SEMESTER PROJECT REPORT

NEMS RESONATOR THERMOMECHANICAL NOISE PREAMPLIFYING CIRCUIT

FAHRADIN MUJOVI

SCIPER: 274633

Special thank you to Professor Guillermo Villanueva for making this semester project possible. This project represents exactly what I wanted to study and learn.

Thank you to Professor Christian Enz for helping me with the noise analysis.

Thank you to Professor Giovanni Boero for giving a good testing method for TIA circuits.

TABLE OF CONTENTS

1 SEMESTER PROJECT PRESENTATION

This semester project was supervised by Professor Guillermo Villanueva. It was separated in two main parts. The first part was to resolve the problems a printed circuit board (PCB) had with a vacuum device.

As the second part was much more interesting than the first part, it also took significantly more time to work on it. It was, at first, the analysis and the testing of an already existing preamplifying circuit made to serve as an interface to a NEMS resonator and a lock-in amplifier. It was then the designing of a circuit that would work and be capable of amplifying very small signals.

In this report, I will describe and summarize the tasks I did during this semester project. I will also expose and explain some problems I encountered to, hopefully, justify some of the choices I made in order to solve them.

2 O-RING PCB

A microfluidic device designed by the NEMS laboratory had to be used in vacuum. A PCB was used to serve as the electrical interface between the vacuumed device and the outside. The vacuum was achieved by using an O-ring with a silicon sealing. The PCB used for the interface was 2-layered, the tracks present on the surface induced important enough prominences for the vacuum sealing to be compromised. The solution to that problem was to transform the 2-layered PCB in a 4-layered PCB to make the hindering tracks go through an intermediate layer when passing though the vacuum sealing location. At first, buried vias (vias starting at an external layer and going to an intermediate layer) were used in the design but those came out to be too expensive. The alternative used was the usage of through vias (vias going through the whole thickness of the PCB) with an epoxy filling to prevent leakage through them.

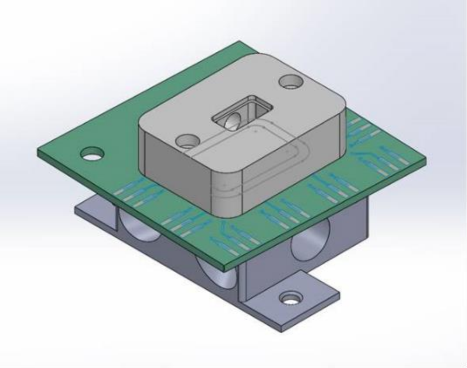


Figure 1 - Representation of the vacuum sealing device put on the PCB. An upper and lower part are needed for the device to work.

In addition of changing the number of layers, there was some light reorganization work to be done on the PCB according to the instructions of the PhD student Maillard Damien. Here is a non-exhaustive list of the tasks I did for this PCB:

- Transition from 2 layers to 4 layers
- Size of the whole PCB changed
- Arranging of the wire-bonded pads to minimize space between them according to Eurocircuits specifications [1]
- Arranging intermediate layer tracks location to ensure 1 $[mm]$ distance from future drilled holes.
- New footprint designing for newly specified SMA connectors (ref. *Amphenol RF 132255* [2]).

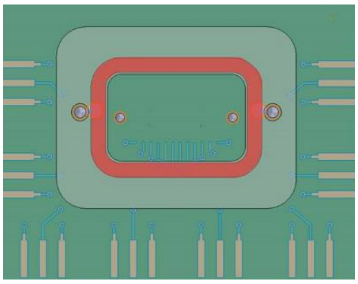


Figure 2 - Representation of the O-ring silicon sealing. The old design had the tracks go under the sealing and compromise it.

This part was secondary in the context of this project compared to the section 3. Piezoelectric NEMS Thermomechanical Noise preamplifying circuit.

3 PIEZOELECTRIC NEMS THERMOMECHANICAL NOISE PREAMPLIFYING CIRCUIT

During this semester project, the goal was to make a preamplifying circuit for a NEMS (Nano-Electromechanical System). The NEMS itself was made to observe and measure the thermomechanical noise present in all systems. As the NEMS system is a resonator, it will amplify all excitations near its resonance frequency. And as the thermomechanical noise has a constant spectral density around the NEMS resonance frequency, it will be amplified by the resonator. This oscillation is then transformed in a signal with piezoelectric materials. This signal is to be observed by, for example, a lock-in amplifier. The problem comes from the fact that the signal is very small: its square root spectral density is assumed to be around 900 $[fA/\sqrt{Hz}]$. At this level, a lock-in amplifier bottom noise level will drown the signal and make it unobservable. A low noise preamplifying circuit is needed to amplify the signal coming from the NEMS resonator enough so that it will be observable with a measuring device.

Every amplifying circuit will have noise. The key in this project is to refer all the system noise to the input of the system and directly compare it to the NEMS resonator's signal level. In the last step of this project, I will only make a theoretical amplifying circuit with some simulations and explain and analyze the different key parameters for the making of a low noise amplifying circuit. It is important to note that a real circuit will have other issues concerning its noise minimization that will need to be taken in account in order for the system to work. Some of these issues will be quickly addressed near the end of the report.

The lumped element model used during this whole project for the piezoelectric NEMS resonator is very similar to a diode model. It consists of a current source, a resistor, and a capacitor in parallel to each other as shown in Figure 3.

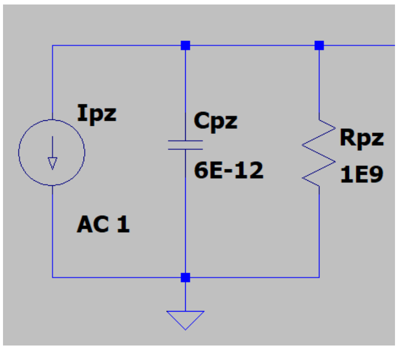


Figure 3 - Lumped element model (LEM) of the piezoelectric NEMS resonator. For this specific case, the capacitance of the model is represented by a 6 $[pF]$ capacitor and the resistance is represented by a 1 $[G\Omega]$ resistor in parallel to the current source.

3.1 INTEGRATOR ANALYSIS

The second part of the project consisted of analyzing an already existing PCB. It was an integrator circuit made by a previous PhD student. It was designed and fabricated but not fully tested. From my understanding, it was not working properly so I was assigned the task to test it and see why it did not work.

