Physics Experiments 2
Practicals manual

Paul Logman

Koen van Deelen

Matthijs Rog

Koen Schouten

Leiden University
2025-2026

Physics Experiments 2: Practicals Manual
Paul Logman, Koen van Deelen, Matthijs Rog, Koen Schouten

First edition (English): March 2021

Acknowledgments

Contents

Introduction

PE2 will expand on the groundwork on Fourier Analysis and automated measurements that has been laid during PE1. Any groundwork is never complete; we will now delve deeper into the details of Fourier Analysis and signal processing.

We will start with focusing on digital artefacts that arise by using automated measurements using the MyDAQ. You may have noticed such artefacts during PE1. Next, we will expand our Fourier analysis to 2D. In the second session we will investigate the various sources of noise: how to characterize them and how to implement noise reduction. We will finish with the third and fourth session by exploring various feedback systems as well as the use and limitations of the Operational Amplifier (OpAmp). Fourier analysis is not just interesting from an analytical viewpoint.

Again, nearly every experiment performed at both LION & STRW incorporates some form of signal processing as described above. These are very important skills to develop and practice.

Because both theory and practice go hand-in-hand, half of PE2 consists of practical sessions. The sessions follow the lectures closely. In the same order and just before each practical session the lectures will also treat digital artefacts, 2D Fourier, noise, feedback and OpAmps. This way, keeping up with the lectures and exercise classes will help you to understand the practicals and vice versa, the practicals will help you to understand the theory!

PE2 will consist of four sessions. In the first one you will learn more about digital artefacts and 2D Fourier Transforms. During the second session you will learn how to characterize and reduce noise in experiments. In the third session you will apply PID-feedback to either a thermal or an optical system. During the final session you will research one of many possible electronic OpAmp feedback circuits. In these four sessions, we will give you very specific instructions but still you can choose various applications within each session. We hope that you will learn a lot and enjoy the PE2 practicals.

0.1 Learning Objectives

General learning objectives

During this course you get the chance to learn how to:

- apply theoretical knowledge while performing experiments.
- comfortably switch between analyzing signals in the time domain or in the frequency domain, and develop accompanying analysis skills.
- expand your Python skills that you can utilise again in other courses (like PE3) or even your Bachelor Project.

During PE2 we will build on your Python skills from PE1, thus make sure you have your PE1 code available during the practical sessions.

Practical learning objectives

After successfully finalizing the practicals you will be able to:

- apply the following concepts in experiments involving various physical phenomena and to set up your own complex experiment.

More precisely you will be able to:

- Predict and measure transfer functions, complex impedance's, Bode plots, and response functions for electronic and mechanical systems.
- Perform simple image processing using 2D Fourier transforms.
- Apply mathematical tools to signals. Those tools include convolution, modulation, the Wiener-Khinchin theorem.
- Name the basic time-domain / frequency-domain Fourier pairs.
- Describe the relation between operations in the time domain and in the frequency domain.
- Analyze linear time-invariant systems using the Laplace transform and the various Fourier transforms.
- Describe the cause, spectrum, and consequences of various sources of noise and propose solutions to reduce noise.
- Analyze the transfer function and stability of negative and positive feedback systems.
- Analyze simple electronic circuits containing an OpAmp.

This will prepare you well for your open experiment in PE3 and your Bachelor Research Project.

0.2 Setup of the practicals

PE2 will consist of 4 sessions. Each session will make use of the theory that has been discussed in the lectures and exercise classes. Make sure that before preparing the practical, you are up-to-date on the exercise classes. These exercises provide you with the knowledge and skills required in subsequent practical sessions. A basic understanding of relevant theory is necessary, and is expanded on during the practical sessions. We think understanding the theory and applying it in practice goes hand-in-hand. This is backed by a strong correlation between exam and practical grade. We also think it's more fun this way.

The structure of the practicals of PE2 closely resembles the structure of the practicals of PE1. You will be provided with a short description of the experiment, including a research question. You will then perform the experiment in order to answer the research question, where we ask you to come up with your own research plan as much as can reasonably expected. Again, you will be provided with `Python`-code and other digital material. You will read and analyze the given code in each session. After using the first part of the session to prove that your program is working as it should, you will proceed with the practical as normal.

We aim to give you choices in the type of experiment to perform during a session. Thus you need to indicate in your preparation which option of experiment you have chosen. When applicable there will be a Google form which you need to fill out as well. You only need to prepare the experiments that are part of your chosen option.

You write a preparation in order to smoothly and safely execute your experiment during the available lab time. While performing the experiment (also during the programming sections), you will keep a lab journal where you record your actions towards answering your research question.

As in PE1 you will perform each experiment as a duo. We do encourage discussing your hypotheses or results with fellow students and with the TA's! Experimental research is never a solo-act, and discussing ideas or interpretations is part of being a researcher!

The first two sessions of PE2 are formative (and busy) sessions: your preparations will be graded but your lab journal will not: it will only be given feedback halfway through each session (ask for feedback whenever you like, even when preparing).

During the third and fourth session, you will perform an experiment on feedback systems (thermal/optical and electronic (OpAmp)). While you will keep a proper lab journal during all these sessions (we will check!), only the lab journal of the last two sessions will be graded.

The rubrics that we will use to grade your preparations and lab journals can be found on BrightSpace and reflect the general learning objectives of practicals.

0.3 Lab Journals

Lab journals are essential for the documentation of scientific results and research. In your first year, you already learned how to keep a lab journal. Here, we only lead you through the basic aspects. We reiterate the goals of a good lab journal:

(a) A lab journal must be readable and understandable for a third party. Someone else must be able to use your lab journal to reconstruct your actions in the laboratory. Even if they do not possess the manual of the experiment! This ensures your experiment is reproducible, and therefore proper science! *In the meantime it also gives us the opportunity to get insight into your learning process and the problems you face.*
(b) A lab journal helps you to perform your experiments in a structured manner, and to work efficiently.
(c) It allows you to retrace your steps in the future, and thus helps you to construct new, effective steps when you get stuck.
(d) It gives you a place to record your smart ideas and insights.
(e) It is one of the documents that helps to guarantee lab safety.

To fulfill all these goals, you will be following an analytic pattern of reasoning and experimentation. By following this pattern, you will be able to efficiently draw new insightful conclusions from an experiment, and easily report your findings in papers, posters and presentations. Even when you need the information from your lab journal many months after you performed the experiment. Note how nearly all these instructions have been taken directly from the first year's syllabus.

While writing your lab journal, you should use the structure from the following flowchart:

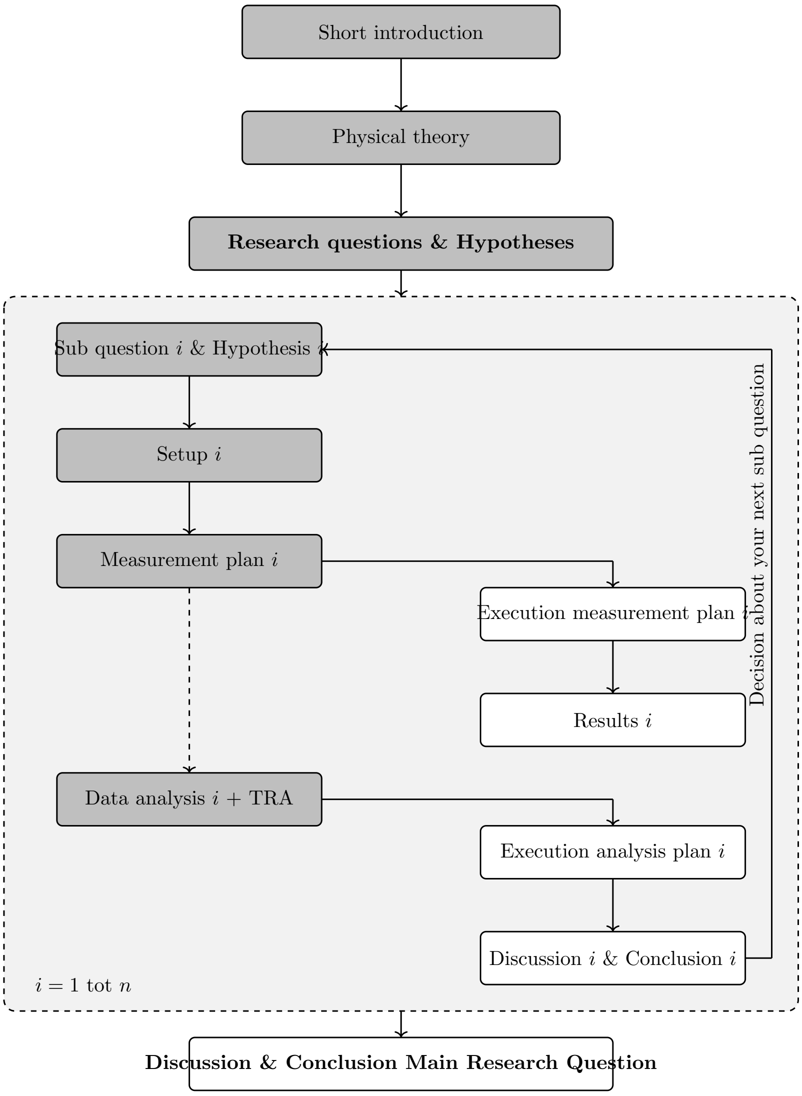


Figure 0.1 – *The (mandatory) structure of a good lab journal. Headings in dark grey are written as preparation before a session; headings in white are written in the laboratory. The headings as well as the research cycle (light grey) must be put into your lab journal* ***explicitly!*** *Use e.g. large, pronounced headers to implement this structure. Of course, you are allowed to add research questions whenever your research calls for them!*

In order to gain time and work efficiently you will **prepare** *all* dark grey steps at home. Your analysis plan is an essential part of these seven steps. Writing a preparation ensures that even if something goes wrong or if something unexpected happens:

1 you understand the relevant theory
2 come up with hypotheses to test
3 execute a thought-out measurement plan
4 analyse your data properly
5 are able to answer any (sub)-research questions

Before starting the practical session make sure that before each session you have handed in your preparation as a pdf-file in BrightSpace. Also make sure that your lab journal is properly set-up in a collaboration space of your choice and send us a direct link to it. At the end of the last session for PE2 you will hand in all your lab journals as a pdf-file in BrightSpace.

To conclude this section, we now give a quick overview of each step in the lab journal and the information we expect to find within:

Personal details

We expect to see your student numbers and names, the name of your group, the date of execution, the name of the experiment and the name of the teacher. You can add more info if you like.

Short Introduction

A short introduction should be clear and concise. It really only takes three sentences: one sentence to provide societal context; one sentence to provide scientific context; one sentence to clearly formulate your scientific goal. Do note that we want to read the goal of your research, not the goal of the course! Anything longer than this takes away from the clarity of your message.

Physical theory

Here you summarize all the physical theory of your experiment: important theories and experiments in your field of research, that are **directly** applicable to your research. You only need to show a derivation if one of the steps in the derivation has fundamental importance to your research. In principle, analysis theory does not need to enter this section. However, if you are researching an analysis method, this would be a natural place to summarize the underlying theory.

Research Questions & Hypotheses

Formulate in a single sentence as concisely as possible the research question of the experiment. The research question should be as clear and scientific as possible. Write the answer that you expect: your hypothesis. A good hypothesis is always falsifiable and based on theory: it can be disproven which may have consequences for either your setup or the theory! Do not forget to give your hypothesis an error margin: when do you prove your hypothesis wrong?

