Lab 1

Analog Filtering

Authors:

Jonas Lussi, Naveen Shamsudhin, Tarik Rifai and Prof. Bradley J. Nelson

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

Date: 2020

Version: 1.5-C

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.

1.1 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.

1.1.1 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 SAM3X8E microcontroller 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.

1.1.2 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 UDOO board 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, you can work at the labs of the student organization BASTLI (`http://www.bastli.ethz.ch/`).

1.2 Prelab Procedure

Note: Prelab assignments must be done before reporting to the lab and must be turned in to the lab instructor at the beginning of your lab session. Additionally, you also have to upload your solution as a single PDF-file to moodle. Make sure to upload the file before your lab session starts, late submissions will not be corrected. The prelab needs to be handed in as a group. All the prelab tasks are marked with **PreLab Qx**.

1.2.1 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 1.1. 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 following questions:

1. How many separate voltage comparators are there on the LM319N? **(PreLab Q1)**
2. To which pin/pins on the LM319N must we connect the supply voltage? **(PreLab Q2)**
3. On which pin/pins can we obtain the output of the LM319N? **(PreLab Q3)**
4. Think of a real world application where you could use the LM319N IC. Explain it briefly (in less than four sentences). **(PreLab Q4)**
5. How much do you think a single LM319N IC costs? **(PreLab Q5)**

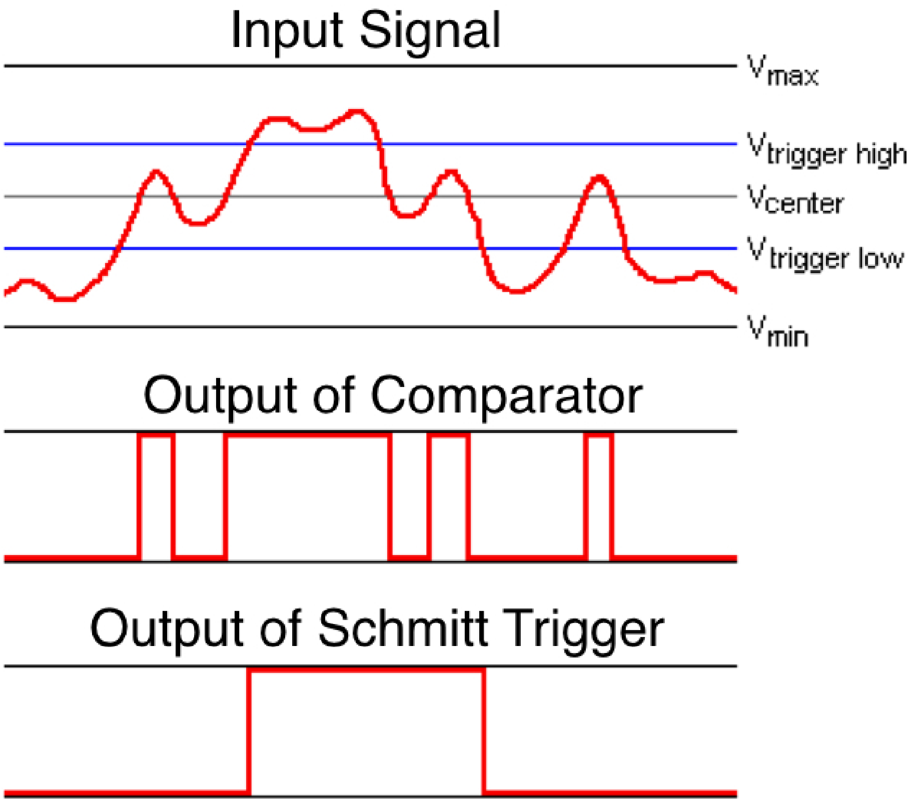

Figure 1.1: The comparator operation is shown in a time-series plot

1.2.2 Low Pass Filter - MATLAB

1. Using MATLAB, plot the bode plot (both magnitude and phase response) of a first order low pass filter with unity gain and a cut-off frequency of 100Hz. Make sure the plots have frequency in unit 'hertz'. **(PreLab Q6)**

2. Plot the impulse and step response of the same transfer function (You can use the functions *impulse()* and *step()*) **(PreLab Q7)**

   - What's the maximum phase shift introduced by the filter? **(PreLab Q8)**
   - What is the gain at the cut-off frequency? **(PreLab Q9)**
   - Can the filter be used to change the input frequency of your signal? Why or why not? **(PreLab Q10)**

1.3 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, inform the assistant immediately!

