

Laboratory Guide of Signals and Mechatronic Systems

for the 4th quarter of 2023/2024

Carlos Carneira, Duarte Valério, José Raul Azinheira, Mário Ramalho

March 28, 2024

Contents

Contents	ii
Introduction	iii
1 Generating and measuring electrical signals	1
1.1 Objectives	1
1.2 Previous study	1
1.3 Introduction to the assignment	1
1.3.1 AD laboratory	1
1.3.2 Multimeter	3
1.3.3 Oscilloscope	3
1.3.4 Function generator	4
1.3.5 Colour code for resistors and capacitors	4
1.3.6 Cables and jump wires	4
1.3.7 Box	5
1.3.8 Arduino Uno	5
1.4 Homework before class	5
1.5 In the laboratory	7
1.5.1 List of materials	7
1.5.2 Experimental procedure	7
2 Operational amplifiers (op-amps)	11
2.1 Objectives	11
2.2 Previous study	11
2.3 Introduction to the assignment	11
2.3.1 Operational amplifiers	11
2.3.2 Potentiometers	12
2.3.3 Tinkercad	12
2.4 Homework before class	12
2.5 In the laboratory	13
2.5.1 List of materials	13
2.5.2 Experimental procedure	14
3 Time responses	15
3.1 Objectives	15
3.2 Previous study	15
3.3 Introduction to the assignment	15
3.4 Homework before class	16
3.5 In the laboratory	17
3.5.1 List of materials	17
3.5.2 Experimental procedure	17

4	Frequency responses	19
4.1	Objectives	19
4.2	Previous study	19
4.3	Introduction to the assignment	19
	4.3.1 Lissajous curves	19
	4.3.2 Pulse width modulation (PWM)	20
4.4	Homework before class	21
4.5	In the laboratory	22
	4.5.1 List of materials	22
	4.5.2 Experimental procedures	24
	4.5.3 Troubleshooting	24
5	Signal filtering	29
5.1	Objectives	29
5.2	Previous study	29
5.3	Introduction to the assignment	29
5.4	Homework before class	30
5.5	In the laboratory	30
	5.5.1 List of materials	30
	5.5.2 Experimental procedure	31
6	Sensors and actuators in a control loop	33
6.1	Objectives	33
6.2	Previous study	33
6.3	Introduction to the assignment	33
6.4	Homework before class	34
6.5	In the laboratory	35
	6.5.1 List of materials	35
	6.5.2 Experimental procedure	36
	6.5.3 Troubleshooting	36
	Glossary	39

Introduction

Who would be satisfied with a navigator or engineer, who had no practice or experience whereby to carry on his scientific conclusions out of their native abstract into the concrete and the real?

John Henry NEWMAN (1801 — †1890)
An essay in aid of a grammar of assent, VIII 1, 2

Scheduling

There are six laboratory assignments, scheduled during April and May of 2024 as shown in the course webpage. Check carefully in which day your shift will have each of the six classes. Because of bank holidays (Freedom day, Labour day, and Corpus Christi) and the anniversary of IST, the different shifts will *not* have the same class on each week. Notice that:

- Monday, Tuesday and Friday shifts have classes during the first six weeks, and none during the last week;
- Wednesday shifts have classes down to the last week;
- Thursday shifts will have *two* classes during the last week, on Monday, Tuesday, or Friday.

Students who missed a class may complete that assignment during one day in June, yet to be announced. Priority will be given to students who missed a class for a reason allowed by law or regulation (e.g. illness or grief). Other students with a missing grade may enrol for the additional vacancies.

Classes

Classes take place in the Control, Automation and Robotics Laboratory, Mechanical Engineering III building, basement floor 1.

Classes must be prepared in advance, according to the instructions found in each chapter, in sections entitled “Homework before class”. Students who show up in class without preparing this homework will not be allowed to carry out the experiments, and will receive a 0 mark in that class.

Grades

Assignments are carried out in groups of two or three students, which will be formed at the beginning of the first class. In each class, the instructor grades each student *individually*. Differences in grade among member of the same group are possible because of different homework, different punctuality, or a clearly different participation in the experimental work.

Each assignment is graded from 0 to 4. The laboratory grade, from 0 to 20, is the sum of each student’s *five best grades*.

Punctuality

Punctuality is required when attending classes. The penalty for arriving late (i.e. more than 10 minutes after the beginning of the class) is a deduction of -0.5 to the mark of that class. The penalty for arriving very late (i.e. more than 20 minutes after the beginning of the class) is a deduction of -1 to the mark of that class.

Students enroled in previous academic years

Laboratory marks obtained in 2022/2023, and those only, remain valid this year, but students may repeat laboratory evaluation if they desire attempting a better grade; if they do, but fail to improve the grade this year, the 2022/2023 grade will be kept.

Older laboratory marks will *not* be considered.

Students entitled to special evaluation period

Students entitled to special evaluation period (and only those) may obtain their laboratory mark during that period. They do not need to attend classes during the semester. Once enroled in the special evaluation period, they must contact Prof. Duarte Valério to signify their interest in laboratory special evaluation and arrange deadlines and class days. The assignments are the same.

Assignments that depend on your student number

Simulation assignments have parameters that depend on the five or six digits of your student number, n_0 to n_5 . Your student number is thus equal to $n_5 \times 10^5 + n_4 \times 10^4 + n_3 \times 10^3 + n_2 \times 10^2 + n_1 \times 10 + n_0$. For instance,

- for student number 109876, $n_5 = 1$, $n_4 = 0$, $n_3 = 9$, $n_2 = 8$, $n_1 = 7$ and $n_0 = 6$;
- for student number 32109, $n_5 = 0$, $n_4 = 3$, $n_3 = 2$, $n_2 = 1$, $n_1 = 0$ and $n_0 = 9$.

Exercises where students fail to use the proper values depending on their student number will not be considered for marks.

Links in this guide

This laboratory guide includes several links to websites where you can further explore the subjects addressed. (Use a PDF viewer that highlights links to see them.)

An important piece of advice for classes

Turn the power *off* before building your circuits. Check connections *before* turning the power on. In case of doubt, call your instructor. In no way can our experiments turn out as bad as that in Figure 1. But still.

Gontran



et les autres

GÉGÉ



POUR LA CENTIÈME FOIS...!

...ET JE NE CESSERAI JAMAIS DE LE RÉPÉTER...
N'EST-CE PAS...



...QUAND ON RÉALISE UN MONTAGE ÉLECTRONIQUE EN "TRAVAUX PRATIQUES", N'EST-CE PAS...?



...ET QUAND CE MONTAGE EST PARTICULIÈREMENT COMPLEXE, N'EST-CE PAS, VOYEZ...?



...EH BIEN! AVANT DE PASSER AUX...
HEIN!



...AVANT DE BRANCHER LE TOUT SUR LE "SECTEUR"? N'EST-CE PAS...?



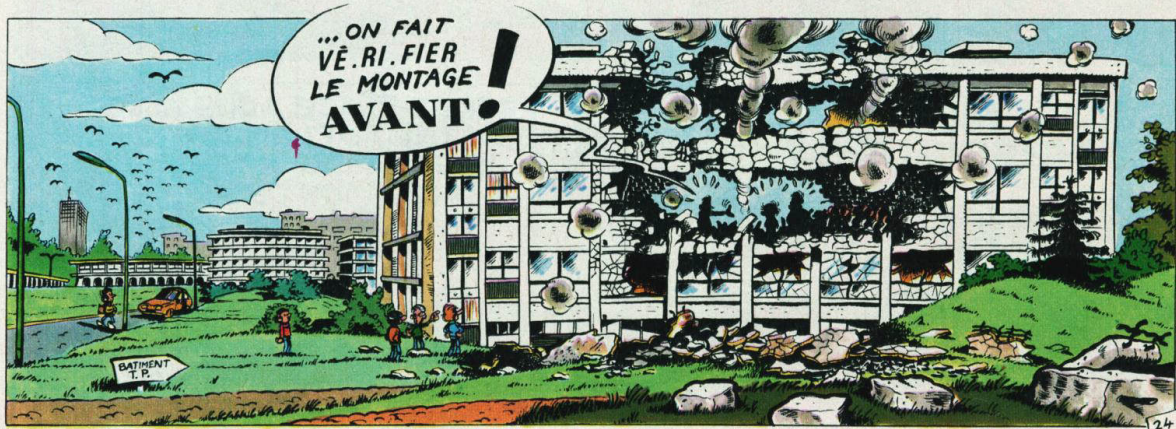
...AVANT TOUT ÇA... HEIN?... JE VOUS LE DEMANDE...?



QUE FAIT-ON?



BEN...
...ON... ON FAIT VÉRIFIER LE MONTAGE AVANT...
PARFAITEMENT...



ON FAIT VÉRIFIER LE MONTAGE AVANT!

GÉGÉ 24

Figure 1: Journal de Spirou, 1980/2/21, numéro 2184, page 21.

Chapter 1

Generating and measuring electrical signals

1.1 Objectives

This Chapter has three objectives:

- to provide a first contact with material used in electronics;
- to acquaint you with measurement errors, their sources, and their acceptable (and unacceptable) values;
- to be a first contact with the Arduino microprocessor.

1.2 Previous study

To finish this assignment, your knowledge previous to this course should suffice.

1.3 Introduction to the assignment

1.3.1 AD laboratory

The AD laboratory (Figure 1.1) has:

- a breadboard, for the prototyping of electrical circuits;
- a voltmeter, to measure electrical potential in a circuit;
- an ammeter, to measure the current in a circuit;
- a DC power supply;
- a function generator;
- and other elements which we will not need.

The breadboard has hundreds of perforations, where components can be inserted. Below the perforations, metallic strips (Figure 1.2) will connect the inserted components. Avoid using perforations where the plastic is burned (i.e. where someone performed a failed experiment that damaged the material).

Some perforations are connected vertically (terminal strips) and some are connected horizontally (bus strips). Figure 1.3 shows both strips and the manner of inserting integrated circuits. Notice that some breadboards have bus strips from side to side; others have separate bus strips for the left and right sides of the breadboard. This is the case of the breadboard in Figure 1.2, and the two separate bus strips are indicated by red and blue lines.

As you should know,



Figure 1.1: AD laboratory.

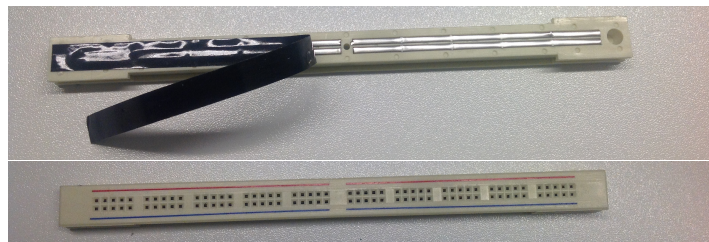


Figure 1.2: How breadboard perforations are connected.

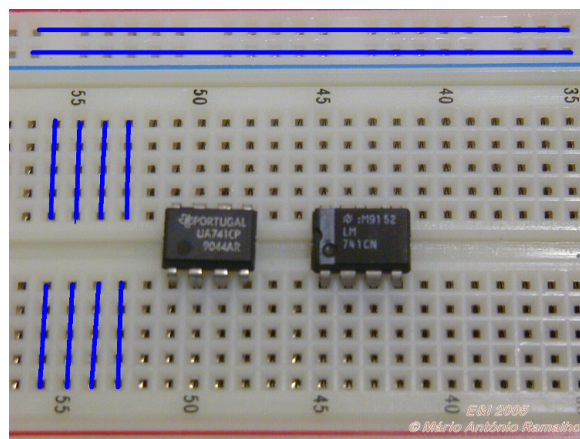


Figure 1.3: Terminal strips (vertical), bus strips (horizontal), and how to insert an integrated circuit.

