Lab course in IE1206 and IF1330

Spring 2021

Carl-Mikael Zetterling
Version: 2020-04-10

Table of Contents

Introduction

Background and motivation
Learning electric circuit theory (IF1330 electrical principles or IE1206 embedded electronics) can be challenging. It involves much mathematics, and you must learn both analysis and design of electric circuits. When designing circuits, approximate expressions and rounded-off values of the components is often necessary. Basic electric measurement equipment like the Multimeter, Oscilloscope and Waveform generator are introduced. For the understanding, it is necessary to do all of hand calculations, simulations, and measurements. The course is thereby important training in practical Engineering skills.

For the spring 2020 we were unfortunately unable to offer the measurement labs in Room 305, and we had to substitute the lab course with simulations only, using the software QUCS. This year (2021) we have developed a new lab kit that can be borrowed and used at home or elsewhere outside a specialized lab room. This involves some compromises in the selection of instruments. The Multimeter is a much simpler, but still an adequate version of what we have in Room 305. The Oscilloscope and Waveform generator is perhaps the biggest compromise. The BitScope in the Lab kit combines these two instruments, and it has severe limitations in bandwidth (max frequency), input voltage range and output impedance. This has made it necessary to select component values that will work properly under these limitations.

Rules
You may borrow a Lab kit with Multimeter, BitScope, and components in a foil bag. As it is difficult to get more BitScopes and Multimeters, we are not sure whether we will be able to offer them for purchase, like we did with the Digital Design Lab kits. If you break or lose the Bitscope you will have to pay for it, 400 kr. At the end of the course we will collect lab kits, date to be announced. Make sure to read through all appendices, and try the instruments out as directed to familiarize yourself with them. Note that you will need two USB ports on your computer, or a USB voltage adapter for the Arduino, for these labs.

Be sure to read the safety rules in Appendix 4 before starting with the measurements. Help will be available in Room 305 all afternoons in the normally scheduled weekdays, 3 to 5 pm. The lab assistants are the same as in Digital Design. If your schedule or other reasons do not permit you to come to Kista, we will also offer Zoom sessions the same hours.

Lab report is due May 17 and May 24
Write one lab report per person for all four labs, and submit it as pdf in Canvas.
You will be peer reviewing each other’s lab reports, and submitting a final version May 24.
Only Lab 4 needs to be shown to a Lab assistant (or recorded and the link sent to Per-Erik).
Write a short text describing your conclusions for each of the four labs.
Include schematics from the simulations, simulation results, measurement results and discussion about your results.

1 DC measurements

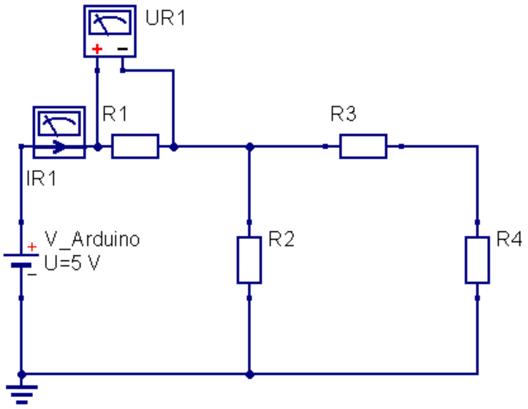
Equipment: Arduino, Multimeter

Components: Resistors, LEDs

QUCS: Simulate part 1.1

Preparation: Read Appendix 1 – 5, especially how to use the Multimeter

1.1 Resistive network with Arduino



Select 4 different resistors R1 – R4 in the range 1 kΩ – 47 kΩ.
Let the smallest be R1 and the largest be R2.
Simulate the voltage across and current through all resistors,
by adding voltage probes and current probes to all resistors

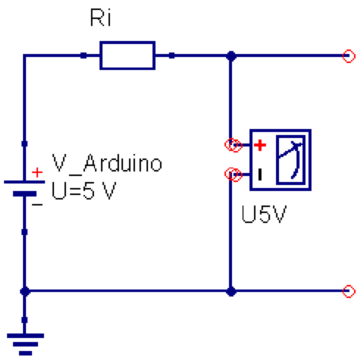
Build the circuit on the breadboard, and connect 5 V from the Arduino.
Using the Multimeter, measure all resistances, voltages and currents, fill in the table.

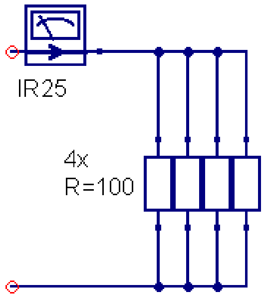
Comp.	Meas (kΩ)	Meas. U (V)	Meas. I (mA)	Calc (kΩ) $R = U/I$	Calc (mW) $P = U\,I$	Calc (mW) $P = U^2/R$	Calc (mW) $P = I^2\,R$
R1							
R2							
R3							
R4							

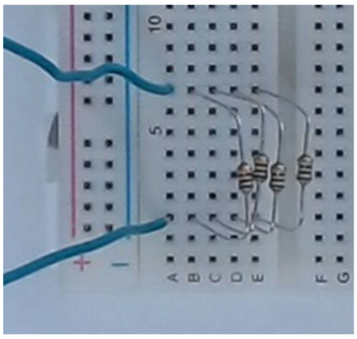
Compare the values to the simulated values, are they the same?
Calculate all resistances from measured voltage and current. Do they agree?
Confirm KVL in the loops U – R1 – R2 AND R2 – R3 – R4.
Confirm KCL in the node between R1, R2 and R3.
Calculate the resistor power losses from voltages and currents.
Calculate the power from the Arduino and confirm the power balance.

1.2 Thevenin equivalent for Arduino 5 V and 3.3 V outputs

Connect four 100 Ω resistors in parallel. What is the resistance value? R =
If 5 V is applied to this resistance, what is the power per resistor? P =





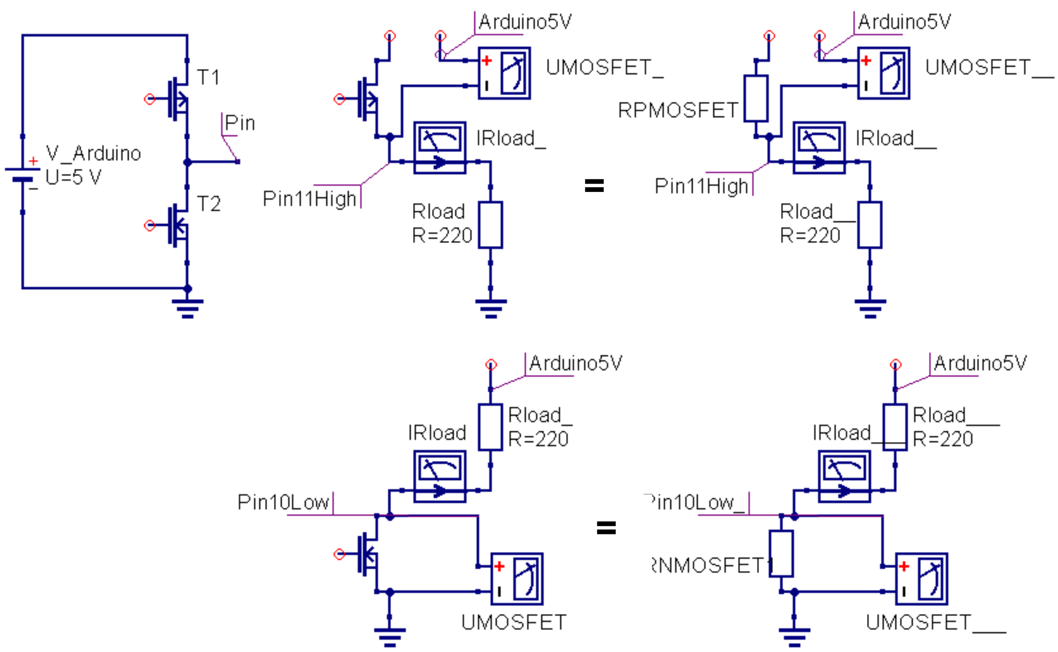


Using the Multimeter, measure the 5 V source on the Arduino, U(5V unloaded) =
Now connect the combined resistor, and measure the voltage again. U(5V load) =
Calculate the unknown internal resistance Ri from the two measurements using voltage division, Ri (5V) =

Measure the current through the combined resistor load, I =
We can now calculate the internal resistance Ri as the voltage difference between the two measurements, divided by the measured current. Ri (5V) =

Now repeat this measurement on the 3.3 V output of the Arduino. What is Ri(3.3 V)?
Why do you think the internal resistance is so low for the 3.3 V source?

1.3 Resistance of MOSFETs on output pins

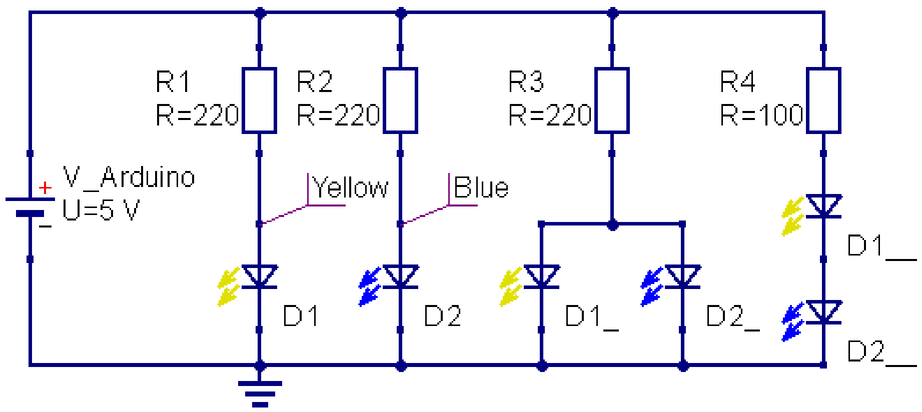


When an Arduino digital pin is designated as an output, it can be modeled as a PMOSFET (T1) and NMOSFET (T2) in an inverter push-pull configuration.
If the output is high, only the PMOSFET is active, and we can measure its on-resistance.
If the output is low, only the NMOSFET is active, and we can measure its on-resistance.
The Arduino outputs are rated for max 40 mA, so we will only load the output with a 220 Ω resistor, as shown in the figure. You also need to set the output pin high or low. Use the example code in the appendix that sets Pin 11 high and Pin 10 low.

Make two separate measurements to determine the resistance of the PMOSFET $R_{PMOSFET}$ and the NMOSFET $R_{NMOSFET}$ as in the circuits above.
The resistors in the right-most figures represents the on-resistance of the MOSFET.

Remember that we can measure the current two ways: with the current range on the Multimeter, or by measuring the voltage across the load resistor and calculating current as voltage divided by the known resistance. The second method is safer since it does not require separating the circuit for the measurement.

1.4 LED circuits



Try the 4 circuits above with one yellow LED and one blue LED (clear plastic).
Measure the voltage across the LED and resistor. What is the current through the LED?
Remember that the current can be calculated from the voltage measured over the resistor.
Explain why the blue LED does not light up when paralleled with the yellow LED.

