

# Laboratory Manual

## TSEI01 Analog Circuits

### Description

The purpose of this series of laboratory exercises is to provide the student with hands-on experience of mainly amplifier structures. We will use the MultiSim tool (Circuit Design Suite) from National Instruments as well as their ELVIS+ platform for measurements. The measurement and simulation systems have the same look-and-feel. More information is conveniently downloaded from National Instruments' home page. For example, you can find a tutorial at:

- <http://www.ni.com/pdf/manuals/374482d.pdf>

In short the labs consist of:

- 1) Simulation in MultiSim and get to know the tool. We will work with one of the tutorial manuals from National Instruments and simulate a variety of test circuits to understand the concepts of for example transient analysis, AC analysis, and possibly DC analysis.
- 2) Simulation in MultiSim: Stability of Operational Amplifiers. Negative feedback.
- 3) Project exercise: Design a RIAA<sup>1</sup> amplifier. During two four-hour sessions you will design an RIAA amplifier, simulate it, build it and demonstrate it.

Notice that the labs do not contain a bunch of **fill-in-the-box-so-that-I-can-go-home exercises**. Instead some of the questions are found in the text flow and some of the questions cannot be answered just with a number or so.

x The scope of the labs are compiled in blue boxes, like these.

x Questions are also found here.

Also notice that you have to **hand in a laboratory plan**, 48 hours before the laboratory starts, to the laboratory assistant. He will then check if you are prepared enough to conveniently manage the lab within the scheduled hours. Handing in the plan and showing up on the labs are part of the obligatory attendance on the laboratories.

Read Lab 0 instructions in this manual in order to know more on how to prepare the laboratory plan. Some comments:

x Tips and important information are given in these kind of boxes.

<sup>1</sup>Recording Industry Association of America (RIAA).



- x Please, please, read the instructions carefully.
- x Also, save a tree - you do not necessarily have to print this manual.
- x Further on, notice that laboratory 1 and 2 are "simple" (give or take), but they are crucial for the preparations of your project that you will do in lab 3.
- x If you finalize laboratories early, why not start with the next one ...

## Outline

### -LAB0-

Introduction.....	4
1 Introduction.....	4
2 Preparations.....	5
3 Examination.....	5

### -LAB1-

Simulation with SPICE.....	7
1 Introduction.....	7
2 Examination.....	8

### -LAB2-

Opamps and Negative feedback.....	9
1 Introduction.....	9
2 Preparations.....	10
3 Simulation setup and running the simulations.....	10
4 Measurements, i.e., the hardware part.....	15
5 Examination.....	22
6 Concluding remarks.....	22

### -LAB3-

RIAA Amplifier Stages.....	23
1 Introduction.....	23
2 Specification.....	24
3 Simulation.....	25
4 Measurements.....	26
5 Resources.....	26
6 Examination.....	26
7 Report template.....	26



## List of Tables

Table 3.1: Example of a laboratory preparation plan..... 6  
 Table 3.1: AC analysis settings..... 12  
 Table 3.2: Transient analysis settings..... 14

## List of Figures

Figure 2.1: Example snapshot of the captured schematics..... 8  
 Figure 3.1: Three-stage amplifier chain. Notice that R1 should be 100 kOhm..... 11  
 Figure 4.1.1: Exaggerated examples of bad (top) and good (bottom) practices..... 16  
 Figure 4.1.2: Photo of the ELVIS+ board top left pin-out..... 17  
 Figure 4.1.3: Photo of the ELVIS+ board lower left pin-out..... 18  
 Figure 4.1.4: Example view of the instrument launcher..... 19  
 Figure 4.1.5: Function generator example window..... 19  
 Figure 4.2.1: Bode plotter screenshot. Remember to set the parameters in the Measurement Settings pane..... 20  
 Figure 4.3.1: Oscilloscope screenshot. Choose channels on top..... 22  
 Figure 2.1: Example of RIAA equalization characteristics [source: wikipedia]..... 25

## History

Rev	Date	Comment	Created/ Issued by
P1A	2014-11-13	Updated for the new labs in the TSEI12 version of the course. New document number and title. Reduces the contents of lab 1 quite significantly. Illustrated lab 0 a bit more clearly. Distributed to students.	J Jacob Wikner
P2A	2014-11-18	Updated the manual with the second laboratory.	J Jacob Wikner
P3A	2014-11-24	Updated the manual with the second laboratory, hardware parts.	J Jacob Wikner
P4A	2014-11-26	Updated with information for the third laboratory.	J Jacob Wikner
P5A	2014-12-01	Finalized the third laboratory.	J Jacob Wikner

# -LAB0-

# INTRODUCTION

## 1 Introduction

This first part of the document just outlines the scope of the laboratories. The first two labs are essentially just to get started with the tool. During the first laboratory we will capture a schematic and simulate it, in the second laboratory we will then get a schematic, enter it into the simulator and verify its functionality. After that you will connect it also with hardware components.

The third laboratory is a miniproject. In this case, you will more or less just get the specification and then it is up to you (with some guidance) to synthesize and design the analog circuit requested.

This manual will not describe conceptual descriptions of simulation, testing, measurements, etc. Those aspects are covered by the main course book, and to be honest, after a few minutes in the laboratory, you will understand what simulations are about.

We will use the Multisim environment from National Instruments and it is installed in the Transistorn and Freja labs.

x Make sure you are able to log on to the windows machines at ISY!



Now, if you can login to the machines, you will quite easily find the tools that we are going to use under the Start menu. Browse through something like: All Programs : National Instruments : Design Suite : Multisim.