- the voltmeter is connected in parallel with the circuit where you measure the electrical potential;
- the ammeter is connected in series with the circuit where you measure the current.

Instead of the AD laboratory's voltmeter and ammeter, it is more expedient to use the multimeter, described in Section 1.3.2. Instead of the AD laboratory's function generator, which provides signals with a poor quality, it is more expedient to use an external function generator, described in Section 1.3.4. However, we will need the AD laboratory's function generator during class 5.

1.3.2 Multimeter

The multimeter (Figure 1.4) can work as

- voltmeter,
- ammeter,
- ohmmeter,
- frequency meter,
- capacitance meter.

Notice that cables must be connected differently for different functions. If you hear the multimeter beep, this is (almost always) because your cables are wrongly connected. Correct your mistake *at once* or you may damage the multimeter.



Figure 1.4: Multimeter.

1.3.3 Oscilloscope

The oscilloscope is used to show a plot:

- of one or two electrical potentials (y -axis, i.e. vertical axis) as a function of time (x -axis, i.e. horizontal axis): see Figure 1.5, left;
- of one electrical potential (y -axis) as a function of another electrical potential (x -axis): see Figure 1.5, right.

Oscilloscopes use BNC (Bayonet Neill-Concelman) connectors; Figure 1.6 shows a BNC tee connector, to connect the same signal to two connectors.

Practical advice:

- Before you use a channel of the oscilloscope, set its selector to GND (ground) and make sure that the channel's offset is set to zero.
- There are buttons for fine tuning the amplitude. If used, you cannot know which scale you are using. So, always make sure they are *not* in use.

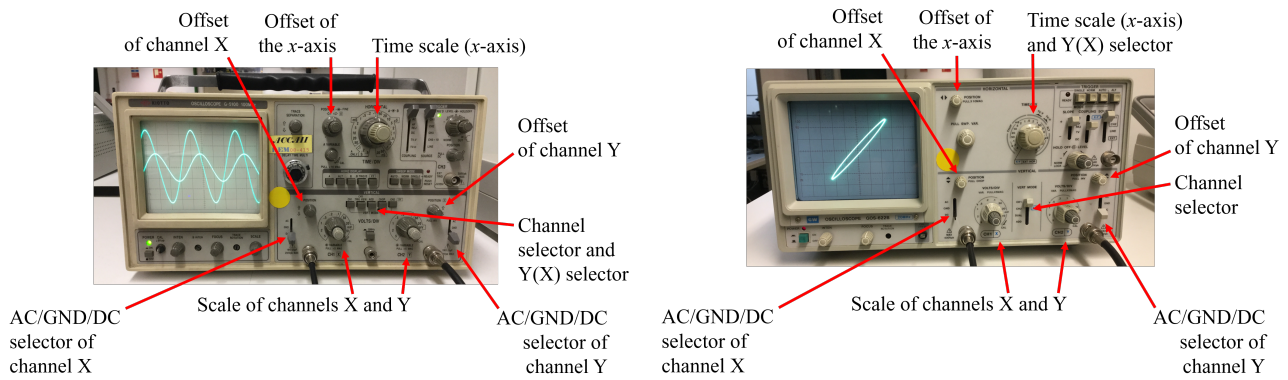


Figure 1.5: Some of the oscilloscopes in the laboratory; left: two sinusoidal signals as functions of time; right: two sinusoidal signals, one as function of the other.



Figure 1.6: BNC tee connector.

1.3.4 Function generator

The function generator (Figure 1.7) can provide periodic variations of the electric potential. The frequency, amplitude and mean value of the resulting signal can be adjusted. The shape of the wave can be sinusoidal, square, or triangular.

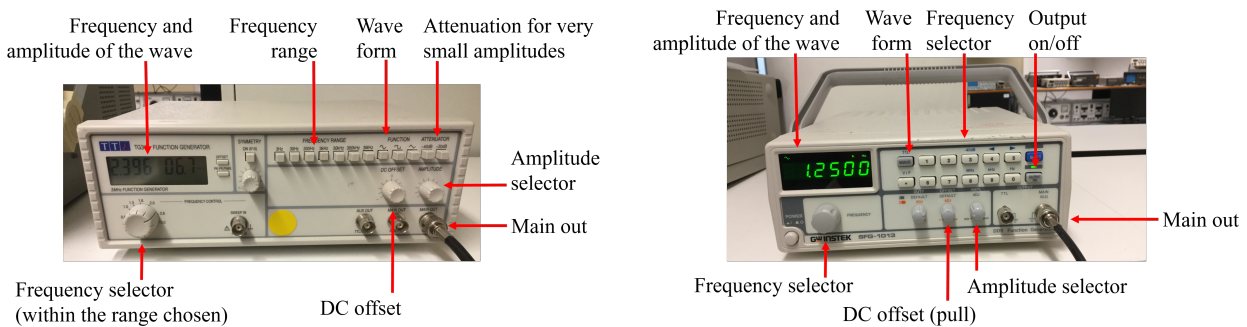


Figure 1.7: Some of the function generators in the laboratory.

1.3.5 Colour code for resistors and capacitors

See Figure 1.8 for the colour code of resistors. Capacitors in the laboratory are not marked with this colour code, but they could be. You can find a more complete version of the colour code here.

1.3.6 Cables and jump wires

See Figure 1.9. Practical advice:

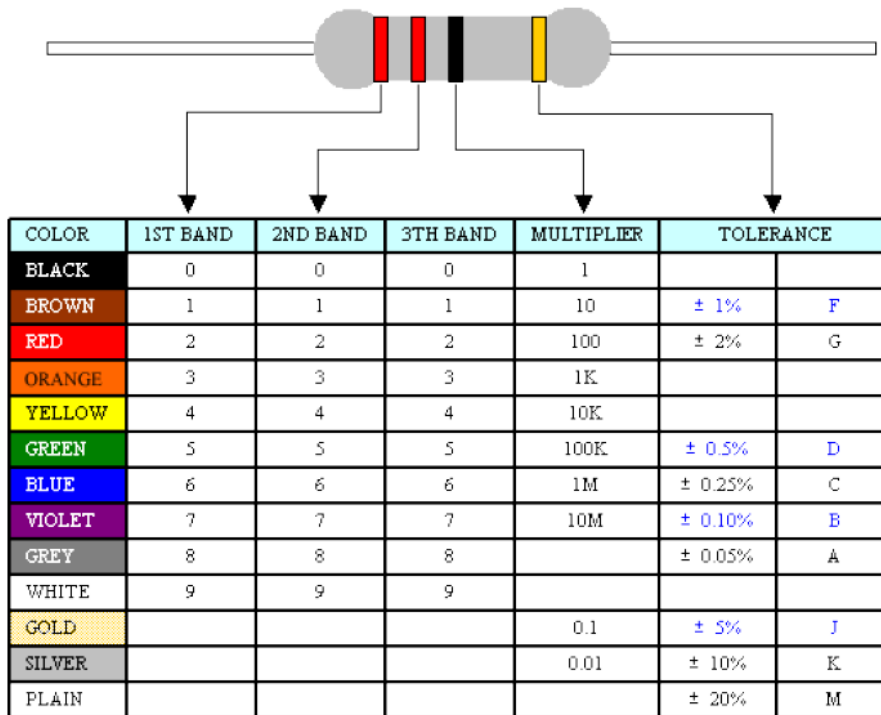


Figure 1.8: Colour code.

- Jump wires or jumpers, used for connections in the breadboard, should have a short stripped wire at the end; if the stripped wire is too long, two metallic parts may touch and spoil your circuits. Use the wire stripper and the pliers to solve this.
- You can replace a BNC–jump wire cable with a BNC–bananas cable and two banana–jump wire cables.

1.3.7 Box

There is one box (Figure 1.10) per workbench with jump wires, pliers and a wire stripper.

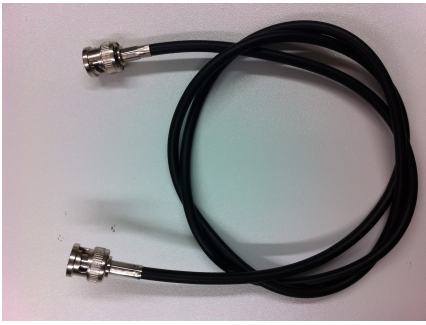
1.3.8 Arduino Uno

Arduino Uno, shown in Figure 1.11, is a general purpose microcontroller. You can view it as a rather simple computer, to which we can connect sensors and actuators, and which we can program using Matlab, or using a variant of the C++ programming language (which you are not expected to know). You will use one of the laboratory’s desktop computers to do so; if you prefer, you may use a laptop you bring, in which case you must configure the software yourself.

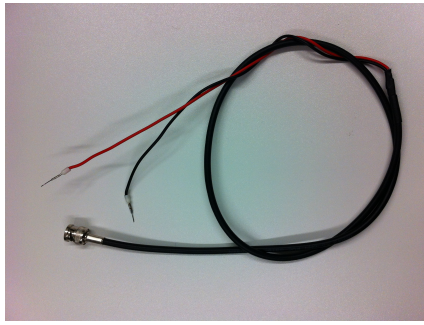
1.4 Homework before class

To prepare this class:

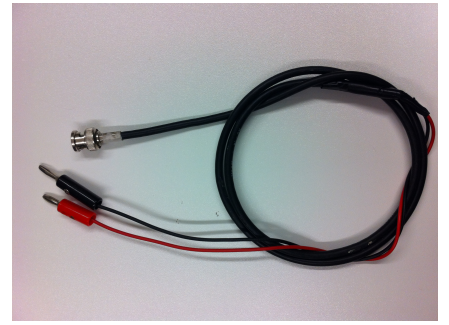
- Study the *Introduction to the assignment* above.
- Then, go to Playlist *Sinais e Sistemas Mecatrônicos, laboratório* and watch the first nine videos (of which one is in English, the others in Portuguese), that is,
 - from video “SSM Lab01 Material do armário”,
 - to video “Lab1 Teste Blink”.
- Study the experimental procedure.



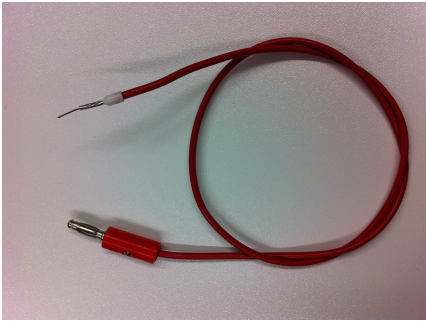
BNC-BNC cable



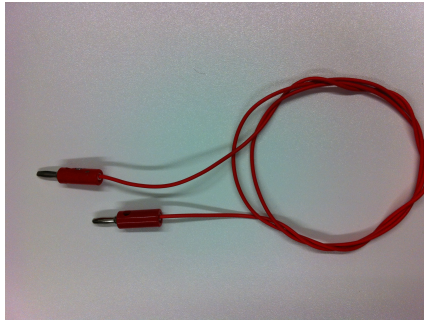
BNC-jump wire cable



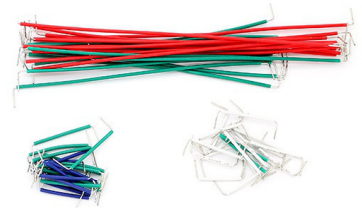
BNC-bananas cable



Banana-jump wire cable



Banana-banana cable



Jump wires or jumpers

Figure 1.9: Cables and jump wires.



