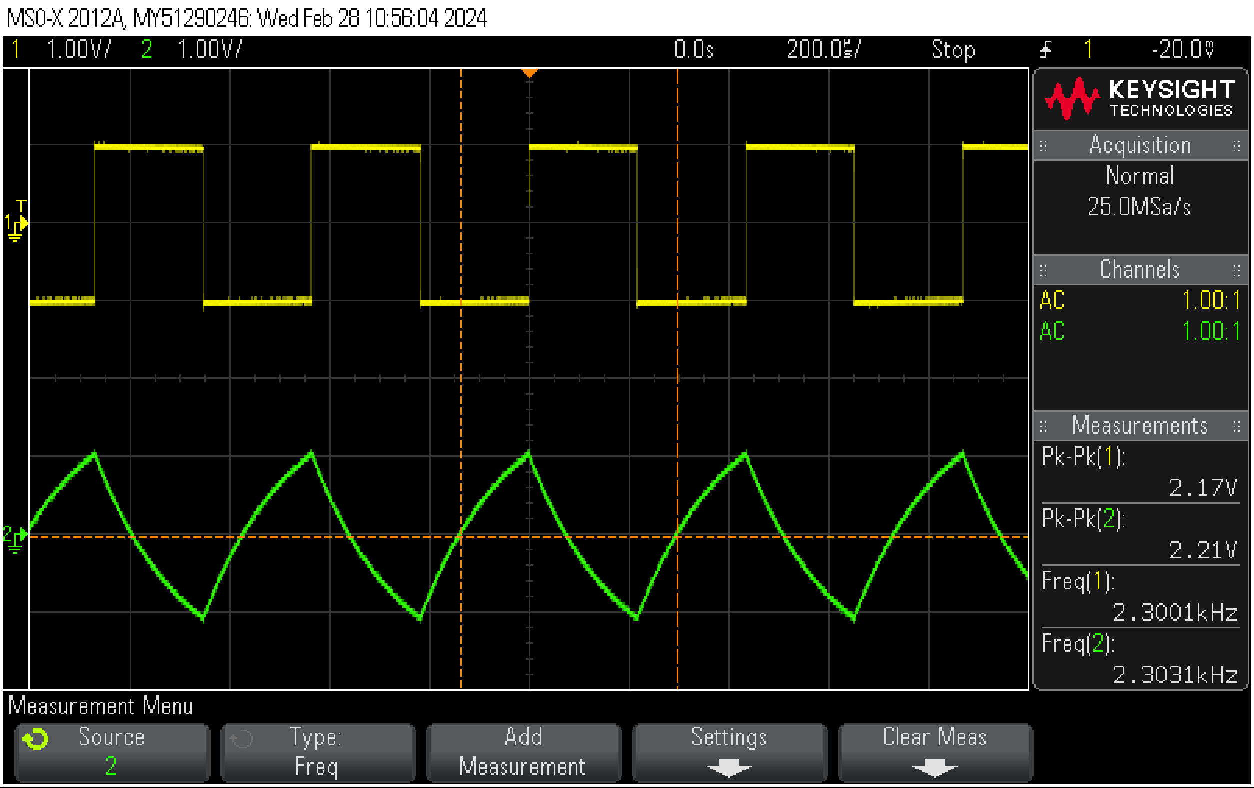
9.3.1. Both inputs with VPP and frequency measurements (total of 4 measurements).