When your research question is very large, it becomes a *main research question*. You then ask *small research questions* in order to answer the main query. We ask that you present your small research questions and their hypotheses **in between** the main research question and its hypothesis. Every research question in your lab journal, no matter how big or small, is subject to the same scientific scrutiny: keep your work falsifiable! Your main hypothesis should be based on and consistent with your sub hypotheses!

Experimental Setup

Here you show and explain the experimental setup you plan to use. Be specific: another researcher repeating your experiment should use exactly the same setup *and materials* as you have! If you re-use the experimental setup of a previous experiment, do not remove this header, but instead explain if, and what kind of modifications you have made.

Every heading should have a text where you explain what you are doing. Here, we also want you to make your plan visual: make sure to provide a structured, well-labeled and properly captioned drawing of your setup.

Measurement Plan

What if your hypothesis is false: how would you test it? What methods would you use for this test? In the methods section, you describe your experimental test. Be as specific as possible, and do not forget to explain how you will collect and store your raw data. You can even put predictions of your raw data!

Experimental Execution (in the lab)

This section is written in the lab. Here, you must write down everything you do while performing your experiment. While you can and must refer to earlier sections, this is no excuse to simply write "I execute my plan!" No plan is ever planned and/or executed perfectly, and to ensure reproducibility, write down your process exactly. This especially includes all the things that go wrong, together with their solutions.

Make sure that you take note of which parameters you use in your measurement (e.g. temperature, voltage, frequency) and why you are stopping or continuing your measurements (e.g. noise below a threshold, weak signal).

Results (in the lab)

Here you show all your *raw* results, that have not yet been analyzed. Make sure to present it properly, preferably in graph-format but make sure to keep your raw data file. Make sure to plot correctly: for instance, we do not want to see line plots anymore! If it is possible to generate raw data errors (and mostly it is), make sure to put error bars on your data points.

Data analysis plan

This plan should lead the reader from raw data to an answer to the research question. Be as precise as possible: which deliverable are you aiming to present, which raw data and formulas do you need for this, which function fit will you do and how do you judge the goodness-of-fit? The most important rule is: we want you to *explain* what you are doing. Preparing Python code at home is super smart, but does not replace an explanation! Always include error calculation (and error propagation) in this step and make this error propagation experiment-specific!

Analysis (in the lab)

Here you explain the execution of your analysis plan. You should be as exact and precise as you were in your experimental execution. Do not forget your error propagation! The steps you take in analyzing must be crystal clear. Be sure not to interpret your data yet; for now, you must remain completely objective and factual. Nobody should be able to argue with you about this section.

Discussion (in the lab)

In this section you interpret your data and your findings. Under 'Results & Analysis' you only present (analyzed) data. You use the 'Discussion' section to give it physical meaning. At the very least, compare your data to the hypothesis and discuss the size of your errors. Discuss the successes and shortcomings of your experiment and/or theoretical knowledge, and advise on future experiments.

Conclusion (in the lab)

Give the conclusions of your experiment concisely. Answer the research question, and repeat the most important findings.

Task-Risk Analysis

In the lab we have to perform several tasks that may include some non-trivial risks. Use the task risk analysis Excel sheet to summarize the possible risks during the practical and determine whether the suggested measures are sufficient. You can find this Excel sheet 'TRA' on BrightSpace together with several safety sheets. You can add it to the end of your preparation and your lab journal. Always update your task risk analysis to the current situation (e.g. the situation on covid is continuously changing).

Session 1: Digital Artefacts

3.1 Introduction

This practical session will consist of 2 sections, which each will take about 2 hours. The relevant theory for this session are exercise classes 1 and 2 (on digital artefacts ant the impulse & step response function). Make sure that you have finalized those exercise classes before starting your prep work for this practical session.

Section 1: Digital artefacts

Section 2: Impulse & step response function

In each section, there are several options listed. You are required to choose one option for each section, but it is highly recommended to finish all. If you have enough time and are ready for a challenge, an optional EXTRA research question is formulated at the end of each option. The description of each section with its options of experiments follows next.

3.2 Section 1: Digital artefacts

This section explores digital artefacts. We advise you to choose the option for which you understand the relevant theory **the least**. We think that during these practical sessions you will gain a better understanding of the relevant theory which in turn will result in a better exam grade.

3.2.1 Digital artefacts

In this section we study several digital artefacts that arise due to measuring a signal using a computer. These artefacts arise because we measure with finite measurement points and finite measuring time and speed. We will use a tuning fork to measure a well-defined frequency. We use a microphone to measure the sound of the tuning fork, and automate our measurements with the MyDAQ. Reuse your code from PE1 to drive the MyDAQ!

We will start with a very small research on aliasing and then move on to study spectral leakage.

Aliasing

Aliasing occurs when measuring a signal containing a certain frequency f_{max}, while using a sample frequency that is too low. The Nyquist theorem states that the minimal sample frequency should be:

$$f_s \geq 2\ f_{max}$$

In this practical we try to find our own limit, as we are working with limited electrical systems. This means that we usually have to *oversample*: have the sample frequency be a multiple of f_{max}.

The research question thus will be: **How much oversampling is required to measure the tuning fork correctly.** Try to formulate your own definition of ‘correctly’ in your analysis plan.

EXTRA
When aliasing *does* occur, we know that we interpret the frequency of the tuning fork incorrectly as a lower frequency, via the relation

$$f_{measured} = \pm f_{signal} \pm n\ f_s$$

n being some integer.

A second research question could thus be: **How accurately can we predict the ‘real’ frequency we measure when aliasing occurs?**

Spectral leakage

We will now study which digital artefacts arise due to using a particular window when measuring a signal. In previous practicals we did not pay any attention to it, but we were measuring all the time using a *square window*: the signal itself is deemed infinite in time, but we measure it only in a certain time-frame (a square window).[1] We can study this behaviour for various window-types and different window-widths by multiplying our measured signal with a window function:

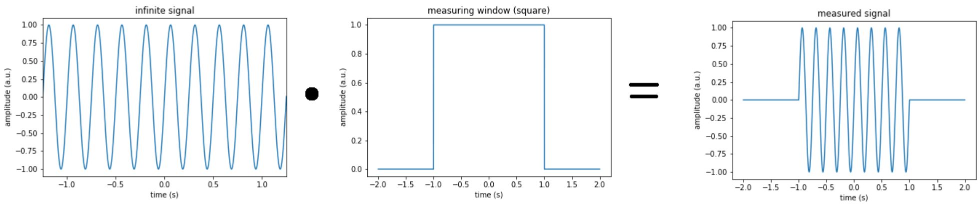
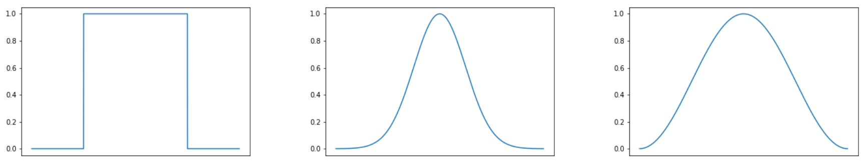


Figure 3.2 – *Measured signal = acquired data · window function. For this example a square window is used.*

The use of a finite time/width window will result in 'spectral leakage' of the signal. The used window function can be anything. One of the most used windows are, as noted in the PE2 reader:



(a) *Square window* **(b)** *Gaussian window* **(c)** *Hanning window*

Figure 3.3 – *Often used measuring windows*

You can write these window functions yourself or, as so often is the case, find a (numpy/scipy) function that does it for you.

Inspiration for your research question will thus be: **Which impact do the window width and window type have on the measured Fourier spectrum?** Start by studying the spectrum when using a square window and then vary the window width. Then try to compare at least two window types. Be sure to use your measured signal (audio of the tuning fork) when analyzing these artefacts. *Hint: try to study the extremes of the system: when is the window too small or large enough?*

[1]Maybe you already noticed these artefacts during PE1 when measuring for a (very) short time.

EXTRA
We can define a **quantified expression** for the amount of 'spectrum that is leaked' and compare window widths and types. A second research question could thus be: **How does the amount of spectral leakage vary with window width and type?**

3.3 Section 2: Impulse & step response function

The Fourier domain can tell us many interesting properties of a system, but studying a signal in the time domain can also be interesting. The impulse response function, for example, fully characterizes the behaviour of a system, and is defined as the output of a system when the input is a delta function. In practise this delta function (also called 'impulse' function) is a very brief signal. We know that the impulse response function $h(t)$ and the transfer function are Fourier coupled:

$$\mathcal{F}(h(t)) = H(\omega)$$

Similarly the step response function fully characterises the behaviour of the system:

$$\mathcal{F}\Big(\frac{d\ s(t)}{dt}\Big) = i\omega S(\omega) = H(\omega)$$

with $S(\omega)$ being the Fourier transform of the step function s(t).

The research question thus is: **With what precision can we characterize the system using the impulse or step response function (in the time domain)?** The system that you will study can be an RC-, LC-, LR-, or RLC-filter, you choose. Try to compare an impulse or step response function of two different systems. For your predictions: you can for example start by writing down the differential equation of your system and predict how it responds to an impulse or step response function.

For your preparation it is necessary that you know which values of R, C and/or L you require for your filter. We will hand them out at the start of the session. Of course it is wise to start with a proof of principle measurement for your system. Ensure that it is properly connected and the MyDAQ is able to send and receive signals properly.

Next, you will investigate the accuracy of the Fourier coupling as described above. Knowing the analytic transfer function of your system, you can determine how well the impulse or step response function models the transfer function. A second inspiration could thus be: **With what accuracy can we recover the transfer function using the impulse or step response function?**.

EXTRA

During the experiment, you may notice a lot of noise in your measured transfer function. One technique to reduce the noise is statistical binning. In essence, you bin the transfer function, and take the average of each bin as the true value at that frequency. Doing so introduces a plethora of assumptions (that will become clear after the lectures on noise) that we will assume to be true for now. Inspiration for an extra research question could thus be: **By how much can we improve the measured impulse or step response transfer function through statistical binning?**

3.4 Mandatory Preparation

Here is a summary of what you need to prepare.

- Write the first (dark grey) steps of your lab journal for each section. We expect you to have a proper preparation for each research question. Of course, the research questions posed here serve as inspiration and a guideline of what we expect you to complete, but you are welcome to introduce a different research question that captures the same spirit. You are also welcome (and encouraged) to divide these research questions into several smaller sub-research questions.
- Brush up the relevant theory and prepare any necessary code, e.g. to drive the MyDAQ for automated experiments. Be sure to have finished the first two exercise classes.
- Prepare your code to drive MyDAQ.

Session 2: Noise Characterization and Reduction

4.1 Introduction

This practical session will again consist of 2 sections, which each will take about 2 hours. In the first section you can try to characterize various types of noise and in the second section you can try to reduce noise (chapter 3 of the PE2 Reader). The necessary preparation for the sections are the exercise classes that discuss the relevant theory: exercise classes 4 and 5. Make sure that you have finalized those exercise classes before starting your prep work for this practical session.

Section 1: Noise Characterisation
Section 2: Noise Reduction

Within each section there are some choices listed. You are required to choose one option for each section. The description of each section together with its options of experiments follows below.

Note that the experiments given below can be both interesting and challenging. Each experiment therefore has 2 research questions; the first aims to explore the experiment, the second is to dive deeper into the specifics. We expect you to answer **at least the first research question** per option of experiment. You should still prepare for both research questions; then while performing the experiment in the lab you can decide whether to continue (if you have time), or to move on to the next section. **Be sure to formulate sub-research questions** to ensure you can stop (or move on) at any time and still have results!

4.2 Section 1: Noise Characterisation

In this section you will choose an experimental setup and characterize its noise. You will see that noise is not only characterized in order to circumvent it: the characteristics of noise often give essential physical information and insights about a system!