3.1.1 INTEGRATOR CIRCUIT FUNCTIONING

Using an integrator was an interesting choice since the piezoelectric read-out of the NEMS system will actually emit charges proportional to the rate of movement of the NEMS device. The current injected in the system will be $I = \frac{dQ}{dt}$. An integrator circuit will integrate this current and allow us to have a signal in Volts directly proportional to the charges injected in the system. This allows to have a charges-to-voltage ($[C]$ to $[V]$) transducing circuit.

To achieve an integrating effect with a circuit, it basically needs to have an early pole in its transfer function to achieve a $-20\ [dB/dec]$ slope on the wanted bandwidth with a gain high enough for the signal to be perceivable.

3.1.2 *ADA4817* AMPLIFIER

The chosen amplifier was the *ADA4817* [3]. I assume it was chosen because of its low input noises ($4\ [nV/\sqrt{Hz}]$ for its input voltage noise and $2.5\ [fA/\sqrt{Hz}]$ for its input current noise) and fast and wideband behavior ($\geq 410\ [MHz]$). I will go into more details as to why these parameters are so important in section 3.2.2. Chosen amplifier .

3.1.3 CIRCUIT EXPLANATION

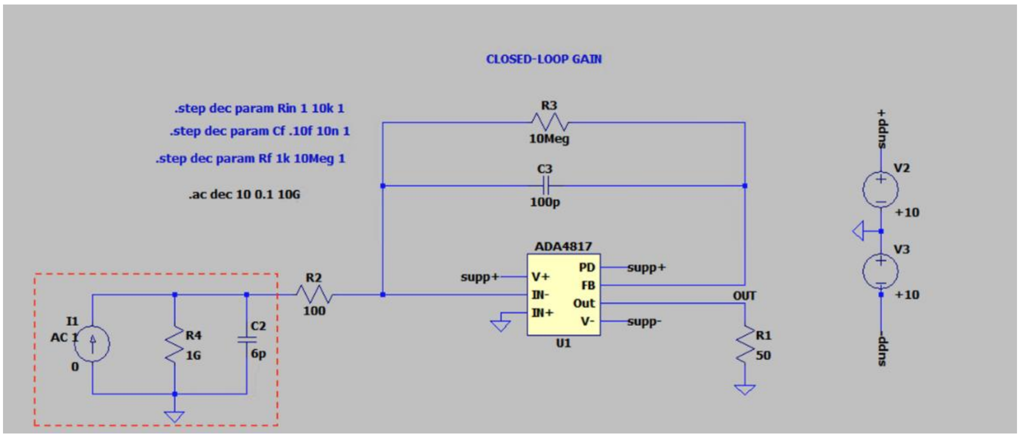


Figure 4- Integrator circuit designed by a previous PhD student. Taken from the LTSpice simulation. The U1 block is Spice auto-generated symbols with the correct model taken directly from Analog Devices. Figure taken from resources provided by Professor Guillermo Villanueva [4].

The circuit from Figure 4 has two poles. The first is at $1\ [kHz]$ and provides the integrating effect with a slope of $-20\ [dB/dec]$. The second pole comes at $1\ [GHz]$ and sets the usable bandwidth. These poles can be seen in Figure 5.

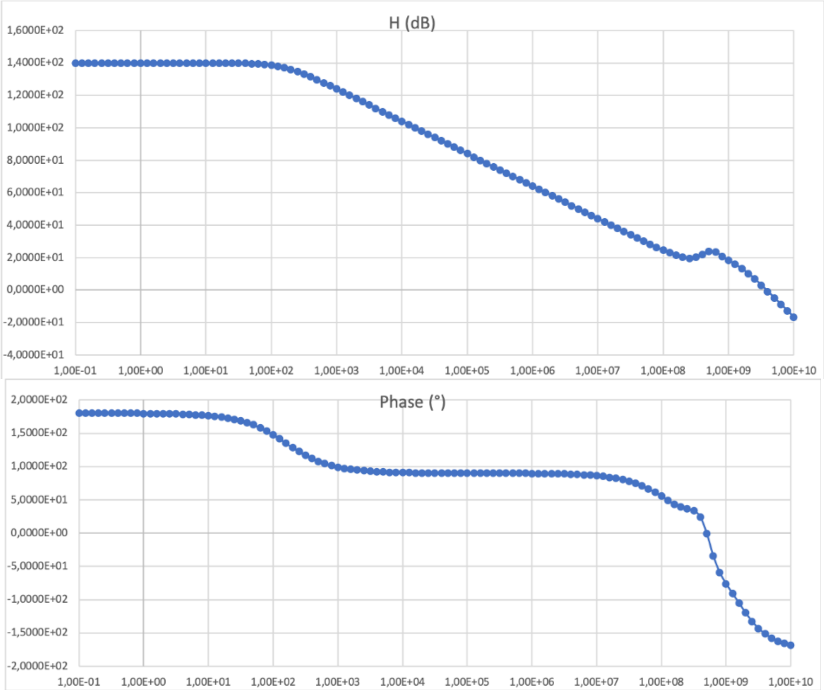


Figure 5 - Graphs showing the transfer function amplitude in [dB] (upper graph) and phase in [degrees] (lower graph). We can easily identify the two poles at 1 [kHz] and 1 [GHz]. We also notice some peaking at the second pole and a steep phase-shift happening around the unity-gain, this might be synonym of instability. Taken from [4].

The following circuit was previously designed. It is a two-stage circuit with the first stage being the integrating circuit seen previously and the second stage being a simple 100-gain stage. The 10 $[\mu F]$ and 0.1 $[\mu F]$ capacitors are advised by Analog Devices to add some filtering to the power supplies [3]. When amplifying low signals, the power supply can be a source of problems even with a high *Power Supply Rejection Ratio* (PSRR).