## 2 Preparations

As mentioned in the lecture and on the course home page. In order for you to be able to start the laboratory, you need to submit a laboration plan report to the laboratory assistant (LA) and examiner (EX) two days in advance of the lab, by the latest. Send them to both:

[jacob.wikner@liu.se](mailto:jacob.wikner@liu.se)

[arta.alvandpour@liu.se](mailto:arta.alvandpour@liu.se)

Make sure to mark the e-mail subject with **TSEI01 : Laboratory Plan 1/2/3**

The report should be something like 1/2 to 1 A4 page where you in text, supported with figures if needed, explain how you plan to execute the laboratory. Also consider the time involved - what can I do in four hours. How do I divide the work.

For example Table 3.1 might give you a hint, or two.

## 3 Examination

In each part of this laboratory manual you will find a section called Examination. That section will clearly define what you need to show the LA in order to pass. Study those also when you prepare your plan.



Topic	Examples
Components	2x Operational amplifier 741, 10 resistors, cables
Equipment	ELVIS platform, Oscilloscope, Digital multimeter
Software	NI Multisim, MATLAB, GoogleDocs
Theory	"Chapters 3, 5, 7" Notice that you have to find the relevant theoretical parts required for the laboratory in the course book yourselves. They are not directly expressed in this manual
Documents	Important documents I need to have ready to support my work. Multisim manual, <a href="http://blahblah">http://blahblah</a> , Lab Manual, blahblah Familiarize yourself with the ELVIS board: <a href="http://www.ni.com/white-paper/14502/en/">http://www.ni.com/white-paper/14502/en/</a>
Expected outcome	What do you expect to see on the screen? What do you expect to learn from the lab? What are you expected to report to /show the laboratory assistant (LA).
Risks and derisking	Assume something goes wrong during the lab. "This part seems complex. What if we do not solve that?" The risk/derisk is a quite good way to get a feeling for if you will be able to complete the lab within time. What if we do not find this result - what should we do? Can we work more efficiently together by splitting up work? Preparing?
Report	How to document the results .GoogleDocs, screen capture of every graph, diary? Remember that some documentation from the first laboratories could be very good to save for the third laboratory (miniproject). Save the schematics and test benches!

Table 3.1: Example of a laboratory preparation plan.



# -LAB1-

## SIMULATION WITH SPICE

### 1 Introduction

In this first laboratory of the TSEI01 course you will simply get used to the simulator and the tool from National Instruments. The software is easily located under the Start menu (Windows 7), Programs, National Instruments, Design Suite.

Open the "Getting Started" [PDF] from the Help menu and simply follow the steps through it. The document is called "NI Circuit Design Suite, Getting started NI Circuit Design Suite". (Notice that there are some different revisions of the document floating around if you would google for it).

x Go through chapter 1 and chapter 2 of the manual.

The latest version of the document can also be downloaded from

- <http://www.ni.com/pdf/manuals/374482d.pdf>

Supporting documents can also be videos and there are plenty of those in various flavours, e.g.:

- <https://www.youtube.com/watch?v=tgCPDBtRcso>
- <https://www.youtube.com/watch?v=pg4DI5s8FNo>
- <https://www.youtube.com/watch?v=QVR86gaLhUc>
- <https://www.youtube.com/watch?v=vZA47ojNOLs>







# -LAB2- OPAMPS AND NEGATIVE FEEDBACK

## 1 Introduction

In this laboratory we will look at different ways of feedback and compensation of your operational amplifier in order to improve the overall performance of the system with respect to stability and linearity. In Bengt Molin's book we can read a about the history of feedback and conclude that it was surprisingly difficult to understand and introduce as a design concept for electrical circuits. Today, even though the maths can be a bit troublesome from time to time, we see it as a quite natural way of stabilizing a circuit.

The task of this laboratory is to

x Dimension the feedback in a three-stage amplifier (using operational amplifiers) such that the total phase margin is  $\varphi_m = 45$  degrees.

x The compensated amplifier's frequency and step responses should be simulated and also measured.

In the first part of the laboratory you will simulate an electronic circuit using a set of operational amplifiers. You will investigate how feedback can be applied and what the consequences are.



The circuit schematic is given to you and can be downloaded from the course webpage:

[http://www.isy.liu.se/edu/kurs/TSEI01/laboration/TSEI01\\_PM\\_laboratory2.ms12](http://www.isy.liu.se/edu/kurs/TSEI01/laboration/TSEI01_PM_laboratory2.ms12)

and reprinted in Figure 3.1 as it looks in the simulator.

In the second part of the laboratory you will also set up the design in a hardware test bench and evaluate it in "real life".

**x Tip!**

You are most likely two people in the lab group. Save yourself some time: one of you will setup the simulator and one of you will start arranging with the hardware platform and connect the circuits. The laboratory assistant provides you with components.

## 2 Preparations

Do not forget to hand in your lab preparation plan! Suggestions on topics to plan are found in Table 3.1. In this laboratory the following things could be discussed (examples)

- Phase margin. What is the concept of phase margin and what is the impact on e.g. the step-response of your amplifier?
- DC gain. What is the total DC gain, in open-loop configuration, of the amplifier in Figure 3.1?
- Feedback factor. What is the relationship between feedback factor and gain, phase margin, and amplitude margin?
- Download datasheets, read chapters in book, refresh lecture notes and lesson exercises.

and more.

## 3 Simulation setup and running the simulations

As mentioned, the amplifier schematics is shown in Figure 2.1. As you see we have prepared most of the schematics for you, but the feedback (right-most part) is not really dimensioned as it should be.

Notice the type of operational amplifier ([UA741CD](#)). Datasheets for that particular type are found on the web and you would there also find descriptions of the pin-outs, open-loop gain, and any other parameter that might be vital for you.

In this case, the feedback factor  $\beta$  is essentially given by how much voltage is fed back from the output to the input of the amplifier by using the resistive divider:

$$\beta = \frac{R_{10}}{R_9 + R_{10}} \quad (2.1)$$

With the switch **S1** in the schematics, the feedback network on the far right can be disconnected such that the open-loop gain  $A_0$  can be determined. Further on, by adjusting resistors **R9** and **R10** we can then also adjust the feedback factor accordingly.

x Dimension  $R_9$  such that the phase margin becomes  $\varphi_m = 45$  degrees.