2 Time-dependent measurements

Equipment: Arduino, BitScope, (Op amp buffer)

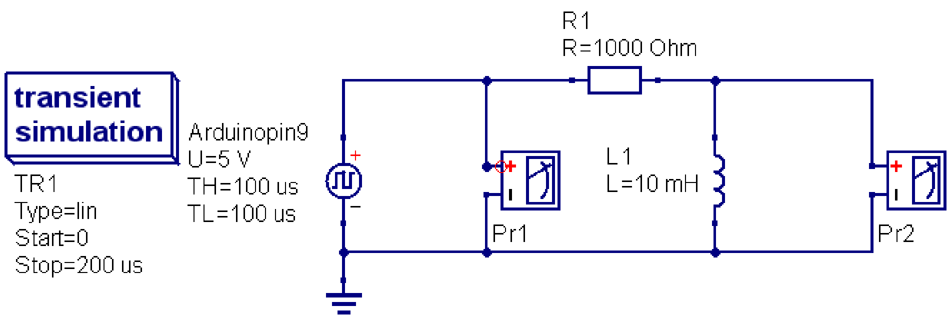
Components: Resistors, capacitors, inductors, diodes

QUCS: Simulate part 2.1 – 2.5

Preparation: Read Appendix 1 – 6, especially how to use the BitScope

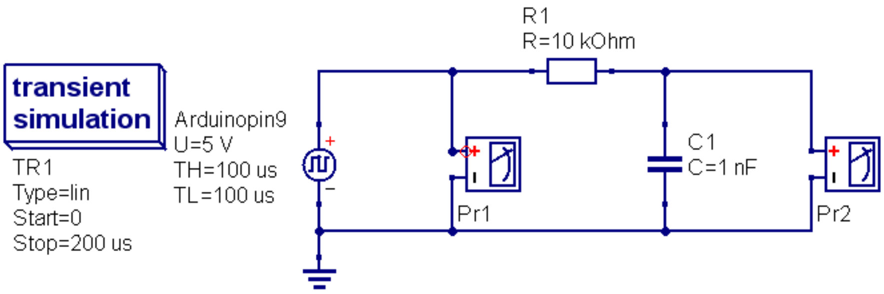
Note that in the text book time dependent circuits are shown using a switch that opens or closes at a certain time. You can try this using the provided switch in the lab kit. However, it is much easier to connect a square waveform generator to our circuit to repeatedly generate the time dependent voltage curves for our RL and RC circuits. We can only measure voltages with an oscilloscope, but currents can be calculated by measuring the voltage over a series resistor.

2.1 Time dependent behavior of RL circuits



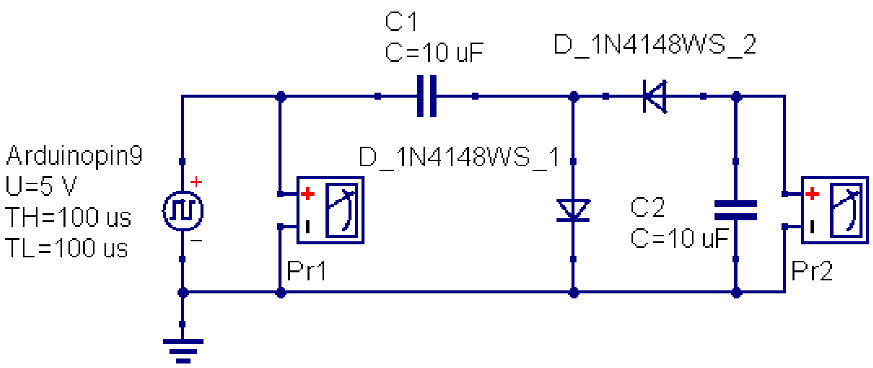
Calculate the time constant τ based on R1 and L1.
Simulate the RL circuit.
Include the simulation curve in your report.
Connect the circuit to Arduino pin 9, connect channel 1 and 2 of the BitScope.
The example code in the appendix outputs a 5 kHz square wave.
Measure the time constant τ using the BitScope.
Add the measurement points to the simulation plot to compare.
Try some different values of R (470, 1000, 2200 Ω).
Also try series and parallel connection of L (you have two inductors).

2.2 Time dependent behavior of RC circuits



Calculate the time constant τ based on R1 and C1.
Simulate the RC circuit.
Include the simulation curve in your report.
Connect the circuit to Arduino pin 9, connect channel 1 and 2 of the BitScope.
The example code in the appendix outputs a 5 kHz square wave.
Measure the time constant τ using the BitScope.
Add the measurement points to the simulation plot to compare.
Try some different values of R (4.7, 10, 22 kΩ).
Also try series and parallel connection of C with two capacitors.

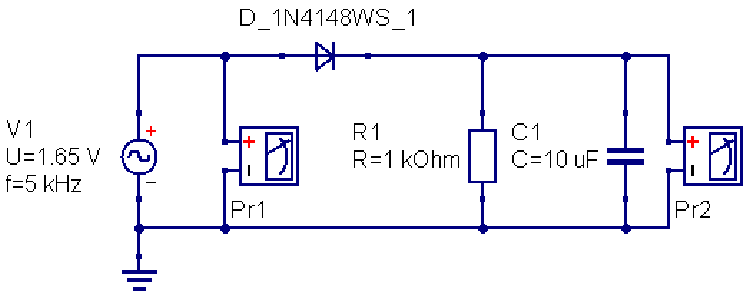
2.3 Negative voltage generator



In Lab 3 we will need a -5 V supply for our Op amps. This circuit shows how -5 V can be generated from +5 V using diodes and capacitors. As input signal use the same square wave as before from pin 9 of the Arduino. Use the blue unpolarized 10 uF capacitors.

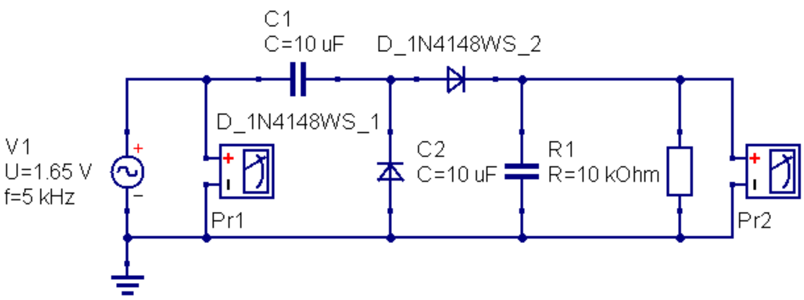
Use the diode model 1N4148WS from the library, and simulate for 100 cycles (20 ms).
Ignore the Warnings.
Include the simulation curves in your report.
What is the voltage at C2 where it connects to the diode?
Measure the output voltage of C2 using the multimeter and compare it to the simulation.
Add the measurement points to the simulation plot to compare.
Why is it not -5V?
Try connecting an LED and resistor.

2.4 Rectifier circuit with capacitor and resistor



Simulate the circuit 1000 us. Try it with and without the capacitor.
Include the simulation curves in your report.
Connect the BitScope waveform generator via the Op amp buffer circuit in appendix 6.
Try it with and without the capacitor.
Add the measurement points to the simulation plot to compare.
Do you get similar behavior?

2.5 Voltage doubler



Simulate the circuit 2000 us. Try it with and without capacitor C2.
Connect the BitScope waveform generator via the Op amp buffer circuit in appendix 6.
Try it with and without the capacitor.
Add the measurement points to the simulation plot to compare.
Do you get similar behavior?

References

https://www.allaboutcircuits.com/projects/build-your-own-negative-voltage-generator/
https://www.electronics-tutorials.ws/blog/voltage-multiplier-circuit.html

3 AC measurements

Equipment: Arduino, BitScope, Op amp buffer

Components: Resistors, capacitors, inductors, piezo speaker

QUCS: Simulate part 3.1 – 3.4

Preparation: Read Appendix 1 – 6, especially how to use the BitScope

3.1 Internal resistance of BitScope waveform generator with Op amp buffer

Derive the equation for internal resistance R_i as a function of

- Unloaded voltage signal $V_{pp,U}$
- Loaded voltage signal $V_{pp,L}$
- Load resistance R_L

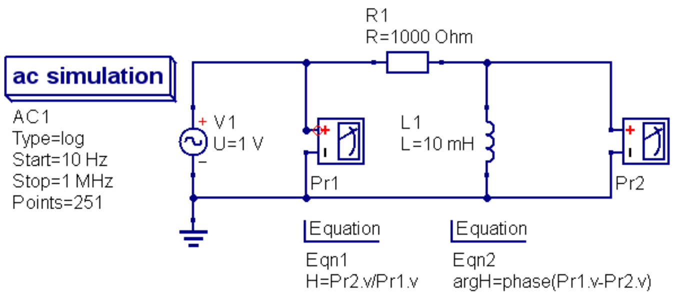
$$R_i = R_L \frac{V_{pp,U} - V_{pp.L}}{V_{pp.L}}$$

Measure the BitScope waveform generator without and with a load resistor.
Start with 1 kΩ. Estimate the internal resistance:

Measure the BitScope waveform generator and Op amp buffer without and with a load resistor. Start with 220 Ω. Estimate the internal resistance:

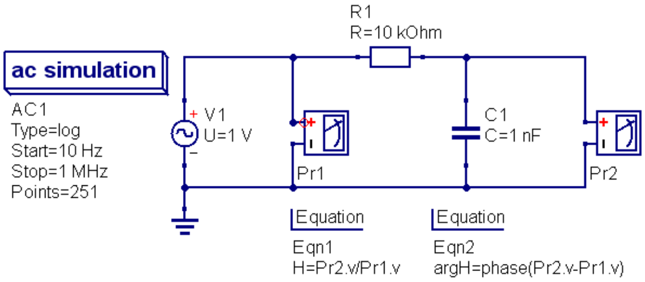
It doesn't matter if you use step (square), ramp (triangle) or tone (sine) wave. See appendix 5 for help with the connection and use of the BitScope.

3.2 RL low pass and high pass



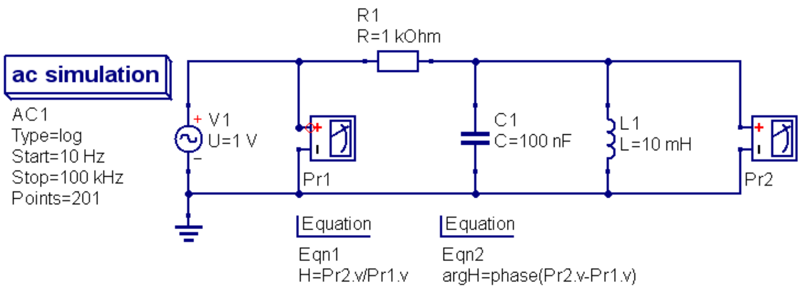
Calculate the cutoff frequency f_c based on R1 and L1.
Change R1/L1 so that your f_c is instead your Month of Birth in kHz (ex: April = 4 kHz).
Simulate the RL circuit vs frequency and plot H = Pr2/Pr1 and argH = phase(Pr1.v-Pr2.v).
Include the simulation curve in your report.
Is it a low pass or high pass filter?
Connect the circuit to the Op amp buffered AWG. V1 = pin 7 of the Op amp buffer.
Connect channel 1 and 2 of the BitScope.
Measure the cutoff frequency f_c using the BitScope by testing some different frequencies.
Click on the frequency and drag left or right to change the frequency.
Add the measurement points to the simulation plot to compare.
Also reverse the connection of R1 and L1 and redo the simulations and measurements.

3.3 RC low pass and high pass



Calculate the cutoff frequency f_c based on R1 and C1.
Change R1/C1 so that your f_c is instead your Date of Birth in kHz (ex: 27th = 27 kHz).
Simulate the RC circuit vs frequency and plot H = Pr2/Pr1 and argH = phase(Pr1.v-Pr2.v).
Include the simulation curve in your report.
Is it a low pass or high pass filter?
Connect the circuit to the Op amp buffered AWG. V1 = pin 7 of the Op amp buffer.
Connect channel 1 and 2 of the BitScope.
Measure the cutoff frequency f_c using the BitScope by testing some different frequencies.
Click on the frequency and drag left or right to change the frequency.
Add the measurement points to the simulation plot to compare.
Also reverse the connection of R1 and C1 and redo the simulations and measurements.

3.4 RLC band pass filter



Calculate the cutoff frequency f_c based on L1 and C1.
Simulate the RLC circuit vs frequency and plot H = Pr2/Pr1 and argH = phase(Pr1.v-Pr2.v).
Include the simulation curve in your report.
Connect the circuit to the Op amp buffered AWG. V1 = pin 7 of the Op amp buffer.
Connect channel 1 and 2 of the BitScope.
Measure the cutoff frequency f_c using the BitScope by testing some different frequencies.
Click on the frequency and drag left or right to change the frequency.
Also measure the bandwidth of the band pass filter.
Add the measurement points to the simulation plot to compare.

4 Operational Amplifiers

Equipment: Arduino, BitScope, Op amp buffer

Components: Resistors, capacitors, inductors

Preparation: Finish Lab 1 – 3 first. Select one of the Labs below.

