M P 1 – Semester 2
2016-2017

L A B O R A T O R Y
P R A C T I C A L S

Analog Electronics

B. VELAY
JC. OLIVIER
G. WASSELYNCK
E. DAVID

PRESENTATION

Observations and Analysis

The following experiments will be carried out during the different practicals.

You will work in pairs. Each pair has to write a report and hand it in at the end of each practical.

Your report will contain:

- The schematics

- The measurements and recordings

- Their analysis

- Your comments and remarks.

Thank you for being clear and concise.

Practicals

Here are the different experimental subjects:

1. OPERATIONAL AMPLIFIER. Real behaviour and limitations of the ideal model
2. FOURIER SERIES EXPANSION. Initiation via simulation

Assessment

The final mark is attributed as follows:

- The work done during the different practicals accounts for 30% of the final mark

- The final exam accounts for 70% of the final mark

# <u>Practical #1:</u> OPERATIONAL AMPLIFIER (Op Amp)
Real behaviour and limitations of the ideal model

AIMS AND OBJECTIVES:

- 📖 Read a genuine data sheet; find relevant data.
- 📖 Perform schematic's wiring, adjustments, recordings and measurements that characterize linear or non-linear behaviour.

APPLIANCES & DEVICES TO BE USED:

- Usual Op Amp with its connecting box, including a symmetric voltage supply +15V / 0 / -15V
- Technical Data on TL081 Op Amp
- Digital Scope
- Multimeter
- Low Frequency Function Generator LFG
- Resistors : 470Ω, 10kΩ, 10kΩ, 100kΩ
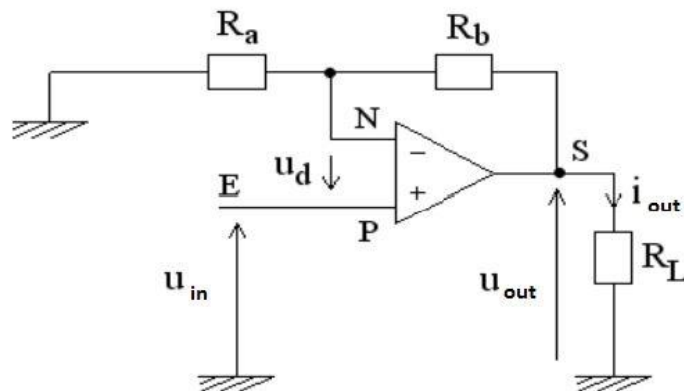
0- BEFORE THE PRACTICAL

Q0.a *Prove that the transfer function of a non-inverting amplifier is* :

$$T = \frac{U_{out}}{U_{in}} = 1 + \frac{R_b}{R_a}$$

Q0.b *Calculate your predictive value of the voltage gain* $T_{th}= G$ *for :*
$R_a = R_L = 10\ k\Omega$ *and* $R_b = 100\ k\Omega$

⇨ Don't forget to measure these values before, using your ohmmeter!

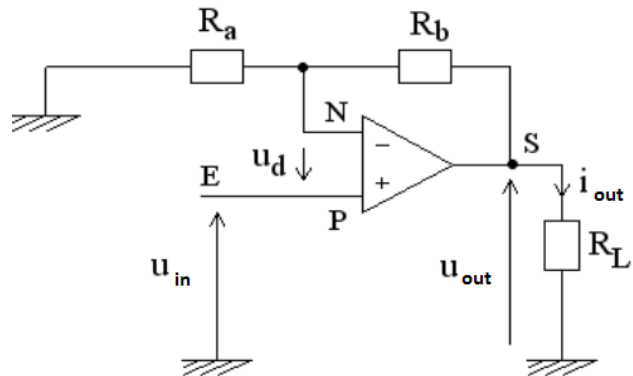*PS: In the general case of a complex function, G is then the modulus of T :* $\left|\underline{T}\right| = G$

NON-INVERTING AMPLIFIER

$R_a = R_L = 10\ k\Omega$

$R_b = 100\ k\Omega$

Voltage $U_e(t)$ will be produced by the LF Function Generator with the following characteristics:

- ✖ Sine voltage,
- ✖ Peak-to-peak value $U_{in\ pp} = 1V$
- ✖ Frequency f = 1kHz,
- ✖ Offset = 0V.

1- Gain measurement

Switch on the Op Amp's DC supply.

⇨ Have a look at the back of the Op Amp's black box!

Q1.a Wire that circuit. Then check precisely if the output signal shows a linear behaviour. Explain in details what criteria(s) you are checking.

*Q1.b Use AUTO scope measurement to get these values **for both U_{in} and $U_{out.}$**:*

- *frequency,*
- *average voltage (« DC component »),*
- *peak-to-peak value*

Q1.c Calculate the experimental voltage gain. Compare to your Q0.b predictive value.

2- Cut-Off frequency fc measurement

Estimated Cut-Off frequency :

Q2.a In the datasheets find the typical value of GBP Gain-bandwidth-Product f_T (= Unity Gain Bandwidth).