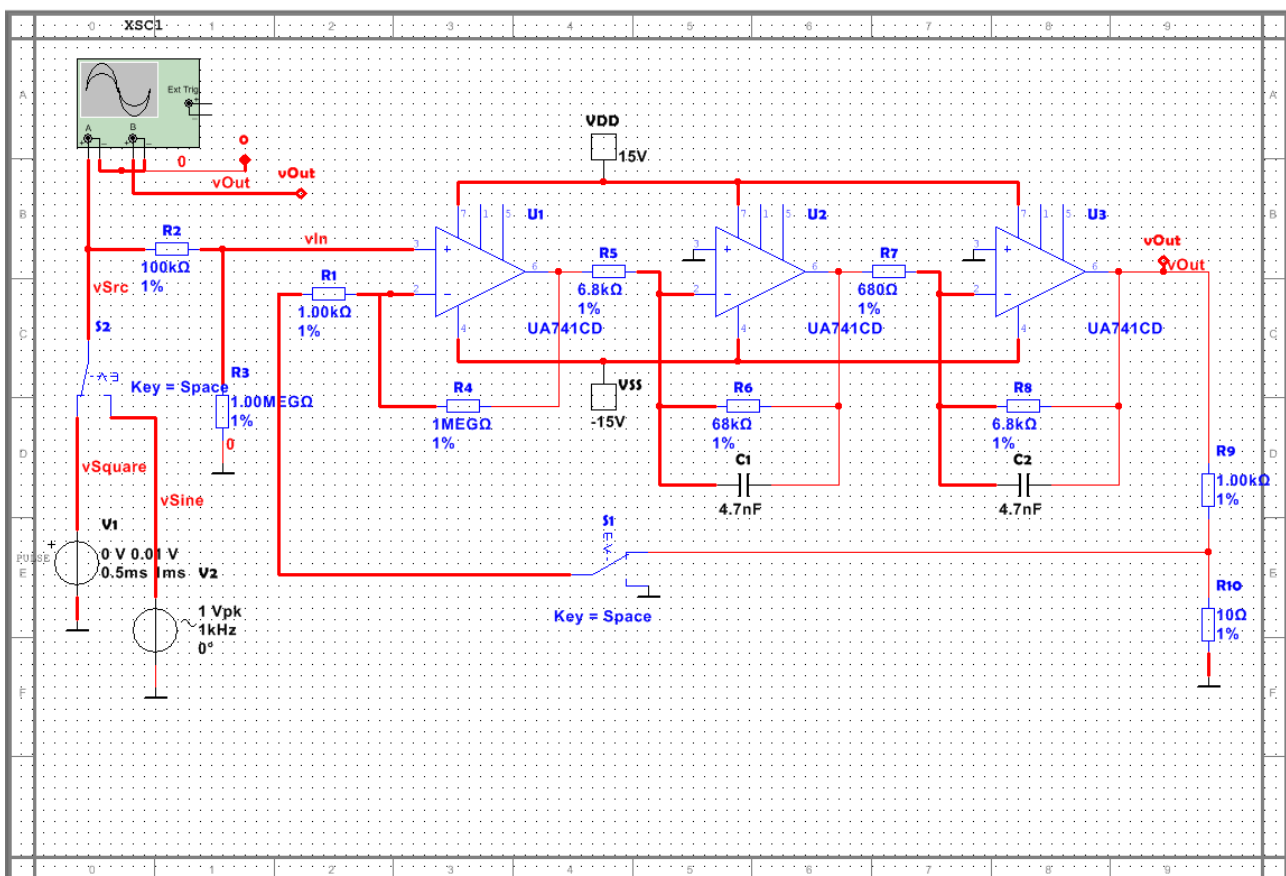


Figure 3.1: Three-stage amplifier chain. Notice that R1 should be 100 kOhm.

The input to the amplifier should be of two different kinds: a sine wave or a square wave. The square wave should have a **10-mV amplitude** and the sine wave should have a **1-V amplitude**. Both inputs should have a frequency of 1 kHz and both of them should have an AC Analysis Magnitude set to 1 V.

### 3.1 Step 1, no feedback

Set the jumper/switch **S1** such that there is no feedback connected to the input of operational amplifier **U1** (left-most). Make sure that the input is grounded by the switch.



When you dimension the feedback you should set  $R_{10}=10$  and  $R_9$  should eventually be a value taken from the standard E12 series [browse the web to find this table of values]. Start your first trials to understand the circuit and check its functionality with a sine wave input signal at a 1-kHz frequency and a 1-V amplitude.

Start the simulation and observe the input and output waveforms in the time-domain by double clicking the oscilloscope.

x What happens with the output signal vOut?

Now configure an AC analysis in the following way.

```
Simulate > Analyses > AC Analysis...  
Frequency Parameters  
Start frequency (FSTART):      1 Hz  
Stop frequency (FSTOP):       100 MHz  
Sweep type:                    decade  
Number of points per decade: 100  
Vertical scale:                logarithmic  
Output  
V(vOut) > Add >  
Simulate
```

Table 3.1: AC analysis settings.

The grapher view window pops up and you can see the magnitude function and the phase diagram, essentially your Bode plot.

x We will determine the feedback factor  $\beta$  for a phase margin of  $\varphi_m=45$  degrees.

To establish the feedback factor, we first find the point where the phase is  $-180-\varphi_m=135$  degrees. In the grapher viewer you can select cursors and add to both the magnitude graph as well as the phase graph. In the phase diagram pull one of them to  $\varphi=-135$ . Find the frequency for which this happens, i.e., the x-value. (Yes, the values in the small white box are a bit cryptical, but move the cursor back and forth and you will see which cursor is affecting which value).

x What x-value, i.e., frequency, do you get?