Select one and find the circuit solutions yourself:

- Build an active filter, Sallen-Key or Butterworth, LP or HP using designs in the book (Nielsen & Riedel) or using TI filter designer or QUCS filter designer
- Build a BP/Notch active filer using designs in book or TI filter designer
- Build a circuit with an op amp to connect a sensor to an Arduino
- Build a voltage regulator with a MOSFET, an Op amp and a LED as voltage reference
- Build an oscillator for sine waves. Use the square wave output from the Arduino to generate triangle and then sine.
- Build a window comparator for CMOS logic levels. Use LEDs of different colors to indicate High, Low and tristate. Use the dual amplifier as comparator.
- Build a Schmitt trigger using an Op amp
- Build an Op amp precision rectifier
- More instructions will be available in Canvas!

Appendix 1 QUCS basics

Download from http://qucs.sourceforge.net/

Start a new project

Draw a schematic

1. At least one source (DC, square wave, AC)
2. Add components like resistors, capacitors and inductors.
3. Diodes (1N4148_WS) and Op amps (TL081) can be found under Libraries
4. Add voltage and current probes
5. Rotate components etc by right clicking before placement
6. Right click a component to edit its properties
7. Connect all components with wires
8. Connect the ground symbol
9. Select a simulation (DC, transient or AC)
10. Set up simulation parameters (Edit properties)
11. Save the schematic
12. Run the simulation (F2)
13. Add graphs in the data window
14. Make changes and re-run the simulation

Right click the schematic and duplicate to make more schematics in the same project.
The schematic can be exported, see File -> Export as image

For component values you can use prefixes: m means milli, u means micro, n means nano, p means pico, k means kilo and M means mega

Some useful things to know for AC simulations
Insert equations to get new plots, for instance adding
H = Pr2.v/Pr1.v allows you to plot the transfer function H
argH = Pr1.v – Pr2.v allows you to plot the phase difference

To make a Bode Plot:
Select logarithmic x axis for the frequency
Use logarithmic left y axis for H (or define dbH = dB(Pr2.v/Pr1.v))
Use linear right y axis for argH

Look at the tutorials available from the sourceforge page or the program Help menu.

Appendix 2 Component knowledge

This appendix will help you identify the different components in the Lab kit. Also refer to the component and equipment list in a separate appendix. Some parts are not included, but are included as background knowledge for further studies.

Resistors

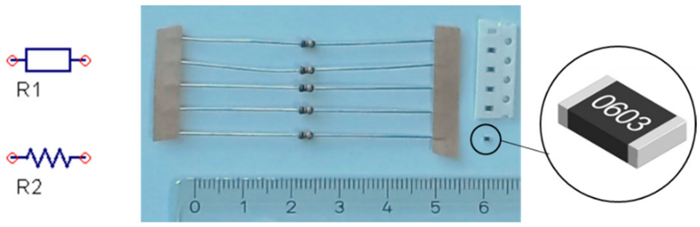
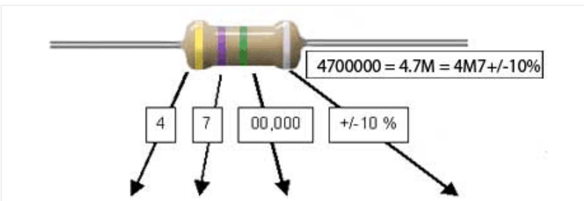


Figure: European (R1) and US (R2) symbols, lab kit and surface mounted resistor

Resistors are used to limit the current, create voltage dividers, or design filters. The unit for resistance is ohm with the symbol Ω. Resistance is created by restricting the current path to a smaller cross sectional area A in a length l of a high resistivity material with electrical resistivity ρ:

$$R = \rho \frac{l}{A} \text{ (unit ohm or } \Omega\text{)}$$

The physical size of a resistor does not correspond to its electrical resistor value, since higher resistance components can be made physically smaller. Size is standardized and is determined by ease of physical handling and power handling. All resistors in the Lab kit have the same physical size, but the resistance is marked on them using a color code. If you are unsure of a resistor value, you can also measure it using the Multimeter in the Lab kit.



Band	1	2	3	4
Meaning	1st Digit	2nd Digit	(No. of zeros)	Tolerance % (No band +/- 20%)
Silver			.00 (divide by 100)	+/-10%
Gold			.0 (divide by 10)	+/-5%
Black	0	0	No Zeros	
Brown	1	1	0	+/-1%
Red	2	2	00	+/-2%
Orange	3	3	,000	
Yellow	4	4	0,000	
Green	5	5	00,000	+/-0.5%
Blue	6	6	,000,000	+/-0.25%
Violet	7	7	0,000,000	+/-0.1%
Grey	8	8		+/-0.05%
White	9	9		

Figure: Color code for 5% resistors (4 bands), values in ohms

The resistors in the lab kit are all selected from the E12 series with 4 color bands and 5% precision in the range 100 Ω to 100 kΩ. For instance, a 1 kΩ resistor has the color bands Brown Black Red Gold. Read about color codes at electronics-tutorials. With 5% precision this means that its value can be between 950 Ω and 1050 Ω. Surface mounted resistors use numbers instead of color codes for identification. Resistors are available in a range from 0 Ω to GΩ, but the common range for electronics is 10 Ω – 1 MΩ.

Resistance in ohms is the ratio of Voltage U (V) to current I (A), R = U/I (Ohm's law)
The other important thing to know when selecting a resistor, is its power rating that can be calculated as

$$P = U \cdot I = \frac{U^2}{R} = I^2 \cdot R$$

All resistors in the lab kit have a power rating of 250 mW. For example, a 100 Ω resistor connected to 5 V, will have a power loss of 5^2 / 100 = 0.25 W, so all resistors can be safely used for the Lab. However, if you build with surface mount resistors, the power rating is smaller, the smaller the physical size, count on 100 mW or less.

Sometimes a precise value of resistance is required, and 1% resistors are not uncommon, but another way is to use a trimmer (smaller) or potentiometer (larger). These are resistors with variable resistance, usually designed so that it can be used as a voltage divider. The Lab kit includes a 10 kΩ trimmer.

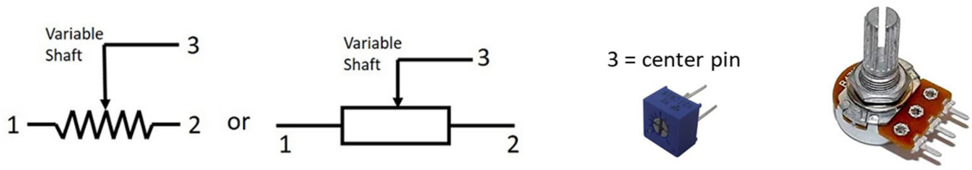


Figure: Symbol for variable resistor (US and Euro), trimmer from Lab kit, and potentiometer

Links
https://www.electronics-tutorials.ws/resistor/res_2.html
https://en.wikipedia.org/wiki/Electrical_resistance_and_conductance

Capacitors

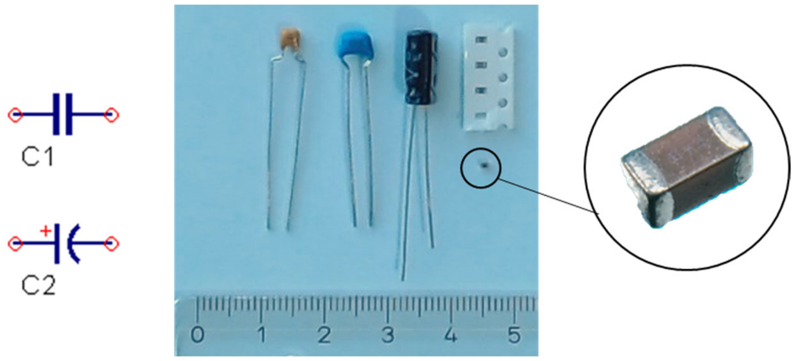


Figure: Unpolarized (C1) and polarized (C2) symbols, lab kit and surface mounted capacitors

Capacitors store electric charge, and are used for decoupling and filters. The unit for capacitance is Farad with the symbol F. Capacitance is created by connecting two conducting metal plates with area A, and separating them with an isolating material (dielectric) with thickness t and dielectric permittivity ε:

$$C = \frac{\varepsilon \cdot A}{t} \text{ (unit F)}$$

The physical size of a capacitor corresponds to the capacitor value since larger plates means higher capacitance. However, this is not obvious if you look at them. Capacitors from 1 pF to 1 µF are almost the same size. Many times the capacitors are made multilayered, and the number of layers are not visible. The Lab kit contains four capacitors of different sizes.

Unlike resistors, color codes are normally not used. Instead a number is printed with the value in pF, for instance 102 is 1000 pF, or 1 nF, and 104 is 100 000 pF or 0.1 µF. Larger values are usually labeled in µF (example 100 µF 10 V) but can also be like 106 which is 10 000 000 pF or 10 µF. Surface mount capacitors usually have lower voltage ratings, and are unmarked.

Some capacitors of large sizes are polarized, meaning that they have a marking how to connect the positive and negative terminal. This is also indicated in the schematic. It is important to follow this. If the capacitor is connected with the wrong polarity it will be damaged. The larger the capacitance, usually the lower maximum voltage rating. The capacitors in the Lab kit are rated between 10 V and 50 V. If you apply a voltage larger than this rating the capacitor can explode, and especially with electrolytic capacitors (the ones that are polarized) you need to be careful.

The precision for most capacitors is 10 to 20%. Unfortunately you can't measure it using the Multimeter in the Lab kit, but there are more expensive multimeters with capacitance measurement. Although there are capacitor trimmers, they are usually mostly used in radio circuits for trimming narrowband filters. Capacitors are available in the range 0.1 pF to 850 F, but the range for most electronics is 1 pF to 1000 µF. The highest values larger than 100 mF are referred to as Supercapacitors and can sometimes replace rechargeable batteries.

Links
https://www.electronics-tutorials.ws/capacitor/cap_1.html
https://en.wikipedia.org/wiki/Decoupling_capacitor
https://en.wikipedia.org/wiki/Capacitance

Inductors

Figure: Inductor symbol, lab kit inductor and structure of inductor

Inductors are used to store magnetic energy, and can be used in transformers, DC/DC converters and filters. Inductors are made by winding a conducting wire in a tight coil. The center of the coil can be air or plastic support, but for larger inductances an iron or a ferrite coil is used to couple the magnetic field more efficiently. The inductance value is not easily calculated since it depends on the radius and the winding of the wire, but it is proportional to the number of turns N squared and thereby the length of the wire used. The unit for inductance is Henry (H).

The inductors practically used are not ideal, they usually have a large resistance associated with the wire used. The current rating is important for inductors. The inductor in the Lab kit is 10 mH, rated for 100 mA, and it has a series resistance of 12 Ω. You can see that it is quite bulky. For large currents and small series resistance the inductor becomes quite bulky. To get a large inductance a ferrite or iron core can be added, but this increases the weight. Surface mounted inductors are usually ferrite based. Inductors are available in the range from 0.1 nH to 10 H, but the common range for electronics is quite small, 1 µH to 100 mH.

Inductors are either marked with its value in H, or a number like 103 for the inductor in the Lab kit, for 10 mH. The precision is usually 10%. We can’t measure inductance with the Multimeter in the Lab kit, but we can measure its series resistance. Inductors are seldom used for audio filters because they are so large. However, for radio frequency signals (RF), inductors are commonly used for filters and matching, and then they can even be made directly in the printed circuit board as a copper trace. A common use is in DC/DC converters, where they store magnetic energy as it is converted from one voltage to another.

Links
https://www.electronics-tutorials.ws/inductor/inductor.html
https://en.wikipedia.org/wiki/DC-to-DC_converter
https://en.wikipedia.org/wiki/Inductance

Diodes

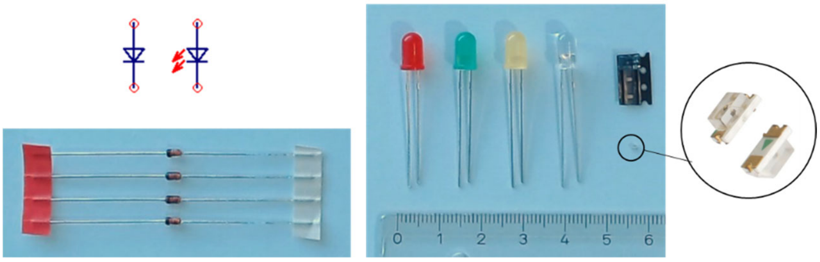


Figure: Diode and LED symbols, lab kit diodes and LEDs, and surface mounted diode

Diodes are used to rectify current. Another common use is in light-emitting diodes, or LEDs. They are made from a semiconductor pn junction, that conducts much more current in one direction than the other. The behavior is nonlinear. The anode (a) should be connected to the more positive voltage, and the cathode (c) to the more negative voltage. Usually there is a black or white band indicating the cathode end of the diode. For LEDs the anode pin is longer and the case may may have a bevel (flat) at the cathode pin.