⇨ It is the Cut-off frequency for the Op Amp used as a voltage follower with G = 1.

For your own value of G, calculate $f_c \approx \dfrac{f_T \times 1}{A}$

The open loop frequency response (schematic #12) shows that the real Op Amp acts as a **1st order Low Passfilter.**

Q2.b What is the formula of the transfer function of a 1st order low-pass filter? Calculate the special value of its modulus (its « gain ») when $\omega = \omega_c$

Cut-Off frequency measurement :

Q2.c   *Explain in details how the Cut-Off frequency of such an amplifier can be measured.*

Q2.d   *Set the values $U_{in\ pp}$ = 2V and Offset = 0V, then measure the Cut-Off frequency f_c (at - 3 dB).*

Q2.e   *Calculate the GBP A $\times f_c$ and compare it to your predictive value obtained in Q2.a .*
      Comment.

3- Rise time Measurement

Estimated rise time:

⇨ When we consider a step function response, the **rise time** is the time required for the output voltage of the Op Amp to increase from 10% to 90% of its steady state value.

Q3.a Find the typical value of the rise time in this Op Amp's datasheet (p 9).
Note that you'll find a specific value when G =1. For any other value of G gain, you have to multiply this reading value by G. Why is it necessary?

Risetime measurement :

⇨ **How To Do with a digital scope? Use <u>Auto measurement</u> or <u>the cursors mode</u>:**
    - Set a square wave with such a voltage that it can spread properly on all the screen
    - Use the menu MEASURE and choose « risetime »…
    - You also can use the cursors mode

Voltage Ue(t) will be produced by the LF Function Generator with the following characteristics :
    ✖ Sine voltage,
    ✖ Peak-to-peak value $U_{in\ pp}$ = 1V

Q3.b Adjust properly the frequency of the input voltage, in order to show a typical exponential shape response.
Measure the rise time.
Analyse and comment (don't forget to justify your own choice of frequency).

NON-LINEAR BEHAVIOR OF THE Op Amp

4- Voltage restriction: the output voltage should NOT saturate

Q4.a *In the datasheets of this Op Amp, find the typical value of the output voltage swing when the signal is saturating (schematic #10) ; read for supply voltage of ± 15 V.*

Q4.b *For a sine wave U_{in} with f = 1 kHz and Offset = 0V, look for the setting of the input signal which induces the output voltage to be right at the saturation limit.*

Then measure the peak-to-peak value of the output voltage and both values of the saturation voltage called +Vsat et -Vsat

⇨ Tip: it might be a bit dissymmetric.

Analyse and comment.

5- Current restriction: if the output current saturates, the output voltage will distort

Q5.a *Still in the datasheets, find the typical figure showing the output voltage swing versus load resistance (schematic #9 p.13).*

According to the schematic, what should happen for a load resistor of 470 Ω?

Calculate the maximum value of the output current (DC case).

Q5.b *Use a resistor of 470 Ω as a load for the Op Amp.*

Measure the output voltage, then estimate the maximum value of intensity for the output current.

Analyse and compare.

Practical #2: FOURIER SERIES EXPANSION
Initiation via simulation

Use of PSPICE, which is the main electronics simulation software, to study the Fourier Series Expansion of periodical signals, for example filtered by a RC 1st order Low-Pass Filter.

Aims and Objectives :

- Discovering the spectral representation of a signal
- Understand how to use simulation and performs some.

To be used :

- PSpice Microsim 8.0 for Windows (student version)
- « Operating instructions manual »

0- BEFORE THE PRACTICAL:

It is desired to produce a periodic square signal e(t) with an amplitude E and a frequency f = 1 kHz from its Fourier series expansion.

⇨ Remember:

The **Fourier series expansion** makes it possible to decompose any periodic signal into a sum of sine and cosine signals (under certain conditions generally verified by the electrical signals)
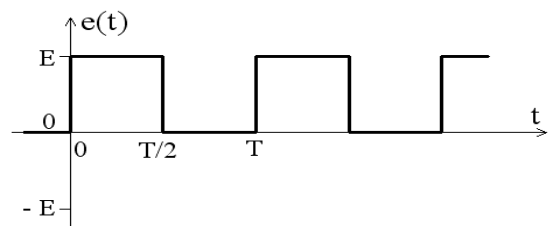
with $\omega = 2\pi f = \dfrac{2\pi}{T}$

$$e(t) = A_0 + \sum_{n=1}^{+\infty} \left(A_n \cos(n\,\omega t) + B_n \sin(n\,\omega t) \right)$$

Calculation of the terms of the Fourier series expansion

The square signal e(t) can be expanded as:

$$e(t) = \frac{E}{2} + \sum_{n\,odd} B_n \; \sin(n\,\omega tn) = \frac{E}{2} + \sum_{n\,odd} \frac{2E}{n\,\pi} \; \sin(n\,\omega tn)$$

Q0.1 For E = 1 Volt, calculate the first 5 coefficients B_n corresponding to the odd-numbered harmonics (n = 1, 3, 5, 7, 9).