While all research questions are equally time-consuming and equally challenging, they are not equivalent. Some experiments have a longer analysis section, while other setups have a more extensive measuring method. Choose an experiment to challenge yourself, so that you can improve your skills!

4.2.1 Option A: Avalanche Noise in Diodes

In the lectures you have already learned about shot noise in electronic circuits. This type of noise is a direct result of the fact that an electric current is made up of discrete charges. Because the electron charge is tiny, this type of noise is extremely small and very hard to measure at room temperature.

In this experiment you will measure a special kind of electric shot noise, called *avalanche noise*, which is closely related to conventional electric shot noise. Your goal will be to measure this noise and describe its properties. Do not be alarmed by the unknown theory: the noise turns out to be remarkably easy to analyze and understand.

Theory: What is Avalanche Noise?

Avalanche noise occurs in special electric components, called *diodes*. Diodes have the remarkable property that they conduct electricity very easily in one direction, but block all current in the opposite direction! Figure 4.4 shows what diodes look like, and how their conductance changes as voltage is applied to them.

Naturally, the ability to block *all* opposing current exists only in theory. In reality, when very large voltages are applied to a diode in the reverse direction, it can no longer withstand the opposing current and breaks down. Diode breakdown happens at a very specific voltage called the *breakdown voltage* V_B.

Being a physicist, you might wonder what causes a diode to break down. The exact answer depends on the type of diode, but for our diodes the answer is simple. Even though diodes aim to block *all* opposing current, some electrons will always find a way to flow across a reversely biased diode. Under normal circumstances the current caused by these electrons is incredibly small, but

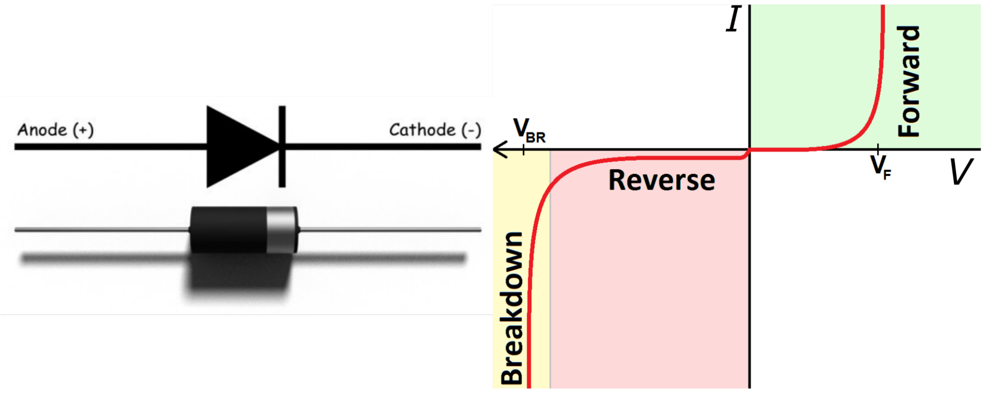


Figure 4.4 – *Two images of diodes.* ***Left:*** *schematic depiction side-by-side with a real diode. Current flows easily from left to right (forward direction). Image adapted from Wikitronics.* ***Right:*** *current as a function of voltage through a diode. Breakdown voltage is labeled on the left. Image adapted from Sparkfun.*

this changes when the diode breaks down. In breakdown, these electrons have so much energy that, when they collide with atoms in the diode, they "punch" bound electrons away from their host atoms. These new electrons then join the current, with enough energy to excite another bound electron, et cetera. In this way, a one-electron current can turn into a million-electron current. Such a sudden amplification is called an **avalanche**, and causes the diode to allow conductance in the reverse direction.

At voltages far above V_B, billions of avalanches occur per second and a continuous current flows through the diode. But when exactly V_B is applied to the diode, the situation is different: only a few ($\sim 10^5$) avalanches can occur per second. Avalanches are very short but carry many electrons. Therefore, you can see each individual avalanche on an oscilloscope!

Research Question

Figure 4.5 shows an example of avalanche noise. As you can see, the avalanches occur randomly, with no fixed time between them. In analogy with photon experiments, we call the number of avalanches in one measurement the *avalanche count*. Because avalanches occur randomly, the avalanche count is randomly distributed according to some distribution. Your research question will therefore be: **Which are the physical properties of avalanche noise in diodes, and how is the avalanche count distributed?**

Of course, it's best to formulate some sub-research questions first, like **At which voltage do these avalanches appear on the oscilloscope?** and think about what the time and voltage scale settings should be on the oscilloscope.

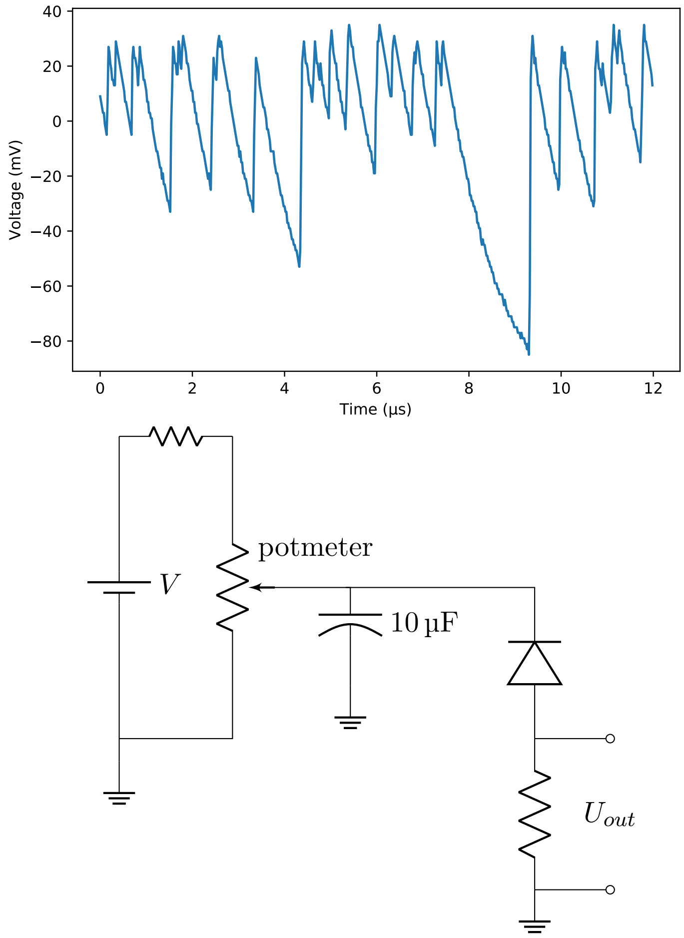


Figure 4.5 – ***Top:*** *Example of avalanche noise in a diode, captured with a Rigol oscilloscope (don't try to recreate it exactly).* ***Bottom:*** *Circuit drawing of a potentiometer and capacitor together making up a voltage divider to carefully control the reverse bias on a diode. What does the capacitor contribute? Why is there a resistor in series with the diode?*

Methods

In order to measure avalanche noise, you first need a setup to carefully set and change the voltage over a diode. You can make one from a power supply, a potentiometer and a capacitor. The potentiometer acts as a voltage divider for the power supply. It can be carefully trimmed using a rotary knob to set the voltage very accurately. If you haven't seen potentiometers yet, do a Google search. A possible setup is shown in the bottom part of figure 4.5.

Note: the capacitor is not bipolar: be sure to connect it in the proper direction (otherwise it may cause a small explosion). Also be sure to check your circuit, because misconnecting the diode or potentiometer may result in a shortcircuit, which potentially heats up and melts (or even sets fire to) the potentiometer. You may prevent this by placing a ($100\,\Omega$) resistor between the $5V$ and the potentiometer. Also be sure to connect all the grounds (to the same ground), as otherwise you may not see any signal. You will thus need to include this risk and how it is mitigated in you task risk analysis!

In the practical, several diodes with V_B ranging from 8 to 12 V will be provided. In order to measure their avalanche noise, you should put them in series with a large resistor. A resistor will convert the current noise to voltage noise through Ohm's law. You can then use an oscilloscope to investigate the noise. For optimal results, make sure your oscilloscope is ac-coupled (removing any dc-offset), so that you can zoom in better.

After finding the noise, take a moment to think about what you see. Can you explain your observations? After reflecting, measure the noise. In order to find the distribution of avalanche counts, you should do many measurements. You can automate your measurements using the `Rigol`-class in `rigol.py` (uploaded to Brightspace and to the T-disk). Below you find some sample code which uses the `Rigol`-class to grab data from the oscilloscope screen:

```
myDevice = RigolOscilloscope()
time, voltage = myDevice.captureChannel1()
```

Be sure that you are using a Rigol Oscilloscope **DS1052E**, as the code isn't able to communicate with other oscilloscope models.

Analysis

In order to find the distribution of avalanche counts, you must first find a way to count the avalanches in one measurement. We advise you to use the function `scipy.signal.find_peaks`. To analyze the distribution of the avalanche count, you can make a histogram. Analyse this histogram to the best of your abilities. The `scipy.stats` module could help you out, but you are not required to use it. Do your conclusions match up with your predictions? Which physical information can you extract from your analysis?

Extra

It is important to take your time to perform the analysis above in depth. Should you still finish early, you can try answering the following research question: **Which are the characteristics of the power spectral density associated with avalanche noise?**

To obtain the power spectral density (PSD) of your noise, it suffices to take the Fast Fourier Transform using NumPy. If you want your PSD to be correctly normalised, `scipy.signal.periodogram` is a nice addition to your toolkit. Before you start measuring, it is important to think about measurement accuracy: what is the most efficient way to accurately measure a PSD?

4.2.2 Option B: Pink ($1/f$) noise of a Thermistor

Pink noise is a type of noise which rapidly increases in magnitude near 0 Hz. Many types of noise are accurately described as pink noise, including mechanical (vibrational) noise and amplifier noise. Because pink noise is so ubiquitous, it makes low-frequency experiments difficult and is therefore of great interest to experimentalists. Additionally, the physical processes underlying pink noise are in many cases not well-understood by theorists, and are still an active field of research in theoretical physics.

In this experiment, you measure low-frequency noise in an NTC thermistor. This is a temperature-dependent resistor, which you may have already used during EN1. Your goal will be to measure its low-frequency noise and describe its properties.

Theory

Physically, pink noise is caused by very diverse physical phenomena. In general, pink noise occurs in devices whose properties can fluctuate rapidly and weakly, or slowly and strongly. For this reason, pink noise is strongly associated with vibrational noise, but can also be found in biology and geography.

With this knowledge, try to argue if and why you would expect pink noise in (the resistance of) an NTC thermistor. You don't need to derive anything; we stress that theoretically, these noise processes are under active investigation.

At low frequencies, pink noise shows up as a straight declining line in a log-log graph of the power spectral density (PSD). The PSD therefore has the form:

$$S(f) = \frac{c}{f^\alpha} \tag{4.1}$$

Here c is some constant, and α quantifies the slope of the line. Originally, the term pink noise was used exclusively to refer to the $\alpha = 1$ case. However, other values are possible too. For more theoretical background, the $1/f$-noise Scholarpedia article has excellent experimental and theoretical references.