Figure 1.10: Box with jump wires, pliers, and a wire stripper.

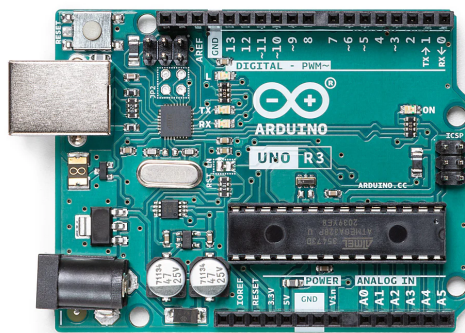


Figure 1.11: An Arduino Uno microcontroller.

- Fill in the following table, ticking the appropriate column for each device:

Device	Generates a signal	Measures a signal
Voltmeter	<input type="checkbox"/>	<input type="checkbox"/>
Ammeter	<input type="checkbox"/>	<input type="checkbox"/>
DC power supply	<input type="checkbox"/>	<input type="checkbox"/>
Function generator	<input type="checkbox"/>	<input type="checkbox"/>
Frequency meter	<input type="checkbox"/>	<input type="checkbox"/>
Capacitance meter	<input type="checkbox"/>	<input type="checkbox"/>
Oscilloscope	<input type="checkbox"/>	<input type="checkbox"/>

- Write the Matlab code to make the Arduino led blink, as specified below for item 8 of the *Experimental procedure*. You have all the important commands below in item 7 of the Experimental Procedure. This assignment depends on your student number, as explained in this Guide's Introduction, page iv.

1.5 In the laboratory

Show the instructor the table above filled in.

1.5.1 List of materials

- AD laboratory.
- Multimeter.
- Function generator.
- Oscilloscope.
- 470 Ω , 1 k Ω and 4,7 k Ω resistors.
- Arduino Uno.
- Cables, jumper wires, BNC tee.

1.5.2 Experimental procedure

1. Insert the three resistors in the breadboard. Use the multimeter to measure the resistance of each resistor. Fill in the upper part of the following table (each group only needs to fill in one table).

Values under ideal conditions			
R_{nominal} colours	470 Ω	1 k Ω	4,7 k Ω
U (V)	5	5	5
I (mA)			
Values measured in the laboratory			
R_{measured}			
I (mA)			
U (V)			
U/I (Ω)			
resistor's tolerance in %			
is U/I within $R \pm \text{tolerance}$?	Yes / No	Yes / No	Yes / No

2. Build the circuit in figure 1.12. Measure the electrical potential and the current for each of the three resistors. Fill in the lower part of the table above and show it to the instructor.

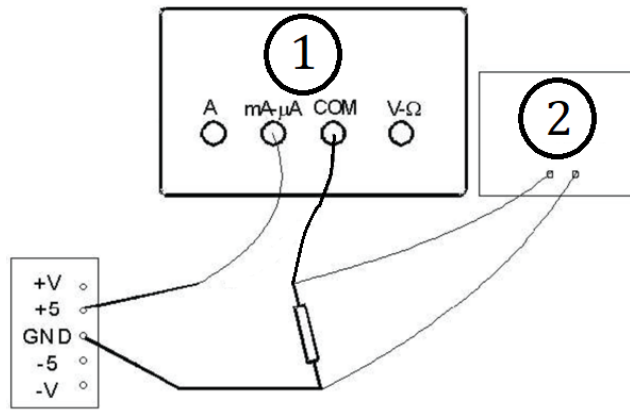


Figure 1.12: Circuit for item 2. ① is a multimeter, working as ammeter. ② is the AD laboratory's voltmeter.

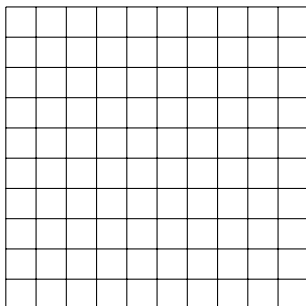
- Connect a BNC-jump wire cable to channel X of the oscilloscope. Set the channel's selector to GND and adjust the offset to zero (i.e. make sure you see a horizontal line on the x -axis). Set the channel's selector to DC and hold the red jump wire with your hand. Adjust the vertical and time scales until you see a roughly sinusoidal signal. Measure its frequency and show the instructor how you found it and where this signal comes from.
- Put a BNC tee at the function generator's output. Connect it to channel X of the oscilloscope with a BNC-BNC cable. Turn the function generator on and generate a sinusoid with 10 V pp (i.e. with a peak-to-peak amplitude of 10 V, i.e. with a peak amplitude of 5 V), zero average, and 100 Hz. Adjust the vertical and time scales until you see two periods in the screen. Use the multimeter as an AC voltmeter, and connect it to the BNC tee with a BNC-bananas cable. Measure the electrical potential. Press the multimeter's "Hz" button to use it as a frequency meter (notice that the simplest multimeters in the laboratory do not have this function; you will have to use one of the most complete). Fill the table below, to compare the values measured with the function generator, the oscilloscope, and the multimeter; then show it to the instructor.

	Function generator	Oscilloscope	Multimeter
Amplitude			
Frequency			

- Keep the BNC tee in the function generator's output. Connect it to both the X and Y channels of the oscilloscope. Set the selector of channel X to DC and the selector of channel Y to AC. Set the function generator for a 2 V pp, 2 kHz sinusoid. Change the function generator's DC offset and notice the settling time. Sketch below what you see.

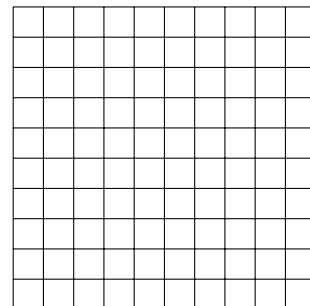
Channel X, set to DC

Oscilloscope's scales: _____



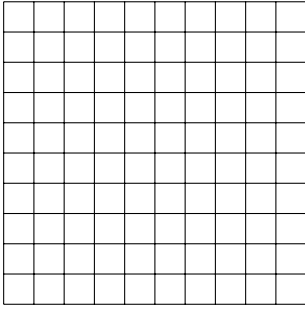
Channel Y, set to AC

Oscilloscope's scales: _____

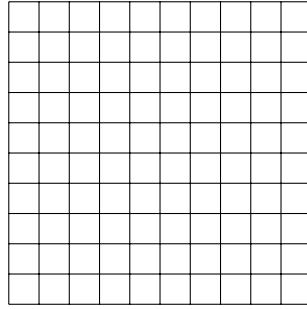


- Keep the same connections, and set the function generator for a 5 V pp square wave without DC offset. Sketch above what you see for frequencies 10 Hz, 100 Hz, and 1 kHz. Call the instructor, show the sketches, and discuss the shape of the output.

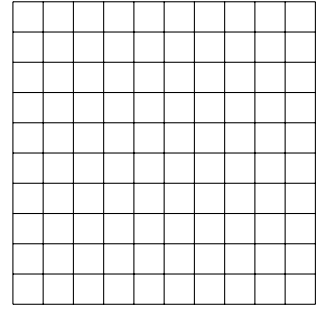
Channels X and Y, freq. 10 Hz
Oscilloscope's scales: _____



Channels X and Y, freq. 100 Hz
Oscilloscope's scales: _____



Channels X and Y, freq. 1 kHz
Oscilloscope's scales: _____



7. Connect the Arduino to the desktop computer using a USB cable, and check in the Device Manager the number of the corresponding COM port. Then run the following commands in Matlab to make sure that everything works:

```
ard = arduino('COM3', 'UNO') % replace 3 with the correct port number  
writeDigitalPin(ard, 'D13', 0); % this turns off the led connected to digital output 13  
writeDigitalPin(ard, 'D13', 1); % this turns on the led connected to digital output 13
```

8. Run the programs written as homework by the group members. The program of each student should make the led blink $n_0 + n_1 + 3$ times, during $\frac{n_2+n_3+1}{2}$ seconds, with $\frac{n_4+n_5+1}{2}$ seconds of interval. Show the instructor the blinking led of the Arduino (with the programs written by each group member).

Chapter 2

Operational amplifiers (op-amps)

2.1 Objectives

This Chapter has two objectives:

- to give you practical knowledge of some of the simplest and most useful circuits you can build with an op-amp;
- to provide a first contact with a simulation software for electronic circuits, Tinkercad.

2.2 Previous study

To finish this assignment, you must first study operational amplifiers and the circuits built with this component:

- in the Lecture Notes of Signals, Systems and Control, section 5.3;
- in playlist Sinais e Sistemas Mecatrônicos, parte 1: modelação, videos SSM T03 04 to SSM T03 09.

2.3 Introduction to the assignment

2.3.1 Operational amplifiers

The operational amplifiers or op-amps that we will use in class are LM741 op-amps. See the integrated circuit's pinout in figure 2.1 and check figure 1.3 again. This is a very common op-amp, but there are other integrated circuits with different specifications (and pinouts) available in the market.

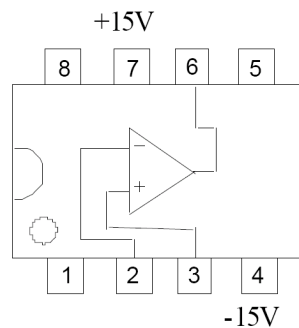


Figure 2.1: LM741 op-amp pinout (notice the indentation on the left side).

2.3.2 Potentiometers

A potentiometer (or variable resistor, or rheostat), shown in Figure 2.2, is a resistor provided with a slider. The left and right terminals are connected to the terminals of the resistor; the middle terminal is connected to the slider. The slider changes the length of the resistance between the middle terminal and each of the other two. Remember that a conductor's resistance is proportional to its length.

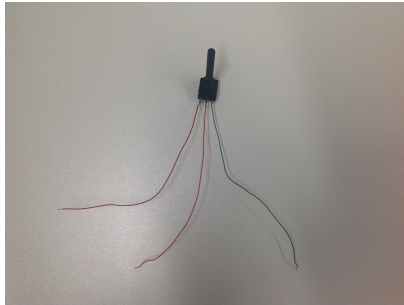


Figure 2.2: Variable resistor, or potentiometer, or rheostat.

2.3.3 Tinkercad

Tinkercad is a free online software that can be used, among other things, to simulate electronic circuits. Visually, it is very similar to what you can see in the laboratory. You need to register a free account at <http://www.tinkercad.com> to use this software. There is a video, mentioned below in the Homework list, which will help you learn how to use it to build simulations of simple electronic circuits.