9.3.2. The square-wave input, and your circuit's output, with VPP and frequency measurements (total of 4 measurements).

9.3.3. A single print with both inputs and the output, all in the same scale.

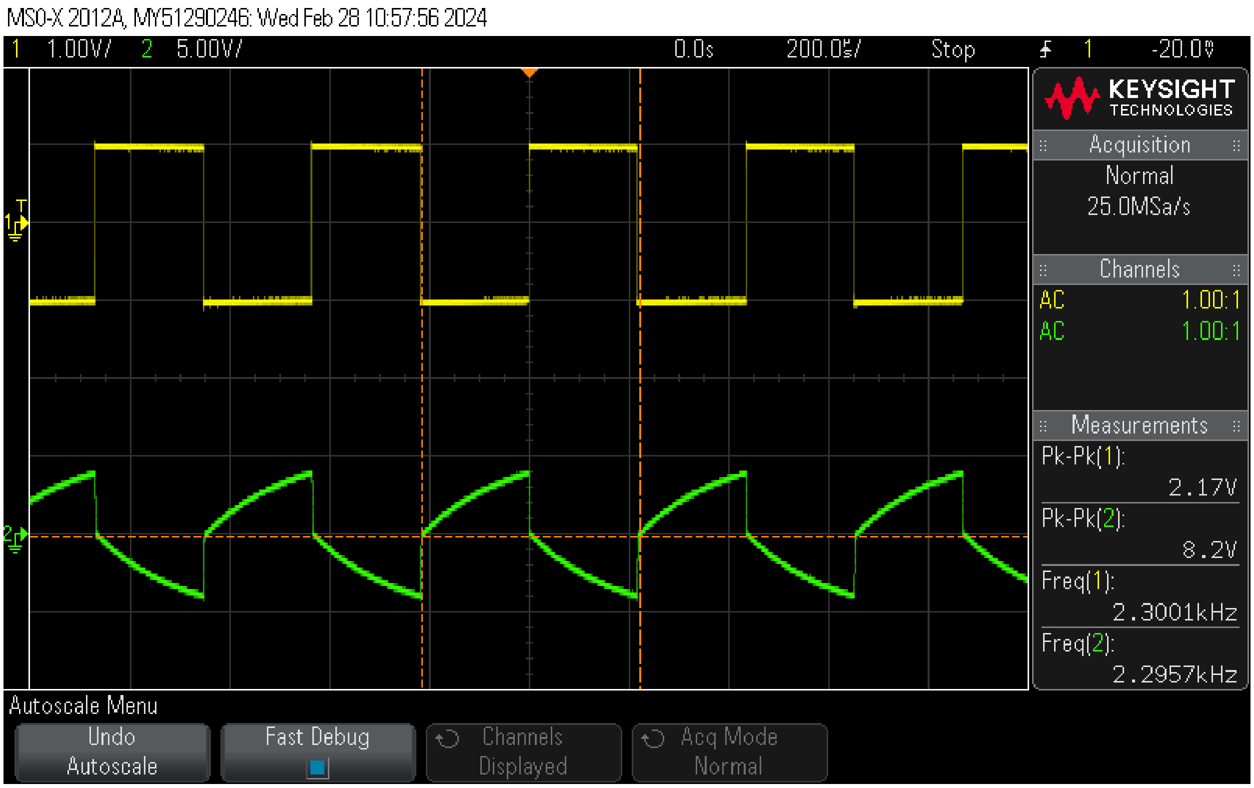
Last update: 2023A 20/10/22

Print: both inputs