Research question

To find out whether NTC thermistors also display pink noise, you will record its low-frequency noise and analyze it. Your research question will therefore be: **Which are the characteristics of the low-frequency power spectral density of the noise in an NTC thermistor, and with which accuracy can it be described as $1/f$ noise?**

Methods

In order to measure the noise in the thermistor you will need a way to record its resistance. While you are free to choose any method you deem appropriate, we advice you to use the NI MyDAQ, since this device can efficiently measure for a long time, up to multiple thousands of seconds. Of course start by doing a proof of principle (measure up to 10 seconds); afterwards you can automate your measurements using the `MyDAQ-Long`-class in `mydaq-long.py` (uploaded to Brightspace and to the T-disk). [2] Some sample code which uses the `MyDAQ-Long`-class to collect data is given below:

```
myDevice = MyDAQ_Long()
time, voltage = myDevice.capture(duration=10, samplerate=10000)
```

A MyDAQ can only record voltages, not resistances. To measure the resistance, make a voltage divider, wheatstone bridge, or think of another creative solution. There will be resistors and breadboards available; be sure to think of appropriate values before starting the session. If your voltage fluctuations are directly proportional to resistance fluctuations, there is no need to perform a conversion between the two. Why will this not change your final results?

We recommend that you think beforehand about the most efficient way to improve the accuracy in your measurement. We also recommend that you first collect a small data set to analyze while you collect a larger data set - should you require one - to ensure your time is used efficiently.

Analysis

To extract a PSD from your raw data, you can use a Fast Fourier Transform. Of course, it suffices to take the Fast Fourier Transform using NumPy. If you want your PSD to be correctly normalised, `scipy.signal.periodogram` is a nice addition to your toolkit.

To accurately answer your research question, perform a function-fit on your PSD. To which accuracy is the low-frequency noise of the form $1/f^{\alpha}$?

Hint: think about what analysis approach is smarter: measuring a longer time and looking at the Fourier transform, or measuring a shorter time but multiple times and averaging the Fourier transforms. How do these approach differ (quantitatively)?

[2] It might be tempting to use your old MyDAQ code from an earlier practical, but note that some fundamental changes must be made to the way you record data, in order to measure for longer than 10 seconds.

Extra

It is important to take your time to perform the analysis above in depth. Should you still finish early, you can try answering an extra research question. Any type of noise, pink or not, can in a small frequency range be described as a power law. So, for any frequency range you can determine an α corresponding to the slope of the power spectral density within that frequency range. In this way, you can map the transition from pink to white noise. You can test your data analysis skills by researching: **How does the slope of the PSD change with frequency?**

4.2.3 Option C: Optical Shot Noise in CCD Cameras

Because of their reliability and sensitivity, charge-coupled device (CCD) cameras have become the standard for scientific imaging in low-light context. All astronomy students should by now have become acquainted with CCD data, but CCD's are also used in many areas of experimental physics.

Because CCD's are so sensitive, each pixel acts as a very rudimentary photon detector. A CCD picture will tell you exactly how many photons the CCD's pixels have detected during integration. Because of this, CCD pictures are subject to many optical sources of noise. In this experiment, your goal will be to measure the shot noise (or *dark noise*) in an Alphalas CCD and describe its properties.

Theory

CCD cameras consist of an array of tiny light detectors, called *pixels*. When a photon falls onto a CCD pixel, it releases an electron.[3] Each pixel contains a charge "bucket" to store these released electrons while a picture is taken. At the end of the integration, the CCD counts the electrons in each pixel and converts this to computer data. In this way, a CCD tells you exactly how many photons have fallen onto each individual pixel while the picture was taken.

Does this mean a CCD is a single-photon detector? Not quite. For one, a CCD image gives no information on the arrival times of the photons; it only counts them. Furthermore, CCD's are sensitive to various sources of unwanted signal.

First of all, the internal electronics of the CCD makes it impossible to ever count zero photons. For every picture, independent of the integration time, a small offset is added to each pixel. This offset is called the *bias count*. A CCD always counts upwards from the bias count and not from zero, regardless of measurement parameters. The bias count is a constant for the pixel.

Secondly, CCD's are subject to thermal noise, which we also call *dark noise*. Because our experiments are performed at room temperature, electrons and photons are sometimes thermally excited inside the CCD camera, even in a pitch black environment. The CCD mistakes these thermal excitations for "real" photons and counts them accordingly.

A CCD picture therefore contains three independent contributions which can be described in a simple formula:

[3] This picture is simplified, yet captures the essence very well.

$$\text{Count} = \text{Bias count} + \text{Dark count} + \text{Signal} \tag{4.2}$$

Research Question

Because dark noise occurs randomly, the dark count in a picture is randomly distributed according to some distribution. Your research question will therefore be: **How is the dark count in a CCD pixel distributed, and what are the associated physical implications?**

Method

The associated measurement setup is rather straightforward. We will provide you with an Alphalas CCD array, which is a one-dimensional CCD. Depending on the CCD type, you will have 2048 or 3600 pixels at your disposal.

In order to find the distribution of dark counts, you should do many measurements. You can automate your measurements using the `AlphalasCCD`-class in `TOO_AlphalasCCD.py` (uploaded to Brightspace and to the T-disk). Again, some sample code which uses the `AlphalasCCD`-class to make a single measurement is given below:

```
myDevice = AlphalasCCD()
inttime = 5000 # integration time in micro-seconds
capture = myDevice.capture(inttime) # Returns an array of
                   # the count value for every pixel; a single
                   # "CCD Picture"
myDevice.close_device()
```

Analysis

To analyze the distribution of dark counts, you can make a histogram. Analyze this histogram to the best of your abilities. Try to find a distribution that matches your histogram. The `scipy.stats` module may help you out, but you are not required to use it. Do your conclusions match your predictions? Which physical information can you extract from your analysis?

Take care when performing a curve fit on a histogram. If your fit parameters are only allowed to be integers, `scipy.optimize.curve_fit` will not be of much help. It is only tailored to continuous parameter spaces. An easy fix is to write code that tries out all possible fit parameters, and then chooses those parameters that best describe the data.

Extra

It is important to take your time to perform the analysis above in depth. Should you still finish early, you can try answering the following research question: **Which are the characteristics of the power spectral density associated with dark noise?**

To obtain the power spectral density of your noise, it suffices to take the Fast Fourier Transform using NumPy. If you want your PSD to be correctly normalised, `scipy.signal.periodogram` is a nice addition to your toolkit. Before you start measuring, it is important to think about measurement accuracy: what is the most efficient way to accurately measure a PSD?

4.3 Section 2: Noise reduction

In this section we will explore how to deal with interference and noise. We will use the MyDAQ to take automatic measurements and analyze the data using Python.

4.3.1 Option A: Finding a hidden signal

Somewhere in the room a hidden signal is generated. We know that this signal only contains a single, well-defined frequency. However, as is often the case, this signal is very weak, and thus likely not visible above the noise if we do a standard/quick measurement using the MyDAQ.

A lab can be a complicated place, with lots of devices and experimental physicists working at the same time. Thus it can be the case that we experience interference very close to our signal, which we will measure but is not part of our desired signal. Interference can come from different sources, like the power supply of any electrical equipment, such as 50Hz from electrical outlets, but it can also be something that we don't expect, such as the kHz electric hum of TL-lightning, an experiment of a colleague or something completely unknown. To simulate this we have placed an interference source that is very close in frequency to our desired signal. The challenge in this experiment is to see the signal and interference separately.

Research Question

The research question will be: **How long do you need to measure to separate the hidden signal from interference to determine its frequency?**

For your hypotheses: assume that the interference is very close to the signal, say between 10mHz and 1Hz.

When you've found the hidden signal and are able to separate the signal from interference, it would be interesting to explore the use of different window types, as this influences the amount of spectral leakage. How does this impact how well you're able to differentiate between signal and interference?

For your predictions: set up a hypothesis on the relation between frequency resolution and measurement time, and how windowing can influence this.

Hint: as the hidden signal is generated somewhere in the lab, you can try to position your microphone such that it can pick up a signal optimally from any direction.

Methods

We try to measure the hidden signal using a microphone and preAmplifier, and automate our measurements using the MyDAQ. Of course it is wise to start with a proof of principle: show that you are able to measure an audio signal and analyze it in the Fourier domain. Start by doing a proof of principle (measure up to 10 seconds); afterwards you can automate your measurements using the `MyDAQ-Long`-class in `mydaq-long.py` (uploaded to Brightspace and to the T-disk), which enables longer measurement times.

Be sure to characterize every significant area or peak in the power spectral density. What do you think is noise and why? Determine a threshold for amplitude, that the signal should exceed to be characterized as our hidden signal. *Hint: the hidden signal contains a very specific frequency (between 500 Hz - 10 kHz) that is a power of 2.*

4.3.2 Option B: Recording white noise by analog filtering

Often noise is considered unwanted and has to be filtered out. However, as Landauer puts it: noise can also be considered as the signal.[4]. Examples of this is Johnson-Nyquist (thermal) noise, or shot noise of a current. As a recent example, shot noise in the STM current is used to determine if the tunnel current comes from single or paired electrons to further study the nature of charge transfer in superconductors.[5] Measuring this noise correctly can be a challenge, as Johnson-noise and Shot noise both are white noise sources, and thus are not limited in frequency. When recording this we use a certain sample rate; however if we don't take precautions we will experience *aliasing*: frequencies higher than half the sample frequency will enter our recording and show up, mistakenly, at low frequencies. This results in the recorded spectrum being higher than we would expect. Thus we need to filter our signal before recording it, to measure the noise power properly.

Theory

White noise can be characterized by a spectral density $S(f)$, that is constant up to some cut-off frequency; this is usually 20 kHz when working with audio signals but can be higher in electrical systems, usually limited by the impedance of the electronics or used cables (MHz–GHz), or is limited by the physical properties of your system.

Suppose we measure a voltage, then the total measured noise is

$$\sigma_V^2 = \int_0^{f_{c,noise}} S_V(f)\ df = S_V \cdot f_{c,noise}$$

as for white noise S_V is constant in frequency.

Suppose we are limited in our sample rate: $f_s << f_{c,noise}$, then we will measure a much higher noise power than expected due to aliasing. This can be prevented if we filter our signal before recording it.

Research Question

The research question thus will be: **What sample-rate and cut-off frequency should we use to measure the total noise power of a white noise source accurately with a microphone?**

Describe how much higher you expect the noise power to be in terms of the bandwidth of the noise $f_{c,noise}$ and the used sample rate f_s if we experience aliasing ($f_s << f_{c,noise}$).

[4]https://www.nature.com/articles/33551
[5]https://pure.tudelft.nl/ws/files/57303686/PhysRevB.100.104506.pdf

Next you can think about how the situation changes when we use a low-pass filter before recording the noise. With an ideal filter, everything above $f_{c,filter}$ if completely filtered out, so we can safely say that our highest frequency is at $f_{c,filter}$. What is the total noise power in this case?

However, we are limited to using first and second order filters. Think about how this would change the measured noise power. What can we do in order to bring the measured noise power closer to the ideal case?

4.3.3 Methods

As a noise source we can use the white noise generator of the function generator, which can output up to 5MHz.

To perform a noise measurement, we can send the white noise from the function generator to a speaker, and record the audio (noise) with a microphone. Be sure to connect the function generator first to a buffer amplifier, and connect that output to the speaker.

We have several first and second order low-pass filters, with cut-off frequencies at 100-1000 Hz. We suggest you first check just the filter with a scope. Think about a quick way to determine the cut-off frequency and order of the filter.

We suggest you first check the white noise from the function generator on a oscilloscope before measuring it with a MyDAQ, before and after the filter.

Try to relate the calculated noise power to the settings of the function generator, or you could even check whether your calculations are correct by analyzing the signal on the oscilloscope. You can use the fact that for Gaussian noise the peak-to-peak value is 6-8 times the standard deviation.

Hint: for easy comparison you can calculate the spectral density of both the generated white noise (theoretically), and the measured noise (practically).

4.4 Mandatory Preparation

Here is summarized what you need to prepare.