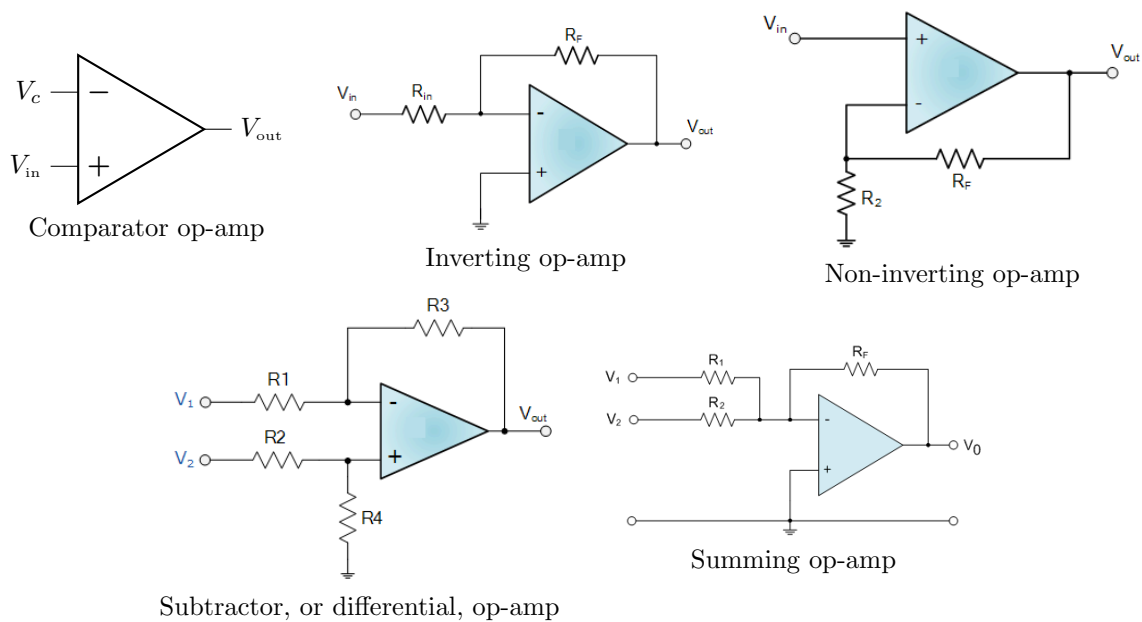


Figure 2.3: Op-amp circuits.

2.4 Homework before class

To prepare this class:

- Study the *Introduction to the assignment* above.
- Then, go to Playlist *Sinais e Sistemas Mecatrônicos, laboratório* and watch the tenth and eleventh videos (in Portuguese), that is,

- from video “SSM Lab10 Montar um circuito com um ampop” (showing what in the past was done in the first laboratory class, and is now part of this second class),
- to video “SSM Lab11 Simular no Tinkercad”.
- Study the experimental procedure.
- Reproduce in Tinkercad two of the circuits that you will carry out in the laboratory, seen in Figure 2.3. The circuits you will implement depend on your student number, as explained in this Guide’s Introduction, page iv:
 - your first circuit is
 - * the inverting op-amp circuit, with $R_{in} = 1 + n_2$ k Ω and $R_F = 2R_{in}$, if n_0 is odd,
 - * the comparator op-amp circuit, if n_0 is even;
 - your second circuit is
 - * the subtractor op-amp circuit, with all resistors equal to $1 + n_3$ k Ω , if n_1 is odd,
 - * the summing op-amp circuit, with all resistors equal to $1 + n_3$ k Ω , if n_1 is even.

For both these circuits:

- demonstrate, in your hand-writing, the input-output relation;
- in your Tinkercad simulation, use a 741 op-amp;
- feed it with +15 V and –15 V, using two power supplies;
- use a function generator as input;
- set the function generator to provide a 2 kHz, 15 V pp sinusoid with a +5 V offset (mean value), save for the comparator, that requires a 0 V offset;
- for the subtractor or summing circuits, which have two inputs, use as second input a $5 + n_0$ V power supply;
- for the comparator, which also has a second input V_c , use a power supply providing a constant $3 + n_5$ V voltage;
- use two oscilloscopes to visualise both the input and the output;
- adjust the time scales of the two oscilloscopes for the same value;
- choose that value so that you can see two or three periods of each signal.

2.5 In the laboratory

Show the instructor

- the handwritten demonstrations, and
- the Tinkercad simulations

asked for in the homework.

2.5.1 List of materials

- AD laboratory.
- Multimeter.
- Function generator.
- Oscilloscope.
- LM741 op-amp.
- Four 4.7 k Ω resistors.
- Two potentiometers.
- Cables, jumper wires, BNC tee.

2.5.2 Experimental procedure

Build the five circuits in Figure 2.3 following these steps:

1. Use the multimeter to set the AD laboratory's DC variable power supply to +15 V and -15 V. These values are needed to supply the op-amp.
2. Use the function generator to generate a 1 kHz, 15 V pp sinusoid, with a 0 V offset (mean value).
3. Build the comparator op-amp circuit. Connect the function generator to V_{in} , the AD laboratory's +5 V power supply to V_c , as shown in figure 2.3, V_{in} to channel X in the oscilloscope, and the circuit's output to channel Y. Adjust the vertical and time scales so that you can see two periods on screen.
4. Call your instructor and show your completed assignment working. Explain what would happen with the output if you could change the value of V_c .
5. Change the function generator offset (mean value) to +5 V. Keep the amplitude and frequency of the sinusoid.
6. Build the inverting op-amp circuit. Use $R_{in} = 4.7 \text{ k}\Omega$ and $R_F = 9.4 \text{ k}\Omega$. (To obtain 9.4 k Ω , should you use two 4.7 k Ω resistors in series or in parallel?) Connect the function generator to the circuit's input, the circuit's input to channel X in the oscilloscope, and the circuit's output to channel Y. Adjust the vertical and time scales so that you can see two periods on screen.
7. Call your instructor and show your completed assignment working. Change the offset of the input and explain the reason why the output is not always a sinusoid.
8. Build the non-inverting op-amp circuit. Use $R_2 = 4.7 \text{ k}\Omega$ and $R_F = 9.4 \text{ k}\Omega$. Connect the function generator to the circuit's input, the circuit's input to channel X in the oscilloscope, and the circuit's output to channel Y. Adjust the vertical and time scales so that you can see two periods on screen.
9. Call your instructor and show your completed assignment working. Change the offset of the input and explain the reason why the output is not always a sinusoid.
10. Build the subtractor op-amp circuit. Use $R_1 = R_2 = R_3 = R_4 = 4.7 \text{ k}\Omega$. Connect V_1 to the AD laboratory's +5 V DC power supply, V_2 to the the function generator and to channel X in the oscilloscope, and the circuit's output to channel Y. Adjust the vertical and time scales so that you can see two periods on screen.
11. Call your instructor and show your completed assignment working. Change the offset of input V_2 and explain the reason why the output is not always a sinusoid.
12. Build the summing op-amp circuit. Use $R_1 = R_2 = R_F = 4.7 \text{ k}\Omega$, employing potentiometers for R_1 and R_2 , and a resistor for R_F . Connect V_1 to the AD laboratory's +5 V DC power supply, V_2 to the the function generator and to channel X in the oscilloscope, and the circuit's output to channel Y. Adjust the vertical and time scales so that you can see two periods on screen.
13. Call your instructor and show your completed assignment working. Change the values of the potentiometers and explain what happens with the output.

Chapter 3

Time responses

3.1 Objectives

This chapter has two objectives:

- to study the time responses of RC and RLC circuits as examples of first and second order systems;
- to give you a practical notion of what the time constant and the damping factor of a system are.

3.2 Previous study

To finish this assignment, you must first study what time responses, and in particular step responses, are:

- in the Lecture Notes of Signals, Systems and Control, sections 10.1 to 10.3;
- in playlist Sinais e Sistemas Mecatrônicos, parte 2: sistemas dinâmicos, videos SSM T07 01 to SSM T08 01.

You can study what the time responses of first and second order systems are:

- in the Lecture Notes of Signals, Systems and Control, sections 11.2 and 11.3, *passim*;
- in playlist Sinais e Sistemas Mecatrônicos, parte 2: sistemas dinâmicos, videos SSM T09 03, SSM T09 05, and SSM T09 07.

3.3 Introduction to the assignment

Figure 3.1 shows a first order system and a second order system. As you know, there are other systems (mechanical, thermal, etc.) with the same transfer functions.

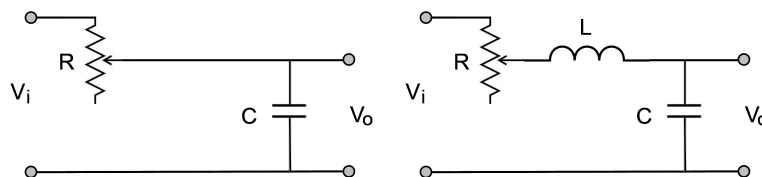


Figure 3.1: Left: a first-order circuit, with a resistor and a capacitor (RC circuit); right: a second order circuit, with a resistor, an inductance, and a capacitor (RLC circuit).

We will study the step responses of these two systems, using a square wave. As you know, if the period is large enough, in each half period the transient response vanishes, and only the steady state response remains. Of course, what a large enough period is depends on the system's time constant.

3.4 Homework before class

To prepare this class:

- Study the *Introduction to the assignment* above.
- Study the experimental procedure.
- Find in the Lecture notes the expression for the time constant of the first order system.
- Find in the Lecture notes the expressions for the maximum overshoot and the peak time of the second order system.
- For the second order system:
 - knowing that you will use two 100 mH inductors in series, find the equivalent inductance;
 - knowing that you will use a 100 nF capacitor, find three values of resistance R , corresponding to
 - * an underdamped system with $\xi = 0.39 + \frac{n_0}{50}$,
 - * a critically damped system, and
 - * an overdamped system with $\xi = 1.2 + \frac{n_1}{50}$;
 - find for each of the three values of R the poles of the system's transfer function;
 - find for the underdamped system the maximum overshoot and the peak time (see Figure 3.2);

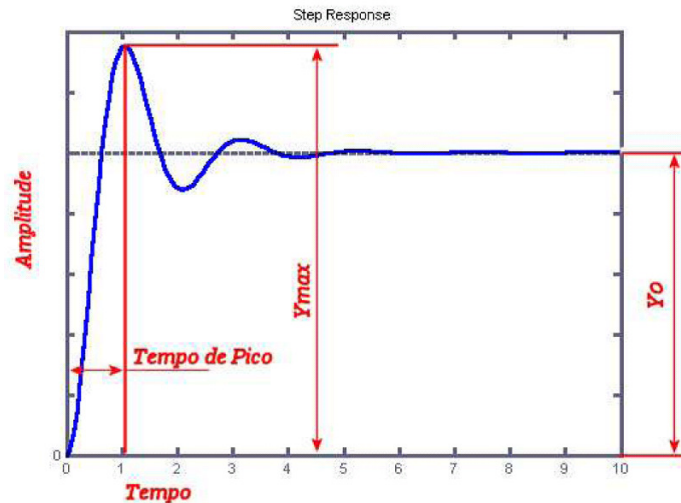


Figure 3.2: Maximum overshoot and peak time of a step response seen on the oscilloscope.

- use Matlab function `step` to plot the step response for the three transfer functions corresponding to the three values of R , and save the plots (in a format such as .png, or pasting them in a .docx document).
- Simulate in Tinkercad two systems that you will implement in the laboratory, with the same inputs that you will use:
 - the first system depends on n_2 :
 - * if n_2 is even, implement the RC circuit described below in point 2,
 - * if n_2 is odd, implement the RC circuit described below in point 4,
 and then check that the time constant (which is equal to the time the output takes to rise up to 63% of the final value) corresponds to product RC ,
 - the second system is the underdamped RLC system described below in point 9, and then check:
 - * if n_3 is even, that the maximum overshoot M_p corresponds to what you calculated,
 - * if n_3 is odd, that the peak time t_p corresponds to what you calculated.

3.5 In the laboratory

3.5.1 List of materials

- AD laboratory.
- Multimeter.
- Function generator.
- Oscilloscope.
- Potentiometer.
- 100 nF capacitor.
- two 100 mH inductors.
- Cables, jump wires, BNC tee.

