Lab 3

Analog Filtering

Authors:

Jonas Lussi, Naveen Shamsudhin, Alexandre Mesot, Michelle Mattille and Prof. Bradley J. Nelson

Multi-Scale Robotics Lab, Institute of Robotics and Intelligent Systems

Date: 2022

Version: 2.1

Summary: In this lab, we will introduce the concept of analog signal processing by building and testing circuits with discrete electronic components.

We will learn:

- How to generate periodic signals of arbitrary magnitude and frequency using the microcontroller.
- How to operate and use a digital oscilloscope to display and study signal waveforms.
- How to use discrete electronic components to build electrical circuits.
- How to design analog low pass filters according to application requirements.
- How to use the LM319N integrated circuit as a signal comparator and convert sinusoidal signals into square-wave signals.

3.1 Background

The perceivable *Newtonian* world around us is inherently analog in nature (discounting quantum-mechanical discretizations). The motion of objects is smooth with an infinite number of steps or in other words with a theoretically infinite resolution. This assumption of inherent-continuity in systems allows us to use a wide repertoire of mathematical tools to study them. In this document, the terms analog systems/signals will be used interchangeably with continuous systems/signals and they refer to electrical waveforms and signals.

Resistors, capacitors, inductors and transistors are the most common computing elements in an analog electronic circuit. Combinations of these computing elements can be used to develop building blocks for signal arithmetic, for signal filtering or to build complete control systems in the analog domain. Understanding the behavior of these blocks in the time and frequency domain is essential. Commonly used analytical tools in analog circuit design include differential equations, Laplace Transforms and bode plots.

The IRM lecture notes on **Noise and Signal Processing** provide additional background information for this lab.

3.1.1 Function Generators and Oscilloscopes

Function generators and oscilloscopes are indispensable equipments for measurement and test of electronic systems. As the name suggests, a function generator can be used to generate an electrical waveform. The minimum functionality of a function generator includes the ability to create periodic signals of a single frequency in sinusoidal, triangular and square patterns. Advanced function generators called arbitrary waveform generators can be programmed to generate electrical waveforms with complex spatial and temporal characteristics. In this lab, we will use the digital to analog convertors on the Featherboard to synthesize analog waveforms.

Oscilloscopes are used to visualize electrical signals on a display for test and analysis. The earliest variant of oscilloscopes called CRO or cathode ray oscilloscopes use electrostatic deflection of an electron beam to generate patterns on a phosphor coated screen similar to a television. The latest generation of oscilloscopes have the ability to display analog and digital signals (Mixed-Signal), to perform mathematical operations, to pause and save signals onto a non-volatile memory or computer for further post-processing. In this lab, we will use a dual-channel digital oscilloscope, Tektronix TBS1052B.

3.1.2 PCBs - Printed Circuit Boards

PCBs or Printed Circuit Boards are platforms to implement electronic circuits. It provides a place to position and mount different discrete components and interconnect them. PCBs are generally made of plastic with tracks of copper used to interconnect the different elements forming a circuit. Components are fixed in position either by soldering or by mounting them on pre-soldered holders. The Featherboard is essentially a PCB. For all general purpose electronic circuitry, the breadboard provides a convenient test and debugging platform. In the mechatronic design flow, once the electronic circuit has been tested and verified on the breadboard, the circuit layout may be transferred to a PCB for deployment.

If you would like to design and build your own PCBs, there are lots of resources available to students, like Autodesk Eagle (now included in Fusion 360) and cheap manufacturing services like `www.jlcpcb.com`.

3.2 Prelab Procedure

Note: The Prelab quiz on Moodle must be done before reporting to the lab. The submission deadlines are on Moodle. Late submissions will not be corrected. Each group member has to complete the quiz on their own, however you are allowed to work with students in your group to solve the quiz (and we absolutely recommend you to do so). Please remember to submit your quiz when you have answered all of the questions.