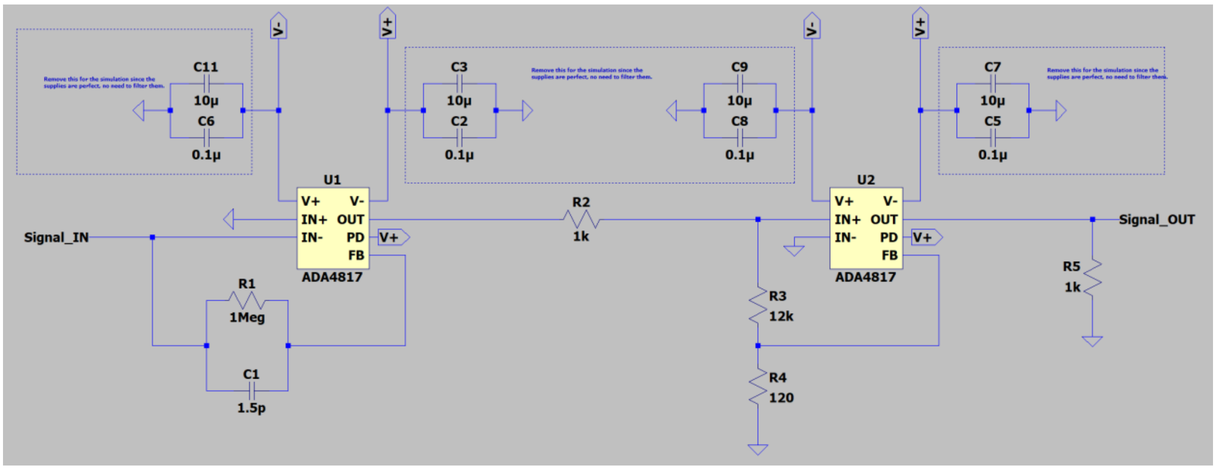


Figure 6 - Complete schematic of the charge amplifying circuit. Two-staged circuit with power supply bypassing. Since this circuit was modelized in LTspice, the power supply bypassing capacitors are not needed as the supplies are perfect models. They were added here for the sake of understanding and showing the complete circuit.

3.1.4 PROBLEMS

This amplifier has most of its specifications suited for this application. For this kind of application, where the currents that the resonator outputs are extremely small, the input bias current of the amplifier used becomes a key parameter. The input bias current of an amplifier is the current that will be taken by its inputs. For a bipolar amplifier, it is the base current taken from its bipolar transistors. For a FET or CMOS transistor, it is a small leakage [5]. For the *ADA3817*, the input bias current is 2 $[pA]$ (20 $[pA]$ at maximum). This amount of current taken by the amplifier is too high. If the input bias current is too high, there will be less current going through the feedback loop, meaning that the amplifier will lose its *ideal-amplifier-like* characteristics.

Another point to be further analyzed might be the stability of the system. There is a higher risk of having an unstable device if the circuit has a capacitive load impedance or a capacitive input impedance. In this case, the input capacitance is equivalent to:

$$C_{in} + C_{PZE} + C_f + C_p = C_{equ,input}$$

With C_{in}, the amplifier differential input capacitance. C_{PZE}, the piezo device LEM equivalent capacitance. C_f, the feedback capacitance and finally C_p, a representation of all the parasitic capacitances in the circuit. Generally, a transimpedance amplifying circuit with a capacitive input will need a feedback capacitor to damp the model enough to avoid instability. The higher C_{in}, C_{PZE} and C_p, the higher will have to be C_f for stabilization. This issue will also be further explained in the section 3.2.4.

Stability of the circuit when showing the design of the proposed amplifying circuit.

This system might be stable since the bigger the feedback capacitor, the greater will be the damping of the system, meaning that it will be less likely to reach instability. In the case of an integrator, the feedback capacitance needs to be of high value since a higher value will move the pole to lower frequencies, enabling the integrating effect earlier as explained before. This issue would need further analysis to confirm the stability.

3.2 TRANSIMPEDANCE AMPLIFIER

A Transimpedance Amplifier (TIA) is a circuit which will generally take a current $[A]$ as the input signal and give a voltage as its output signal $[V]$. Its gain is expressed as the $output/input$ in $Ohms$ since $[V/A = \Omega]$. Since it is expressed in $[A]$, the noise analysis will be expressed as a current noise and referred to the input of the amplifier since we want to compare it directly to the signal levels coming from the resonator.

3.2.1 WHY USE A TIA?

I chose to use a TIA since they are widely used for diode amplifying circuits. This implies that TIA circuits are generally well documented, and many examples already exist. As we saw in section 3. Piezoelectric NEMS Thermomechanical Noise preamplifying circuit, the resonator model is very similar to a diode model. Using a TIA was helpful since many advices and interesting points are often mentioned in the respective datasheet.

3.2.2 CHOSEN AMPLIFIER

A TIA circuit is straightforward, it consists of an amplifier negatively counter-reacted with a high resistor in its feedback loop. The feedback resistor R_f is the DC gain of the circuit. As we briefly saw previously, we generally add a feedback capacitor C_f to ensure better stability if there is a notable input capacitance.

The chosen amplifier was the *LTC6268-10*, an ultra-low noise FET input amplifier adapted for TIA circuit applications [6]. Many of its key characteristics will be reviewed in this report to show the good compatibility for this application.

Input Voltage and Current noise:

Its low voltage and current noise, respectively of 4 $[nV/\sqrt{Hz}]$ and 7 $[fA/\sqrt{Hz}]$, are adequate for this application. The current is low enough to be neglected in this case. The voltage noise is also very low and will only be relevant at high frequencies (≥ 1 $[MHz]$).

Input Bias Current:

As explained previously, the input bias current is the amount of current that will go in the amplifier's inputs due to its imperfections (i.e. the non-infinite input resistances). For this amplifier, its input bias current is equal to $I_B = \pm 3$ $[fA]$. It is three orders of magnitudes lower that the input bias current of the ADA4817.

Bandwidth:

The *LTC6268-10* has generally a very wide bandwidth due to its very high gain bandwidth product (GBW) of 4 $[GHz]$. It achieves this very high GBW due to the fact that it is a decompensated amplifier. In normal compensated amplifiers, the open-loop bandwidth is already stably fixed. However, the "trade-off" for this wideband application comes from the fact that this amplifier will only be stable at a gain ≥ 10 [6].