Now, determine the open-loop gain  $A_0$  when the phase is at  $\varphi=-135$ . This can be done by placing the mouse pointer in the magnitude diagram and move the cursors to the same x-value frequency.



x What  $A_0$  value do you get for the point where  $\varphi = -135^\circ$  ?

According to theory (you have read the chapters in the book?) we should now let the feedback factor be a function of the amplitude margin

$$\beta = \frac{1}{A_0} \text{ (c.f. amplitude margin) and } \beta = \frac{R_{10}}{R_9 + R_{10}} \quad (2.2)$$

which, if we assume  $R_{10} = 10$ , can give us the value of  $R_9$ . Round the value to the closest value in the E12 series.

x Which  $R_9$  value do you get?  
x Which value do you take from the E12 series?

### 3.2 Step 2, with feedback

Now set jumper/switch in a position such that feedback is guaranteed and apply the feedback settings derived from the previous part of the lab.

Re-run the simulation and observe the results in both time domain and frequency domain.

x What overall DC gain do you get?  
x What bandwidth?  
x What unity-gain frequency do you get?  
x What does the sinusoid look like now?

Toggle the input signal to use a pulse train with a 1-kHz frequency and 10-mV swing and run a transient analysis. In this case use the grapher viewer rather than the oscilloscope. The settings are found in Table 3.2.

Apply a step response and observe and capture the behavior.

x How large is the overshoot? How does this correspond to theory?



```
Simulate > Analyses > Transient Analysis ...

Analysis parameters
Initial Conditions           User defined
Start Time (TSTART)         0
End Time (TSTOP)            0.002 s
Maximum time step (TMAX)
Minimum number of time points 1000

Output
V(vOut)> Add >

Simulate
```

Table 3.2: Transient analysis settings.

### 3.3 Intermediate examination

Before you get too wild you might want to double check with the laboratory assistant that you have everything set up correctly in order to continue with the hardware lab (Section 4).

- x Reasonable value on R9 found?
- x All the questions in the blue boxes answered?

## 4 Measurements, i.e., the hardware part

You now also have to measure the amplifier setup in real life. For this purpose you will use the ELVIS+ board. There is plenty of information on the web about the platform as well as tutorials on National Instruments home page and included also in the software. We will not repeat them in this document, nor will we describe the way a component is physically placed in detail. Trial-and-error is a good strategy also here. The ELVIS+ board is more or less bullet-proof and it is easy to pick-and-place components and move them around. If you by accident short to signals, ELVIS will not mind.

- x Read more about the hardware platform! Identify components, knobs, buttons, what-have-you on the board. Notice that there are two ON-buttons.
- x Understand how all the holes on the ELVIS+ board are connected "underneath".

Once you have a pretty good understanding, you will start to connect your amplifier, more or less as it is shown in Figure 3.1, on the board by placing all the components. Jumpers/switches are of course not there, supplies and signals are taken from the ELVIS+. The oscilloscope is also provided to you by the ELVIS+ board.

- x Show the assistant that you have ELVIS+ up and running and then ask the laboratory assistant for components that you need to hook up your circuit.

The most complex component is probably the operation amplifier and you need search for the [UA741CD](#) component on the internet to find the pin-out, for example

- <http://cdn.instructables.com/FXE/J8RT/GAFDTBQ1/FXEJ8RTGAFDTBQ1.LARGE.gif>

or simply the datasheet from the hardware provider. At this point in the laboratory series, you probably do not have to care for the offset adjustment pins (5 and 1) unless your specific sample of operational amplifier happens to have bad quality. The supplies should connect to the fixed +15/-15 available on the ELVIS+ board. Use the long +/- vertical rails to distribute the voltages across your design.

### 4.1 Connecting the cables

It is then time for you to start orienting the devices and connect them with cables. Some tips:

- Do not let the cables fly around in a mess.
- Do not use too long cables,
- Press them down, and massage them a bit to mimic the schematics with straight angles.

- Do not use cables for interconnects provided to you locally by the board.
- Use color-coded cables, supply/ground/input/output

Figure 4.1.1 shows some bad and good examples for inspiration. Use common sense and create a circuit that you can proudly present: "this is my design".