1.3.1 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.

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 UDOO.

4. What is the peak-to-peak amplitude (in volts) of the generated signal? (Hint: Use Measure Option) **(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 the 'while' loop ? (Hint: Use Horizontal display panel) **(PostLab Q4)**

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)**

1.3.2 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 1.2), 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 1.2.
   - 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. 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. **(PostLab Q7)**
   - Describe what would happen if you switched the positions of the resistive and capacitive elements? What does this circuit do? **(PostLab Q8)**

1.3.3 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.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.

1. Use the microcontroller to generate a sinusoidal waveform of arbitrary frequency.

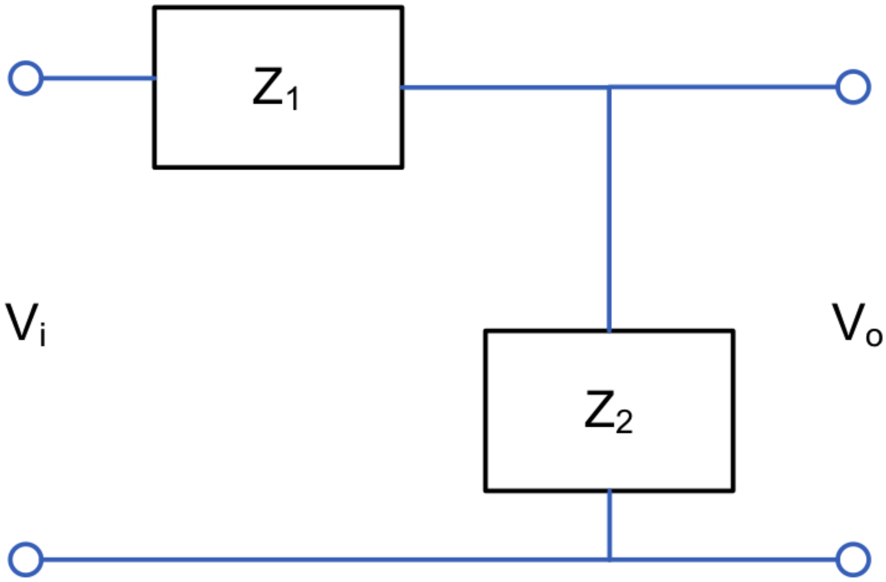

Figure 1.2: Generic Voltage Divider Circuit. Vi is the input voltage and Vo is the output voltage

2. Ask the assistant for a LM319 IC. The assistant will give you instructions on how to discharge static charge from your body so as not to damage the IC when handling it.
3. 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.
4. We need to ensure that the LM319 receives its power. Locate the +5V and GND terminals located on the UDOO and connect them to the respective pins on the LM319. Make sure that V_{EE} is additionally connected to the ground.
5. Connect the 1.65V reference as well as your input signal to the correct terminals. Make sure to connect all ground's correctly. To obtain and view the comparator output signal, we need an additional resistor connecting the comparator output to +5V. The signal can then be directly read from the Output terminal.
6. Please take a screenshot of the input and output signals displayed on the oscilloscope for your post lab report. **(PostLab Q9)**

1.4 Postlab and lab report

1. Show your working system to the teaching assistant.
2. Upload a single PDF-file with your solution to moodle. The file should contain your answers (including plots, images and MATLAB code) to the postlab questions Q1-Q15. Please upload your solution in time (before next lab session), late submissions will not be corrected.
3. Print the PDF-file and hand it in at the beginning of the next lab session.
4. Come prepared with the Prelab procedure for the next lab.