- Write the first (dark grey) steps of your lab journal for each section. Note which option of experiments you will perform. We expect you to have a proper preparation for each research question. Of course you may divide these research questions into several smaller sub-research questions. You do not need to prepare experiments that you did not choose.

- Brush up the relevant theory and prepare any necessary code, e.g. to drive the MyDAQ for automated experiments, or to perform and analyze an FT of your data. Be sure to have finished exercise classes 4 and 5.
- Prepare your code to drive the MyDAQ.
- Prepare your code to perform your Fourier analysis.

4.5 Extra / Ideas for PE3

For PE3, you may want to consider other options of dealing with noise that have not (yet) been implemented in the above practicals, like:

- auto correlation
- lock-in amplifier

If you would like to practice with these outside the scope of PE2 and/or to prepare yourself extra for the exam or PE3, do not hesitate to contact us because we do have some assignments for this. Of course this is completely voluntary!

Furthermore, to practice extra for PE2 at home, you may want to consider the following options:

- Characterize the noise in an old 2D television image (is it really white noise?) and try to improve the SNR of the image by applying your knowledge of noise reduction.
- Measure any possible signal using the app PhyPhox on your phone. You can export your data (csv) and import it in a python programme. Analyze your signal using Python (which types of noise does your data contain?) and try to improve the SNR of your signal by applying your knowledge of noise reduction.
- Even more challenging could be to access the data of your webcam using Python and see whether you can characterize its dark noise. Webcams have all kinds of built-in filters that will make your analysis extra difficult/interesting but we can help you accessing your webcam.

Again, do not hesitate to contact us for any help you need to become a skillful researcher!

Session 3: Feedback systems

5.1 Introduction

In this practical session we will focus on feedback and more specifically PID-feedback.

This practical session will consist of only 1 experiment, which therefore may last the whole session. It will involve feedback in general, and more specifically PID-feedback (chapter 4 of the PE2 reader). The necessary preparation for the sections are the exercise classes that discuss the relevant theory: exercise class 6 introduces feedback systems and exercise class 7 explores PID-feedback. Make sure that you have finalized these exercise classes before starting your prep work for this practical session.

Within this session you are required to choose one option out of the three given options. If you can think of any other physical quantity that you would like to subject to your PID-feedback you are most welcome to suggest it to us (well in advance though). The description of each given option follows below.

Note that the options given below can be both interesting and challenging. Each experiment therefore has 2 research questions; the first aims to explore the experiment, the second is for delving deeper into the specifics. We expect you to answer **at least the first research question**. You should still prepare for both research questions. While performing the experiment in the lab you can then decide whether to continue (if you have time), or to move on to the next research question. **Be sure to formulate sub-research questions** to ensure you can stop (or move on) at any time and still have results!

Make sure you keep a structured lab journal containing a description of the experiment, goals to achieve, (clever) notes on everything you do and interpretations of your results based on theory. During PE2 we focus on the analysis and discussion of the results you find (e.g. What happens? Is my result reasonable? What does this implicate? What do I expect from theory? etc...). Reflect on your work!

5.2 (PID-)Feedback options

You can choose a slow or a fast experimental setup and analyze its stability from the following options:

- Option A: a slow thermostat system containing a heater and an NTC in a confined space of various size
- Option B: a fast optical system containing a LED and a photodiode
- Option X (setups only on request): another system involving a different physical quantity (e.g. an acoustic system, a mechanical system, etc.)(inform us well in advance)

These options emphasize real experimental problems involving PID-feedback adding 'applying theory to reality' to your skills set (i.e. slow/fast, thermal/optical systems but this can be transferred to any physical quantity). In the last section (5.7) we simulate a physical system with PID-feedback. This simulation enables us to make accurate predictions for which values of P, I, and D a system is stable. You can challenge yourself by including this section in your preparation, adding some simulation skills to your skills set! Always choose an experiment to challenge yourself, so that you can improve your skills!

5.3 Theory on PID-feedback

This section will focus on the theory behind PID-feedback and provides clickable links (underlined in the text) for you to explore more.

Feedback systems consist of a system of which the output has some influence on the system's input. One of the easiest examples is a thermostat that turns a heater on whenever the temperature falls below a certain threshold and turns it off when it's above. Such a simple system does contain feedback (on/off-feedback) but no PID-feedback. If you turn the described system on it will generally overshoot the desired temperature in the beginning and after that turn on and off in a certain pattern. You can use PID-feedback to control the overshoot (minimize it) and the subsequent on/off pattern.

Below you find some example code to explain the on/off feedback described above:

```
# Measure temperature
curt = ReadTemp()
# Calculate desired output
out = (5 if curt<setpoint+273.15 else 0)
"""You can make sure that certain values do not cross
certain minimum or maximum values."""
out = np.clip(out, 0, 5)
# Write desired output to heater
WriteVoltage(out)
```

PID-feedback is an engineering feedback trick to make systems that are inherently unstable, stable again. The 3 factors P, I, and D stand for Proportional feedback, Integral feedback and Differential feedback and can be described by the following equation:

$$u(t) = -\left(K_\mathrm{P}\ e(t) + K_\mathrm{I}\ \int_0^t e(\tau)\,\mathrm{d}\,\tau + K_\mathrm{D}\ \frac{\mathrm{d}\,e(t)}{\mathrm{d}\,t}\right) \tag{5.3}$$

where:

- $u(t)$ is the overall control function and $e(t)$ is the error value: the difference between a desired setpoint and a measured process variable.
- K_P, K_I, and K_D are all non-negative and denote the coefficients for the proportional, integral, and derivative terms respectively, and thus describe how strong the influence of the proportional, integral, and differential feedback is.

Of these types of feedback, proportional feedback is the easiest to understand: in the case of the thermal system this means that the heater cannot only be turned on or off but heating power can be adjusted to any value proportional to the difference between the desired temperature and the measured temperature. Now try and think what kind of effects integral and differential feedback would have on a thermal system.

For an optical system you cannot turn the light on/off as is done in the thermal system above because the optical system reacts way too fast. For an optical system the easiest feedback would be something like adding a certain amount of extra power to your light source whenever the light intensity where you are measuring is lower than what you would like it to be. Or decreasing it by a certain amount when it is too bright. You can again make sure that certain values do not cross certain minimum or maximum values. Again, this is a way to provide simple feedback (up/down-feedback), not P-, I- or D-feedback. What would P-feedback look like for an optical system?

Below you find some example code to explain the simple feedback described above:

```
if read_value < set_value:
    led_voltage += .01
elif read_value > set_value:
    led_voltage -= .01
if led_voltage > 4:
    led_voltage = 4
print("LED:", led_voltage)
```

It will prove to be quite easy to program proportional (P-)feedback. To implement integral (I-)feedback you may want to look into the np.trapezoid function. To implement differential (D-)feedback we can approximate the time derivative using finite difference coefficients.

Because PID-feedback is an engineering trick you need to play around with the 3 constants to find optimal values for K_P, K_I, and K_D. It is easiest to find these for when the system is critically damped. When implementing PID-feedback, we suggest you start with the proportional component, and then start to include the integral and/or differential component. When you're finished studying a critically damped system, you can move to the other regimes (stable, unstable).

To quantify the stability of your system you can look for example at the first and second moment of the probability distribution of your measurements compared to the desired value for your system's physical quantity (see among others Diffusie and Statistical Physics 1: An Introduction to Statistical Mechanics and Thermodynamics by Swendsen). What should the first and second moment of the probability distribution be in a perfectly stable system?

5.4 Research questions

The above leads us to the following research questions which are the same for all options within this session:

- Research question 1: How do K_P, K_I, and K_D separately influence the stability of the system? To do this, describe the initial stability as well.
- Research question 2: Which values for K_P, K_I, and K_D result in a stable / unstable / critically damped system?

For research question 2: We advise you to start with the critically damped system. It may be that you can only write a (falsifiable) hypothesis for research question 2 after you have finalized your research on question 1, but you should still be able to make some predictions.

5.5 Method

We use the MyDAQ to write a signal to the system and to measure the output of the system. In order to provide feedback, the output will be determined by the measured signal. We thus have to read & write for a small time, so that we have a fast response feedback system. We could implement this by using the read-write function that you've already made and used a lot. A downside to this code is that it is slow, as each time it's run it generates a new readtask and a new writetask, which takes some 100 ms. Think about whether that could form a problem to your experiment and how you could alter this code. A solution would be that we generate a readtask and writetask once, which we then can use multiple times in succession. A basic version of this can be found on Brightspace: 'MyDAQ_feedback.py'. Check out the comments given in the code before the practical so you understand how to expand on it. Be sure to write down in your preparation whether this code is necessary for your experiment (and **why**)!

When adjusting your MyDAQ code, it may be useful to consult the documentation.

Make sure to critically discuss and reflect on the role of the MyDAQ: would you recommend its use as a PID-controller and under which circumstances?

5.5.1 Option A: Thermostat

A nice example of a feedback system in everyday life is the functioning of a thermostat. Naturally, when the temperature is below a desired temperature, the system needs to heat up. Once the desired temperature is reached however, you do not want the system to overshoot. Therefore, it is essential to build / program a system which produces feedback in order to get a smooth transition to the desired temperature and keep it there as constant as possible.

Method

We already know how to use the MyDAQ to generate input values and measure output values for any system. We also know how to generate transfer functions (Bodeplots and polar plots) from the acquired data.

In this experiment you will use the MyDAQ to control a TOO Temperature Control Box (see figure 5.6) which contains a heater and an NTC. If you're interested in for example the temperature range and resistor inaccuracy of the NTC you can look up its data sheet [6]). The heater and NTC may be covered with a plastic see-through box to keep a certain volume of air at a certain temperature. We have 2 sizes of see-through boxes.

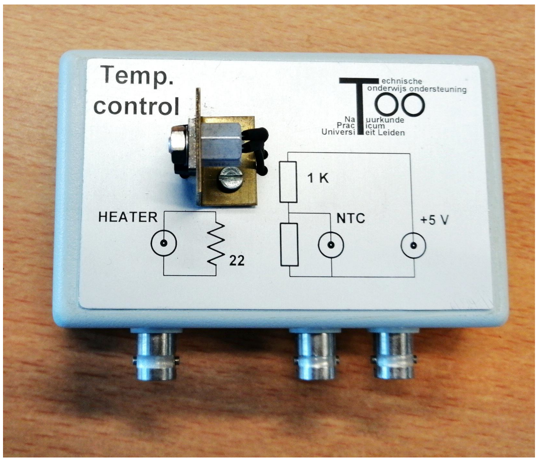


Figure 5.6 – *TOO Temperature control box.* ***Left:*** *Here you find the heater and at the bottom the connector for it.* ***Right:*** *Here you find the NTC in a voltage divider, the connector to measure the voltage over the NTC and the connector to put 5V over the voltage divider.*

Because the heater draws quite a lot of current from the MyDAQ (which the MyDAQ cannot deliver) you will want to use a buffer to control the heater.

[6]type number NTCASCWE3102J with data sheet at `https://shop.griederbauteile.ch/info/2/2381-640.pdf`

To measure the temperature with the MyDAQ you can use the NTC on the TOO Temperature Control Box. The NTC sits in a voltage divider together with a resistor of $1\,\mathrm{k\Omega}$. To make the voltage divider work it needs power from a power supply. Therefore, also connect the 5V input on the Temperature Control Box to the 5V output of the MyDAQ (AGND and +5V).

Besides measuring the voltage over the NTC with the MyDAQ you can visually keep track of its signal using an oscilloscope or virtual bench (or write a GUI). When you use an oscilloscope set it to ‘Roll’ (you can retrieve how to do this by searching for ’Roll’ in the online manual for your oscilloscope (e.g. Rigol 1052E)).