LEDs are made from direct bandgap semiconductors. Since the wavelength of the emitted light is inversely proportional to its energy, different semiconductors are needed for different colors. The forward voltage drop when the LED emits light is also different for this reason. Silicon diodes intended for rectification typically have a voltage drop of 0.7 V, and this can be measured using the diode range of the Multimeter.

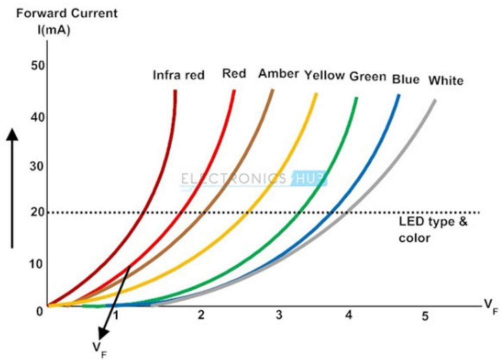


Figure: LED color versus forward voltage

In the Lab kit you have small signal rectifying silicon diodes called 1N4148, and LEDs of four different colors: red, yellow, green and blue. Be careful when testing the LEDs, a series resistor is necessary to limit the current. Use 220 Ω to 1 kΩ. The current or necessary resistor value can be calculated if you know the forward voltage drop of the diode at 10 – 20 mA, usually around 2 V for most LEDs, except the blue or white LEDs that are 3 V or higher. Example: with Vcc 5 V, and a 470 Ω resistor, the voltage over the resistor is 5 – 2 = 3 V, and the current will be 3 / 470 = 6.4 mA. A current of less than 20 mA is recommended. LEDs are available in several sizes, but the 3 mm and 5 mm round LEDs with clear or diffused/colored lens are the most common. There are also surface mounted LEDs and diodes.

Links

https://www.electronics-tutorials.ws/diode/diode_8.html
https://en.wikipedia.org/wiki/Light-emitting_diode
https://en.wikipedia.org/wiki/Diode

Operational amplifiers

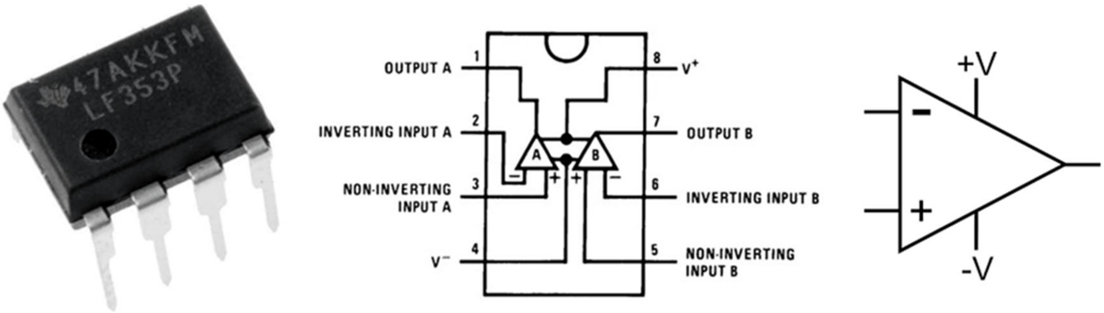


Figure: Lab kit, pinout and symbol for operational amplifier LF353

The operational amplifier, or op amp or opamp, is an integrated circuit consisting of a mix of JFET, MOSFET and bipolar transistors, diodes, resistors and usually one capacitor. In total there can be 30 – 60 components. In this course we will use it as an ideal amplifier without learning the details about the circuits inside or the transistors. If you want to learn more we recommend the course IE1202 Analog Electronics.

Op amps are found in IC packages of 8 – 14 pins, containing 1, 2 or 4 op amps. In the course we use the LF353 which is a dual JFET input op amp. The op amp has an output, and two inputs called the non-inverting (+) and inverting (-) input. To operate it a dual supply voltage of at least +5 V for Vcc and -5 V for Vee is required. Unused amplifiers should normally be connected so that they are inactive and draw less current. The LF353 will use 3.6 mA. We will use a special circuit to create -5 V from the +5 V from the Arduino.

Extra op amp information
Some non-idealities of the op amp are worth knowing. Most important is the limitation in output voltage. Most op amps are not rail-to-rail, meaning that the output voltage can only be between Vcc – 1.5 V and Vee + 1.5 V. For our op amp configuration it means +3.5 and -3.5 V. Bipolar op amps have an input current that makes the op amp present 10 MΩ at the input, but the LF353 has an input impedance of 10^12 Ω. The output resistance can be around 50 Ω, and maximal output current can be 20 mA. All op amps have a frequency limitation, the LF353 operates up to a unity gain bandwidth of 4 MHz. Output slew rate limits large signal output swings to 13 V/us.

There are other operational amplifiers that are called single supply, that only needs 5 V and ground. The drawback is that the output voltage swing is between 0 and Vcc, negative voltages are not positive. Therefore all signals must have a DC level centered around a virtual ground reference of half the supply voltage. One amplifier and two resistors can be used to create this virtual ground reference, but circuit design requires some extra care.

Links
https://www.electronics-tutorials.ws/opamp/opamp_1.html
https://en.wikipedia.org/wiki/Operational_amplifier
https://www.ti.com/lit/pdf/slos012 (LF353 Datasheet)
https://www.ti.com/lit/pdf/sloa030 (Single-Supply Op Amp Design Techniques)
https://www.ti.com/lit/pdf/sboa059 (Single-Supply Operation of Operational Amplifiers)

ICL7660 voltage converter

The integrated circuit ICL7660 that we use to convert +5 V to -5V for the op amp supply Vee uses external capacitors and internal switches and diodes. It operates by chopping up the 5 V supply at around 4 – 6 kHz, and this can generate noise with an amplitude of 20 mV on the power supply. Decoupling capacitors on the op amp are needed, and the larger the external capacitors, the better. It can only supply around 10 mA on the -5 V. See Appendix 6 for use.

Links
https://www.renesas.com/www/doc/datasheet/icl7660.pdf

Other ICs

Comparators
The comparator has the same symbol as the operational amplifier, but it has a simpler output stage that consists of one transistor that is on or off. We can see it as a digital output that is one if the + input is larger than the – input, and zero otherwise. A pull-up resistor is usually required to get the high voltage output of a one. An op amp without any feedback can be used as a comparator as well but is not as fast.

Digital 74HC series and Arduino
You used these in the course IE1204 Digital Design. The output of an inverter is similar to the outputs of the Arduino in that it has a PMOSFET connected to 5 V and an NMOSFET connected to GND. The Thevenin equivalent of digital outputs usually have quite low series resistance, and the 74HC 04outputs can drive 4 mA. The Arduino can drive 40 mA.

Voltage regulators
Linear voltage regulators are integrated circuits with a op amp and other components that can stabilize an output voltage to for example 3.3 V, if the input voltage is larger. The Arduino has a 3.3 V voltage regulator. The output resistance is very low as long as currents less than 150 mA is used.

Other parts

Figure: Piezotransducer and tactile switch with symbols

The piezotransducer has a sticker that states “Remove seal after washing” that is normally removed after soldering on a pcb and cleaning. It can be left in place (with lower volume) or removed (for higher volume). If you find the noise too loud you can connect a 100 Ω resistor in series. The long pin is +. The tactile switch is the same type that we used in Digital Design.

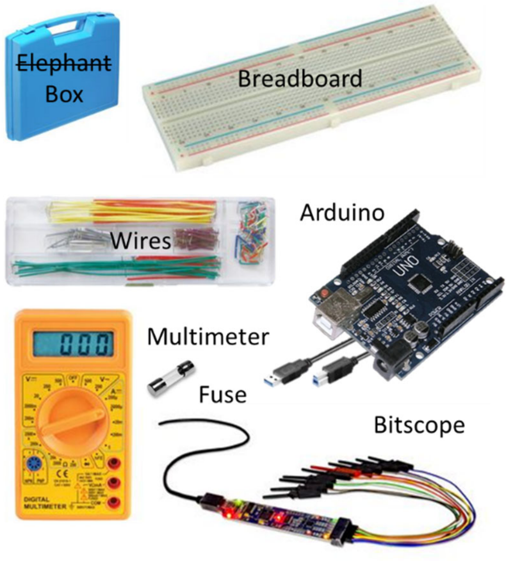
Appendix 3 Contents of the Lab kit

From the Digital Design Lab kit we will use

- Plastic Box
- Breadboard
- Jumper Cables
- Arduino with USB cable

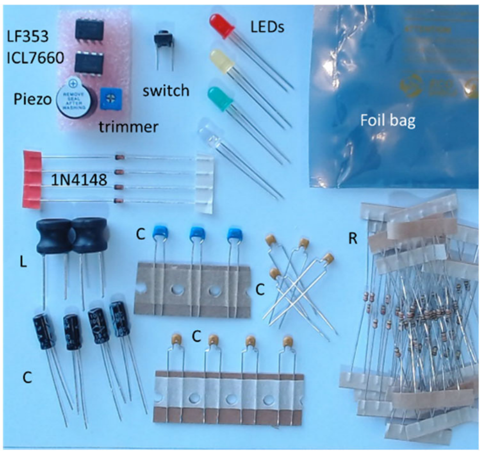
Two instruments are used in the Labs,
read Appendix 5 for instructions.

- Multimeter with manual and two test cables
- Bitscope with USB cable and 10 test clips



Components in foil bag, see Appendix 2 for more figures

#	Description	Comment
1	LF353P Op amp	on pink foam
1	ICL7660 (-5 V generator)	on pink foam
1	Piezo Transducer	on pink foam
1	Trimpot 10 kΩ	on pink foam
1	Tactile switch	
1	Red LED 5 mm	red lens
1	Yellow LED 5 mm	yellow lens
1	Green LED 5 mm	green lens
1	Blue LED 5 mm	clear lens
4	1N4148 diode	on tape
2	Inductor 10 mH 100 mA 12 Ω (103)	
3	Capacitor 10 uF 25 V (106) unpol	blue
4	Capacitor 100 uF 10 V	black
4	Capacitor 100 nF 50 V (104)	yellow
4	Capacitor 1 nF 50 V (102)	yellow on tape
5	Resistor 100 Ω 250 mW	on tape
5	Resistor 220 Ω 250 mW	on tape
5	Resistor 470 Ω 250 mW	on tape
5	Resistor 1 kΩ 250 mW	on tape
5	Resistor 2.2 kΩ 250 mW	on tape
5	Resistor 4.7 kΩ 250 mW	on tape
5	Resistor 10 kΩ 250 mW	on tape
5	Resistor 22 kΩ 250 mW	on tape
5	Resistor 47 kΩ 250 mW	on tape
5	Resistor 100 kΩ 250 mW	on tape
0	Extra fuse, 250 V 250 mA Fast, 20x5	Ask lab assistant



Appendix 4 Measurement safety

Caution. Do not connect any of the components or instruments in the Lab kit to a wall socket with 230 V AC! You can suffer electrical injury or death.

The exercises in these Labs all use +5 V from the Arduino, and sometimes -5 V generated on the breadboard. This voltage is safe to touch with your fingers and there is no safety hazard involved. You can hold the Multimeter probes to any component pins and it is not dangerous. However, to avoid breaking a component, disconnect +5 V when you are reconnecting your circuits.

More about electrical injury
The domestic power supply in our wall sockets is 230 V AC (alternating current) at 50 Hz. This voltage is large enough for currents of 50 mA or more through your skin, and that level is lethal if it passes through your heart. These current levels will cause your muscles to cramp so that you can't let go. Therefore, if you see someone get an electric shock and get stuck, first find a way to turn of the electricity (pulling the plug), otherwise you can get stuck too. Most wall sockets have a ground fault protection that triggers at 30 mA within 25 ms, before you sustain dangerous injury, but it only works if you touch one of the wires. If you touch both wires the electricity doesn't stop until you break the fuse at 10 A. Even if the ground fault protection triggers, you can sustain an injury if you fall down from the shock, for instance if you were fixing a ceiling lamp. Current levels as low as 10 mA are still very painful. If you have ever licked the terminals of a 9 V battery you know it is painful, and that is probably less than 1 mA. Never perform repairs if you are unsure.