Input Capacitance:

The last key parameter is the input capacitance of the amplifier itself. For low noise amplifiers, the input capacitance is generally low, but the *LTC6268-10* manages to have very low input capacitance of 0.1 $[pF]$. This implies that the main focus for the circuit input capacitance will be on the circuit and PCB design that could add some parasitic capacitances. Other capacitances will have to be avoided such as cable capacitances.

3.2.3 CIRCUIT DESIGN

This transimpedance circuit has a $10\ [M\Omega]$ gain. Such a high resistor was chosen, firstly, to augment the gain (an explanation is given in the section 3.2.4. Stability of the circuit for the feedback resistor and capacitor values). Secondly, a high feedback resistor value will minimize the Johnson Noise coming from it. To justify this, a full noise analysis was done to verify what were the main noise sources in a typical TIA circuit.

Every noise source was transformed in its equivalent current noise and referred to the input of the circuit. Generally, every resistive component will generate a noise, called the *Johnson Noise*. In this case, we will ignore the noise that the Resonator's equivalent resistor R_{PZ} (from Figure 3) will give since it is part of the signal we want to analyze. So, the only other resistive noise we will take into account comes from the feedback resistor of our circuit R_f, called i_{n,R_f}. Its expression is as follows:

$$i_{n,R_F} = \sqrt{4k_bT/R_f}$$

With k_b, the Boltzmann constant and T the temperature. We obtain, for a resistor of $10\ [M\Omega]$, a current noise of $i_{n,R_F} = 40.69\ [fA/\sqrt{Hz}]$.

We then need to consider the noise that the amplifier itself will be adding to our circuit. As mentioned in the previous section, the *LTC6268-10* is an ultra-low noise amplifier. Meaning that its input referred current and voltage noise are very small. For the current noise source, we can directly add it to the total noise source, we will see later that it will be negligible compared to other sources. On the contrary, the voltage noise source will be applied to all the passive components and give an equivalent current dependent on the frequency. The impedance that the voltage noise sees is the following (see Figure 7):

$$Z_{PZ}//Z_f = \frac{Z_f \cdot Z_{PZ}}{Z_{Pz} + Z_f}$$

By Ohm's law we deduce that the equivalent current noise of the amplifier's voltage noise is:

$$i_{n,equ} = e_{n,Amp} \cdot \frac{Z_f + Z_{PZ}}{Z_{Pz} \cdot Z_f}$$

To add all of these contributions we need to take their squared values:

$$i_{n,tot} = \sqrt{i_{n,R_f}^2 + i_{n,Amp}^2 + i_{n,equ}^2}$$

Or more interestingly:

$$i_{n,tot} = \sqrt{i_{n,R_f}^2 + i_{n,Amp}^2 + e_{n,Amp}^2 \cdot \left[\left(\frac{1}{R_{PZ}} + \frac{1}{R_f}\right)^2 + \omega\left(C_{PZ} + C_{in} + C_f + C_p\right)^2\right]}$$

We see that the contribution of the resistor is constant. Since $R_{PZ} \gg R_f$, we can ignore R_{PZ}'s contribution. The noise contribution of the capacitors is dependent on the frequency. Their contribution will begin to be relevant at high frequencies. Fundamentally, the main contribution around $200\ [kHz]$ is from the feedback resistor. We see here that the greater its value, the lower its current noise will be. This motivates the use of higher value resistors and permits the usage of higher gains in a TIA circuit.

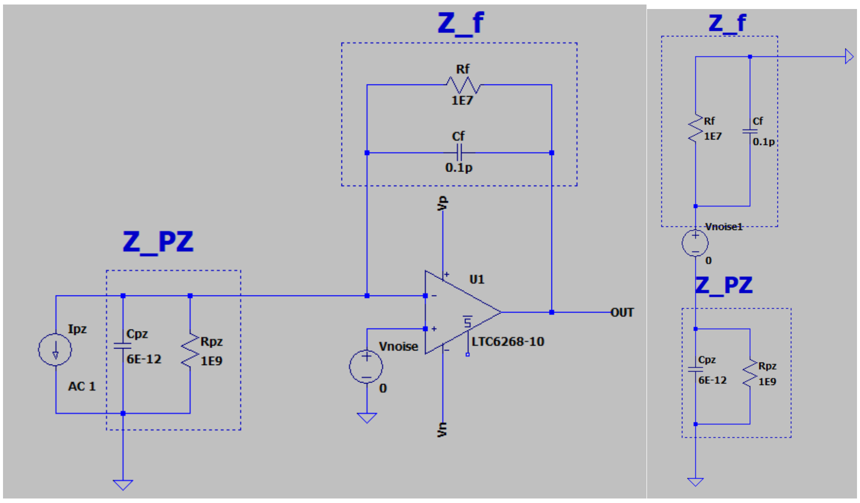


Figure 7 - Complete circuit on the left side. Equivalent circuit seen by the Noise Voltage source on the right side. To get the equivalent impedance seen by the noise voltage source, we simply need to ground all the other contributions. We see that the impedances Z_{PZ} and Z_f will be in parallel.

The mathematical model can be compared with an LTspice simulation to see if it is reliable.

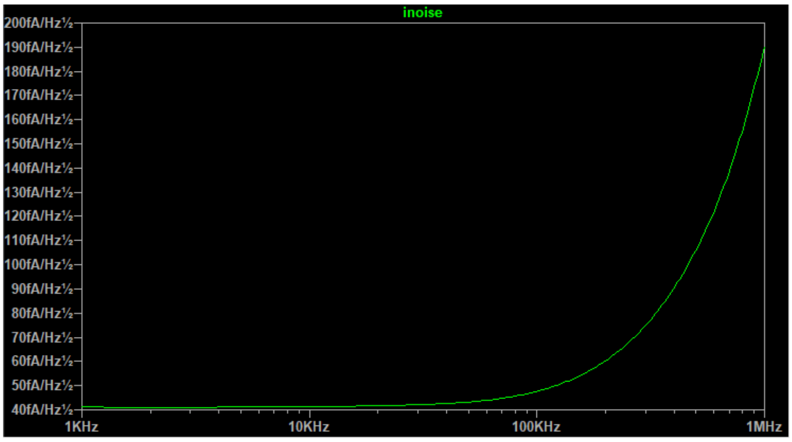


Figure 8 - Input referred current noise spectral density of the circuit simulated on LTspice.