An electronic circuit of the setup is given in figure 5.7.

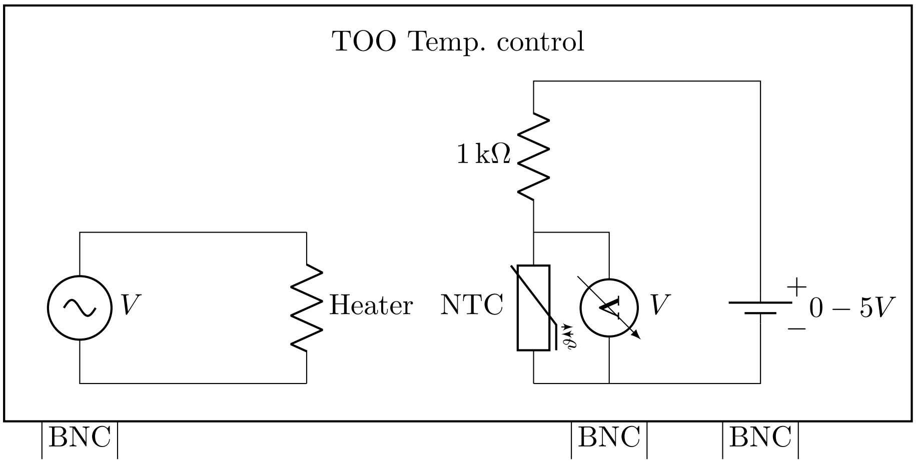


Figure 5.7 – ***Left:*** *Voltage V from the MyDAQ (after going through a buffer) is applied to the Heater.* ***Right:*** *NTC is placed close to the heater, and is connected in series with a* $1\,\mathrm{k\Omega}$ *resistor and a 5V power supply. The voltage over the NTC can be measured (using an oscilloscope/virtual bench/MyDAQ).*

A snippet of code is given to convert the voltage over the NTC to a temperature:

```
def TempFromVolt(Untc):
    Tc  =  3528.0
    Tref = 25.0 + 273.15
    R0 = 1000.0
    R1 = 1000.0
    Uin = 5.0
    return Tc/(np.log((Untc/(Uin-Untc))*(R1/R0)) + (Tc/Tref))
```

Check if things work. When you write a voltage to the heater the copper plate should start heating up. **Note that it will NOT stop heating, so make sure you set the voltage back to zero or disconnect the power supply to the heater. DO NOT exceed 55 °C!**

Again, once you've finished your measurements, make sure to stop the heating by e.g. shutting down its power supply.

Analysis

It can be insightful to plot the effect of the three (PID-)components separately, in order to tweak the parameters.

Implement a way to calculate the first and second moment of the probability distribution of your data to quantify the stability of your system.

It may also be insightful to calculate Bodeplots and polar plots for your system for various combinations of K_P, K_I, and K_D.

5.5.2 Option B: Optical system

A nice example of a feedback system in everyday life is the functioning of an automated camera. When the light intensity is below a desired level, you would like more light to fall onto your pixels (photodiodes). On the other hand, when the light in a situation is too bright you may want to diminish the light that falls onto your pixels. Once the desired intensity is reached however, you do not want the system to overshoot. Therefore, it is essential to build / program a system which produces feedback in order to get a smooth transition to the desired light intensity and keep it there as constant as possible.

Method

We already know how to use the MyDAQ to generate input values and measure output values for any system. We also know how to generate transfer functions (Bodeplots and polar plots) from the acquired data.

In this experiment you will use the MyDAQ to control a LED using a photodiode to measure the light intensity at a certain point (see figure 5.8). If you want, the LED and photodiode can be covered with some heat shrinking tubing to diminish outside influences. However, the challenge lies in overcoming those outside influences and keep the area where the photodiode is located to have the desired light intensity! If you build a very good PID-feedback system you may even be able to determine the outside influences to some extent.

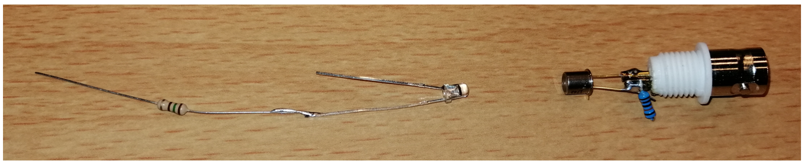


Figure 5.8 – *The optical feedback system.* ***Left:*** *Here you find an unconnected LED in series with a* $100\,\Omega$ *resistor.* ***Right:*** *Here you find the photodiode BPX-65 parallel with a* $100\,\mathrm{k}\Omega$ *resistor both soldered onto a BNC-connector.*

Such a combination of a LED and a photodiode is often used as an optocoupler to transfer signals between two systems without the need for an electronic connection between those two systems. This can be very useful when you want to couple several systems, without letting them influence each other.

Because the LED draws quite a lot of current from the MyDAQ (which the MyDAQ cannot deliver) you will want to use a buffer to control the LED. However, using a buffer makes it possible for large currents to run through the LED and that may breakdown the LED. Therefore, put a resistance of $100\,\Omega$ in series with your LED (see Fig. 5.9 Left) and make sure that the voltage *over the LED* remains below $4V$.

To measure the light intensity with the MyDAQ you can use the photodiode (BPX 65). A photodiode emits a current proportional to the light intensity that hits it, thus a photodiode is a current source. To convert the current into a measurable voltage it is enough to place a resistor (100 kΩ) over it and measure the voltage over the resistor (see Fig. 5.9 Right). Besides measuring this voltage with the MyDAQ you can visually keep track of the signal using an oscilloscope or virtual bench (or write a GUI).

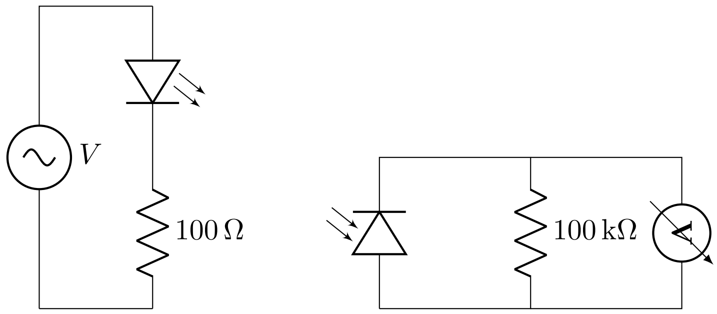


Figure 5.9 – ***Left:*** *LED in series with a* 100 Ω *resistor.* V *denotes the output of the buffer.* ***Right:*** *photodiode with a* 100 kΩ *resistor placed over it. Both connected to an oscilloscope/virtual bench.*

Check if things work. When you write a voltage to the combination of LED and resistor, the LED should light up. **Note that the LED can break down, so make sure the voltage *over the LED* does not exceed 4 Volt.** When you move the LED to or from the photodiode you would expect variations in the measured voltage over the photodiode.

Analysis

It can be insightful to plot the effect of the three (PID-)components separately, in order to tweak the parameters.

Implement a way to calculate the first and second moment of the probability distribution of your data to quantify the stability of your system.

It may also be insightful to calculate Bodeplots and polar plots for your system for various combinations of K_P, K_I, and K_D.

5.5.3 Option X: PID-feedback systems involving other physical quantities

Please read option B because the work that needs to be done will be very similar. *You can also keep your thoughts on feedback for other physical quantities to PE3!*

5.6 Mandatory Preparation

Here is summarized what you need to prepare.

- Write the first (dark grey) steps of your lab journal for each section. Note which option of experiment you will perform. We expect you to have a proper preparation for each research question. Of course you may divide these research questions into several smaller sub-research questions. You do not need to prepare experiments that you did not choose.
- Brush up the relevant theory (e.g. exercise classes 6 and 7)
- Prepare your code to drive the MyDAQ: e.g. for automated experiments, or to provide short response feedback.

5.7 Extra / Ideas for PE3

Simulation of an Inverted pendulum

For extra depth, or for extra ideas for PE3, it may be interesting to explore the total feedback transfer function (with PID-feedback) for a simulation of a physical system. In a practical session last year we simulated the following (unstable) system: an inverted pendulum, which we can describe as balancing a pencil on our finger. When no feedback is applied, the pencil will swing around and fall off our finger. We can apply feedback by moving our finger, to ensure the pencil stays in its upright (desired) position, thus stabilizing the system.

Method

The JuPyter notebook in the supplied zip-file 'Inverted_Pendulum.zip' contains the described simulation, in which some steps still need to be filled in. We start by viewing the simulated movement of the pendulum when no feedback is applied. We can explore the system by tweaking some initial parameters and describe the resulting motion. Next the total transfer function of the system with feedback is analyzed by looking at its poles: we can predict from the values of the (complex) poles whether the system is stable for certain settings of feedback parameters K_P, K_I, K_D. We can then predict (for certain values of K_P, K_I, K_D) when the system is (critically) damped or unstable. Finally we check our predictions by looking at the simulated motion of the pendulum when PID-feedback is applied.

Research questions

- Research question 1: How do K_P, K_I, and K_D separately influence the stability of the system? To do this, describe the initial stability as well.

- Research question 2: Which values for K_P, K_I, and K_D result in a stable / unstable / critically damped system?

For research question 2: We advise you to start with the critically damped system. It may be that you can only write a (falsifiable) hypothesis for research question 2 after you have finalized your research on question 1, but you should still be able to make some predictions.

Session 4: OpAmp feedback circuits

6.1 Introduction

In this practical session we will focus on the use of OpAmps. We will explore 1 (or more) out of various options for the many uses of OpAmps.

This practical session will consist of only 1 experiment, which therefore may last the whole session. We will research an application of using an OpAmp in experiments (Chapter 5 of the PE2-reader). Time permitting, you can choose to research two applications of OpAmps. The necessary preparation for this session are the exercise classes that discuss the relevant theory: exercise classes 8 and 9. Make sure that you have finalized those exercise classes before starting your prep work for these practical sessions.

Within this session you are required to choose one option out of the given ones or come up with an equivalently challenging circuit involving an OpAmp yourself. In that case you will have to contact us with your proposal a few days in advance. The description of each given option follows below.

Note that the experiments given below can be both interesting and challenging. Each experiment therefore has 2 research questions; the first aims to explore the experiment, the second is for delving deeper into the specifics. We expect you to answer **at least the first research question** per option of experiment. You should still prepare for both research questions; then while performing the experiment in the lab you can decide whether to continue (if you have time), or to move on. Do write this decision (and what it is based on) in your lab journal!

Make sure you keep a structured lab journal containing a description of the experiment, goals to achieve, (clever) notes on everything you do and interpretations of your results based on theory. During PE2 we focus on the analysis *and discussion of the results that you find (e.g. What happens? Is my result reasonable? What does this imply? What do I expect from theory? etc. . .). Reflect on your work!*

6.2 OpAmp feedback circuits

For this assignment we have come up with 7 options. Choose 1 of the options A-G and perform the accompanying assignment. Of course, you can also research an equivalently difficult circuit involving an OpAmp. In that case you will have to contact us with your proposal a few days in advance (via PE2@physics.leidenuniv.nl).