Q0.2 What is the expression of the transfer function modulus of a 1st order RC low-pass filter?

1- SYNTHESIS OF A PERIODIC SQUARE SIGNAL WITH THE PSPICE SOFTWARE

You should find tips in the « Mode d'emploi pratique : Utiliser et exploiter PSpice »

Use the components of the libraries and draw this simulation diagram.
It permits to visualize the result of the synthesis of e(t) from the harmonics B1 and B3 (calculated in the O Homework part).
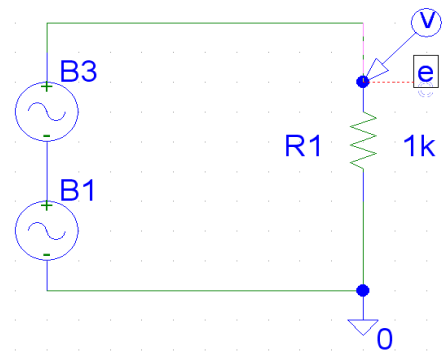
Use two VSIN sources, one R resistor, one GND-ANALOG analog ground and one MARKERS / MARK VOLTAGE voltage probe. Generate the DC component A0 as an DC option of B1.

Q1.1 Perform a simulation in TRANSIENT mode; represent the synthesis signal in PROBE. Then comment upon the curve you have obtained.
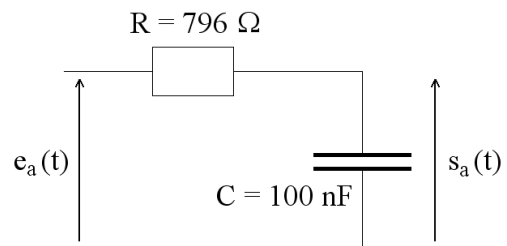
Q1.2 Modify the previous diagram, in order to visualize the result of the synthesis of the signal e(t) from the first 5 harmonics calculated in O part.
Perform the simulation and compare to the result of question Q1.1.

2- LOW-PASS FILTERING OF A PERIODICAL signal

2.1- Calculation of s_a(t) from the transfer function

$$e(t) = A_0 + B_1 \sin(\omega t) + B_3 \sin(3\omega t)$$
$$+ B_5 \sin(5\omega t) + \cdots$$
$$\dots B_7 \sin(7\omega t) + B_9 \sin(9\omega t)$$

R = 796 Ω

$e_a(t)$ C = 100 nF $s_a(t)$

The signal e(t) is approximated by the signal e_a(t) *("a" stands for "approximation")*. It is the result of the synthesis made with the first 5 non-zero harmonics of e (t).

This reconstituted signal e_a(t) is then filtered by a simple RC low-passfilter.
The goal is to determine the response s_a(t) of the filter, then to compare it to the simulation result s(t) directly performed by Pspice on the circuit.

Q2.1a   Calculate the transfer function V : $\underline{T} = \dfrac{u_{out}}{u_{in}}$

Q2.1b   Show that: $T = |\underline{T}| = \dfrac{1}{\sqrt{1 + 2.5\ 10^{-7} f^2}}$ and

$$\varphi_{out/in} = arg(\underline{T}) = -arctan(5\ 10^{-4}\ f)$$

Q2.1c   *Complete the table below for harmonics 1 to 9:*

Harmonic of rank n	DC or « n = 0 »	1	3	5	7	9
Frequency (Hz) f_n	0	1000	3000	5000	7000	9000
Amplitude B_n de e_a (t)						
Gain at f_n : T_n						
Phase shift at f_n : $\varphi_n = \varphi_{s/e}(f_n)$						
Amplitude S_n de s_a (t)						

Derive the expression of $s_a(t)$ using the form:

$$s_{a\,calculate}(t) = \ S_0 \ + S_1 \sin(\omega t + \varphi_1) + S_3 \sin(3\omega t + \varphi_3) + \cdots$$

$$\dots + S_5 \sin(5\omega t + \varphi_5) + \ S_7 \sin(7\omega t + \varphi_7) + S_9 \sin(9\omega t + \varphi_9)$$

2.2- s_a(t) Signal Spectrum

The PROBE software makes it possible to draw curves and to visualize the spectrum of a signal *(see the « operating instructions manual »).*

Add capacitor C to your simulation diagram; perform the simulation with C = 100nF.

Q2.2a   *Trace the signal $s_a(t)$ graph, then do so for the spectrum of the signal s_a on another graph.*

Q2.2b   *Analyse the spectrum of the signal s_a.*
Compare the amplitude of the lines with the expression of $s_{a\,calculé}(t)$.
What is the advantage of knowing the spectrum of a signal?