From Figure 8, we see that at low frequencies (i.e. from $1 - 100\ [kHz]$), the dominant noise source is mainly i_{n,R_f} at $40\ [fA/\sqrt{Hz}]$. At higher frequencies, we see that the voltage noise of the amplifier dominates.

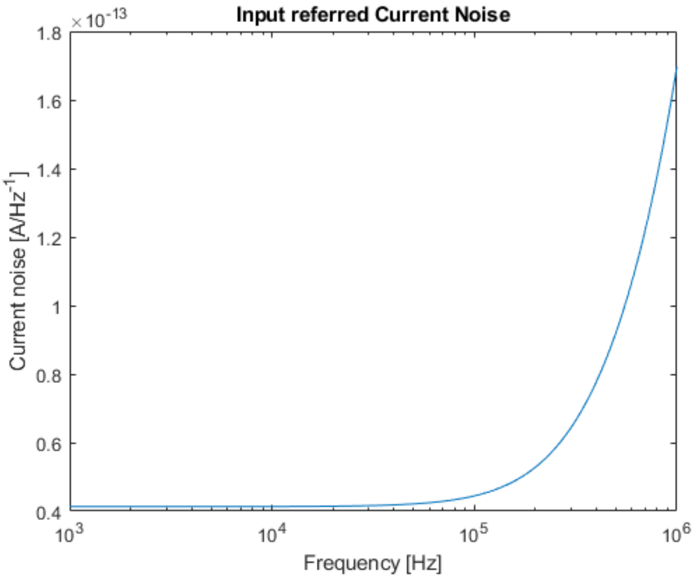


Figure 9 - MATLAB computation of the input referred current noise mathematical model obtained previously.

Figure 9 show the results obtained from the mathematical noise model. We see that the model fits very well the simulation with a slight difference at high frequencies: the simulation shows 5% more current noise. This is certainly due to a simplification made in the mathematical model used.

3.2.4 STABILITY OF THE CIRCUIT

As seen briefly previously, an amplifying circuit might be instable if it has a capacitive input. At first glance, an instability can easily be noticed if a circuit has very high peaking in his transfer function. In the case of this circuit, we can easily see in the Figure 11 that the peaking is greatly reduced with the addition of a 0.1 $[pF]$ feedback capacitor. The drawback is that the bandwidth suffers from the addition of a capacitor in the feedback loop. In our case, the bandwidth isn't very important as long as the gain is important enough.

Another way to verify that is to look at an amplifying circuit as a block system (see Figure 10). Xi is the input signal and Xo is the output signal. We have the following relations:

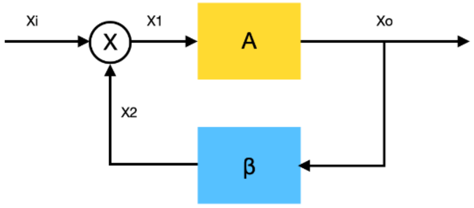


Figure 10 - Block diagram equivalent of an amplifying circuit. Taken from the OneNote file [4]. A is the amplifier open-loop gain and β is the feedback factor. The product Aβ is called the loop gain.

$$Xo = X1 \cdot A, \quad X2 = Xo \cdot \beta, \quad X1 = Xi - X2$$

Using these equations, we can deduce the closed-loop gain of the circuit:

$$A_f = \frac{A}{1 + A\beta}$$

For the gain to be instable, we see that two conditions are needed (see OneNote [4] and TI document [7] for more documentation regarding this topic):

1. The magnitude of the loop gain $|A\beta| = 0dB$
2. The phase of the loop gain $A\beta = \pm 180°$

When these conditions are met, the system reaches instability. On a simulation program like LTspice, we can graphically compute the product $A\beta$ and see if the result is stable. The stability of the circuit is guaranteed according to the datasheet of the LCT4248-10 [6], see section 4.1. Technical Comments for more details.