- Option A: Optocoupler/IV-convertor (transimpedance amplifier) *(follows up on PE2 session 3, option B)*
- Option B: Amplifier (non-inverting)
- Option C: Prevention of loading using a buffer *(e.g. improving Q factor for LC filter, as a function generator always has output impedance: follows up on PE2 session 1)*
- Option D: Adder (summing amplifier)
- Option E: Subtractor (differential amplifier)
- Option F: Differentiator
- Option G: Integrator

For the practicals we will mostly be using the OP07D OpAmp. Other OpAmps available in the lab are TL081CP and LM741. When you look at the datasheets for the various OpAmps you can find the differences between them. In the datasheets you can also find descriptions of each pin of the OpAmps. *Note that we purchase these OpAmps for about* 70 *cents per device, and so can you!*

For some of the options you will need resistors and capacitors. We have resistors in the $10\,\Omega$ to $1\,\mathrm{M}\Omega$ range and capacitors in the $10\,\mathrm{pF}$ to $10\,\mu\mathrm{F}$ range. *For completeness we add that the lab also has inductors available. We have small inductors in the* $1-500\,\mu\mathrm{H}$ *range with low resistances. We also have larger inductors in the* $1-100\,\mathrm{mH}$ *range, which have a lot of copper inside them. As a result, they come with relatively high resistances of* $10-500\,\Omega$*).*

6.3 Theory

Feedback systems

For feedback systems there are two flavours: positive and negative feedback. Positive feedback is used for example in an amplifier. Negative feedback is more commonly used to stabilize systems. A general system with the more common negative feedback is given in figure 6.10.

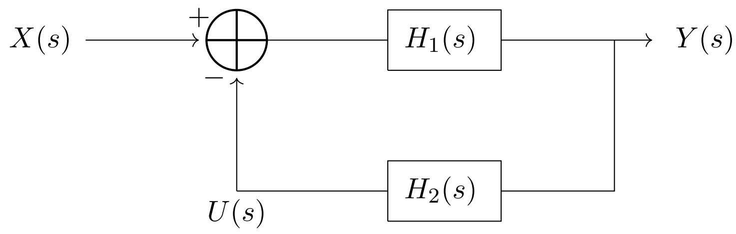


Figure 6.10 – *A general system with negative feedback. The output signal is subtracted, via the transfer function $H_2(s)$, from the input signal $X(s)$. $U(s)$ is called the error signal.*

In Chapter 4 of the PE2-reader the following corresponding transfer functions are derived.

For positive feedback the transfer function is:

$$H(s) = \frac{H_1(s)}{1 - H_1(s)H_2(s)} \tag{6.4}$$

For negative feedback the transfer function is:

$$H(s) = \frac{H_1(s)}{1 + H_1(s)H_2(s)} \tag{6.5}$$

Ideal OpAmp

Together with the 'golden rules of the ideal OpAmp' (see Chapter 5.1 of the reader: **read that page!**), you can use these equations to predict the specific transfer function for the (ideal) OpAmp application of your choice.

Non-ideal OpAmp

In reality however, ideal OpAmps do not exist. A non-ideal OpAmp can be modelled as in Figure 6.11.

Non-ideal OpAmps suffer from various deficiencies compared to ideal OpAmps (see Chapter 5.3 of the PE2-reader): e.g. frequency dependent gain, limited gain-bandwidth product, noise, non-zero current, offset voltage, and phase shift. All of these discrepancies can be predicted to some extent using the non-ideal OpAmp model of Figure 6.11. Using these physical quantities the differences between ideal OpAmps and real OpAmps can be researched quantitatively. Choose one of these physical quantities/concepts and investigate

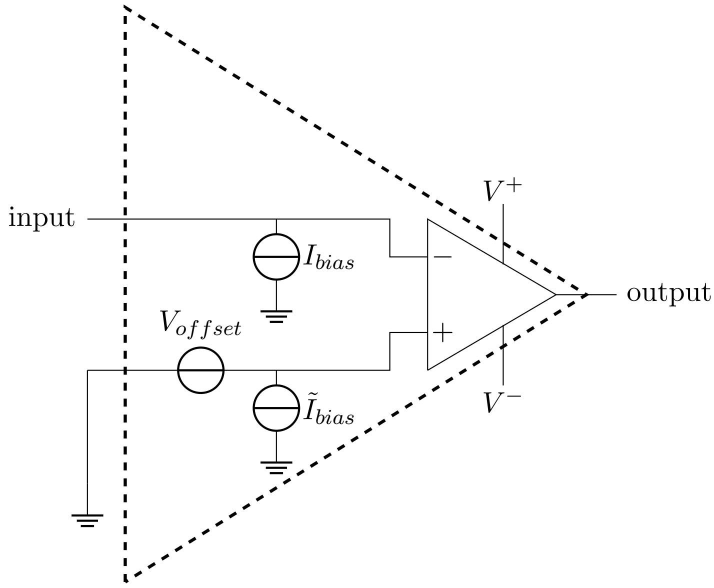


Figure 6.11 – *A realistic OpAmp suffers from, among others, an offset voltage V_{offset} and current leakage I_{bias}. You can take these effects into consideration in your system by using the above circuitry. Check Chapter 5.3 of the PE2-reader to remind yourself what effects these components have.*

it online: e.g. gain-bandwidth product. You may also find information about your chosen quantity in the datasheets of the various OpAmps.

6.4 Research questions

The above leads us to the following research questions which are the same for all options within this session:

- Research question 1: What is the transfer function (both Bodeplots and the polar plot) of the OpAmp system of my choice?
- Research question 2: How does the measured transfer function compare to the theory of the ideal OpAmp and the non-ideal OpAmp? *Base this on at least one of the following physical quantities: frequency dependent gain, gain-bandwidth product, noise, non-zero current, offset voltage, or phase shift. Choose one of these and investigate the physical quantity/concept online. And then rewrite the research question into your version of it.*

You may find the program LTspice useful to make more precise predictions for your chosen system. A quickstart manual is given on BrightSpace and on the creator's website as well.

Think about for which frequencies you are expecting interesting behaviour. Remember that the MyDAQ is able to measure up to 100 kHz, while the VirtualBench and Rigol Oscilloscope can measure up to 30-50 MHz.

6.4.1 Option A: Optocoupler/IV-convertor (transimpedance amplifier)

Introduction

An optocoupler is often used when connecting two electronic systems between which you do not want an electronic connection because of possible interference.

When reading out the current through a photodiode, or any current for that matter, you want to convert it to a voltage. In the Optical system option from PE2 practical session 3, this was done by putting a very (very!) large resistor in series with the photodiode, but this is often disastrous to the performance of your circuit. The underlying reason is that every photodiode comes with a certain (small) capacitance, and that adding a large resistor makes the RC-time of your circuit dramatically large. As a good approximation, you can compare this with the low-pass cutoff-frequency of your system becoming dramatically small. [7]

For this reason, fast current measurements require a measuring device with near-zero input impedance, large bandwidth and large amplification. One such device is the IV-convertor, or transimpedance amplifier (TIA).

Theory

Use the general theory for an ideal OpAmp to predict the transfer function for this setup. Of course, for the complete optocoupler you would like the transfer function to be 1 for all frequencies. *If you want to predict the transfer function for the non-ideal OpAmp, you can use LTspice.*

Method

In figure 6.12 you find the electronic circuit for an IV-convertor (transimpedance amplifier). Build the circuit and measure the transfer function (plot both Bodeplots and the polar plot). Just like in PE2 session 3 you can illuminate the photodiode using a LED. *Make sure that while driving the LED with some frequency it remains on all the time! Also we don't want any background noise, so make sure it is shielded from ambient light properly.*

Analysis

Compare your results quantitatively to your prediction using an 'ideal' OpAmp.

[7] Recall that for a resistor and capacitor in series, there is a characteristic charging time $\tau = RC$ and a characteristic cut-off frequency $f_c = \frac{1}{2\pi RC}$.

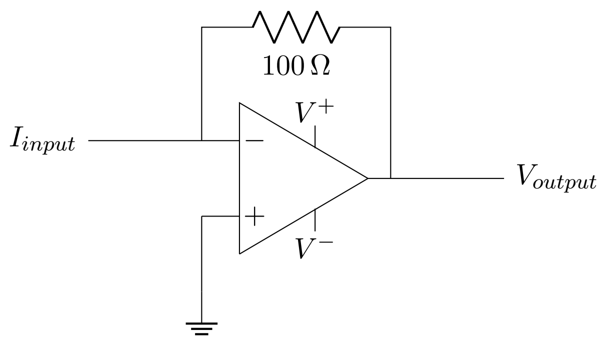


Figure 6.12 – *The schematic for an IV-convertor or transimpedance amplifier (you may want to play around with the value for the resistor). Note that the input is a current source and the output is a voltage. In the case of an optocoupler you will use the photodiode as your current source.*

Extra

Try various OpAmps and compare the resulting outcomes.

6.4.2 Option B: Amplifier (non-inverting)

Introduction

An amplifier is often used when signals are very weak. Of course, you would like the amplification to be the same for all frequencies. For various reasons this can very difficult, e.g. because a (non-ideal) OpAmp has a frequency dependent gain.

These amplifiers are for example used when recording Johnson noise (see session 2) or shot noise, or whenever the noise in the experiment *is* the desired signal. John van Noort's group records the motion noise of a bead attached to a DNA molecule and from that can calculate the stiffness of the DNA. This makes it possible to study the sequential unwrapping of DNA when pulling on the bead with increasing force [8].

Of course we want to have a high SNR, so we want to measure across a broad frequency bandwidth, so we require a high (enough) gain for a broad bandwidth.

Theory

Use the general theory for an ideal OpAmp to predict the transfer function for this setup. Of course, you would like the transfer function to be as flat as possible for all frequencies. *If you want to predict the transfer function for the non-ideal OpAmp, you can use LTspice.*

Method

In Figure 6.13 you find the electronic circuit for a non-inverting amplifier (gain $G_0 = 10$). Build the circuit and measure the transfer function (plot both Bodeplots and the polar plot). Of course it's interesting to explore this circuit for several different values of gain.

Analysis

Compare your results quantitatively to your prediction using an 'ideal' OpAmp.

Extra

Try various OpAmps and/or various gains and compare the resulting outcomes.

[8] see for example this paper

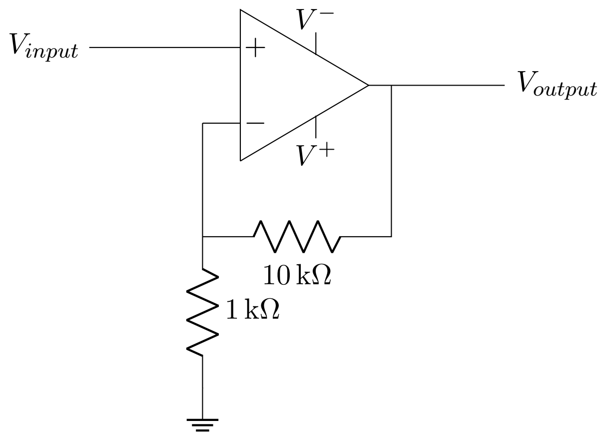


Figure 6.13 – *The schematic for a non-inverting amplifier. You can vary the gain by changing the values for the 2 resistors.*

6.4.3 Option C: Prevention of loading using a buffer

Introduction

All AC-sources such as function generators come with a built-in input impedance. While often convenient, this significantly hampers current-focused, or high Q-factor experiments like the NMR experiments on the 8^{th} floor. The series resistance makes it difficult to supply current to mechanical equipment. It also makes it hard to load (i.e. supply current to) a magnetic coil, since coils tend to have low DC resistance. Another use is to improve the Q-factor for a resonant filter (e.g. LC filter), as the applied input to the circuit, e.g. a function generator, always has output impedance. [9]

These problems are often resolved using a buffer, which you have already met many times during EN and PE. In this option you will build your own buffer and apply it between 2 loading RC-filters.