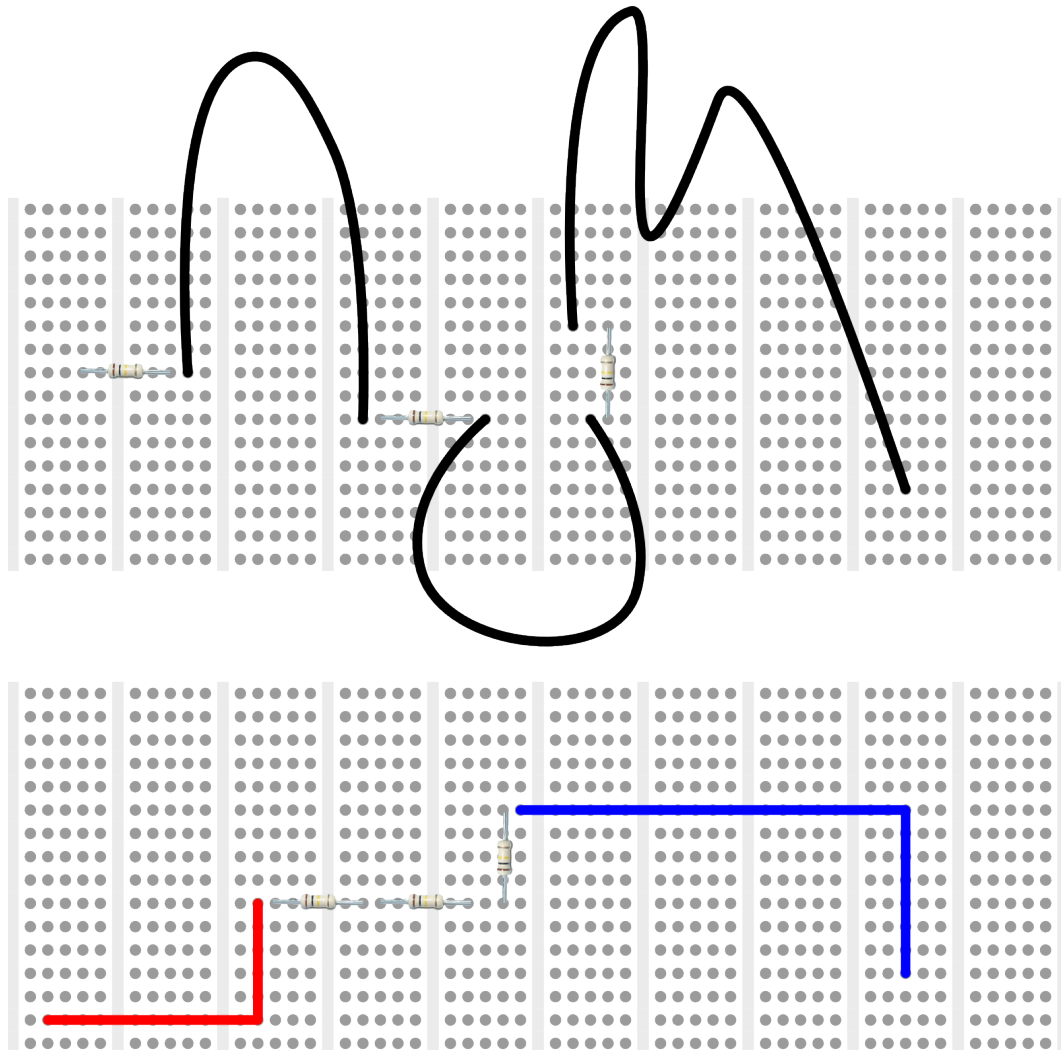


Figure 4.1.1: Exaggerated examples of bad (top) and good (bottom) practices.

On the left-hand side of the ELVIS+ board you have quite a few built-in, bidirectional ports (numbered). By looking at the list below and the following photos, quite a few of these are indirectly explained.



Connect them according to this:

x Obviously there are many ways to connect input and output signals, but in our case, to simplify debugging if needed, please use the same settings as recommended in the lab manual.

- AI0+ (row 1) should be connected to the function generator (FGEN, row 33)
- AI0- (row 2) should be connected to ground (AIGND, row 18)
- AI1+ (row 3) should be connected to the output of the amplifier, vOut in Figure 3.1.
- AI1- (row 4) should be connected to ground (AIGND, row 18)
- AI0+ (row 1) should also be connected to the input of your amplifier, i.e., vSrc in Figure 3.1.

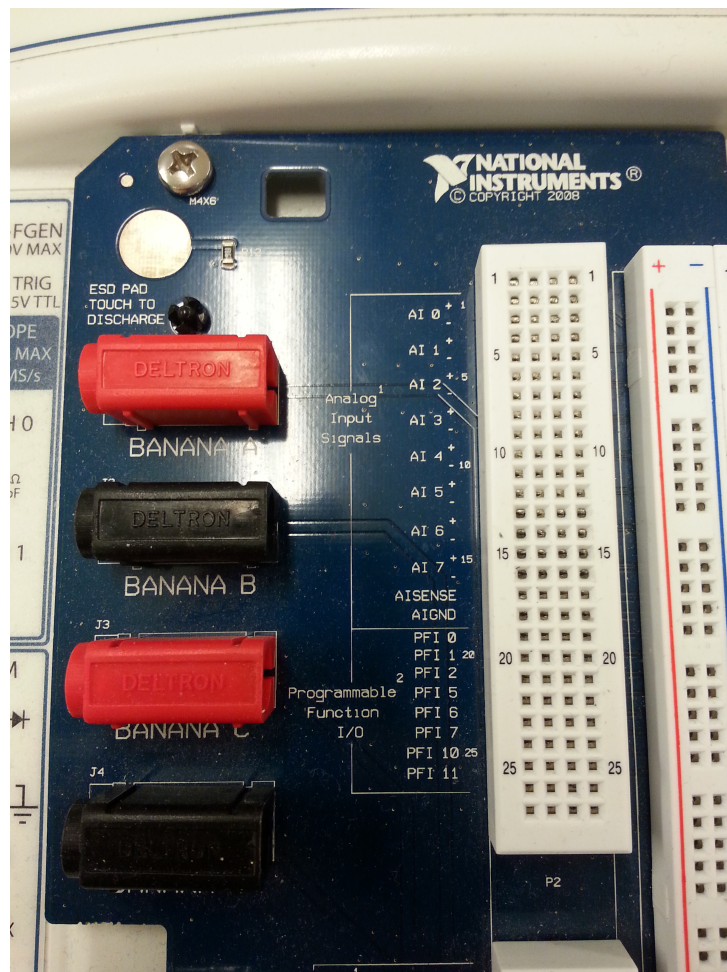


Figure 4.1.2: Photo of the ELVIS+ board top left pin-out.