⇨ *Remember:*

1-   When you want to perform a spectrum in Probe, you must set the time to **a multiple of the time period**.
   If you don't set it this way, the spectrum will be wrongly calculated and the shape will be distorted (that's maths, folks! ☺)

2-   PROBE normally uses **a triangular shape** for each line of its spectrum curve. Obviously, it's easy to see if 1- is not properly set!

3-   Read the frequency and the amplitude of a harmonic <u>at the top of the triangle</u>.

⇨ *How to do: Getting a spectrum with* **PROBE**

- The menu « Plot/Add Plot » permits to add other graphs to the same page (two is a good choice, three or more gives graphs which are too small to read).
- The menu « Plot/Unsync Plot » allows you to "unsynchronise" the x-axis : this way, you can get two axes with different scales of the same variable, **t** for example. Or you can get one time axis and another frequency axis.
- The menu « Plot/X Axis Setting » allows different settings. You can also obtain easily a spectrum of the f(t) spectrum: tick « Fourier » box (on the right). Remember point 1-above…

See « operating instructions manual » for more.

3- LOW-PASS FILTERING OF A NOISY PERIODICAL SIGNAL

3.1- Description of the noisy signal

The noisy signal will be built with:

- a square signal e(t) with amplitude E = 1 V and frequency f = 1kHz as the useful signal,
- a sine signal n(t) with amplitude N = 0,2 V and frequency f_n = 1 MHz as the "non-useful" part of the signal.

The square signal is made with a source VPULSE with these settings:
V1 = 0   V2 = 1   TD = 1ns   TR = 1ns   TF = 1ns   PW = 0.5ms   PER = 1ms

with:

V1: Low Level	TF: Fall time
V2: High Level	PW: Pulse Width
TD: Time Delay	PER: Period
TR: Rise time	

Q3.1 *Adapt your schematic: add a VSIN source to describe the noise n(t), then to set that source.*
Perform a simulation in Transient mode.
Trace the noisy signal. Do it also for its spectrum.
Analyse and comment these graphs.

3.2- Filtering that noise

Use a simple RC Low-Pass filter with R = 1 kΩ.

Q3.2 *Find a value of the capacitance C which allows the filter to attenuate the noise by a factor of 10, while maintaining the amplitude of the first 5 harmonics of the signal e(t).*
Add trace for the filtered signal. Do the same for its spectrum.
Compare the filtered signal and its spectrum to the noisy signal.
Comment upon these curves.

<u>Practical #3:</u> DIODES &VOLTAGE REGULATOR
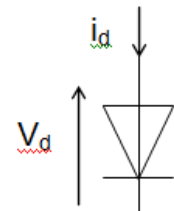
AIMS AND OBJECTIVES :

    ☞   Trace a characteristic diagram of Si diode

    ☞   Understand what is a threshold voltage and a wave rectification

    ☞   Choose a filter capacitor to minimize the peak-to-peak ripple voltage

1- CHARACTERISTIC DIAGRAM OF A DIODE

1.1- What is the proper measurement circuit?

We want to trace the characteristic diagram of a diode, $i_d = f(V_d)$:
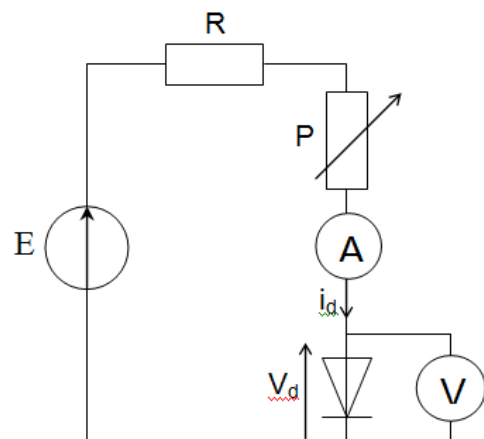
Q1.1a Give two schematics which might allow you to measure enough values

    of i_d and V_d to trace the characteristic curve, using :

–   *One voltmeter and one ammeter,*

–   *A variable resistor,*

–   *A DC voltage source E= 5V.*

Q1.1b Which of these two schematics should give the best results? Explain that point precisely.

1.2- Performing measurements

E = 5V
R = 100Ω
P: set of resistor decade box

WARNING: use **only** the x100Ω, x10Ω, x1Ω

      setting

Q1.2a   Get correct measurements to trace *id = f(Vd)   with 50µA < i_d< 40mA.*

    You may complete the OpenOffice Calc file which is provided.