Theory

Use the general theory for an ideal OpAmp to predict the transfer function for this setup. *If you want to predict the transfer function for the non-ideal OpAmp, you can use LTspice.*

Method

In figure 6.14 you find the electronic circuit for a buffer.

[9] A commonly made mistake during PE3 is to use the buffer as a one-size-fits-all solution to every loading problem. Not all OpAmps are stable in the buffer configuration, and not

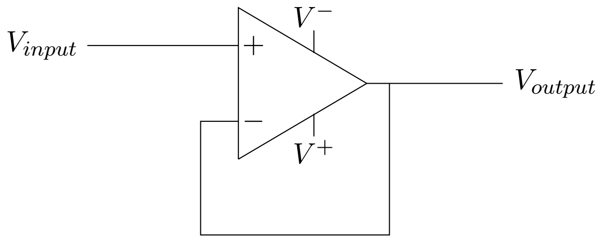


Figure 6.14 – *The schematic for a buffer circuit. Normally, you would connect a function generator directly to an RC-filter and then another RC-filter and measure the voltage over the capacitor or the resistor to investigate loading. With a buffer, you can connect a function generator to the first RC-filter. Connect the output of that filter to the input of the buffer and then put the second RC-filter after the output measuring again the voltage over the capacitor or the resistor of the RC-filter. This way you can investigate loading for these filters without and with the buffer.*

Build the circuit including the function generator and the first RC-filter before the input and the second RC-filter after the output and measure the transfer function (plot both Bodeplots and the polar plot). Of course, do the same also without a buffer in-between.

Analysis

Compare your results quantitatively to your prediction using an 'ideal' OpAmp.

Extra

Try various OpAmps and/or various RC-filters or even an LC-filter and compare the resulting outcomes.

---

all OpAmps can supply more than a few mA of current.

6.4.4 Option D: Adder (summing amplifier)

Introduction

It is often useful to combine two signals at the input of an experiment. When doing measurements on materials with interesting properties, it is often required to run a current through your material while performing your measurement. Moreover, in radio measurements, you might want to combine signals at different frequencies.

While there are many easy ways in electronics to add two voltages, a summing amplifier excels because of two reasons. First of all, it resets the output impedance of the source to zero. In this way, your experiment only "feels" one source, while there might actually be many, perhaps with nasty impedances. Secondly, you can amplify your signal while summing, which is not possible with passive electronics.

Theory

Use the general theory for an ideal OpAmp to predict the transfer function for this setup. *If you want to predict the transfer function for the non-ideal OpAmp, you can use LTspice.*

Method

In figure 6.15 you find the electronic circuit for a summing amplifier, $V_{output} = -(V_{input,1} + V_{input,2} + V_{input,3})$ (note the minus sign: this is an inverting summing amplifier).

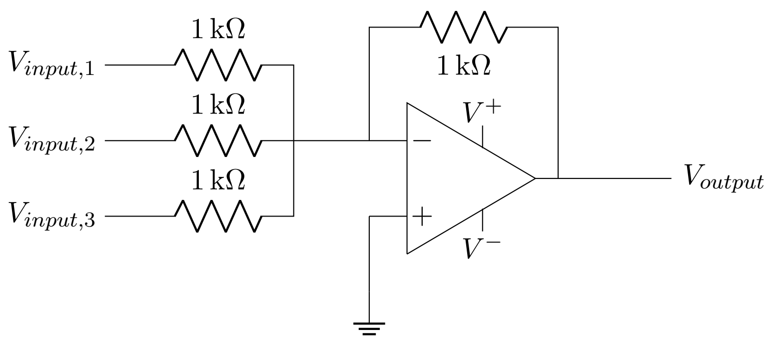


Figure 6.15 – *The schematic for a summing amplifier circuit. You can add multiple inputs in a similar fashion.*

Build the circuit with **only 2 inputs** and measure the transfer function (plot both Bodeplots and the polar plot).

Analysis

Compare your results quantitatively to your prediction using an 'ideal' OpAmp.

Extra

Try various OpAmps and/or adding more than 2 inputs and compare the resulting outcomes.

6.4.5 Option E: Subtractor (differential amplifier)

Introduction

Most signals in PE are referenced to ground, by which we mean that we measure the signal with respect to 0 V. In experiments, however, this is often not the case. Sometimes, an interesting signal is superimposed on top of a less-interesting DC-offset. Or you might be interested in measuring and/or amplifying the voltage difference between two non-grounded points in a circuit, for instance when reading out a Wheatstone bridge. In such situations, physicists use differential amplifiers.

With some changes and additions you can also use a differential amplifier to implement a form of analog P-feedback. However, you would need an integrator-circuit (option G) as well to be able to apply it to e.g. a thermostat.

Theory

Use the general theory for an ideal OpAmp to predict the transfer function for this setup. *If you want to predict the transfer function for the non-ideal OpAmp, you can use LTspice.*

Method

In figure 6.16 you find the electronic circuit for a differential amplifier ($V_{output} = V_{input,2} - V_{input,1}$).

Build the circuit and measure the transfer function (plot both Bodeplots and the polar plot).

Analysis

Compare your results quantitatively to your prediction using an 'ideal' OpAmp.

Extra

Try various OpAmps and/or subtracting multiple inputs and compare the resulting outcomes. *Or add an integrator and try to build an analog P-feedback circuit to drive the thermostat.*

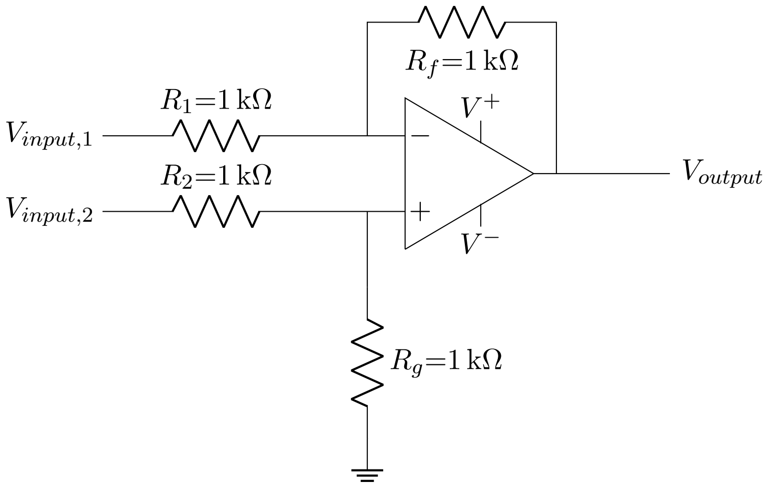


Figure 6.16 – *The schematic for a differential amplifier circuit. The output for this system is the difference between $V_{input,2}$ and $V_{input,1}$.*
While making sure that $R_1 = R_2$ and $R_f = R_g$ you can vary $\frac{R_f}{R_1}$ and implement a form of analog P-feedback. However, you would need an integrator-circuit (option G) as well to be able to apply it to e.g. a thermostat.

6.4.6 Option F: Differentiator

Introduction

The output of the differentiator is proportional to the time derivative of the input function. This makes the differentiator an essential component when dealing with peak and pulse detections. Since they react strongly to rising and falling edges, they may be used in sensitive timing circuits, oscilloscope triggers, and single-quantum measurements.

You may think V_{output} can now be used to provide D-feedback. However, here V_{output} is just the time derivative of the input signal, whereas for D-feedback we need the time derivative of the error signal. Therefore, we need an additional circuit, like a subtractor (option E). You do not need to worry about all this for now. Here, you are just researching the differentiator.

Theory

Use the general theory for an ideal OpAmp to predict the transfer function for this setup. *If you want to predict the transfer function for the non-ideal OpAmp, you can use LTspice.*

Method

In figure 6.17 you find the electronic circuit for a differentiator.

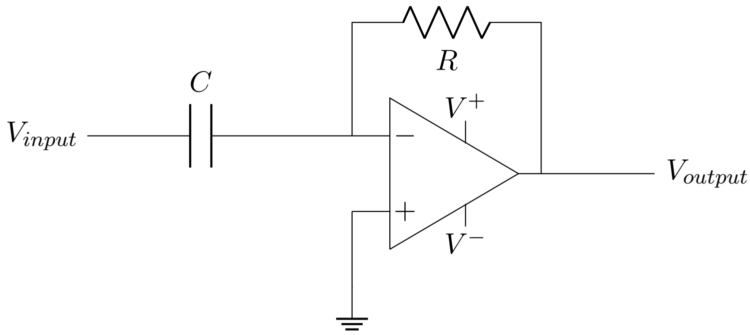


Figure 6.17 – *The schematic for a differentiator circuit. The output for this system is the time derivative of the input.*

Build the circuit (choose realistic values for R and C) and measure the transfer function (plot both Bodeplots and the polar plot).

Analysis

Compare your results quantitatively to your prediction using an 'ideal' OpAmp.

Extra

Try various OpAmps and/or various values for R and C and compare the resulting outcomes.

6.4.7 Option G: Integrator

Introduction

The output of the integrator is proportional to the integral of the input function over time. This makes the integrator very useful as a pulse stretcher for very fast signals. A sensitive integrator can be used to finely tune the voltage on a component with a very coarse controller. Such tricks are extremely useful when dealing with precise experimental setups such as STMs. For the optical system in session 3, the MyDAQ served as an integrator for the LED.

You may think V_{output} can now be used to provide I-feedback. However, here V_{output} is just the integration of the input signal (during a certain integration time), whereas for I-feedback we need the integration of the error signal. Therefore, we need an additional circuit, like a subtractor (option E). You do not need to worry about all this for now. Here, you are just researching the integrator.

Theory

Use the general theory for an ideal OpAmp to predict the transfer function for this setup. *If you want to predict the transfer function for the non-ideal OpAmp, you can use LTspice.*

Method

In figure 6.18 you find the electronic circuit for an integrator.

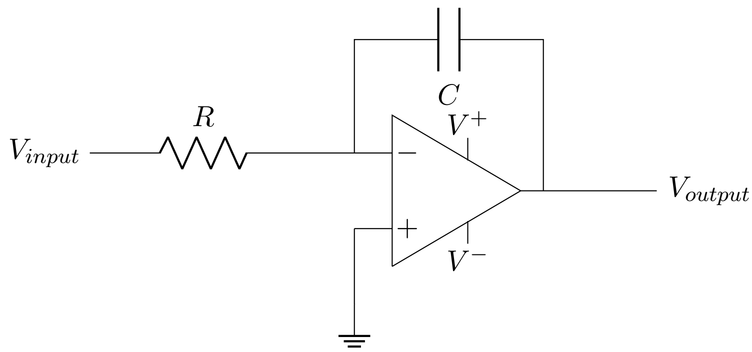


Figure 6.18 – *The schematic for a differentiator circuit. The output for this system is the time derivative of the input.*

Build the circuit and measure the transfer function (plot both Bodeplots and the polar plot). Make sure to choose realistic values for R and C, such that your expected output is not too small.

Analysis

Compare your results quantitatively to your prediction using an 'ideal' OpAmp.

Extra

Try various OpAmps and/or various values for R and C and compare the resulting outcomes.

6.5 Mandatory preparation

Here is summarized what you need to prepare.

- Write the first (dark grey) steps of your lab journal for each section. Note which option of experiments you will perform. We expect you to have a proper preparation for each research question. Of course you may divide these research questions into several smaller sub-research questions. You do not need to prepare experiments that you did not choose.
- Brush up the relevant theory on OpAmps and prepare any necessary code, e.g. to drive the MyDAQ for automated experiments, or to perform and analyze an FT of your data. Be sure to have finished exercise classes 8 and 9.
- Prepare your code to perform your Fourier analysis and generate Bode plots and polar plots.