3.2.1 Comparator LM319N

The LM319N is an IC (integrated circuit) chip that is typically used as a voltage comparator. In the simplest configuration, it compares the amplitude (voltage) of an input signal with a reference/threshold voltage, and whenever the input crosses the reference , the output signal switches between 0 volts and +5 volts (assuming our connected power supply is + 5 volts). In this lab, we configure the LM319N as a comparator with one reference voltages. A comparator with multiple reference voltages has a hysteresis and is called a Schmitt Trigger. The advantages of adding hysteresis is visually represented in Fig 3.1. In a Schmitt Trigger, a small hysteresis band with two reference voltages **Vtrigger HIGH** and **Vtrigger LOW** is added instead of a single reference voltage as in a simple comparator. However, in our case a simple comparator is sufficient.

Read the data sheet (can be found on moodle) of the LM319N and answer the questions about it on the Moodle quiz.

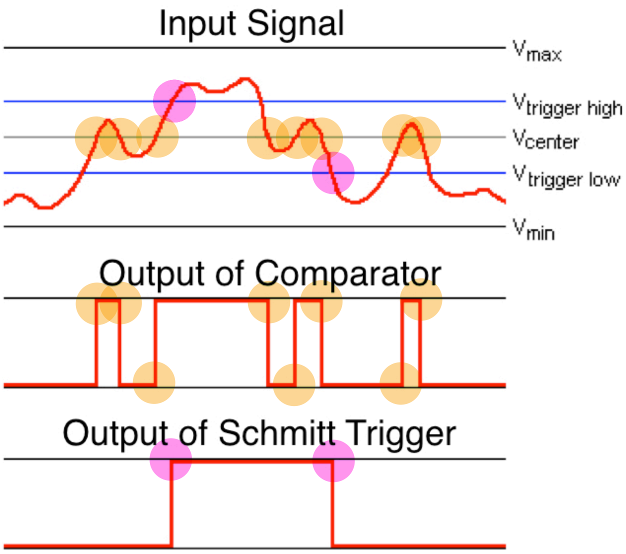

Figure 3.1: The comparator operation is shown in a time-series plot

3.3 Lab procedure

Note: Questions marked with **Postlab Qx** have to be answered as part of **PostLab 03**.

Note: Electronic equipment and components can be damaged if input voltage requirements are not met! Please be careful and follow the instructions. If you detect any burning smell, disconnect the featherboard and inform the assistant immediately!

3.3.1 Signal Generation and Display

1. Switch on the digital oscilloscope and examine its front panel. Locate the two signal input ports and the signal display panels 'Vertical', 'Horizontal' and 'Trigger'.
2. The Lab03.zip file contains code that will access the digital to analog convertor (DAC) on the microcontroller and generate continuous periodic waveforms. Compile and upload the given .ino file onto the microcontroller.

   Hint In order to be able to analogwrite on your feather board you need to install a new library in the Arduino IDE. Go to **Sketch -> Include Library -> Manage Libraries...**. Search for the **ESP32 AnalogWrite**, install it and close the window. This library will include the commands and functions that you need to send analog signals through the DAC.
3. The generated signal can be monitored on the oscilloscope. For this, connect the BNC cable to Channel 1 of the oscilloscope and connect the other end of the cable to DAC1 and GND on the Featherboard. Press the button *Autoset*, and the Oscilloscope will try to locate the signal. You should now see a stationary sine-wave on the oscilloscope screen.
4. What is the peak-to-peak amplitude (in volts) of the generated signal? (Hint: Use *Measure* Option and set the desired measurement quantities for the connected channel) **(PostLab Q1)**
5. What is the DC offset (in volts) of the signal? **(PostLab Q2)**
6. What is the frequency (in kHz) of the generated periodic signal? **(PostLab Q3)**
7. What is the execution speed (in kHz) of one iteration of the 'while' loop ? (Hint: Take the measured frequency of the sinusoid and combine this information with the sample number of the oscillation that you can find in the code) **(PostLab Q4)**

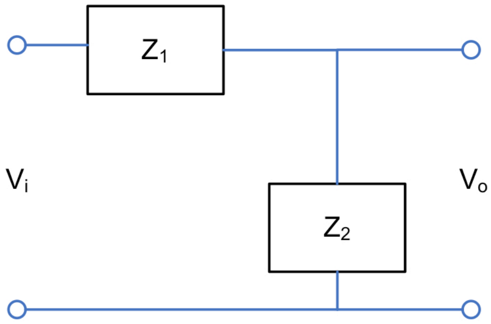

Figure 3.2: Generic Voltage Divider Circuit. Vi is the input voltage and Vo is the output voltage

8. What is the minimum and maximum amplitudes of the generated signal? Consequently what is the output voltage range and voltage resolution of the DAC? **(PostLab Q5)**

3.3.2 Implementing Analog Filters

In this part, you will implement analog filters using combinations of resistors and capacitors to modify a given input signal.