Links
https://en.wikipedia.org/wiki/Electrical_injury

Appendix 5 Instruments

Multimeter

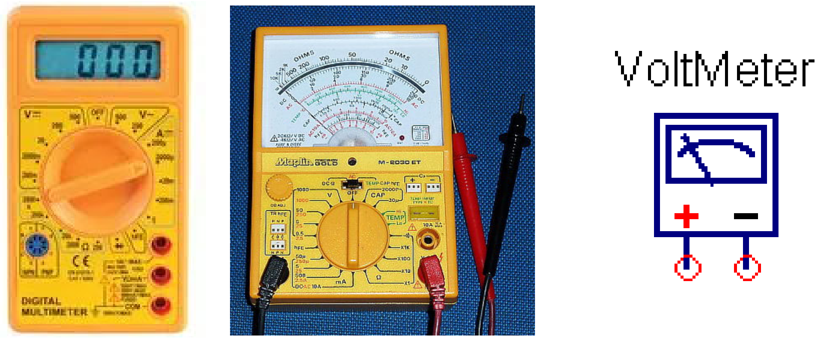


Figure: Lab kit Multimeter, old fashioned analog multimeter, and QUCS voltmeter symbol

The Multimeter is a versatile tool for several electrical measurements. In these labs you will measure voltage, current and resistance using a Digital Multimeter (DMM). Previously, analog multimeters were used, with a needle indicator, which explains the QUCS symbol.

The basis of the instrument is a voltmeter. It has a high input impedance, greater than 1 MΩ, so that it doesn't affect most measurements. You can measure voltages up to 500 V DC or AC, but unless you know what you are doing, keep to the 20 V DC range.

Current measurements are performed by connecting an internal shunt resistor $R_{instrument}$ in the Multimeter into the circuit as follows, and measuring the voltage across it with the voltmeter.

Range	Inserted resistance
200 mA	1 Ω
20 mA	10 Ω
2000 uA	100 Ω
200 uA	1000 Ω

There is a 500 mA fuse in the Multimeter in the current measurement mode. If the instrument is overloaded the fuse breaks, and you have to replace it to make current measurements.
All other ranges still work.

Resistance measurements are performed by forcing a small known current through the unknown resistance R_x and measuring the voltage with the voltmeter. A typical value of the current is 1.6 mA in the 100 Ω range, and then successively lower.

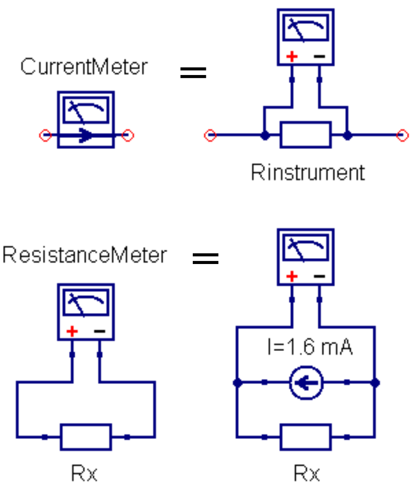


Figure: Internal circuits for current and resistance measurements

When measuring resistance, the resistor R_x must be disconnected from the circuit.

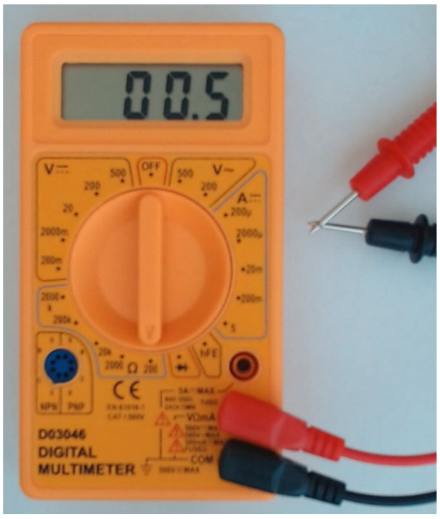
There is also a range to test diodes. It will show the forward voltage drop at 1 mA for signal diodes (1N4148, around 700 mV). LEDs can be tested, they shine dimly, but indication is "1". It is also useful for testing surface mounted LEDs. The hFE range is a bipolar transistor testing range, you will not need it. Expensive Multimeters may also have the possibility to measure frequency, capacitance and even inductance.

Testing the Multimeter

Connect the test leads, Red to “VΩmA” and Black to “COM”. As you switch on the Multimeter or switch between ranges it may briefly show an incorrect full scale value.

Figure: Multimeter briefly illuminates all segments when turned on

You can test the multimeter by setting the range to 200 Ω and shorting the probes together. It should then show around 0.5 Ω. Connect a known resistor (here 1 kΩ) to test resistance.



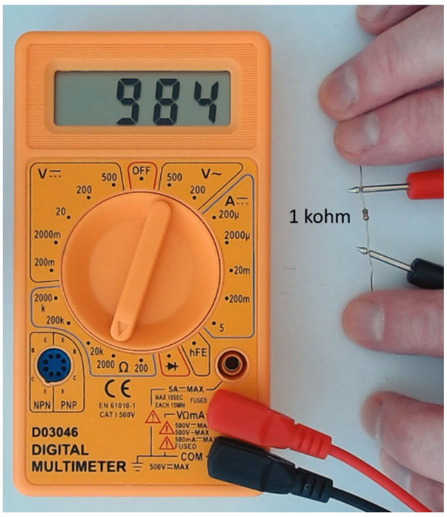
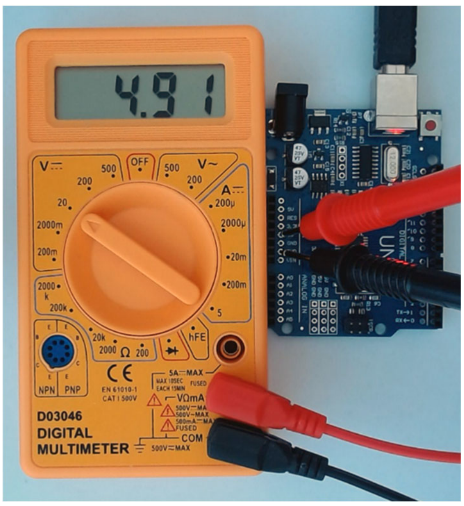


Figure: Connect probes together to test resistance meter, then use to measure resistors

Test the output voltage of the Arduino 5V in the 20 V range. It is not possible to test with the probes in the pin headers, but you can test it on the Arduino board as shown. Easier is to connect some wires and components to your breadboard. It can be difficult to measure in the holes in the breadboard, try to connect the probes to a component pin.



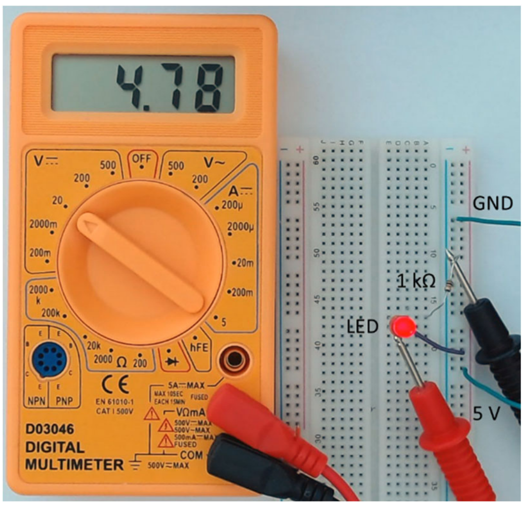


Figure: The Arduino 5 V can be measured on board, or using the breadboard and components

To test the current measurement, use a circuit with a LED and a 1 kΩ resistor. With the Multimeter in current range 20 mA, connect the resistor and LED. It will show around 3 mA.

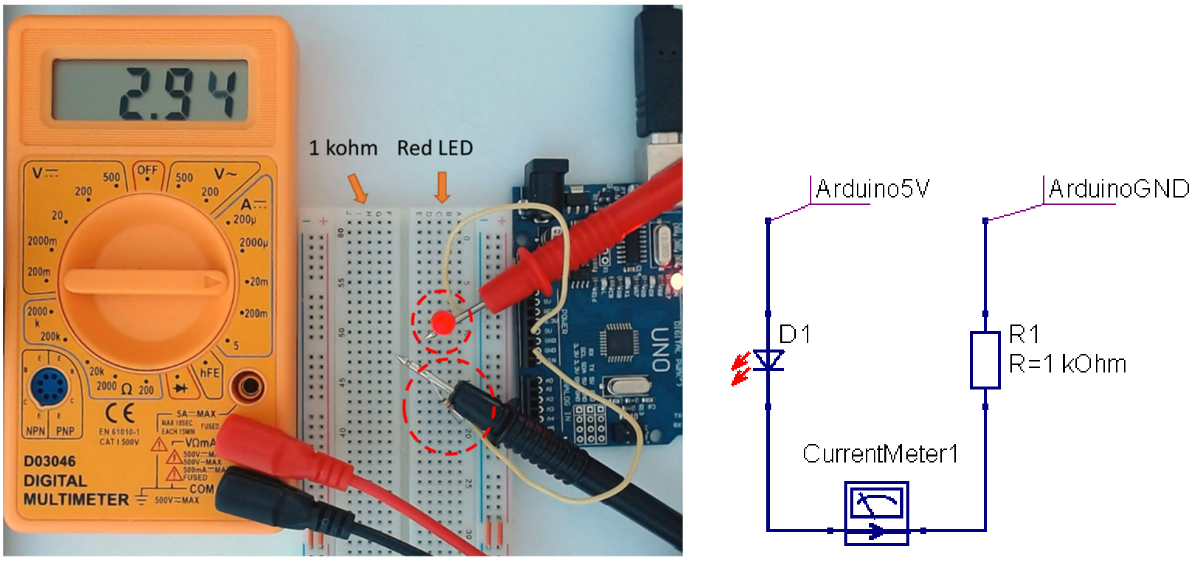


Figure: When measuring current, the Multimeter is part of the circuit

Practical Multimeter measurements

Try to make the measurements at the largest range possible so that you get more accurate results. As a rule of thumb you can't trust the last digit in the reading. Here are some common examples:

Nominal Voltage for Arduino: 5 V

Range (V)	Displayed voltage (example)	Comment
500	^{HV} 005	No need to use this range
200	04.9	Ok
20	4.90	Ideal range
2000 m	1 .	Over range
200 m	1 .	Over range

Output of Op amp while trimming Zero offset (no waveform generator input): 3 mV

Range (V)	Displayed voltage (example)	Comment
500	^{HV} 000	Too small for this range
200	00.0	Too small for this range
20	0.00	Too small for this range
2000 m	003	Ok
200 m	02.9	Ideal range

Resistance for a 1 kΩ resistor in Lab kit: 1000 Ω

Range (Ω)	Displayed resistance (example)	Comment
2000 k	001	Too small for this range
200 k	01.0	Too small for this range
20 k	0.98	Ok
2000	980	Ideal range
200	1 .	Over range

If you have the Multimeter in current or resistance range, and you connect it to a voltage, the instrument may be damaged or the fuse broken. As a habit, turn off the Multimeter between measurements to save battery, or set it to the 20 V range. If the fuse breaks while measuring current you can no longer measure current, but all other ranges still work on the Multimeter.

Remember that we can measure the current two ways: with the current range on the Multimeter, or by measuring the voltage across a resistor in the circuit and calculating current as voltage divided by the known resistance. The second method is safer since it does not require separating the circuit for the measurement.

Ask a lab assistant for help with a new fuse.