3.5.2 Experimental procedure

1. Use the multimeter to measure the capacitor and set the potentiometer to 4 k Ω .
2. Build the RC circuit from Figure 3.1. Use the function generator to provide a 60 Hz square wave with 6 V pp as input V_i . View the in channel A of the oscilloscope, and the output V_o in channel B. Use a time scale that shows one period of the square wave.
3. Measure from the response the time constant of the system, knowing that it is equal to the time it takes to rise up to 63% of the final value. Compare it to the product RC (using the measured values, not the nominal ones). Show the results to the instructor, comparing them with what you obtained in your homework.
4. Change the potentiometer to 8 k Ω , and repeat all the steps above. Show the results to the instructor, comparing them with what you obtained in your homework.
5. Inductors are not ideal, so they have not only an inductance L but also a resistance R_L . Measure R_L with the multimeter and account for it in all that follows, subtracting it to the desired values of R to find the correct resistance for the potentiometer.
6. Change the potentiometer so that R will correspond to the value you determined for an overdamped system. (Each group does this for only one value, used by one of its members.)
7. Build the RLC circuit in Figure 3.1, with the two inductors in series. Use as input V_i a 200 Hz square wave with 6 V pp. Show the result to the instructor.
8. Repeat the above for the critically damped system. Show the result to the instructor.
9. Repeat the above for the underdamped system. Measure maximum overshoot M_p and peak time t_p , comparing them with the values you obtained in your homework. Show the results to the instructor.

Chapter 4

Frequency responses

4.1 Objectives

This chapter has three objectives:

- to help you understand the notion of frequency response (not so easy to grasp as that of time response);
- to study the frequency responses of RC and RLC circuits as examples of first and second order systems;
- to provide an introduction to pulse width modulation (PWM).

4.2 Previous study

To finish this assignment, you must first study what frequency responses are:

- in the Lecture Notes of Signals, Systems and Control, sections 10.4 and 10.5;
- in playlist Sinais e Sistemas Mecatrônicos, parte 2: sistemas dinâmicos, videos SSM T08 02 to SSM T08 05.

You can study what the frequency responses of first and second order systems are:

- in the Lecture Notes of Signals, Systems and Control, sections 11.2 and 11.3, *passim*;
- in playlist Sinais e Sistemas Mecatrônicos, parte 2: sistemas dinâmicos, videos SSM T09 04, SSM T09 06, and SSM T09 08.

4.3 Introduction to the assignment

The circuits you will use are again those of Figure 3.1.

4.3.1 Lissajous curves

As you know, since the systems we are studying are linear, when the input is sinusoidal, the steady-state output is sinusoidal too, and has the same period of the input. This is shown in Figure 4.1.

Oscilloscopes can show the output as function of the input. A curve parametrised as a sinusoid as function of another sinusoid is called a Lissajous curve. It is easy to see that, when the two sinusoids have the same period, the Lissajous curve becomes an ellipse.

- If the two sinusoids have the same amplitude, and the scales of the two axis are equal, we will see a circle.
- The scales of the axis are set by the V/div buttons of the oscilloscope.
- If the two sinusoids are in phase, the axis of the ellipse will be aligned with the vertical and horizontal axis. Otherwise, the ellipse will be rotated.

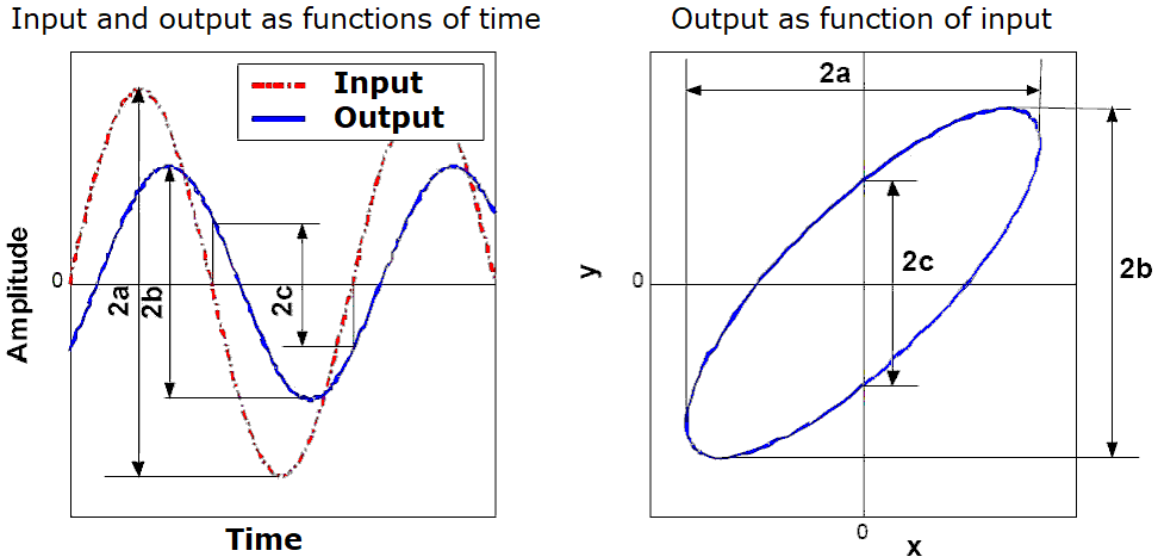


Figure 4.1: Left: input and output of a linear system. Right: corresponding Lissajous curve.

- In Figure 4.1, the input V_i and the output V_o are given by

$$V_i = a \sin(\omega t) \quad (4.1)$$

$$V_o = b \sin(\omega t + \phi) \quad (4.2)$$

- Notice that
 - $2a$ is the peak-to-peak amplitude of the input,
 - $2b$ is the peak-to-peak amplitude of the output,
 - $2c$ is measured from the output at the zero-crossing of the input.
- The gain of the system is

$$|G(j\omega)| = \frac{2b}{2a} \Leftrightarrow |G(j\omega)|_{\text{dB}} = 20 \log_{10} \frac{2b}{2a} \quad (4.3)$$

- The phase of the system is found from

$$2c = 2b |\sin \angle G(j\omega)| \quad (4.4)$$

4.3.2 Pulse width modulation (PWM)

A digital output can only provide two values:

- one corresponding to logical value 0 (or false),
- the other corresponding to logical value 1 (or true).

In the case of voltage digital outputs, the value for 0 is always 0 V, measured in relation to the ground; in what follows, we will let the value for 1 be 5 V, which is a typical possibility in electronics (though surely not the only one).

It is possible to change the output between these two values so that the mean value of the output will be any value U between 0 V and 5 V. At any instant, the output is either 0 V or 5 V; the changes are a sequence of steps such that

- the output is periodic,
- during a fraction of time $1 - \frac{U}{5}$ the output is 0 V,

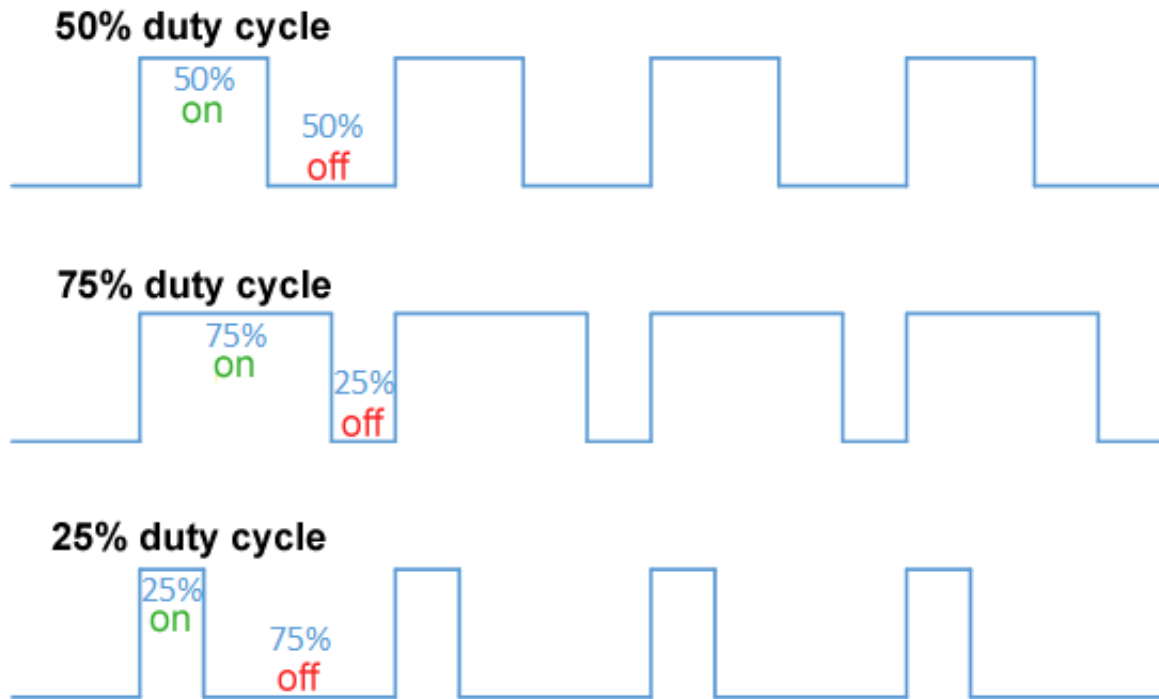


Figure 4.2: Pulse width modulation, with three different values of duty cycle. (Source: Wikimedia.)

- during a fraction of time $\frac{U}{5}$ the output is 5 V. This fraction of time is called duty cycle.

The successive steps define a sequence of pulses, and it is the width of the pulses that determines U . Thus, this strategy is called pulse width modulation (PWM). See Figure 4.2, in which, the instantaneous output being either 0 V or 5 V,

- a duty cycle of 50% corresponds to an average output of 2.5 V,
- a duty cycle of 75% corresponds to an average output of 3.75 V,
- a duty cycle of 25% corresponds to an average output of 1.25 V.

Varying the duty cycle with time, it is possible to emulate a sinusoid with a digital curve. The Arduino Uno has no analog outputs, so using PWM to emulate a sinusoid using a digital output is the best we can do. Notice that:

- the sinusoid can be recovered filtering the PWM, as we will do in the laboratory;
- there are several different ways of determining how the duty cycle should vary with time. For instance, a very coarse, but simple method, consists in approximating the sinusoid as a sequence of steps, and then approximating each step using PWM. In the laboratory, we will use code which has already been developed, so you do not have to worry about this problem.

See Figure 4.3. Obviously, the period of the sinusoid must be clearly higher than the period of the PWM pulses, otherwise it would not be possible to approximate the sinusoid. To ensure that the frequency of the pulses is high enough, we can no longer control the Arduino using Matlab, since communications would take too long: we have to compile a program (written in a variant of the C++ programming language, which you do not have to know), download it to the Arduino, and run it there.

4.4 Homework before class

To prepare this class:

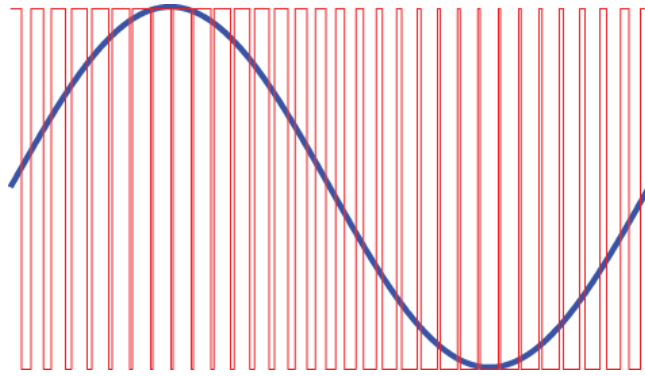


Figure 4.3: How a sinusoid can be approximated with PWM. (Source: https://ctms.engin.umich.edu/CTMS/index.php?aux=Activities_BoostcircuitB.)