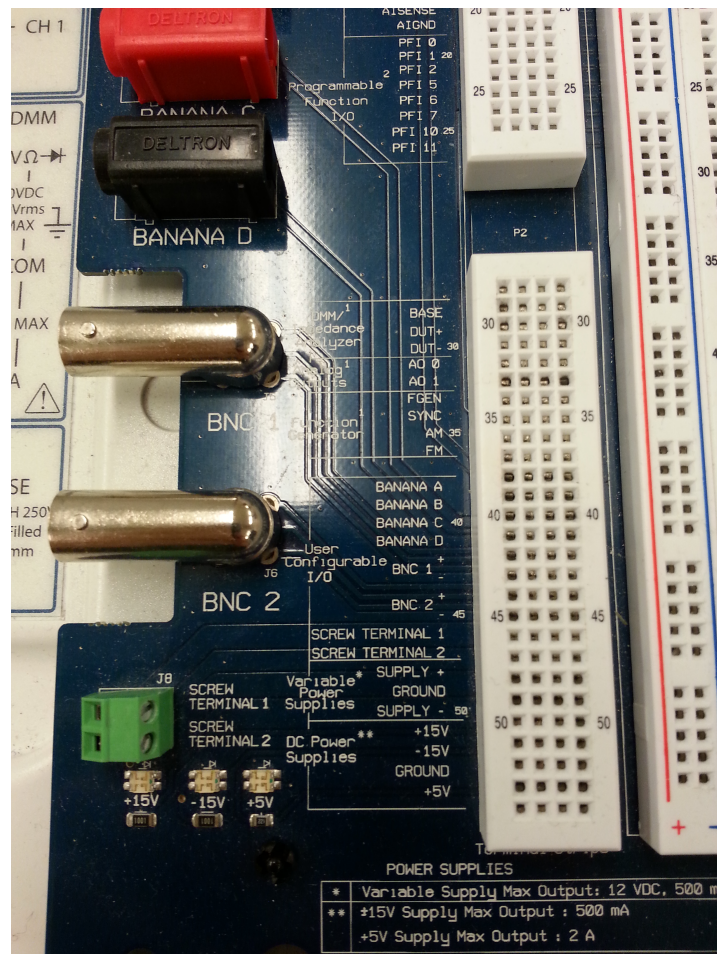


Figure 4.1.

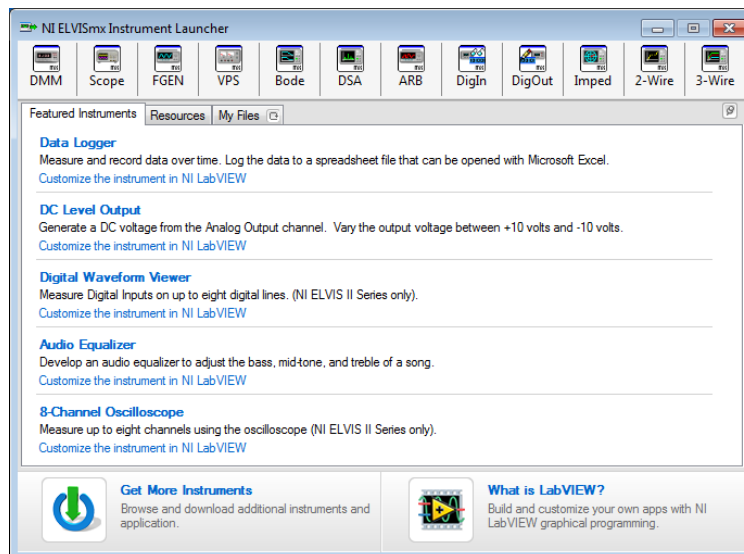


Figure 4.1.4: Example view of the instrument launcher.

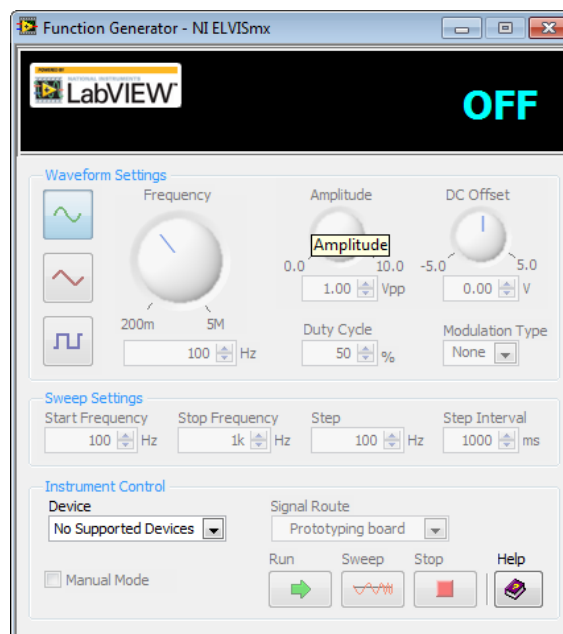


Figure 4.1.5: Function generator example window.

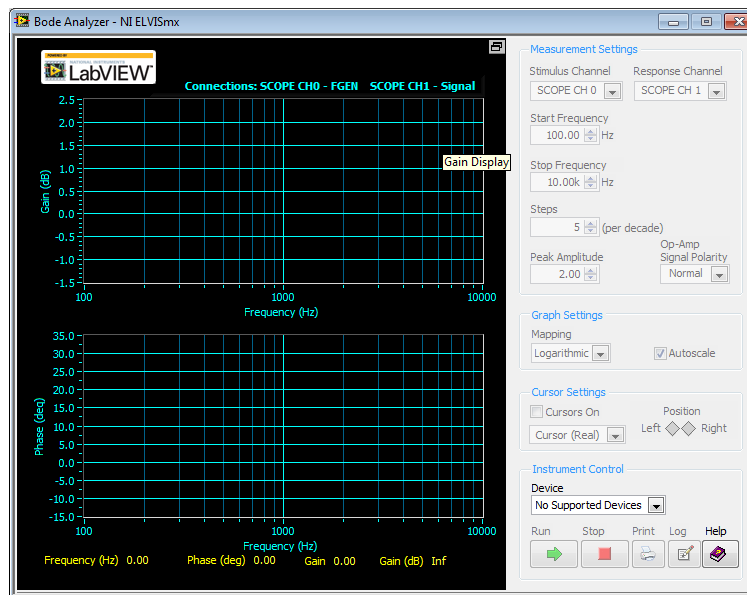


Figure 4.2.1: Bode plotter screenshot. Remember to set the parameters in the Measurement Settings pane.