1. Turn off the Multimeter
2. Remove the back lid with the two screws
3. The replaceable 500 mA fuse is indicated in the figure, remove it
4. You can check the fuse with a Multimeter in the 200 Ω range. It should show 0.5 Ω

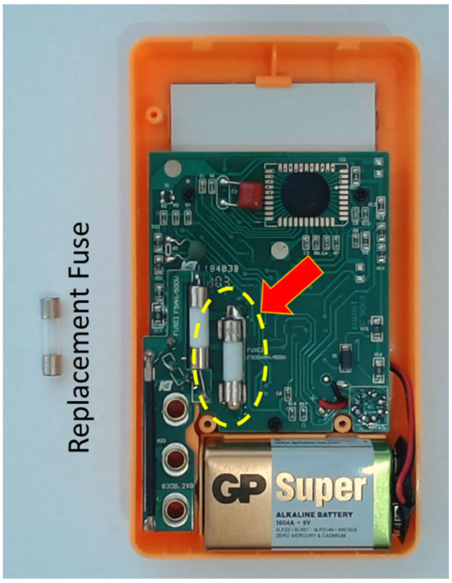


Figure: Replaceable fuse in Multimeter. The replacement is glass cased.

Links

https://en.wikipedia.org/wiki/Multimeter

Arduino

The Arduino will be used as a 5 V power supply. We will measure the internal resistance of its digital outputs. It can also be used to generate a square wave signal for lab 2. If you want to use it for embedded system applications, sensors can be connected to its analog input pins.

The 3.3 V pin comes from a 3.3 V regulator and is very stable. It has a very low equivalent internal resistance. The 5 V comes from the USB connector and can come either from the computer USB port, or from a USB adapter, so the internal resistance will vary. However, if the Arduino is powered from an external 9 V adapter, the 5 V comes via a 5 V voltage regulator with much lower internal resistance.

The digital pins D0 to D13 can be configured in software to be inputs or outputs.
When used as digital output pins, they switch 5 V or ground via a PMOSFET or a NMOSFET, respectively. They can drive up to 40 mA. You will measure the MOSFET resistance in Lab 2.
Don't use D0 and D1, as they are used to communicate to the computer via the USB.
D3, 5, 6, 9, 10 and 11 can have PWM outputs, or be used as normal input or output pins.
D13 is connected to an LED on the Arduino.

The Analog input pins A0 to A5 can be used for voltage measurements, with 10 bits resolution, resulting in a range 0 – 1023. They can also be programmed to be digital pins.
If the sensors have voltages << 5 V you may need an amplifier before connecting it to the analog pin. If the voltage is larger than 5 V you risk damaging the Arduino, unless you connect a voltage divider to protect the Arduino.

If you have never connected and programmed an Arduino from your computer, go to Arduino.cc and download the integrated development environment (IDE) and install it.
There are many tutorials and example codes here, or just a google away.

Arduino code to use in the lab to set pin 11 high, pin 10 low, and generate 5 kHz on pin 9:

```
void setup() {
  // initialize digital pins 9 - 11 as outputs.
  pinMode(11, OUTPUT);
  pinMode(10, OUTPUT);
  pinMode(9, OUTPUT);
  digitalWrite(11, HIGH); // turn pin 11 on (HIGH is the 5 V voltage level)
  digitalWrite(10, LOW);  // turn pin 10 off (LOW is the 0 V voltage level)
}

void loop() {
   tone(9, 5000); // generates a 5000 Hz square wave on pin 9
}
```

You may also use digitalWrite() and delayMicroseconds() on pin 9, but it is less precise:

```
  digitalWrite(9, HIGH);   // turn pin 9 on (HIGH is the voltage level)
  delayMicroseconds(100);  // wait for 100 microseconds
  digitalWrite(9, LOW);    // turn pin 9 off by making the voltage LOW
  delayMicroseconds(100);  // wait for 100 microseconds
```

BitScope

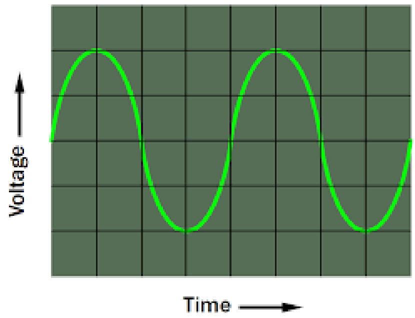
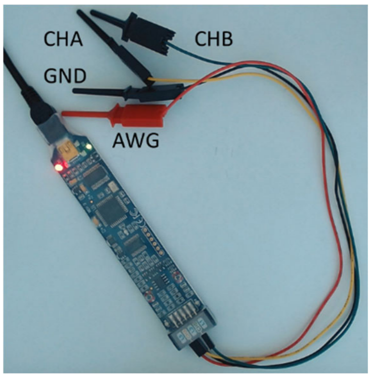


Figure: Ancient Oscilloscope > 20 kg, typical oscilloscope voltage vs time display

Oscilloscopes used to be large, heavy and expensive instruments, based around a cathode ray tube (CRT), same as in old televisions. A voltage could be measured, and via amplifiers it could deflect the electron beam of the CRT in the y direction. A separate voltage sweep from a time base varied the position on the x scale. The measurement is done repetitively to generate the picture, but the screen also has a phosphor to maintain the image. This way the voltage signal versus time could be studied. In modern digital versions several voltage measurements are stored in memory, and plotted on the computer screen, and the instruments can be made very small and inexpensive.

The BitScope is a dual channel digital storage oscilloscope. It also has digital inputs (with serial bus decoding), and a waveform generator. It connects to your computer via USB. You have to install the BitScope DSO software from http://bitscope.com/
If it doesn't work immediately, install the USB driver: http://bitscope.com/software/driver/
This is needed for Windows 10 for instance. There are also tutorials and manuals on this site.



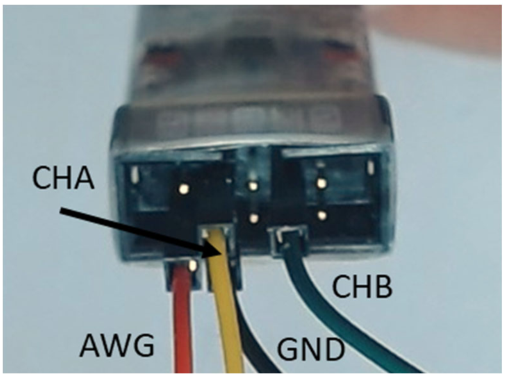


Figure: Bitscope with measurement clips connected

Apart from the two measurement channels CHA and CHB, we will also use the built in waveform generator (or AWG - Arbitrary Waveform Generator). Connect some of the BitScope probes to the pin header. Use Red for the AWG = Arbitrary Waveform Generator, Yellow for Channel A, Green for Channel B, and Black for GND (there are two GND pins, use either).

In the lab instructions they are referred to as channel 1 = CHA and channel 2 = CHB

BitScope specifications

The input amplifiers of the BitScope has standard input impedance of 1 MΩ and 10 pF. It is possible to connect 10:1 oscilloscope probes to the BitScope if an adapter is used. The BitScope is limited in input voltage range, so you have to be careful not to overload it.

Input voltage range: -6.5 V to +9.2 V
Resolution: 12 bits

Vertical scaling can be set from 20 mV/div up to 2 V/div. The cursors follow CHA scaling, so set CHA and CHB to same scaling if you want to measure CHB. These channels can also be run through a FFT to display the frequency spectrum of the signals.

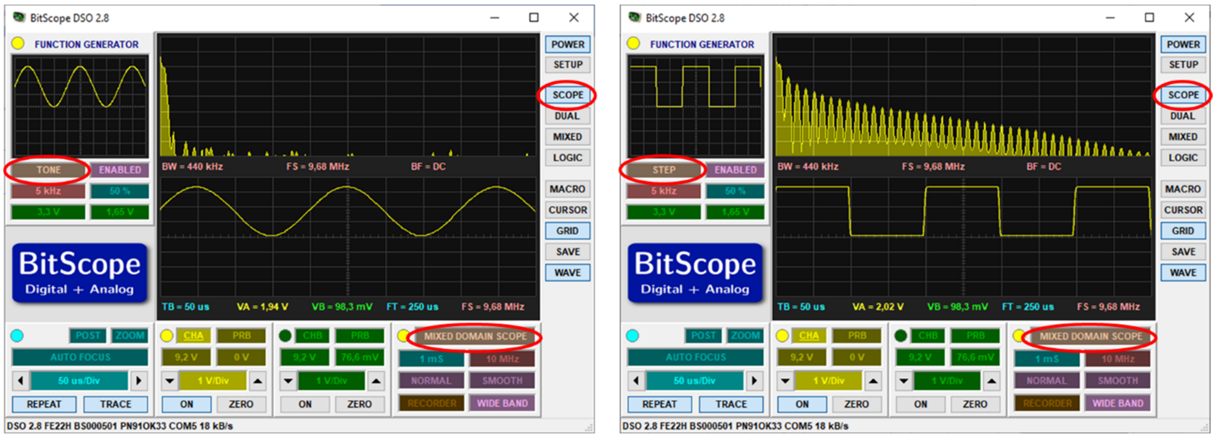


Figure: Frequency spectrum of sine wave and square wave in the BitScope

The digital channels L0 – L5 have 100 kΩ and 10 pF input impedance, and are limited to CMOS/TTL logic levels and a range of 0 - 5 V. L4 is used for AWG and L5 can be used as a digital clock signal. These inputs can also decode serial bus standards: UART, SPI and I2C.

The BitScope has a bandwidth of 20 MHz, and the time base can be varied from 100 ms/div to 1 us/div. However, the sensitivity is not as high in the highest frequencies, so I would use it mainly below 1 MHz.

The built in waveform generator is useful but slightly limited. The output impedance is around 500 Ω, but with the Op amp buffer it can be lowered to 50 Ω. The output amplitude is limited to positive voltages between 0 and 3.3 V, but this can also be adjusted with the Op amp and a DC block capacitor. This is discussed further in Appendix 5 and 6. Although the frequency range is nominally 5 Hz to 250 kHz, it is not usable above 50 kHz since it becomes heavily distorted. Connect it to CHA and test it. You can also connect the Piezo transducer to the AWG to listen to it in the audible range (20 Hz – 20 kHz) but the transducer responds best to the STEP (square signal). If it is too loud, leave the tape on the transducer, or connect a 100 Ω resistor in series.

Links

http://bitscope.com/ (main page)
http://bitscope.com/support/?p=quick (quick start)
https://docs.bitscope.com/KLTUKBXN/ (DSO manual)
http://bitscope.com/product/BS05/?p=specs (specifications)

Testing the BitScope

Connect a 100 kΩ resistor to the breadboard as shown. Connect the GND probe to one end, and the AWG and CHA probes to the other end. The 100 kΩ resistor will not affect measurements, so it can be left in place to hold the BitScope probes. Turn on Wave and right click the frequency to set 5 kHz. Right click to select TONE (sine wave), STEP (square wave) or RAMP (triangle wave). Turn on the Cursors and drag to aid in measuring the curves. The frequency is measured if you set the cursors on two equal locations of the waveform.

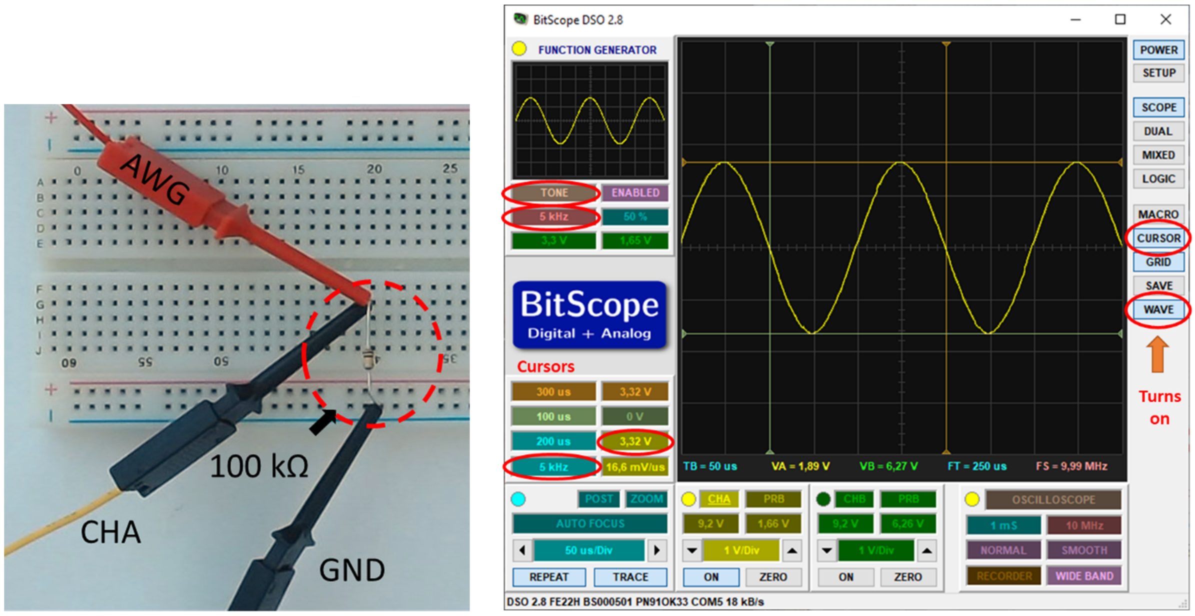


Figure: Breadboard with 100 kΩ resistor, BitScope DSO screen

You will measure the internal resistance of the waveform generator (Lab 3.1). This is done by connecting a 1 kΩ resistor in parallel with the 100 kΩ resistor. Note that the amplitude is reduced. Measure the peak-to-peak voltage (Vpp) using the cursors.