- Study the *Introduction to the assignment* above.
- Then, go to Playlist *Sinais e Sistemas Mecatrônicos, laboratório* and watch the last three videos (of which one is in Portuguese, the other ones in English), that is, all videos you have not seen yet:
 - from video “*What is PWM?*”,
 - to video “*What is Pulse Width Modulation? (PWM)*”.
- Study the experimental procedure.
- Modify the Matlab program of Figure 4.4 so that its first section will plot the Bode diagram of
 - the first order system you studied in Chapter 3,
 - the underdamped second order system you studied in Chapter 3.
- Complete the program so that its second section will plot the experimental points of the Bode diagram you will measure in class. Use (4.3)–(4.4).

4.5 In the laboratory

4.5.1 List of materials

- AD laboratory.
- Multimeter.
- Function generator.
- Oscilloscope.
- 10 k Ω potentiometer.
- 4.7 k Ω resistor.
- 100 nF capacitor.
- two 100 mH inductors.
- Arduino Uno.
- Cables, jump wires, BNC tee.

```

%% 1) Plot the expected Bode diagram.
s = tf('s');
% change this line into the transfer function of the first order system you studied
G1 = 1/(s+1);
% change this line into the transfer function of the underdamped second order system you studied
G2 = 1/(s^2+0.5*s+1);
% use G = G1 or G = G2 to use either transfer function in what follows
G = G1;
% 401 logarithmically spaced frequencies from 10^1 rad/s to 10^5 rad/s
w_rad_per_sec = logspace(2, 6, 401);
% frequency response of G
G_freqresp = squeeze(freqresp(G, w_rad_per_sec));
G_gain = 20*log10(abs(G_freqresp));
G_phase = rad2deg(unwrap(angle(G_freqresp)));
figure, subplot(2,1,1), semilogx(w_rad_per_sec, G_gain)
grid on, xlabel('frequency [rad\cdots^{-1}]'), ylabel('gain [dB]')
subplot(2,1,2), semilogx(w_rad_per_sec, G_phase)
grid on, ylabel('phase [^\circ]')

%% 2) Plot the expected Bode diagram plus the experimental data.
w_lab_Hz = [20 50 100 200 500 1000 2000 5000 10000 20000];
w_lab_rad_per_sec = w_lab_Hz * 2*pi;
% fill in this vector in the laboratory with 2a measured in V
measured_2a = [NaN NaN NaN NaN NaN NaN NaN NaN NaN NaN];
% fill in this vector in the laboratory with 2b measured in V
measured_2b = [NaN NaN NaN NaN NaN NaN NaN NaN NaN NaN];
% fill in this vector in the laboratory with 2c measured in V
measured_2c = [NaN NaN NaN NaN NaN NaN NaN NaN NaN NaN];
% put here the correct expression to find the gain, in dB, from 2a, 2b, and 2c
G_gain_measured = [];
% put here the correct expression to find the phase, in degrees, from 2a, 2b, and 2c
G_phase_measured = [];
figure, subplot(2,1,1), semilogx(w_rad_per_sec, G_gain, w_lab_rad_per_sec, G_gain_measured, 'r.')
grid on, xlabel('frequency [rad\cdots^{-1}]'), ylabel('gain [dB]')
subplot(2,1,2), semilogx(w_rad_per_sec, G_phase, w_lab_rad_per_sec, G_phase_measured, 'r.')
grid on, ylabel('phase [^\circ]')

```

Figure 4.4: Matlab code to be used in the laboratory to plot frequency responses.

4.5.2 Experimental procedures

1. Build again the first order system you studied in Chapter 3, using a potentiometer and the capacitor.
2. Measure its frequency response, for the frequencies listed in the Matlab program of Figure 4.4. Use any input amplitude you see fit, and set the oscilloscope to visualise the Lissajous curve as shown in Figure 4.1. Remember that you can change the scales of the axis, that you can shift the ellipse along both axis with the position buttons, and that values of $2a$, $2b$ and $2c$ all have Volt units. For high frequencies, you will surely have to change the scale of the B channel (or Y channel, depending on the model of the oscilloscope). Register the values in the program of Figure 4.4.
3. Run the program and compare the expected Bode diagram with the experimental points. Pay particular attention to the phase. Do you need to change your code to get correct values? Call the instructor and discuss the matter.
4. Make a copy of the file from Figure 4.4, and repeat the points above for the underdamped second order system you studied in Chapter 3.
5. This time, are the phase values shown by the Matlab program correct when the frequency is above the system's natural frequency? Do you need to change your code to get correct values? Call the instructor and discuss the matter.
6. You will now use a first order system to filter a PWM output of the Arduino. Build the first order system with the $4.7\text{ k}\Omega$ resistor and the potentiometer in series. Use the Arduino's digital output 5 as input (i.e. as V_i in Figure 3.1), and connect the output (i.e. V_o in Figure 3.1) to the oscilloscope. Run the following commands in Matlab:

```
ard = arduino('COM3', 'UNO') % replace 3 with the correct port number
writePWMDutyCycle(ard, 'D5', 0.5); % 0.5 is a 50% duty cycle
```

7. Change the duty cycle so that the output will be 2 V. Adjust the potentiometer to see in the oscilloscope a voltage as constant as possible. Call the instructor and discuss the effect of changing the value of the resistance.
8. Remove the $4.7\text{ k}\Omega$ resistor from the circuit, close Matlab, and open Arduino IDE.
9. On menu Tools, submenu Port, choose the correct port number for the Arduino; then, on submenu Board, choose the Arduino One board (see Figure 4.5).
10. Copy the code of Figure 4.6, that varies the duty cycle to emulate a sinusoid, to Arduino IDE (you may copy it from its original website, and then change the output pin in line 3). Click the \checkmark button to compile the code, and the \rightarrow button to download it to the Arduino.
11. Adjust the potentiometer to see in the oscilloscope a sinusoid. Measure on the oscilloscope screen the frequency of the sinusoid, and the frequency of the PWM square wave. Call the instructor, show how you measured the two frequencies, and discuss the effect of changing the value of the resistance.

4.5.3 Troubleshooting

If the code in Figure 4.6 does not compile, and the error is that file TimerOne.h does not exist, check if the library is installed, clicking the Library button on the left (as seen in Figure 4.7) and then using the first menu to see the list of installed libraries. If TimerOne is missing, download it in a .zip file from here, and then go to menu Tools and choose Manage Libraries... to install the .zip file you downloaded.

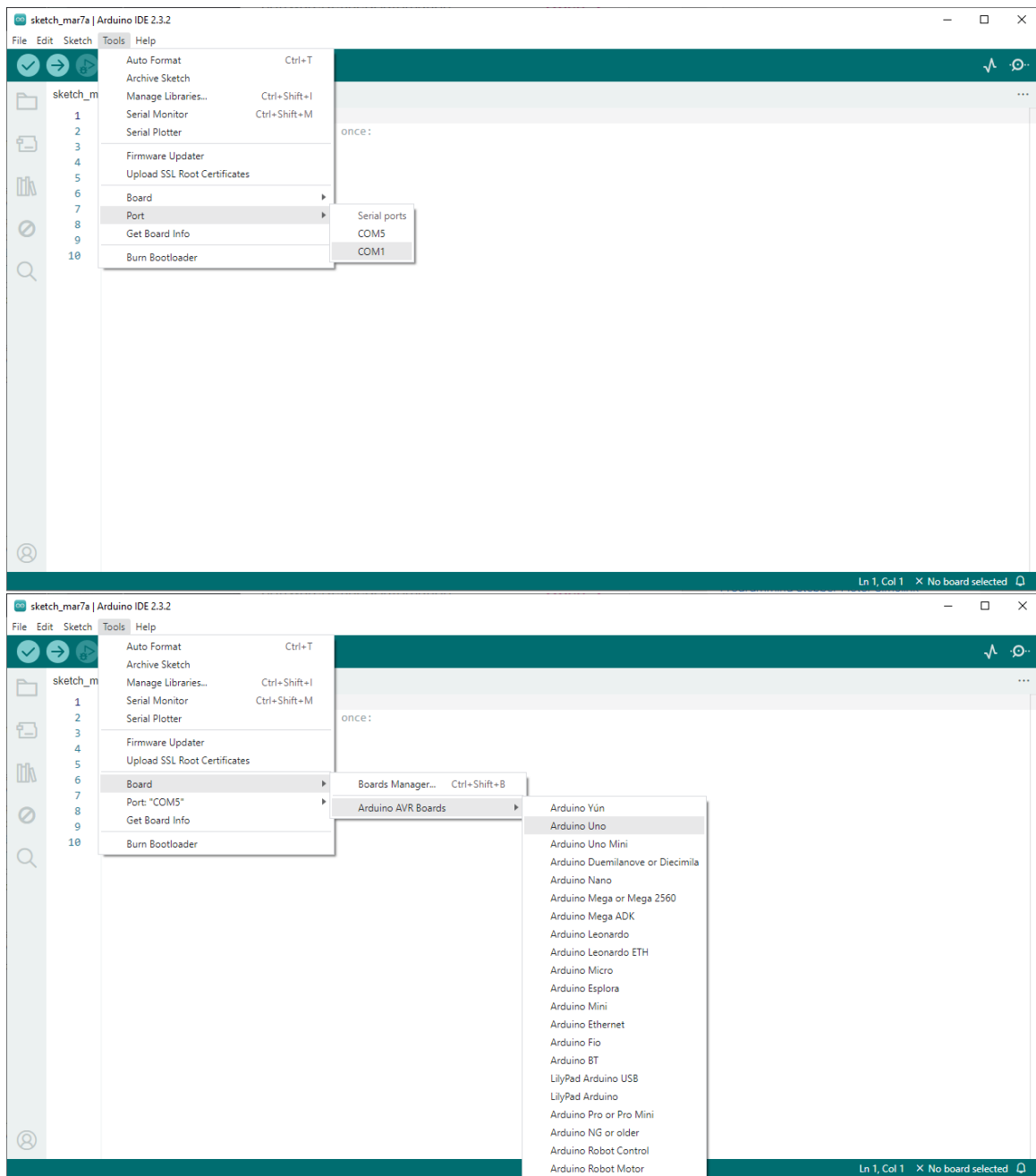


Figure 4.5: Setting the correct COM port (top) and the correct Arduino board (bottom) on Arduino IDE.

```

#include <TimerOne.h>

#define outPin 5
#define delayPin 11 // this pin is just used to test the compute time

const float pi = 3.14159 ;
const float T = 100 ; // sample time in microseconds
const float freq = 1000 ; // frequency of tone in hertz
const float omega = 2*pi*freq ;
const float A = 490 ; // amplitude
// next line initializes oscillation with amplitude A
float a[]={0.0, A*sin(omega*T/1000000.0),0.0};
// c1 is the difference equation coefficient
const float c1 = (8.0 - 2.0*pow(omega*T/1000000.0,2))/(4.0+pow(omega*T/1000000.0,2));

void setup () {
  Timer1.initialize(T); // set sample time for discrete tone signal
  Timer1.pwm(outPin, 0,T); //generates pwm signal on outPin(9), with 0 duty cycle
  //and sample period of T=100us
  Timer1.attachInterrupt(compute); //attachInterrupt(function, period) calls
  //function with optional period
  pinMode(delayPin, OUTPUT);
}

void loop() {

}

void compute()
{
  digitalWrite(delayPin, HIGH);
  a[2] = c1*a[1] - a[0] ; // recursion equation
  a[0] = a[1] ;
  a[1] = a[2] ;
  Timer1.setPwmDuty(outPin, int(a[2])+512);
  digitalWrite(delayPin, LOW);
}

```

Figure 4.6: Code, written in a variant of the C++ programming language, for the Arduino to be used in the laboratory to emulate a sinusoid with PWM. This code was taken from this website, changing the output pin in line 3 to digital output 5, and uses the TimerOne library, which should be already installed in Arduino IDE.