1. Resistive Divider Circuit
   - Use the color-coded resistors given to you to build a voltage divider circuit (as shown in Figure 3.2), which reduces by half the amplitude of the voltage waveform generated by the microcontroller. For obtaining the resistance in ohms from the color-code, you may refer to: `http://www.hobby-hour.com/electronics/resistorcalculator.php`. Observe the input and output waveforms on the two-channel oscilloscope.
   - Generate sinusoidal signals at frequencies of 20Hz, 40 Hz, 60 Hz, 80Hz, 100 Hz, 200 Hz and measure the output of the circuit at each of these frequencies. Write down the peak-to-peak values of voltage that you measure at the output. What kind of trend do you observe in the response of the circuit to inputs of varying frequency? Plot the ratio of output voltage to input voltage as a function of frequency. Also explain how/why you chose the delay in your program to achieve these frequencies. **(PostLab Q6)**
2. Resistor-Capacitor Network
   - Insert the given combination of resistor (Z_1) and capacitor (Z_2) as shown in Figure 3.2. Use the resistor with $R = 50\ \text{k}\Omega$.
   - Generate sinusoidal signals at frequencies of 20Hz, 40 Hz, 60 Hz, 80Hz, 100 Hz, 200 Hz and measure the output of the circuit at each of these frequencies. Write down the peak-to-peak values of voltage and the phase shift (in millisecs with respect to the input signal) that you measure at the output terminal. You can use the second channel to measure the phase shift. Plot the ratio of output voltage to input voltage as a function of frequency. Similiarly plot the Phase Shift in degrees as a function of frequency. Explain the trend observed in each plot. (Hint: Never trust what's written on components. Always measure to be sure. Also take the offset bias of the multimeter into account.) **(PostLab Q7)**
   - Describe what would happen if you switched the positions of the resistive and capacitive elements? What does this circuit do? **(PostLab Q8)**

3.3.3 Using an Integrated Circuit Chip - LM319 dual voltage comparator

We will use the LM319 as a signal comparator. The LM319 can convert sinusoidal signals into square waves. You will compare the sine wave from the Featherboard with the reference voltage of 1.65 volts.

1. The reference voltage for comparison is derived from the 3.3V supply voltage. Build a resistive divider circuit to generate a reference voltage of approximately 1.65 volts.

2. Use the microcontroller to generate a sinusoidal waveform of arbitrary frequency.
3. Take the LM319 IC from the components box. Take care not to damage the legs when plugging and unplugging it from the breadboard.
4. Plug the IC into the breadboard. Make sure the IC is properly oriented (note the dimple on the IC as shown in the data sheet). You may have to apply a slight even pressure with your fingers to bend the legs of the IC to fit it into the socket.
5. Locate the GND pin on the Featherboard and connect it to the respective pin on the LM319.
6. Connect the V_{EE} pin on the LM319 to the ground as well.
7. We need to ensure that the LM319 receives its power. Locate the `+5V` terminal on the Featherboard and connect it to the Vcc pin on the LM319.
8. Now, connect the 1.65V signal from your voltage divider as a reference signal to the LM319.
9. Next, you can connect your input signal containing the sine wave to the correct terminal.
10. Connect with a resistor (R = 50kΩ) the `OUTPUT 1` on the LM319 to +5V.
11. You can now observe the signal at the `OUTPUT 1` with your oscilloscope.
12. Please take a screenshot of the input and output signals displayed on the oscilloscope for your post lab report. **(PostLab Q9)**

3.4 Postlab and lab report

1. Upload a single PDF-file with your solution to moodle. The file should contain your answers (including plots and images) to the postlab questions Q1-Q9. Please upload your solution in time (before next lab session), late submissions will not be corrected.
2. Come prepared with the Prelab procedure for the next lab.