Q1.2b   *Print this characteristic diagram.*

Find on that print what could be « the » threshold voltage of this diode (is it a precise value?).
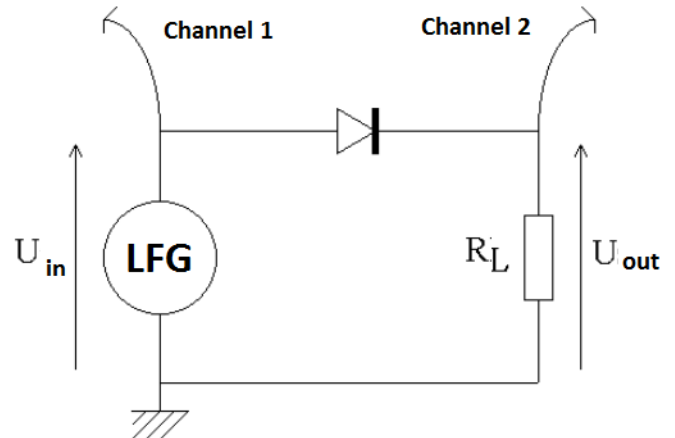
Analyse your choice, using the technical data of this diode.

2- HALF-WAVE RECTIFICATION

2.1- Rectification : threshold level

The input voltage U_{in} will have these settings:

✗   Sine wave,

✗   Peak-to-peak value U_{epp}= 4V,

✗   Frequency   f = 100Hz,

✗   Offset = 0V.

⇨ *Ask for a special wire including a fuse (**for LFG protection**).*

Low level measurement ($U_{in\ pp}$ = 4V)

Q2.1a   Wire this schematic, then use a scope *(= oscilloscope ☺)* to study *Ue and Us.*
Explain precisely the shape of the output voltage Us.

Q2.1b   *Measure with your scope the Peak-to-Peak value, the average voltage and the RMS voltage, both for input and output signals. Compare them to the theoretical values (you should look at your course).*

High level measurement ($U_{in\ pp}$ = 20 V)

Q2.1c   *Set 20 V, then measure with your scope the Peak-to-Peak value, the average voltage and the RMS voltage, both for input and output signals. Compare them to the theoretical values.*

Q2.1d   *Which condition makes it realistic to neglect the threshold voltage of the diode?*

2.2- Voltage filtering by capacitor

The input voltage U_{in} will have these settings:

- ✖ Sine wave,
- ✖ Peak-to-peak value $U_{in\,pp}$= 4V,
- ✖ Frequency   f = 100Hz,
- ✖ Offset = 0V.

Add successively different capacitors of capacitance C <u>in parallel to</u> the load resistor R_L, according to the available values.

Q2.2a   *Group in a table for each of the available capacitors:*

   - a figure giving the appearance of the output voltage after any smoothing,

   - the peak-to-peak amplitude of the residual ripple voltage

   - the average voltage value,

   - the significant ratio (in %) of these last two.

Q2.2b   *What is the C's role in this filtering? Which minimal capacitance seems to be necessary?*

⇨ ***Remember****: your digital scope has both DC and AC setting modes on each channel. Think about that.*

3- VOLTAGE STABILISATION : voltage regulator

According to the technical data of 7805, we must keep in mind that such a voltage regulator is designed to keep its output voltage as close as possible to a specific steady value, even when the output current changes significantly or when the input voltage is not steady (i.e. when the input voltage fluctuates).

So, we must consider both cases.

3.1- Stability according to the change of the load resistor

A voltage regulator is an integrated circuit (IC) exploiting the properties of a zener diode. It is designed so that its output current can be large without making the output voltage vary. This one is a 7805.
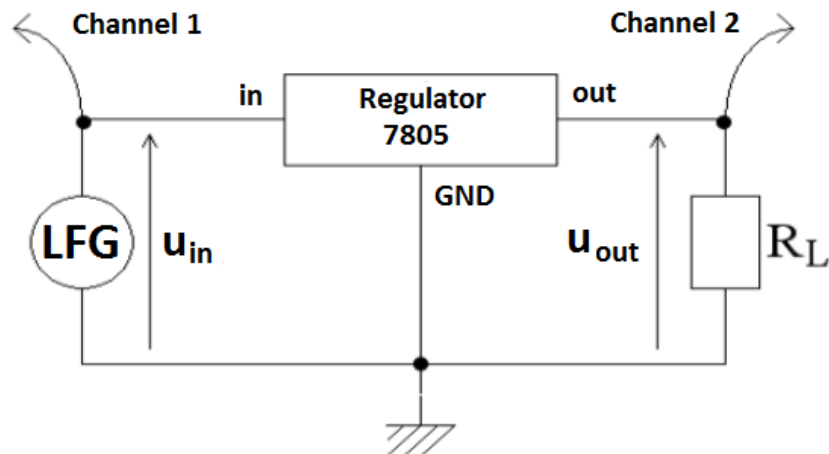
The function generator will provide U_{in}= 8,5 V which is a DC voltage (only a DC component ; use Main Out with a DC setting ; the DC value is set with « Offset »).

Q3.1 Measure the output average voltage for the available values of R_L.

Analyse and comment.

3.2- Stability according to the change of the input voltage

Input voltage fluctuations will be simulated by a sine voltage added to the previous DC voltage:

- Choose $R_L = 470\ \Omega$ as a load resistor
- Set the LF Generator to deliver on Main Out both a 8,5 V DC component and a sine voltage with frequency f = 100 Hz and amplitude V_F (with F as fluctuation).

$$u_{in}(t) =\ 8.5 +\ V_F\ sin(2\pi\ ft)$$

Q3.2 Measure the peak-to-peak amplitude for both input and output voltages, when $V_F =$ 1V, 2V and 3V.