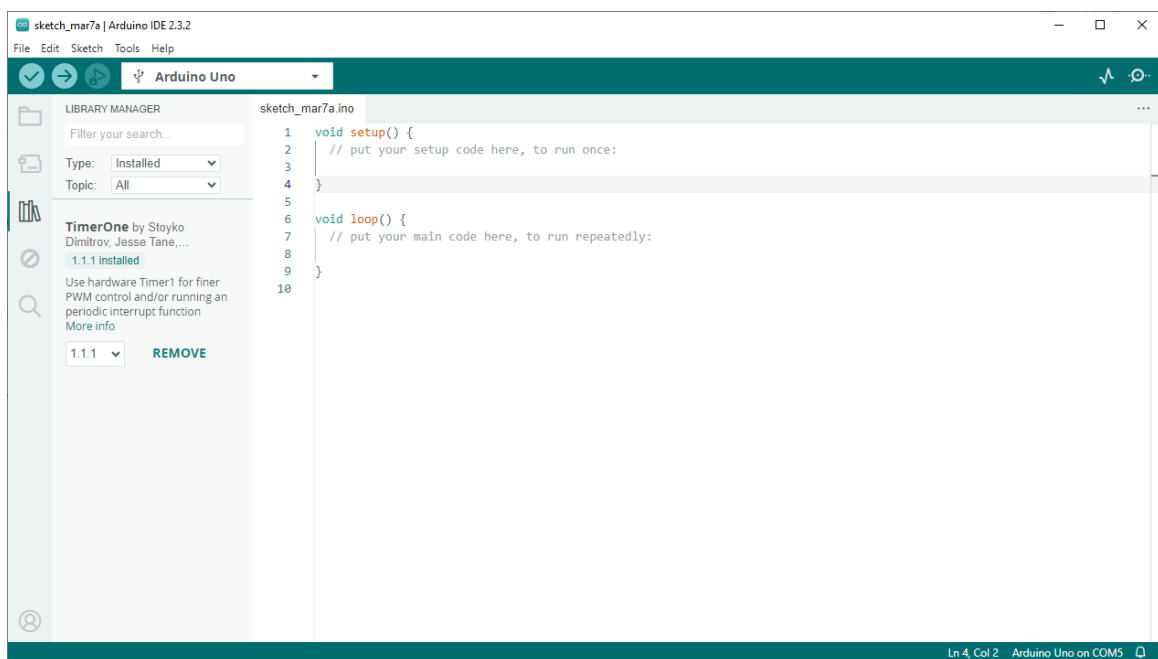


Figure 4.7: Checking if the TimerOne library is installed in Arduino IDE.

Chapter 5

Signal filtering

5.1 Objectives

This chapter has two objectives

- to make you understand what a filter does and what it can be used for;
- to give you further acquaintance with simple op-amp circuits.

5.2 Previous study

To finish this assignment, you must first study filters:

- in the Lecture Notes of Signals, Systems and Control, sections 12.2 and 12.3;
- in playlist Sinais e Sistemas Mecatrônicos, parte 3: cadeia de medida e cadeia de atuação, videos SSM T13 03 to SSM T13 06.

5.3 Introduction to the assignment

Figures 5.1 and 5.2 show how first-order low-pass and high-pass filters can be implemented. In each case, there is a passive implementation (i.e. using only passive components, viz. a resistor and a capacitor), and an active implementation (i.e. with an active component, viz. an op-amp).

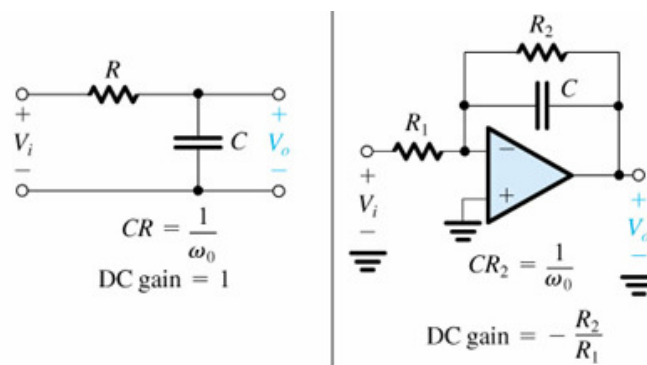


Figure 5.1: First-order low pass filters; left: passive filters; right: active filters.

We will use these filters to separate the components of a signal: we sum two sinusoids, and build a filter to eliminate one of them and pass the other. While this is just an exercise, in practice separating a signal's components is important to eliminate noise from experimental measurements.

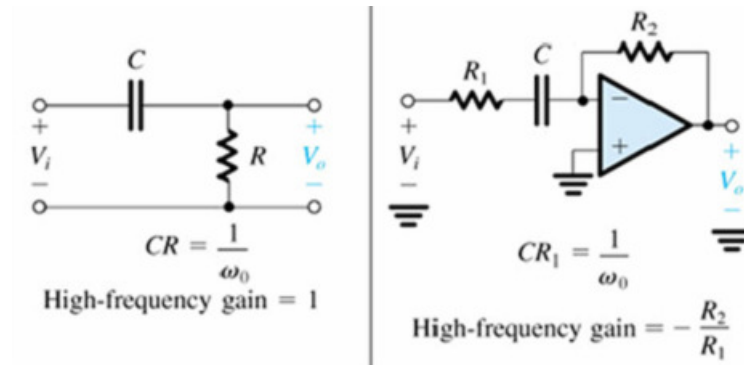


Figure 5.2: First-order high pass filters; left: passive filters; right: active filters.

5.4 Homework before class

- Study the *Introduction to the assignment* above.
- Study the experimental procedure.
- Demonstrate, in your hand-writing, the input-output relation of each of the four filters (two of them passive, two active) that you will build first using Tinkercad, and then in the laboratory using actual components.
- Reproduce in Tinkercad two of the experiments you will carry out in the laboratory, with the following modifications:
 - use a function generator to generate a 5 kHz, $1 + \frac{n_2}{10}$ V pp sinusoid,
 - use another function generator to generate a $20 + 10n_5$ Hz, $6 + \frac{n_3}{10}$ V pp sinusoid,
 - sum them with a summing ampop circuit, using $2 + n_4$ k Ω resistors,
 - build both filters with a resistor rather than a potentiometer, since in Tinkercad the value of the resistance can be easily changed (which of course is not the case in real life),
 - the filters from Figures 5.1 and 5.2 you will implement are
 - * the passive low pass filter, and the active high pass filter, if n_1 is odd,
 - * the passive high pass filter, and the active low pass filter, if n_1 is even.

For the Tinkercad simulation, you can tune the resistance by trial and error, or analytically. Be advised that in the laboratory you will have to calculate and justify the filter's cut-off frequency.

5.5 In the laboratory

Show the instructor the handwritten demonstrations and your Tinkercad file.

5.5.1 List of materials

- AD laboratory.
- Multimeter.
- Function generator.
- Oscilloscope.
- Two LM741 op-amps.
- Two 10 k Ω potentiometers.
- One 100 nF capacitor.

- Three resistors with the same resistance.
- Cables, jumper wires.

5.5.2 Experimental procedure

1. Use the function generator to generate a 5 kHz, 1 V pp sinusoid.
2. Use the AD laboratory's function generator to generate a 20 Hz, 5 V pp sinusoid. Use the multimeter as frequency meter to check the frequency.
3. Build the summing op-amp circuit in figure 2.3, using the three resistors, and use it to sum the two sinusoids of the two function generators. Do not forget to supply the op-amp with -15 V and $+15\text{ V}$ from the AD laboratory's DC power supply.
4. Connect the summing op-amp circuit's output to channel X of the oscilloscope. Adjust the scales so you can see one period of the lower frequency sinusoid.
5. Build the passive low-pass filter in figure 5.1 using the capacitor and the potentiometer. Connect the summing op-amp circuit's output to its input, and its output to channel Y of the oscilloscope. Vary the resistance of the potentiometer to eliminate the high frequency oscillation, without altering the low frequency oscillation.
6. Measure the resistance of the potentiometer and calculate the filter cut-off frequency.
7. Call your instructor and show your completed assignment working, including the calculations of the cut-off frequency. Explain why the filter is, or is not, doing its job properly, and in which circumstances it would work even better.
8. Switch the potentiometer with the capacitor, thus obtaining the passive high-pass filter from figure 5.2. Vary the resistance of the potentiometer to eliminate the low frequency oscillation, without altering the high frequency oscillation.
9. Measure the resistance of the potentiometer and calculate the filter cut-off frequency.
10. Call your instructor and show your completed assignment working, including the calculations of the cut-off frequency. Explain why the filter is, or is not, doing its job properly, and in which circumstances it would work even better.
11. Repeat points 5 to 7 above with an active filter. (You will now need two potentiometers.) Call the instructor, show your completed assignment working, and explain the differences you see in the result.
12. Repeat points 8 to 10 above with an active filter. Call the instructor, show your completed assignment working, and explain the differences you see in the result, in particular when one of the summed sinusoids has an amplitude larger than 15 V.

Chapter 6

Sensors and actuators in a control loop

6.1 Objectives

The objective of this class is to show you how measurement, actuation and signal processing components are used in a control system. In this way you are expected to realise how the things you have been studying in classes and doing in the laboratory can be used for in real life.

6.2 Previous study

To finish this assignment, you must first study signal conditioning:

- in the Lecture Notes of Signals, Systems and Control, chapter 12;
- in playlist Sinais e Sistemas Mecatrónicos, parte 2: sistemas dinâmicos, videos SSM T13 01 to SSM T13 06.

You can learn more about the potentiometric sensor and the DC motor you will use:

- in the Lecture Notes of Signals, Systems and Control, sections 13.2 and 14.2;
- in playlist Sinais e Sistemas Mecatrónicos, parte 3: cadeia de medida e cadeia de atuação, in the videos of classes 14 and 19.

6.3 Introduction to the assignment

We will build a closed-loop control system for a DC motor as follows:

- Controlled variable: angular position of a shaft
- Sensor: potentiometer to measure the angular position of the shaft, providing a signal in the -5 V to $+5$ V range
- Analog-digital converter: National Instruments USB-6008 AD/DA converter, reading signals in the -10 V to $+10$ V range
- Controller: Computer running Matlab and Simulink, implementing a proportional controller (control action directly proportional to the closed-loop control error)
- Digital-analog converter: National Instruments USB-6008 AD/DA converter (again), outputting signals in the 0 V to $+5$ V range
- Actuator: DC motor, receiving a signal in the -10 V to $+10$ V range (actually it could receive up to ± 12 V, but we will not push it to the limit)

It is clear from this description that you will have to build a signal conditioning system, converting the AD/DA converter's 0 V to +5 V output into a signal in the -10 V to $+10\text{ V}$ range. If not, the shaft would rotate in one direction, but not on the other, and it would not reach its maximum rotation speed.

We will connect the sensor directly to one of the AD/DA converter's analog inputs. We will lose resolution, of course, but the resolution we have is good enough already.