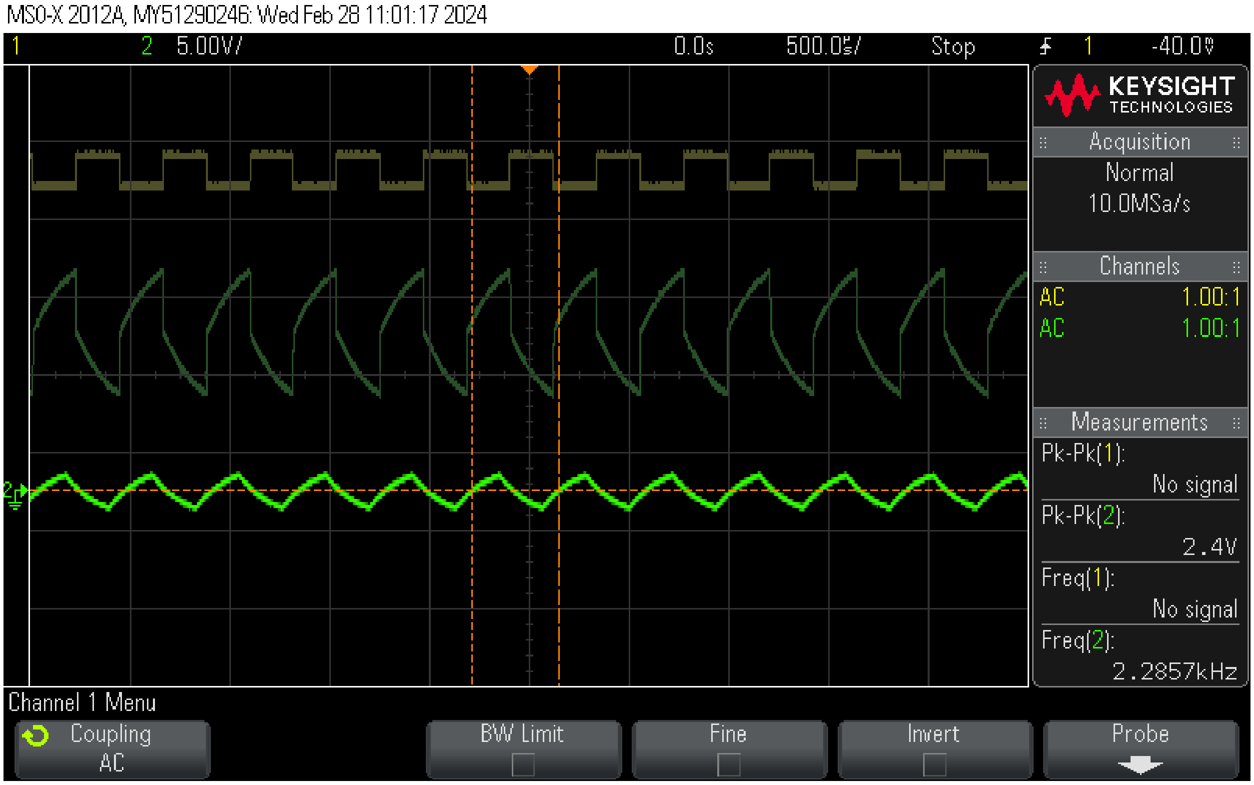
Print: square-wave input and output

Last update: 2023A 20/10/22



Print: both inputs and output

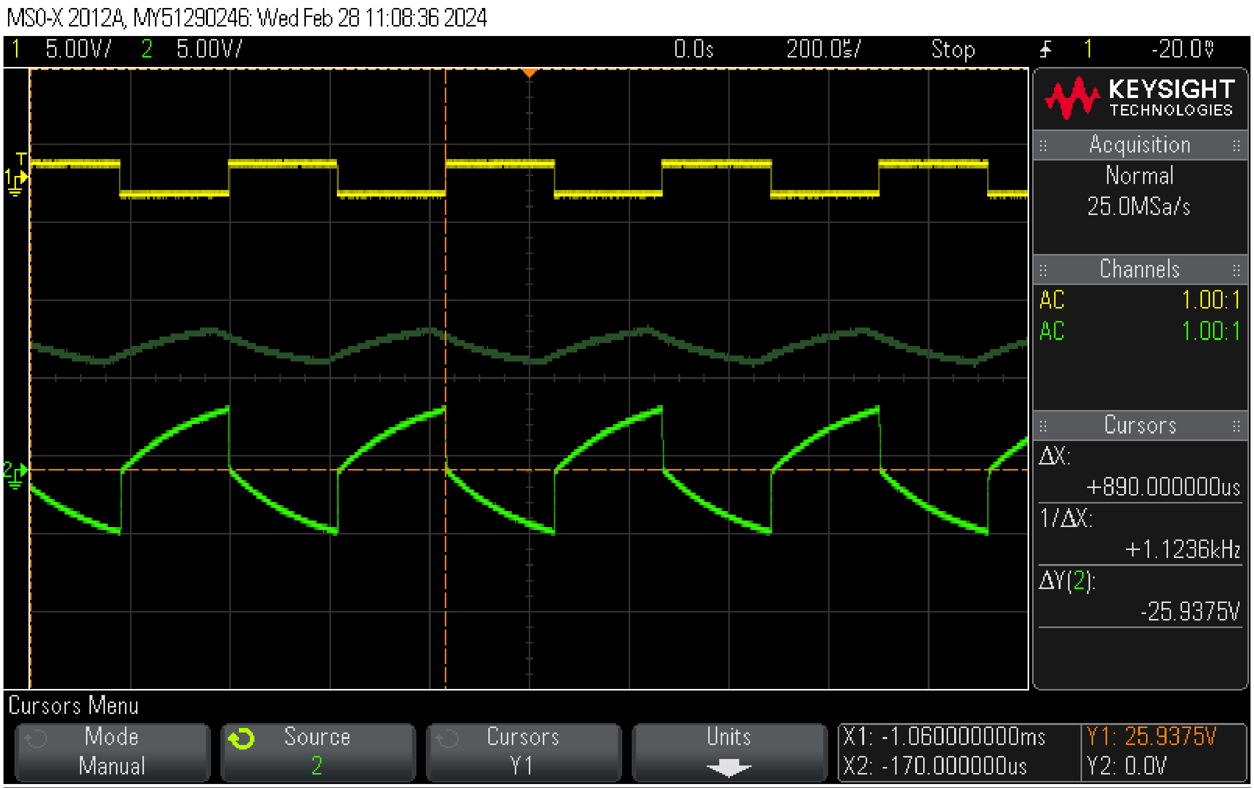
Last update: 2023A 20/10/22



9.4. Attach several prints, as you see fit (at least one, up