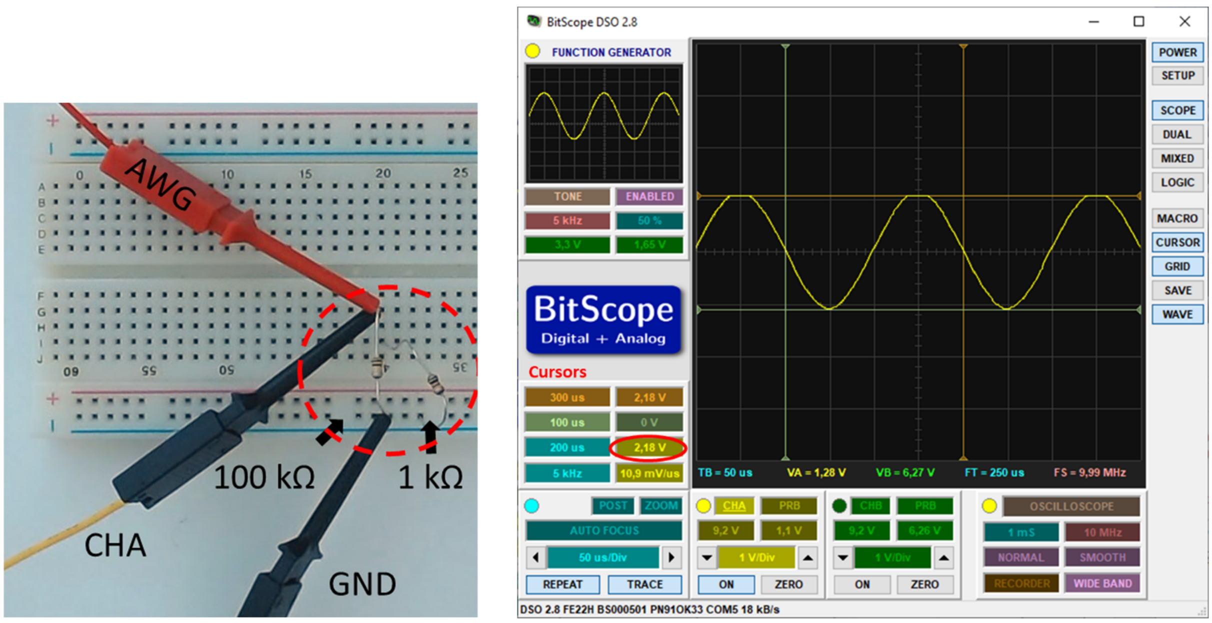
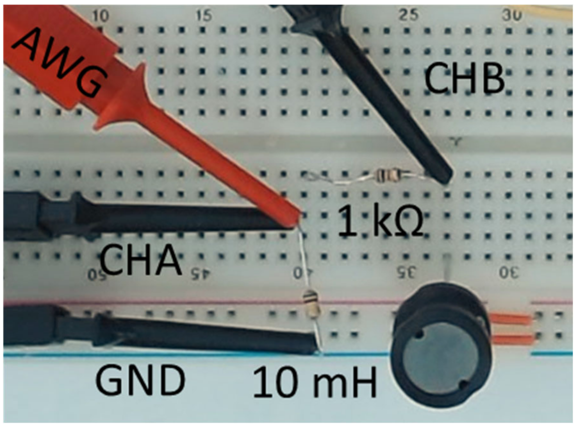


Figure: Breadboard with 100 + 1 kΩ resistor, BitScope DSO screen

Practical BitScope measurements

Now connect the circuit in Lab 2.1 and add CHB to measure the voltage across the inductor. Set BitScope to Dual to see CHB. Set the reference level to REF by right clicking (it is otherwise set to MEDIAN). If we use the Wavefunction generator in STEP mode, we can see that the input signal is affected (there are peaks on the CHA signal).



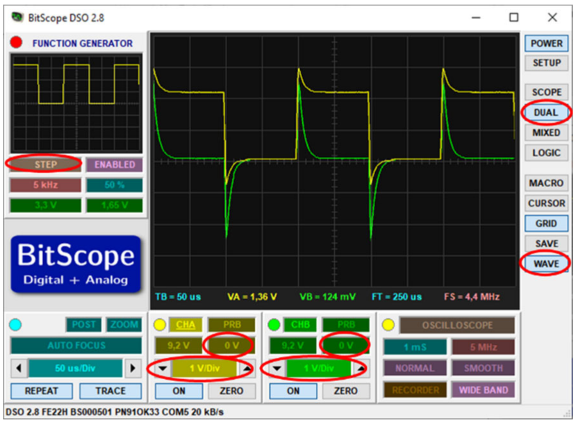


Figure: Breadboard with Lab 2.1 circuit connected to AWG of BitScope

This is because of the internal resistance, and we can model it in QUCS. We can also add the parasitic resistance of the inductor, R_L, even if it doesn't affect these measurements.

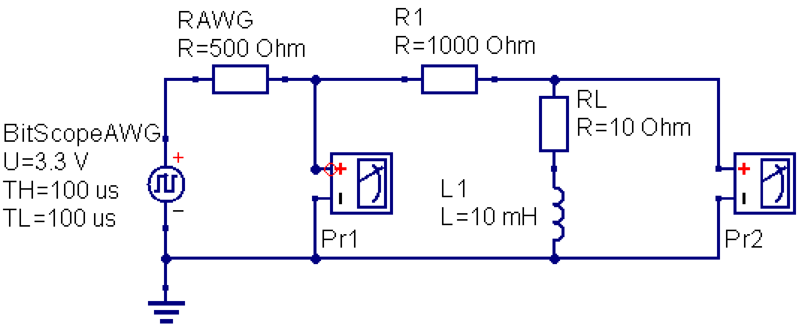
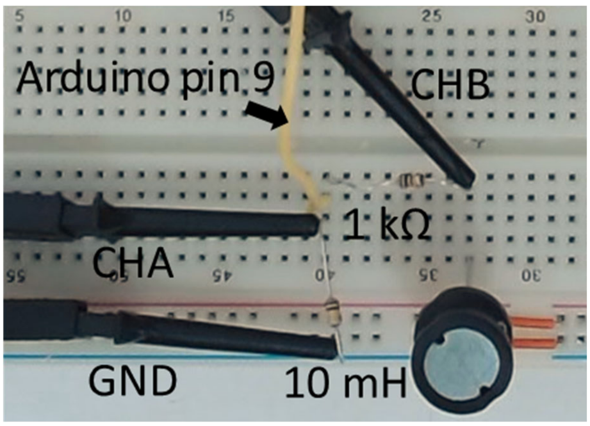


Figure: equivalent circuit for AWG with internal resistance, and inductor with resistance

Therefore we connect the Arduino pin 9 instead of the AWG. Set CHA and CHB to 2V/div. The Arduino has much lower poutput resistance, you measure it in Lab 1.



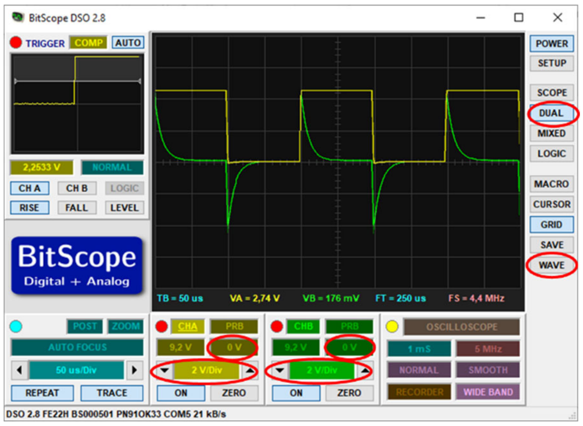


Figure: Breadboard with Lab 2.1 circuit connected to pin 9 of Arduino

For the AC measurements in Lab 3 we will use the BitScope AWG connected to the Op amp buffer circuit in Appendix 6. This can also be used for Lab 2. Note that the amplitude of the buffered signal is only 3.3 Vpp, but it is almost 5 V for the Arduino pin 9.