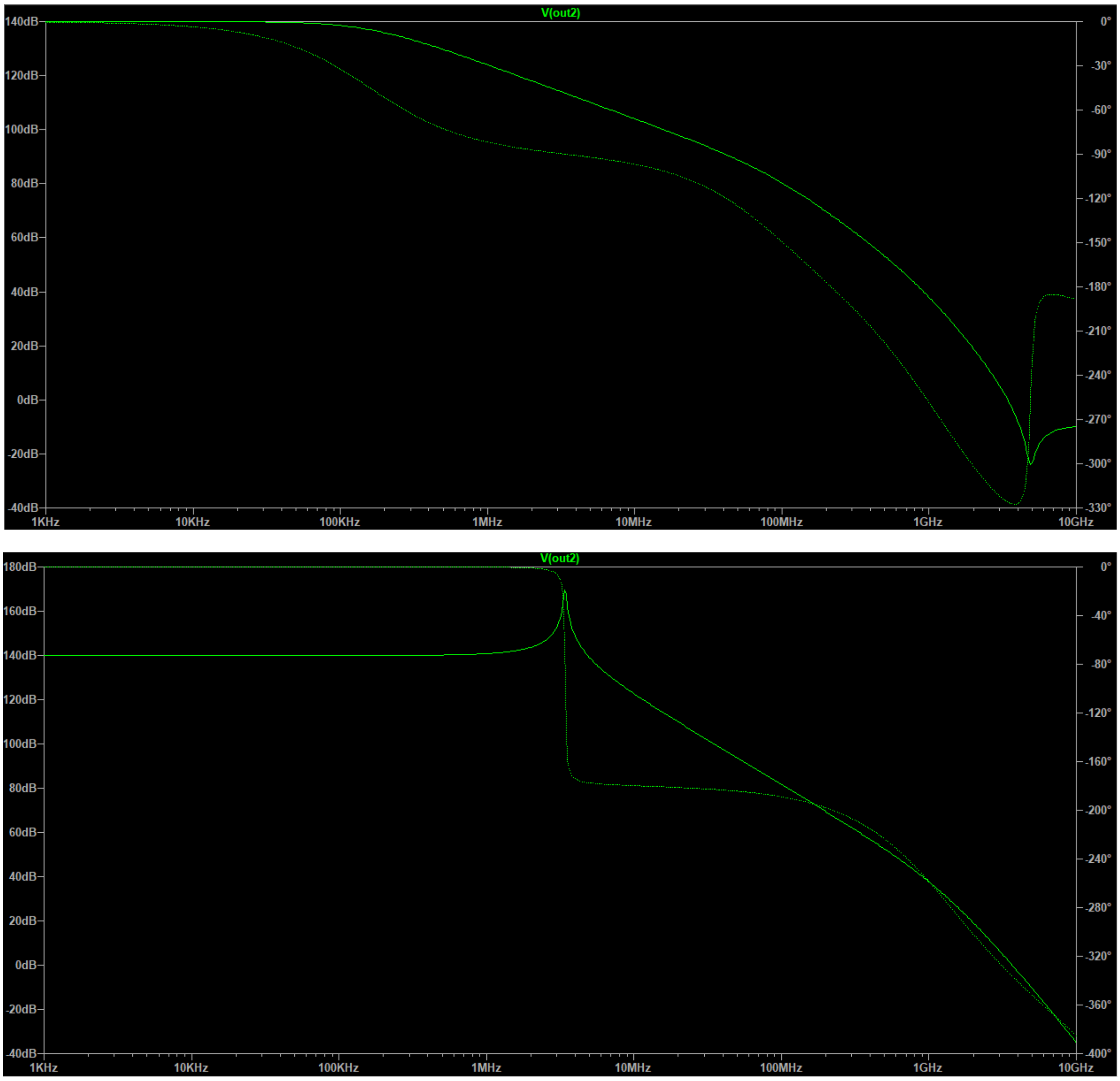


Figure 11 - Transfer function of the TIA circuit. Upper figure is with a compensation feedback capacitor. Bottom one is without a feedback capacitor. We notice a very high peaking in the transfer function without a feedback capacitor.

It is possible on simulation programs to plot separately the loop-gain A and the inverse of the feedback factor $1/\beta$ and look at the intersection of the two curves. We know that in a Bode diagram, a slope of $-20\ [dB/dec]$ gives a phase shift of $-90°$. If we have a slope of $-40\ [dB/dec]$, it implies that there is a high risk of having a $-180°$ phase shift and making the system instable. In the Figure 12, we see that the two curves intersect at approximately $2.5\ [MHz]$. If we look at that intersection, by subtracting the $1/\beta$ to the open loop gain (to obtain the equivalent of the $A\beta$ product in a linear scale), we need the result to have a minimum of $-20\ [dB/dec]$ to ensure stability. In this case, we see that the resulting slope will be equal to $-20\ [dB/dec]$.

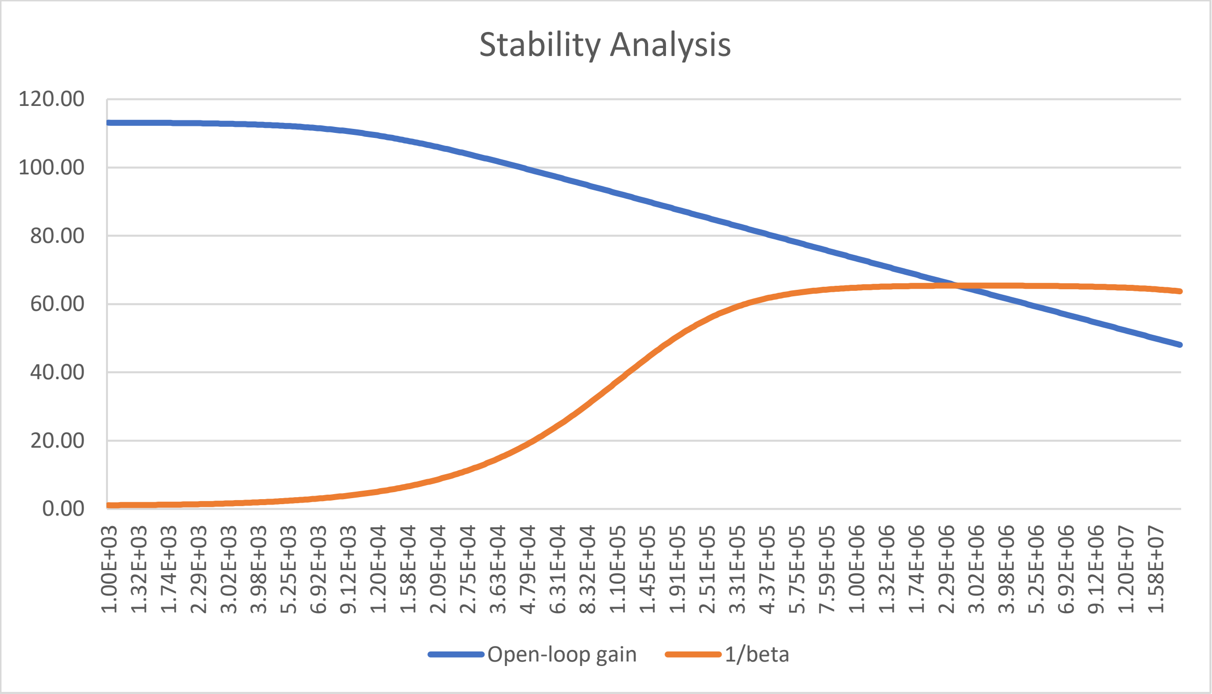


Figure 12 - Stability analysis plot. The first intersection of the two curves help us tell if the circuit is stable. When we subtract the orange curve to the blue to obtain the loop gain, we need to have at most an overall $20\ [dB/dec]$ *slope.*

When studying stability in LTspice, there are special methods to achieve simulating and viewing these functions. Analog Devices made a quick tutorial on how to get the open-loop gain (see ref. [8]). The $1/\beta$ function is simply the gain seen at the input of the amplifier compared to the output. One simple way to do so, is to use a voltage source at the input of the amplifier to compute the transfer function of the system. It is important to also consider the noise present in the system since it can also participate in the instability of the system. Fortunately, LTspice has a noise analysis which can also give the gain of $1/\beta$. For more details, see Appendix 5.2. Feedback factor curve plotting on LTspice.

3.2.5 PROBLEMS

With each amplifier circuit having a capacitive input, there are some points to consider when designing the PCB. It generally comes down to the parasitic capacitances introduced by the PCB.