## 4.2 AC analysis (frequency domain)

In the Bode plotter, you will soon simply press the play button and you should get the frequency characteristics of your system. However, first make sure that you have right stimulus and response channels:

- Set the stimulus channel to the input of your circuit, i.e., `vSrc/AI0+/FGEN`, if you have used the same pin settings as we suggested earlier.
- Set the response channel to the output of your circuit, i.e., `vOut/AI1+` if you have used the same pin settings as we suggested earlier.

and that you have reasonable start and stop frequencies. Set the number of steps per decade to something like 5 the first time to verify that your test works. The start and stop frequency could be 10 and 1 MHz, respectively, in the first trial. Then expand the frequency range and number of points per decade until you get a nice, smooth graph from which you can extract details and compare with your simulated results. Experiment also with the Peak Amplitude settings, start with a low value and work yourselves upwards.

Notice that sometimes one can get rather cryptical results on the Bode plotter due to conflicts between the instruments and or incorrect connections. The Bode plotter will generate a sinusoid input to your circuit at different frequencies and then compare that with the output sinusoid. If something prevents it from generating that sinusoid, the output will obviously be strange too.

x Save the graph and show the lab assistant.



- x Measure the frequency characteristics of the closed loop.
- x What is the DC gain? What is the bandwidth?
- x Are there any differences between simulated and measured? Why?

### 4.3 Transient analysis (time domain)

In the FGEN (Figure 4.1.5) select a pulse wave form and set up the system as shown in the figures below. More importantly, make sure that you set the instruments up such that you can observe the frequency response properly as well as capture the overshoot of your system.

- x Do not forget to press the play buttons to start the instruments!

Unfortunately, due to reasons mentioned above, the Bode plotter cannot be used at the same time as the oscilloscope. Therefore close the Bode plotter once you are done with the AC analysis and instead use the oscilloscope (Scope, Figure 4.3.1, from instrument launcher) to look at the waveforms.

For the oscilloscope you want to select the Channel 0 to look at AI0+ signal (i.e., the vSrc) and Channel 1 to look at AI1+ signal (i.e., the vOut). Choose timebase to 200 us/div and scale can be something like 200 mV/div.

Verify that the circuit behaves as expected by comparing simulated and measured results.

- x Measure the overshoot in the step response.
- x Are there any differences between simulated and measured? Why?

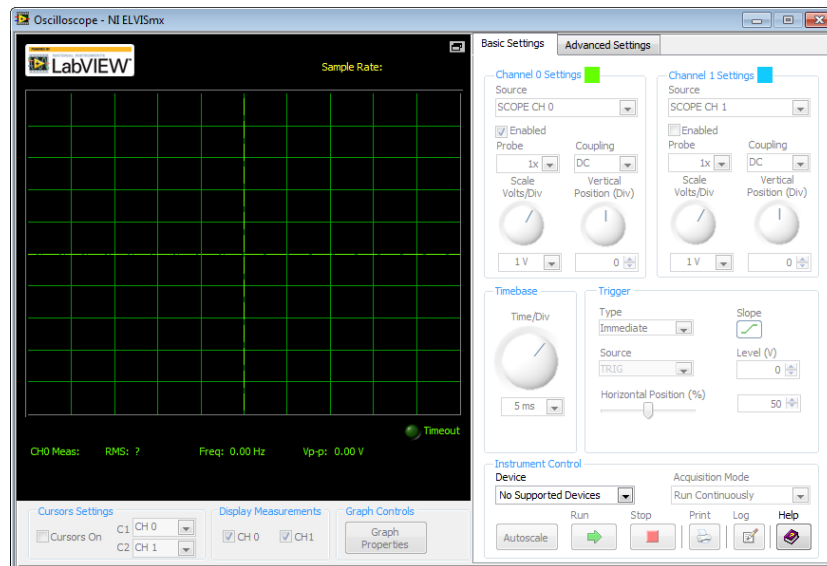


Figure 4.3.1: Oscilloscope screenshot. Choose channels on top.

## 5 Examination

To pass the laboratory, you need to show the following to the LA.

- There is a set of questions in the manual (blue boxes). Show the answers.
- Show all the captured waveforms and your setups, both software and hardware.
- Report the resistor value of R9 that gave you a phase margin of 45 degrees.
- How large, in per cent, is the overshoot in the pulses? Does this comply with theory and simulated results?

x As mentioned before, copy the waveforms into a nice document where you can quickly show the results without having to rerun everything.

## 6 Concluding remarks

Now, before laboratory 3, you should know how to:

- Run the simulator and understand transient and AC analysis
- Set up a hardware prototype and apply stimulus and measure outputs
- Dimension a feedback
- Derive phase margin, bandwidth and DC gain, in both time and frequency domain
- Document your results





# -LAB3-

## RIAA AMPLIFIER STAGES

### 1 Introduction

In modern stereo amplifiers you rarely find an analog input for a record player (phono) any more. Your exercise is therefore to design and build an RIAA-compensated<sup>2</sup> amplifier which then can be connected between the record player and the normal line input of the stereo.

x Notice that in this lab, you will not find the schematic per se in this manual, but a strategic search for e.g. "non-inverting shelving filters" would be a good start.

You will design an amplifier containing three amplifier stages. The first stage should be a transistor-based stage. The second and third should be opamp-based. It would also be practical to use two operational amplifier stages that are architecturally identical, but with different component values. Even though it is not optimum from a noise point of view, set them to have the same gain (approximately).

x Work in parallel! You are at least two people in the group - work in parallel to speed up the process and improve the learning curve.

---

<sup>2</sup>Search on the internet why an equalization is needed - it is quite a fascinating story.