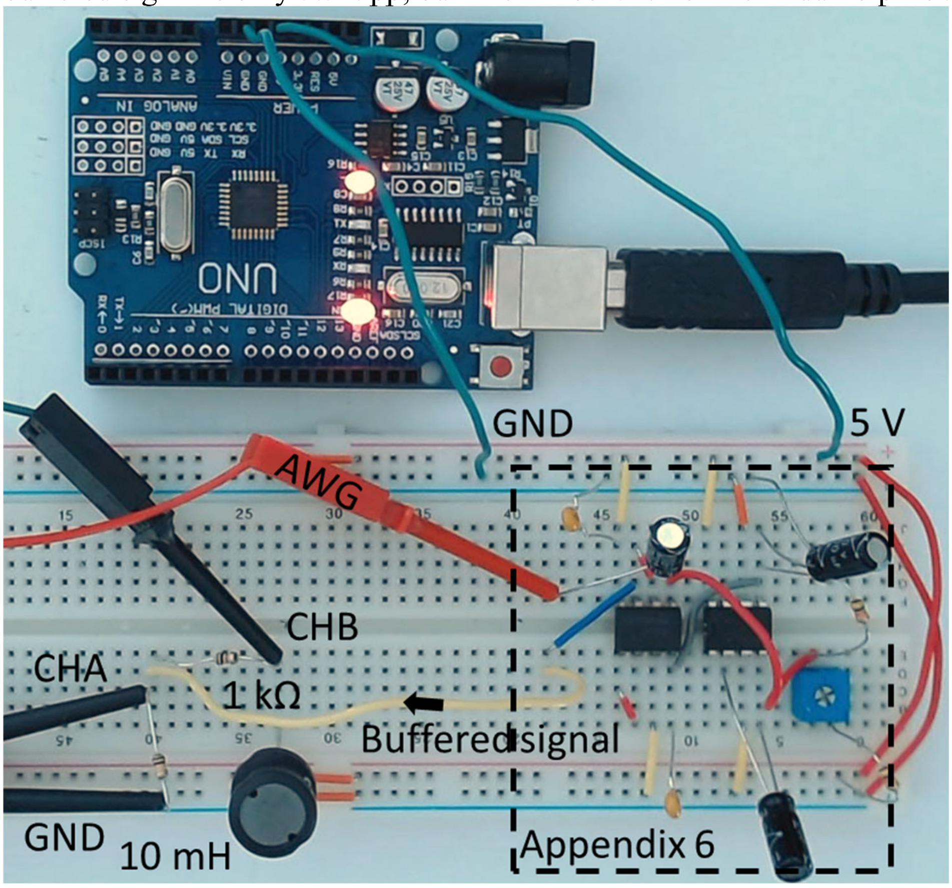


Figure: Breadboard with Lab 2.1 circuit connected to AWG and buffer

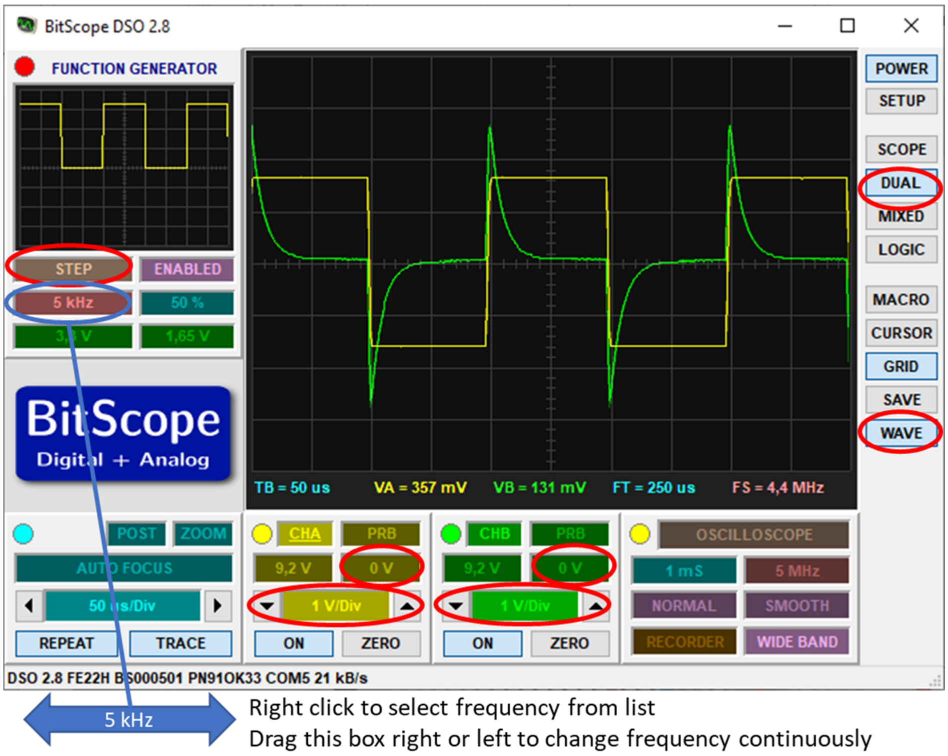


Figure: AWG signal buffered by Op amp can be used for Lab 2 – Lab4

Appendix 6 Measurement circuits

The waveform generator of the BitScope has three drawbacks:

1. High internal output resistance (you will measure it in 3.1)
2. A 1.65 V DC level
3. A maximum frequency of 250 kHz, but the sine wave looks bad already at 100 kHz

We can't do anything about the last issue except do our measurements up to 100 kHz.
The internal output resistance and DC level we can fix using an Op amp buffer.
However, the Op amp needs -5V as well as +5 V, we create this using an extra IC.
We are using a low power dual Op amp, so we have one more Op amp for the lab.

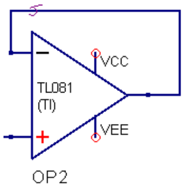


Figure: Op amp buffer

The Op amp buffer increases the current drive and reduces the output resistance.
You will measure it in Lab 3.1.

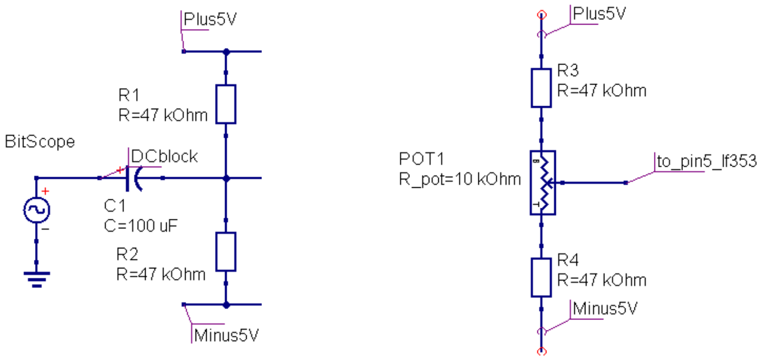


Figure: DC block and reference level restoration

The DC level can be removed with a large capacitor, but then the Op amp needs a resistor network to assure a DC path and correct reference level.
Two equal resistors to +5 V and -5 V work ok, but since the voltages aren't exact, adding a trim resistor we can set the output of the buffer to exactly 0 V without input signal.

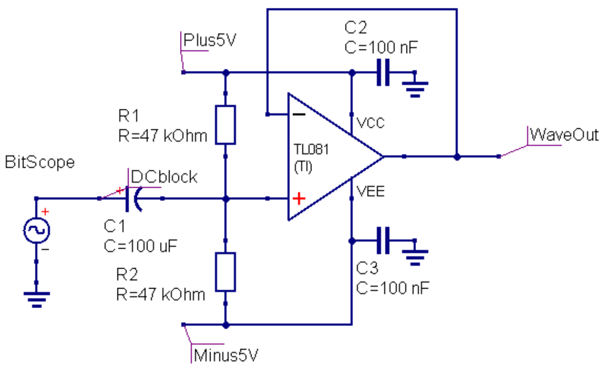


Figure: Complete circuit

Building the -5 V voltage supply using the ICL7660

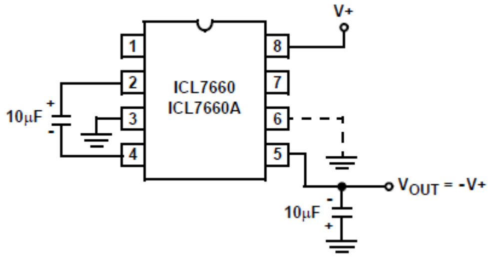


Figure: Connection of ICL7660 and two capacitors

Connect the ICL7660 and two capacitors on the breadboard.
Pin 1 and 7 should not be connected
Pin 2 should be connected via a capacitor to pin 4, “-“ to pin 4
Pin 3 and 6 should be grounded (0 V)
Pin 5 is the -5 V output, connected to “-“ of the other capacitor connected to ground (0 V)
Pin 8 should be connected to +5 V from the Arduino
See breadboard figure on next page.

We will use 100 uF capacitors instead of 10 uF to improve the performance.
Make sure you connect the capacitors with their correct polarity, otherwise they will break.
Measure the 5V from the Arduino using the Multimeter.
Measure the voltage on pin 5 versus ground, is it -5 V?
The ICL7660 has limited current drive, so don’t connect any LEDs as power monitor.

Building the Op amp buffer using the LF353

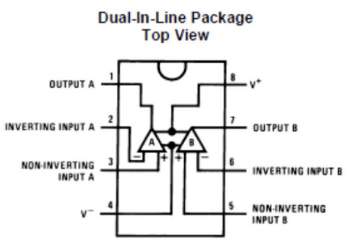


Figure: Pinout for LF353 Dual Op amp

While connecting the circuit, disconnect +5 V from the Arduino.
Pin 4 should be connected to -5 V (pin 5) from the ICL7660
Pin 8 should be connected to +5 V from the Arduino
There is no ground connection to the Op amp required
Connect a 100 uF capacitor as DC block to the non-inverting input pin 5,
observe the polarity, 2-2 to Op amp pin 5 (see figure).
Connect two 100 nF decoupling capacitors between ground and pin 4 and pin 8
Connect two 47 kΩ resistors from pin 4 and pin 8 to pin 5.
Connect the inverting input pin 6 to the output of the Op amp pin 7.
For the unused Op amp that we will use in Lab 4:
Connect the inverting input pin 2 to the output of the Op amp pin 1.
Connect the non-inverting input pin 3 to ground.
See breadboard figure on next page.
NOTE: use TL081 in QUCS simulations as there is no model for LF353

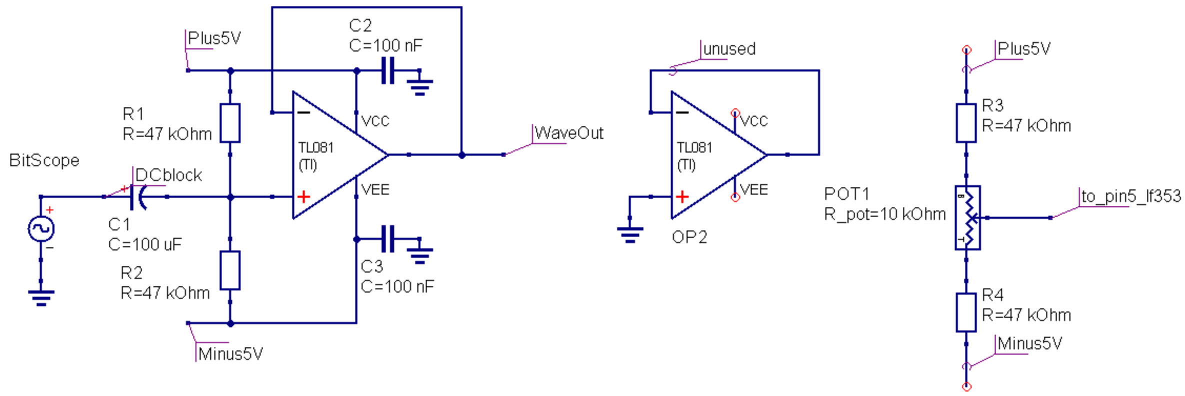


Figure: Complete schematic

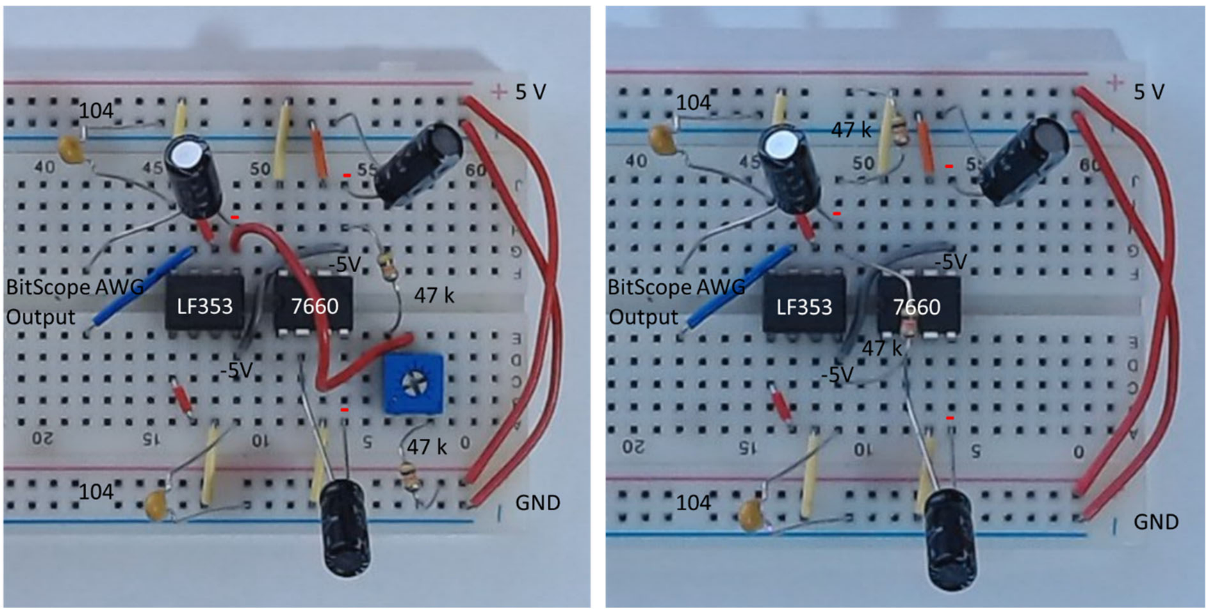


Figure: Example connections on Breadboard, with and without trimmer

Connect +5 V from the Arduino.
Measure the output of the LF353 Op amp buffer pin 7 with the Multimeter (without the BitScope waveform generator connected to the DC block capacitor).
If it is more than 50 mV, also connect the trim potentiometer according to the figure.
Use a small screwdriver to trim the output voltage of the Op amp pin 7 to less than 5 mV.
Now connect the BitScope waveform generator to the DC block capacitor.
See Appendix 5 for some example measurements you can do to test this circuit.

Note
The Op amp buffer with DC resistor divider can also be used to provide a virtual ground reference for single supply Op amp circuits, see the reference from Texas Instruments.

Links (available in Canvas)
https://www.renesas.com/www/doc/datasheet/icl7660.pdf (ICL7660 Datasheet)
https://www.ti.com/lit/pdf/slos012 (LF353 Datasheet)