You may find a block diagram of this control loop in Figure 6.1.

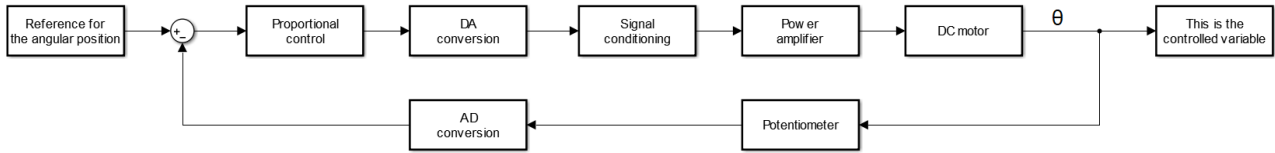


Figure 6.1: Block diagram of the control loop.

The motor is part of transducer and instrumentation trainer DigiAC 1750 (see Figure 6.2).

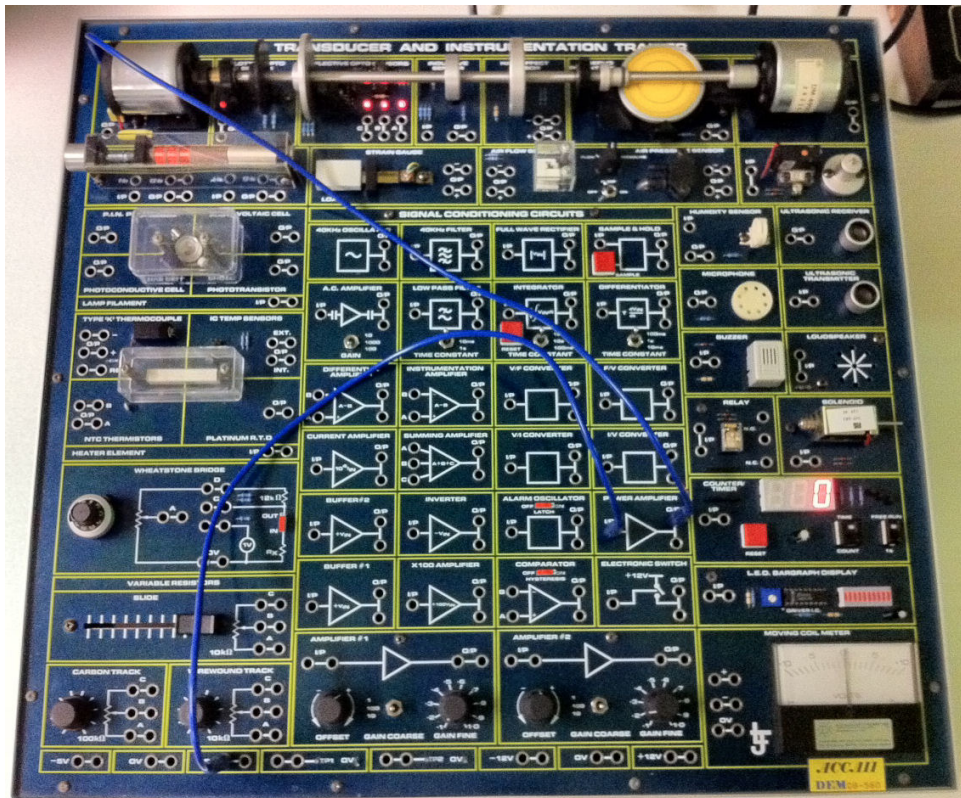


Figure 6.2: The DigiAC 1750.

6.4 Homework before class

To prepare this class:

- Study the *Introduction to the assignment* above.
- Study the experimental procedure.
- Project the signal conditioning system you will implement for the actuation, and draw the corresponding circuit diagram. Notice that:
 - you will need the AD laboratory's variable DC power supply to supply the op-amps with $\pm 15\text{ V}$;

- the op-amps will, consequently, saturate at those values of electrical potential;
 - you also have available ± 5 V at the DC power supply;
 - when using Tinkercad you can easily have any constant voltage you want simply using a power supply, but things are *not* so in the laboratory.
- Answer the following questions, in your handwriting:
 - Knowing that the AD/DA converter uses 12 bits for the output, find the resolution of the actuation, in Volt per bit, including your signal conditioning.
 - Find the resolution of the reading, in degrees per bit.
 - Suppose that, instead of connecting the potentiometer directly to one of the AD/DA converter's analog inputs, we would bother to pass it through a non-inverting op-amp circuit with two equal resistors. Find how much the resolution of the reading, in degrees per bit, would improve.
 - Find in the experimental procedure below the signal that we will use as a reference for the control loop. Knowing that the -5 V to $+5$ V signal of the potentiometer corresponds to an entire rotation of the shaft, find the amplitude of the rotation (in degrees) that the reference will correspond to.
 - Reproduce in Tinkercad the signal conditioning you will implement in the laboratory. Follow these steps:
 - Use as input a sinusoid assuming values between 0 V and $+5$ V, with an arbitrary frequency (say, 50 Hz). This sinusoid will play the role of the AD/DA converter's output (i.e. input of the signal conditioning).
 - Verify with an oscilloscope that the output of the signal conditioning assumes values in the -10 V to $+10$ V range. (If it does not, something is wrong.)
 - Do not use power supplies with voltages that you will not have available in the laboratory.

6.5 In the laboratory

Show the instructor

- the project of the signal conditioning,
- its circuit diagram,
- its Tinkercad implementation, and
- your handwritten answers to the questions in the homework list.

6.5.1 List of materials

- AD laboratory.
- DigiAC 1750.
- Multimeter.
- Function generator.
- Oscilloscope.
- Cables, jumper wires, BNC tee.
- Components for your signal conditioning system.

- see if the name of the converter is Dev1;
- if not, delete the converted with the name Dev1, rename the converter which is connected as Dev1, and click Save.

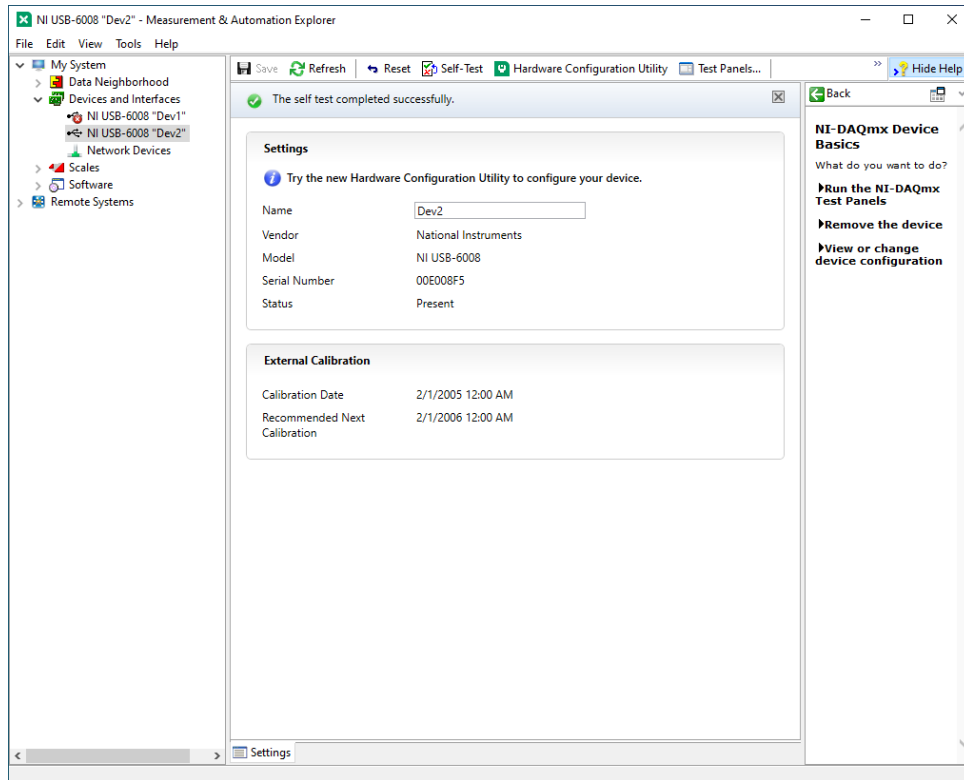


Figure 6.4: The NI MAX software, after selecting an AD/DA converter and running a self test. Notice that its name is Dev2. Thus, it is necessary to delete Dev1, marked in red because it is not connected, rename Dev2 as Dev1, and then click Save.

Glossary

“You just come along with me and have a good time. The Galaxy’s a fun place. You’ll need to have this fish in your ear.”

“I beg your pardon?” asked Arthur, rather politely he thought.

Ford was holding up a small glass jar which quite clearly had a small yellow fish wriggling around in it. (...)

“Listen!”

“But I can’t speak Vogon!”

“You don’t need to. Just put that fish in your ear.”

Ford, with a lightning movement, clapped his hand to Arthur’s ear, and he had the sudden sickening sensation of the fish slithering deep into his aural tract. Gasping with horror he scrabbled at his ear for a second or so, but then slowly turned goggle-eyed with wonder. He was experiencing the aural equivalent of looking at a picture of two black silhouetted faces and suddenly seeing it as a picture of a white candlestick. Or of looking at a lot of coloured dots on a piece of paper which suddenly resolve themselves into the figure six and mean that your optician is going to charge you a lot of money for a new pair of glasses.

He was still listening to the howling gargles, he knew that, only now it had taken on the semblance of perfectly straightforward English.

Douglas ADAMS (1952 — †2001), *The Hitchhiker’s Guide to the Galaxy*, 5

actuator	atuador	frequency	frequência
ammeter	amperímetro	frequency meter	freqüencímetro
amplitude	amplitude	function generator	gerador de sinais
analog-digital converter	conversor analógico-digital	ground	terra
banana	banana	high-pass	passa-alto
BNC tee	T BNC	inverting op-amp circuit	circuito inversor
breadboard	placa de ensaio, <i>breadboard</i>	jump wire, jumper	fio
cable	cabó	Lissajous curve	figura de Lissajous
capacitance meter	capacímetro	low-pass	passa-baixo
capacitor	condensador	mean value	valor médio
closed-loop	anel fechado, malha fechada	multimeter	multímetro
comparator op-amp circuit	circuito comparador	non-inverting op-amp circuit	circuito não-inversor
controlled variable	variável controlada	operational amplifier (op-amp)	amplificador operacional (ampop)
controller	controlador	oscilloscope	osciloscópio
current	corrente	overdamped	sobreamortecido
DC motor	motor de corrente contínua	peak amplitude	amplitude
DC power supply	fonte de tensão contínua	peak-to-peak amplitude	amplitude pico a pico
differential op-amp circuit	circuito subtrator	pliers	alicate
digital-analog converter	conversor digital-analógico	potentiometer	resistência variável, potenciômetro, reóstato
duty cycle	razão cíclica	power amplifier	amplificador de potência
electrical potential	potencial elétrico, tensão	pulse width modulation (PWM)	modulação por largura de pulso
filter	filtro		
fine tuning	ajuste fino		
first order	primeira ordem		

resistor	resistência	tuning	ajuste
rheostat	resistência variável, potenciômetro, reóstato	triangular wave	onda triangular
sampling time	tempo de amostragem	underdamped	subamortecido
second order	segunda ordem	variable resistor	resistência variável, potenciômetro, reóstato
sensor	sensor	voltmeter	voltímetro
sinusoidal wave	onda sinusoidal	wire stripper	descarnador
square wave	onda quadrada		
subtractor op-amp circuit	circuito subtrator		