## 2 Specification

For the RIAA equalization (once again search on the web resources for a closer description) we require the following specification:

- **Input sensitivity:** 1 mV
- **Input resistance:** > 4 kOhm
- **Total gain at low frequencies:** 64 dB
- **Output resistance:** < 1 kOhm
- **Supply voltages:** -/+ 15 V

- The amplifier should be designed with - in mentioned order - **one transistor stage** (NPN, BC546B) and **two operational amplifier stages** (UA741CD).

The RIAA compensation/equalization, i.e., placement of important poles and zeros, should be accomplished with the operational amplifier stages (i.e., do not include the single-transistor stage in this case). See Figure 2.1 for an example of the RIAA equalization curve. It is perhaps not very obvious from this picture, but it contains two poles and one zero.

- x Where are the poles and zero located in the RIAA equalization curve?
- x How do you guarantee the specified output resistance?

The rest of the gain and suitable input resistance is accomplished by the transistor stage, i.e., the transistor stage should be placed at the input of your design.

- Use the BC546B BJT transistor
- Use the uA741 operational amplifier.

- x Think of how you should isolate the potential DC offsets between amplifier stages (where applicable)!

- x The design must first be verified in the simulator, otherwise debugging can become very, very tedious.



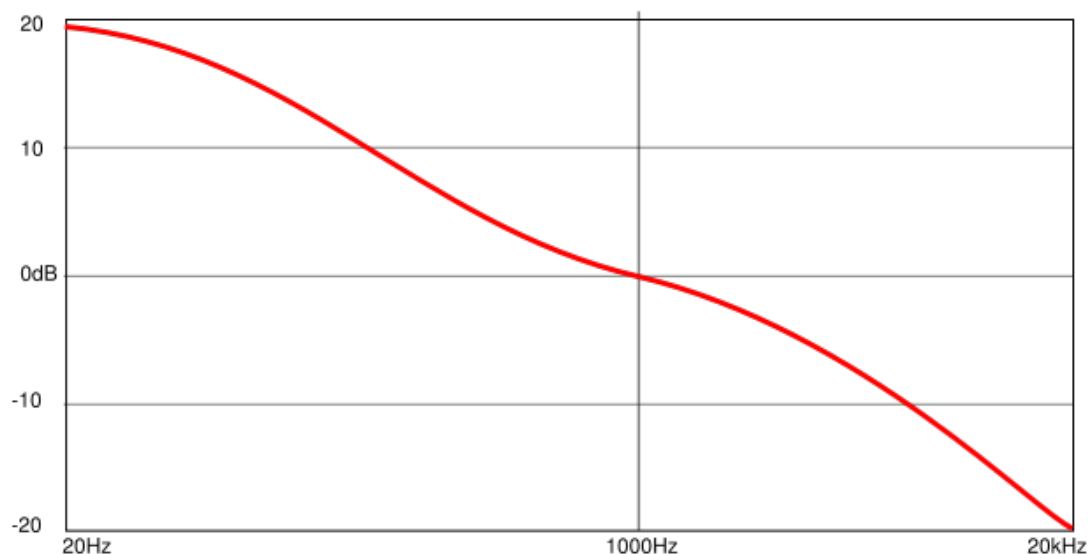


Figure 2.1: Example of RIAA equalization characteristics [source: wikipedia]

### 3 Simulation

The concept of verifying your design is quite clearly given by the previous labs. You need to define the amplifier stages in Multisim and verify the transfer function as in for example Figure 2.1 (spiced with the specification bullets in Section 2.

- x How can you measure the input resistance in the simulator?
- x How can you measure the output resistance in the simulator?

Dimension the (overall) amplifier such that you get as close as possible to the specification - with margin. Make sure the circuit is not oscillating. Aim for a reasonable phase margin.

#### 3.1 Bill of material

Create a BOM, bill-of-material, i.e., a list of components and hand to the lab assistant such that components can be "ordered". The BOM can be generated by the tool itself and will give you an excel spreadsheet that you can show the assistant.

- Capacitance values are taken from the E6 series (Google)
- Resistance values are taken from the E12 series.



## 4 Measurements

You have access to the positive and negative +/-15 V supplies on the ELVIS+ board.

In the measurements verify the functionality with sine wave generator (FGEN) in ELVIS+ and the Bode plotter. Very same way as you did in Lab 2.

Once you have verified that theory, simulation, and measurements agree, ask the lab assistant for help to connect your amplifier to the stereo such that your design can be compared to a reference amplifier.

## 5 Resources

Much of the information is most conveniently found on the internet.

- Google: [RIAA equalization Bode plot](#)
- [http://www.onsemi.com/pub\\_link/Collateral/BC546-D.PDF](http://www.onsemi.com/pub_link/Collateral/BC546-D.PDF)
- [http://pdf.datasheetcatalog.com/datasheet/philips/BC546\\_547\\_3.pdf](http://pdf.datasheetcatalog.com/datasheet/philips/BC546_547_3.pdf)
- <http://www.ti.com/lit/ds/symlink/lm741.pdf>

## 6 Examination

### 6.1 Step 1

In order to avoid coming back to the lab on off-hours in case something was forgotten, you would have to show your intermediate results such that the lab assistant can guide you towards a robust design (i.e., show the BOM).

Further on, address the blue boxes in the manual.

In order to eventually pass the laboratory/mini project, the amplifier must work sufficiently good and a report must be handed it containing schematics, chosen values, measured and simulated results as well as motivation, conclusions, etc. See below.

## 7 Report template

There is no template for the report. You have to convince the examiner that you should pass the course. However, one should obviously follow good writing procedures, such as for example:

- <http://www.liu.se/cmts/kursutbudet/tfke33/lathund-for-rapportskrivning/1.71024/Lathund2006-01-20.pdf>



Make sure to document as much as possible. Take window snapshots and save graphs, etc., to include in the report. You have to supply theoretical results, simulated results, and measured results. You should also comment on the audio quality.

x Insert an executive summary of your results on the first page of your report!