⇨ *Remember that you can use the Average Acquisition option on your scope if the signals are too noisy.*

Calculate a significant ratio (in %) of these peak-to-peak amplitudes. Analyse and comment.

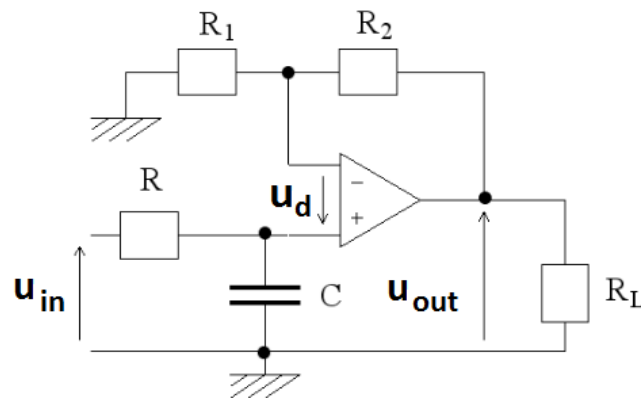# <u>Practical #4:</u> EXPERIMENTING A LOW-PASS FILTER

AIMS AND OBJECTIVES:

- Study experimentally the gain and phase Bode curves of an active first-order low-pass filter.
- Highlight its characteristics (static gain, cut-off frequency and bandwidth at -3 dB, slope in dB / decade).
- Create a simulation of this filter with PSPICE software.

0- BEFORE THE PRACTICAL:

$R = R_1 = R_L = 10 \ k\Omega$

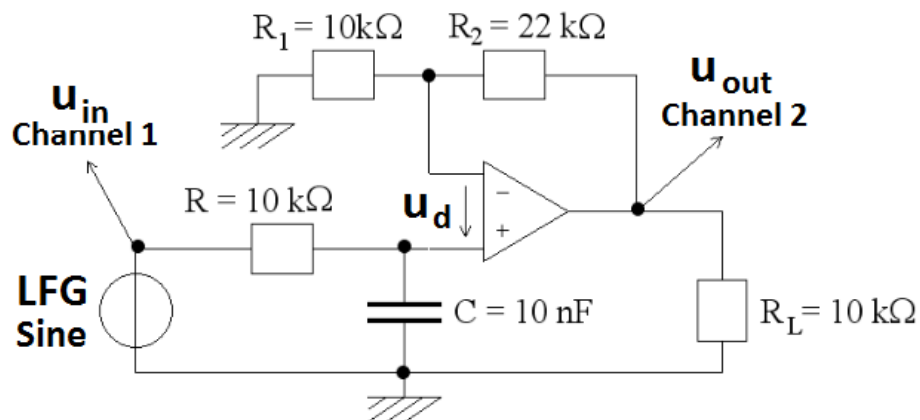$R_2 = 22 \ k\Omega$

$C = 10 \ nF$

Q0.1 *Find the expression of the transfer function of this filter.*

Q0.2 *Draw roughly the shape of the Bode response of this filter.*
Calculate the static gain and the Cut-Off frequency (at -3 dB).

1- REAL EXPERIMENT: BODE RESPONSES IN GAIN AND PHASE

1.1- Experimental circuit

The input voltage U_{in} will have these settings:

- ✖ Sine wave,
- ✖ Peak-to-peak value $U_{in\,pp}$= 4V,
- ✖ Frequency   f = 100Hz,
- ✖ Offset = 0V.

1.2- Qualitative checking of filter behavior

Q1.2a *Check that the output voltage is maximum at low frequencies, and that it decreases as the frequency increases.*

Q1.2b *Measure the cut-off frequency of the filter f_c. Give and justify your procedure.*

1.3- Tracing the Bode responses

Q1.3a *Prepare a table of about ten measurement relevant frequencies, explaining your choice of frequencies.*

Q1.3b *Complete the table with the measurements values of U_{in} and U_{out} , the calculation of the gain G and the phase shift φ between these signals.*

Q1.3c *Trace the gain G versus log (f) on a semi-log paper sheet.*

Q1.3d *Trace the phase shift φ versus log (f) on a second semi-log paper sheet .*