to four), with cursor measurements proving that your design works.

Print: with cursors

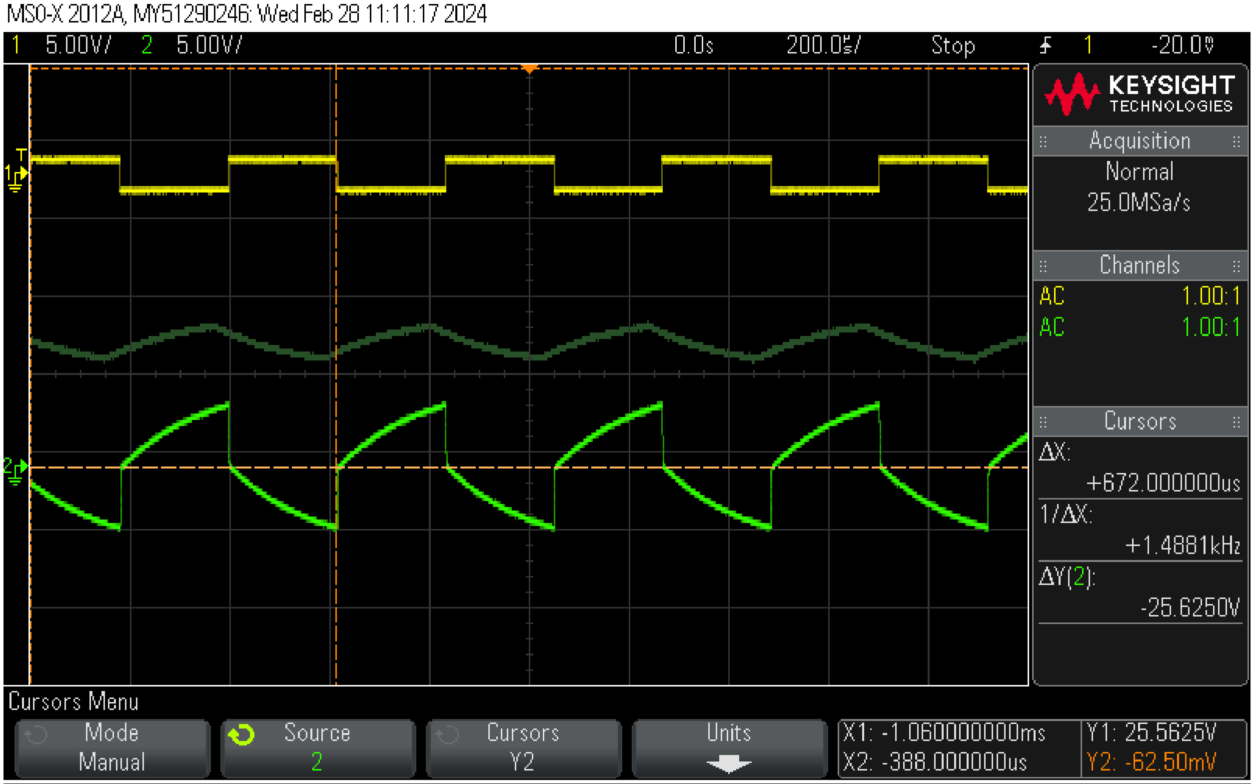
Last update: 2023A 20/10/22



נראה כי הגרף העליון זו הכניסה Vin1, האמצעית Vin2 והאחרונה - המוצא. ערך שתי הכניסות הוא 2Vpp וערך המוצא הוא 0 כפי שנדרש.