Adding cables between the NEMS resonator and the amplifying circuit will introduce a supplementary capacitive load on the input of the circuit, meaning that the amplifier will be more likely to be instable. The circuit can take a heavier capacitive input load up to certain point and still remain stable, but precautions should be taken during the design of the PCB to minimize that.

Another sensitive point is the feedback capacitor. Introducing too much parasitic capacitance in the feedback loop could cripple the bandwidth of the circuit. It should not change too much the stability of the circuit if there is more capacitance added it the feedback but since it is a gain-of-10 stable capacitor, it is still needed to have $C_f < C_{in}/10$ (see *LTC6268-10* datasheet [6]).

Finally, the input bias current needs to be kept as low as possible in this application. To do so, Analog Devices have given multiple design methods in the datasheet of the LCT6268-10 [6]. Some more information is given in the reference document [9] where a full transimpedance amplifying PCB is shown and analyzed. In their example, the feedback parasite capacitance is reduced to some femtofarads.

4 CONCLUSION

This semester project was extremely interesting and enriching. I managed to deepen my knowledge in the making of amplifying circuit. Many concepts were not introduced, or only briefly introduced, during my bachelor electronics courses. Nearly all of the concrete and real imperfections of an Op Amp were new to me. This project was also the opportunity for me to put into practice the freshly acquired knowledge for noise in such circuit from the course *Smart Sensors for IoT* from Professor Ionescu and Professor Enz. This project will contribute to completing the bridge between my theoretical knowledge and the practical world.

I also had the opportunity to deepen my practical knowledge for different programs. The most interesting things I learned were certainly with the program LTspice. I learned how to make a noise analysis, which is often needed for amplifying circuit. I also could learn how to better study the stability of a circuit. As a circuit becomes more complex, it becomes also extremely difficult to study its stability by hand so the stability study with simulation programs becomes essential.

4.1 TECHNICAL COMMENTS

At first, I put a priority on keeping a constant gain up until $1\ [MHz]$ since it was part of the specifications I was given. After discussing with Professor Villanueva, I understood it was more important to prioritize low noise and high gain. In that regard, I decided to put a $10\ [M\Omega]$ resistor instead of a $1\ [M\Omega]$ in the feedback loop. With a 10-times bigger value for the resistor, the gain will be 10-times higher and the noise coming from it will approximately be 3 times smaller. It is a significant amelioration since we want to minimize the input noise of our circuit as much as we can. With this setup, we can nearly reach a 10-times lower input referred noise current compared to the signal we want to measure. The trade-off is that the circuit will begin decreasing at a $-20\ [dB/dec]$ at $100\ [kHz]$ instead of $1\ [MHz]$. I understand that the circuit will have a variable gain at higher frequencies, but this should not be a problem since the ultimate goal is to check the frequencies at which the NEMS will resonate, the measured amplitude will not be important.

I also added some practical information in the report and more particularly in the 5. Appendix. After discussing with Professor Villanueva, we agreed that it would be helpful to add such information for the next students that would continue the project and eventually need to use the same tools. Personally, I learned many different things while using LTspice during this project and I hope these bits of information will be helpful.

The *LTC6268-10* has a section in the datasheet explaining the different conditions to fulfill for a stable circuit. In the TIA circuit, all these conditions were verified. A special care needs to be taken for the ratio between the capacitance C_{in} and C_f. Since we are using a gain-of-10 stable Op Amp, the capacitance C_{in} needs to be at least ten times bigger than the C_f capacitance. In my opinion, it would have been enough to simply use that part of the datasheet to ensure the stability of the circuit, but the analysis method explained here was done by choice, to learn at least one proper method to verify the stability of a amplifying system.

5 APPENDIX

5.1 NOISE SIMULATION ON LTSPICE

A noise simulation spice command exists in LTspice. The command $.noise$ and has the following structure:

.noise V(*out_label*) *input_source type_of sweep points_per_decade start_freqency stop_frequency*

However, LTspice does an analysis at only specific points of the circuit. The first one is the where the selection of the output voltage was done (at the label *out_label*) and the second point is the place where the input source is selected (at the voltage/current source *input_source*).

By convention, in amplifying circuits, the noise source is added at the positive input of the amplifier. Generally, the current and voltage noise inputs of an amplifier is included by the provided fabricant's model. To analyze a circuit efficiently, a simple $0\ [V]$ voltage source can be added to positive input of the circuit. In the case of our TIA circuits, we can use the current source used to simulate the PZE resonator as the *input_source*. If the noise is not included in the Op Amp model, a workaround is possible (see Figure 13).

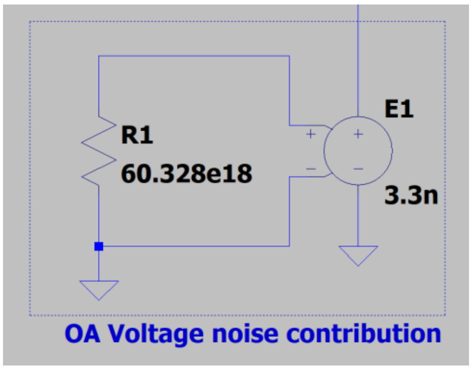
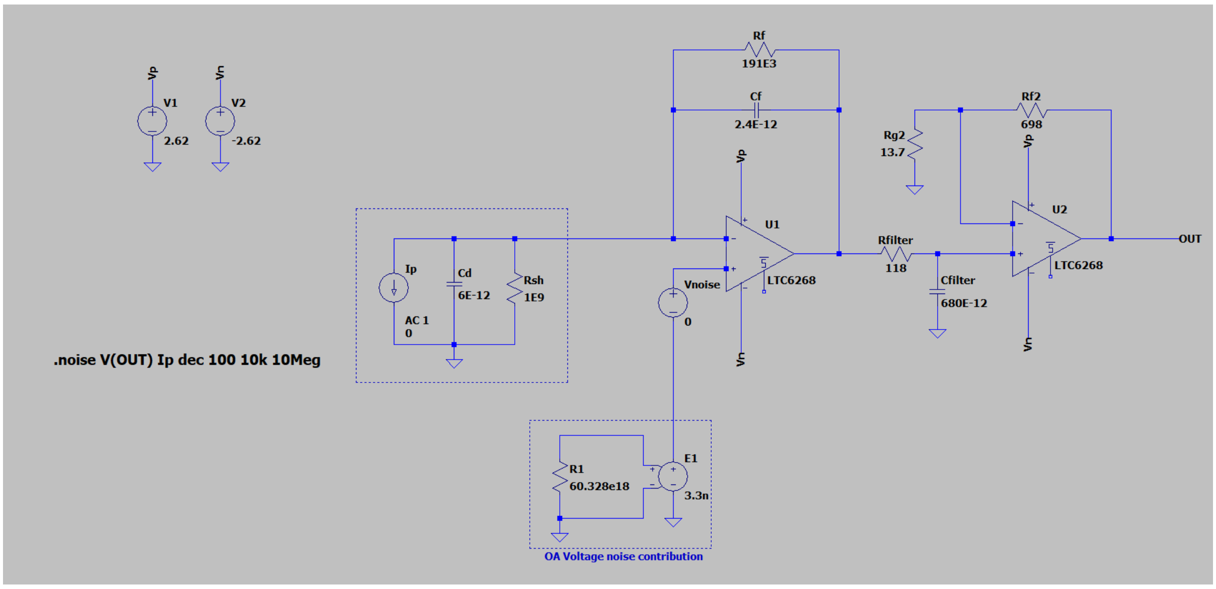