1.4- Analysing the Bode responses

Q1.4a *Find on the gain diagram : static gain, Cut-Off frequency f_c and the slope in dB/decade .*
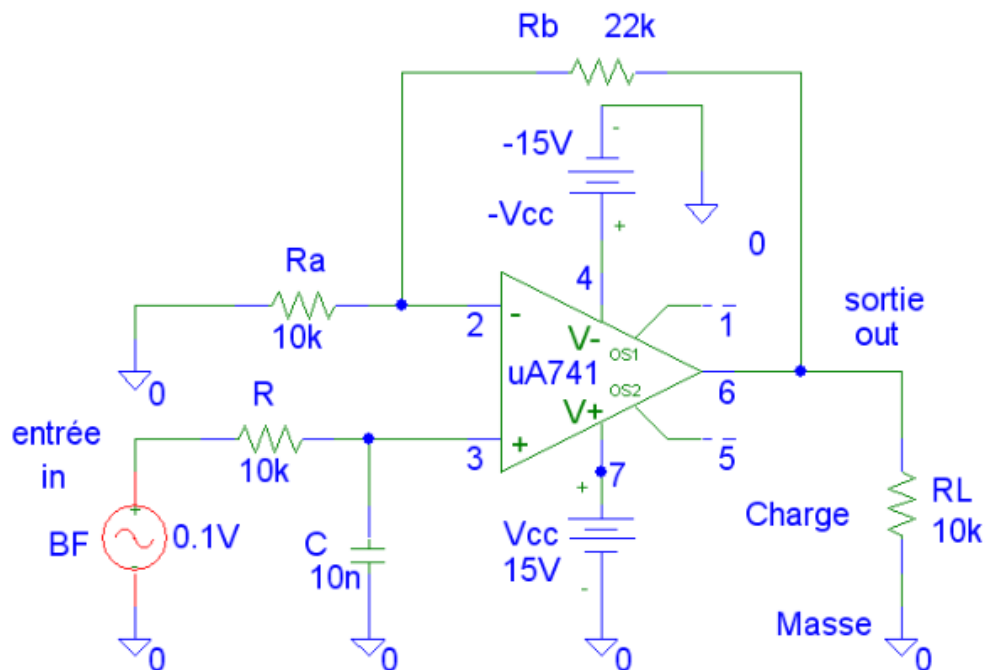
Q1.4b *Find on the phase shift diagram : limits values at low and high frequencies, Cut-Off frequency f_c and the slope of the asymptote between 10 f_c and f_c / 10.*

Q1.4c *Compare the obtained values to the theoretical values.*

2- VIRTUAL EXPERIMENTS

SIMULATION OF BODE RESPONSES IN GAIN AND PHASE

2.1- Use PSPICE to simulate the frequency behaviour of the filter

Find the schematic presented above in the *passe_bas.sch* file.

Set the input voltage at 0.1 V

Set up a frequency analysis over a suitable frequency range (AC Sweep, 10Hz to 1 MHz, 100 points per decades)

*Q2.1a   Simulate and trace the **Gain** Bode response of that filter.*

Q2.1b   Measure the cut-off frequency in cursors mode. Print and comment.

Q2.1c   Simulate and trace the Phase Shift Bode response of this filter.

2.2- Influence of the capacitor value on the cut-off frequency of the filter

One method consists in observing the influence of the tolerance on the capacitance of the filter on the shape of the Bode curve and the associated cut-off frequency (for example for a C value known with a 20% uncertainty).

Use the PSPICE Manual *(cf. binder)* to perform a Monte Carlo simulation on V(OUT): you will set and perform a series of responses when choosing a capacitance value at random, within the range C± 20 % of C.
Then make relevant measurements on PROBE graphs with cursors.

*Q2.2 Investigate the influence of the capacitor on the frequency response of the amplifier.
       Estimate the relative uncertainty on f_c, if you assume 20 % for the C value.
       Print the corresponding graph.*

3- WORK ANALYSIS AND SYNTHESIS

Q3a   Compare the cut-off frequencies that you have obtained.

Q3b   Compare the obtained Bode responses (in gain and phase) for ideal and real models.
.

Practical #5: EXPERIMENTING A SELECTIVE BANDPASSFILTER

Aims and Objectives:

You will perform a study of the frequency response of a bandpass filter [filtre passe-bande] and understand how to use such a filter to get the frequency spectrum of a periodic signal.

0- BEFORE THE PRACTICAL:

This **bandpass filter** is an active filter; its **centre frequency** is simply controlled by a clock signal frequency:

⇨ Note on the schematic:

    - Supply voltage +15V, 0 V and -15 V (active filter)

    - Input voltage u_{in} and Output voltage u_{out}

    - Clock voltage for setting the centre frequency f_0
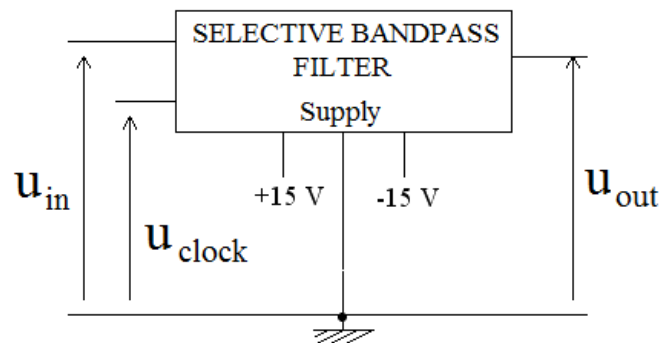
The **frequency response** of this bandpass filter is given by the amplitude transfer function (the modulus of the complex function):

$$A = \frac{U_{out}}{U_{in}} = 1 \Big/ \sqrt{1 + \left[Q\left(\frac{\omega}{\omega_0} - \frac{\omega_0}{\omega}\right)\right]^2}$$

The **quality factor** of the filter is defined by $Q = f_0 \big/ \Delta f$ as the ratio of the center frequency and the filter bandwidth.