Print: with cursors

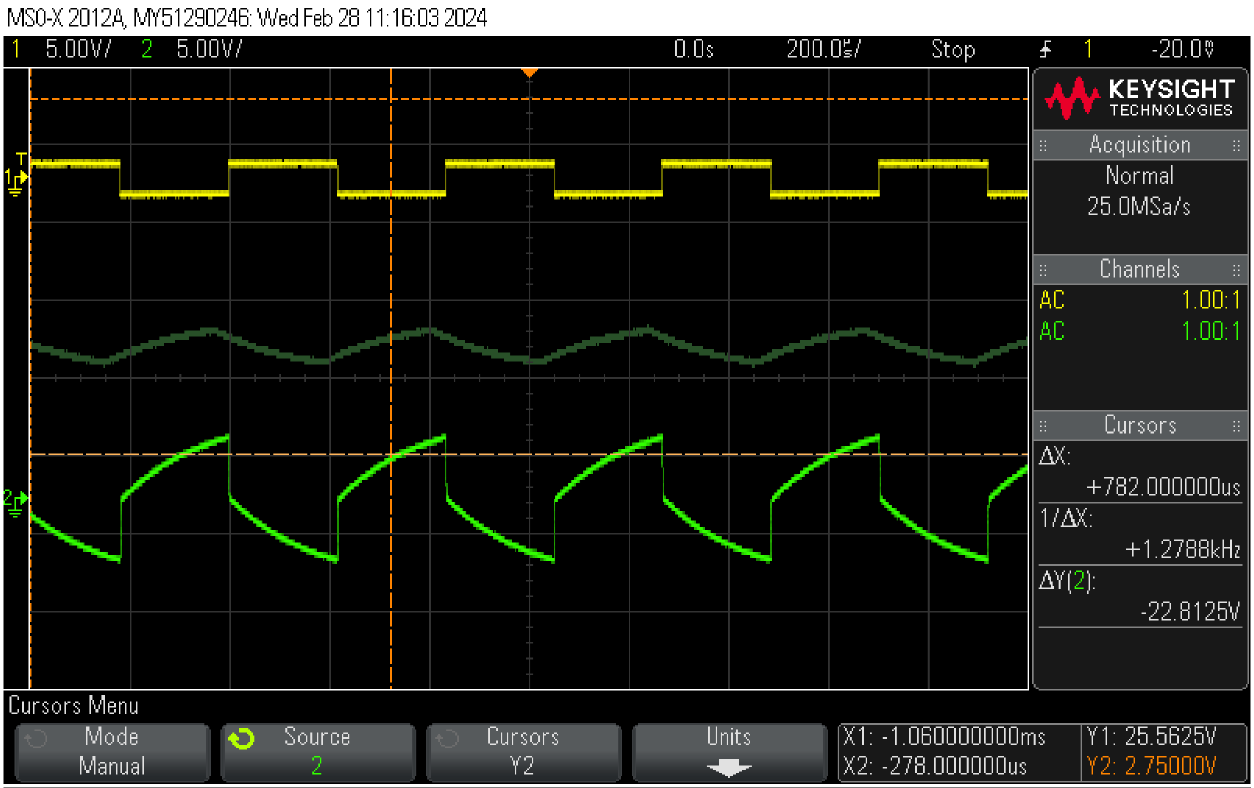
Last update: 2023A 20/10/22



גם כאן – ערך הכניסות הינו 2Vpp וערך המוצא כמעט 0 (עד כדי שגיאה במדידת הcurser) כפי שנדרש

Print: with cursors

Last update: 2023A 20/10/22



כאן נראה כי Vin1 הינו 2Vpp, הכניסה השנייה מאופסת ובמוצא קיבלנו 2.75V שזה קרוב מאוד לדוח המכין.

Print: with cursors

This concludes experiment #1.

You have reached to end of this experiment: ask the lab guide to write down the time. Hand in the preliminary report, and present the complete preliminary report of the next experiment before starting it.