Figure 13 - Voltage noise equivalent in LTspice. The commanded voltage supply E1 (found in the LTspice library by typing "e") will output a voltage similar to what it sees on its inner input nodes. This works on the fact that LTspice simulates the Johnson Noise coming from resistors. With the right value put in the resistor, it becomes possible to have it simulate a voltage noise of $1\ [V/\sqrt{Hz}]$*. The value put in the commanded voltage source will take that proportion of the Voltage Noise seen at its input. In this case, E1 will emits a Voltage Noise of* $3.3\ [nV/\sqrt{Hz}]$*. The same can be done for a Current Noise supply (by typing "g" in the LTspice library).*



```
.noise V(OUT) Ip dec 100 10k 10Meg
```

Figure 14 - Typical noise analysis full LTspice circuit. Here, I assumed that the LTC6268 model didn't have its voltage and current noise input simulated so I added the voltage noise source as seen in Figure 13. The current noise source can be omitted here since it is generally negligible compared to the other sources (especially the feedback resistor of the first stage R_f*).*

To see the input referred noise in LTspice, it is simply done by right clicking on the plot plane (after running the noise simulation), selecting Add Traces and finally choosing the *inoise*.

5.2 FEEDBACK FACTOR CURVE PLOTTING ON LTSPICE

The feedback factor is simply the gain perceived if a voltage source is applied just before the feedback loop; it is the gain of the feedback loop. In the case of the circuit in Figure 14, we would need to apply a voltage source at one of the amplifier's input (since they virtually have the same voltage in a normal functioning state). The noise analysis is useful for this since we can use the same voltage source to analyze the gain perceived by feedback loop. When doing the noise analysis, a *gain* trace can be added on the plot space in LTspice (by right clicking on the plot space and selecting *Add Traces*.

5.3 POSSIBLE TESTING SOLUTIONS

To test these circuits, Professor Giovanni Boero advised the use of a Photodiode + LED setup. The photodiode will act as a current source while the LED can give a signal to test the circuit gain and the wanted frequencies. Some key elements need to be carefully chosen when selecting the right photodiode and LED.

Capacitance:

The photodiode and LED will have an equivalent capacity, it is important to select diodes that will have a low nominal capacitance (without applying a negative bias).

Speed:

The diodes will have a certain rise time (and fall time, generally the same). They will define the bandwidth of the diode. It needs to be small enough to cover the bandwidth of the circuit. The speed of a diode is proportional to its capacitance. A lower capacitance means the diode will tend to be faster.

Dark current:

Dark current appears in diodes when they are negatively biased. A negative bias will reduce the equivalent capacitance of the diode and thus make it faster. The dark current coming from a negative bias is generally too high for this application and risks of giving too much noise at the input of the circuit in the case of the photodiode.

A 3D printed box model was made to separate the Photodiode + LED system of ambient light. This is not an optimized setup since cables will be needed between the photodiode and the circuit. It will still be enough for some basic testing.

6 REFERENCES

[1] Eurocircuits, "PCB Design Guidelines – SOLDERMASK," 2020. [Online]. Available: https://www.eurocircuits.com/pcb-design-guidelines-soldermask/.

[2] Digi-key, " Amphenol RF 132255," [Online]. Available: https://www.digikey.ch/product-detail/en/amphenol-rf/132255/ACX1424-ND/1989873.

[3] Analog Devices, "Data Sheet ADA4817-1/ADA4817-2, Low Noise, 1 GHz FastFET Op Amps," [Online]. Available: https://www.analog.com/media/en/technical-documentation/data-sheets/ADA4817-1_4817-2.pdf.

[4] G. Mongardi, "Charge amplifier schematics and simulation from Onenote".

[5] P. H. a. W. Hill, The Art of Electronics, US: Cambridge University Press, 2015.

[6] Analog Devices, "LTC6268-10/LTC6269-10 – 4GHz Ultra-Low Bias Current FET Input Op Amp," 2015. [Online]. Available: https://www.analog.com/media/en/technical-documentation/data-sheets/626810f.pdf.

[7] Texas Instrument, "Stability Analysis of Voltage-Feedback Op Amps," [Online]. Available: https://www.ti.com/lit/an/sloa020a/sloa020a.pdf?ts=1622134516697&ref_url=https%253A%252F%252Fwww.google.com%252F.

[8] Analog Devices, "LTspice: Stability of Op Amp Circuits," [Online]. Available: https://www.analog.com/en/education/education-library/videos/5579254320001.html.

[9] Linear Technology, " Transimpedance Amplifier LTC6268 and LTC6268-10 3-Channel SOT-23," [Online]. Available: https://www.analog.com/media/en/technical-documentation/user-guides/DC2414AF.PDF.

[10] Texas Instrument, "Compensate Transimpedance Amplifiers Intuitively," March 2005. [Online]. Available: https://www.ti.com/lit/an/sboa055a/sboa055a.pdf?ts=1621974493125&ref_url=https%253A%252F%252Fwww.google.com%252F.