For that peculiar IC [Integrated Circuit], Q is given by:

Q = R₁ / R₂ with R₁ = 10 kΩ and R₂ = 330 Ω.

This specific IC has been specially suggested to make the conception of the different 2nd order filter quite easy.

MF10 includes a technology called « commutated capacitor » which allows a very efficient way of controlling the main filter's parameter f_0 :

The **center frequency f_0** is set when choosing the clock frequency f_{clk} as simply as $f_0 = \frac{f_{clk}}{100}$.

Q0.1a   Calculate Q, then Δf when $f_0 = 1$ kHz

Q0.1b   Describe the shape of the amplitude transfer function of this filter.

1- CHECKING HOW THE FILTER WORKS

That IC has been used to make audio filters. It will work fine in the audio band, a few kHz.
For example, for a choice of $f_0 = 1$ kHz , you must set your frequency with $f_{clk} = 100 f_0 = 100$ kHz.

1.1- Plugging & setting

AGILENT 33500B will be involved in the setting of the parameter which requires t**o be sharply set**. In the first experiment §1.2, it will set the input signal frequency, in the second one §2.2 it will set the clock frequency.

- *One function generator will give the Clock signal*
⇨ *Please be awake and DO NOT use the AGILENT one.*
  Link the TTL output to the left channel of the Scope. Check the signal .
- *Set a frequency of 100kHz.*
- *Then move the cable and plug the clock signal on the clock enter of the test card.*

- *Now link the usual output of the function generator AGILENT 33500B to the Input of the filter. Set a sinus signal (2V peak-to-peak, no DC component, frequency $f = 1000$ Hz)*
- *Check both Input and Ouput signals of the filter on the scope (Input on the left-hand Channel, Ouput on the right-hand one),*
- *Use AutoMeasurement for frequency and amplitude. Look for a max for output signal. What can you deduce?*

1.2- Measuring the Bode amplitude function

The center frequency of this filter should be close to $f_0 = 1$ kHz.

⇨ You will have to choose a good set of test frequency values to obtain a useful curve.

Q1.2a *For 12 pertinent values of Input frequency around f_0, measure both Input and Output voltage, the calculate the Gain A. Complete the set with 10 f_0 and f_0 /10*

Q1.2b *Trace the gain A versus the test frequency in the usual way. Then trace the Bode function*
$G = 20\ Log_{10}(A)$ versus $Log\ f$

Q1.2c *Define the -3dB bandwidth for both curves. Measure the experimental values of the bandwidth, of f_0 and then Q. Is it a selective filter?*

2- ANALYSIS OF THE SPECTRUM OF A INPUT SIGNAL

A triangle signal is a good one for testing the efficiency of this so-called « selective » passband filter.

2.1- What should be measured?

According to the Fourier theory, such a triangle signal should be described by this series, with A = 8V:

$$u_{triangle} = \sum_{p=0}^{+\infty} C_p\ cos\big((2p+1){\times}2\pi ft\big) = \sum_{p=0}^{+\infty} \frac{4A}{\pi^2}{\times}\frac{1}{(2p+1)^2}\ cos\big((2p+1){\times}2\pi ft\big)$$

Q2.1a *Calculate the numerical values of the first eight harmonics. Mind these ones:*
C_1 : harmonic n°1 when $p = 0$ and $f_1 = (2p+1)f = f$,
C_3 : harmonic n° 3 when $p = 1$ and $f_3 = (2p+1)f = 3f...$
What is C_2 ? C_4 ? And so on.

Q2.1b *Trace the theoretical spectrum.*

2.2- Measurement using the selective filter as a frequency scanner

Set the test signal with the usual function generator (don't use Agilent):

- *Set a triangle signal (it has to be very symmetric)*
- *Peak-to-peak value: 8 V.*
- *DC component : 0V*
- *Triangle frequency : 500 Hz*

Looking for harmonic component n° q = 2p+1 means that you will have to scan the output signal when the clock frequency is around q x $f_{triangle}$.

Q2.2a   *Start with f_{clk} = 45 kHz, then increase the clock frequency until you obtain some useful signal.*

         You are trying to get information about the first eight harmonics (if you can!). Describe each local max (frequency, shape of the signal, amplitude C_q)

Q2.2b   *Complete your graph with this experimental spectrum. Compare them with the theoretical one.*

Q2.2c   *What do you think of this simple analog frequency scanner?*