Circuits and Signals
Lab manual

Marek Nałęcz
Marek Rupniewski

Institute of Electronic Systems
Faculty of Electronics and Information Technology
Warsaw University of Technology

Warszawa, January 12, 2023

Contents

Chapter 1

Circuit elements and circuit laws

1.1 Introduction

1.1.1 Kirchhoff's current law (KCL)

We assume that nodes of the electric circuits do not accumulate electrical charge. Therefore the total current entering each node is constantly equal to zero. This characteristics of electric circuits was first stated in 1845 by the German physicist Gustav Kirchhoff.

Kirchhoff Current Law (KCL). The sum of the currents entering any node of an electric circuit equals the sum of the currents leaving the node.

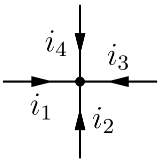


Figure 1.1: Currents entering (converging to) a node

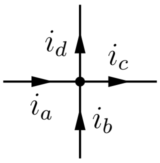


Figure 1.2: Currents entering and leaving a node

For example, for the nodes depicted in Fig. 1.1 and Fig. 1.2 we have, respectively,

$$
\begin{aligned}
i_1(t) + i_2(t) + i_3(t) + i_4(t) &= 0, \\
i_a(t) + i_b(t) &= i_c(t) + i_d(t).
\end{aligned}
$$

We also assume that none of the electric circuit elements accumulates electrical charge. Therefore, the sum of the currents entering an element (through all of its terminals) equals the sum of the currents leaving the element.

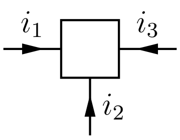


Figure 1.3: Three-terminal device

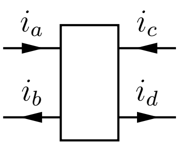


Figure 1.4: Four-terminal device

Thus for the three- and four-terminal devices depicted in Fig. 1.3 and Fig. 1.4, we have:

$$i_1(t) + i_2(t) + i_3(t) = 0,$$
$$i_a(t) + i_c(t) = i_b(t) + i_d(t).$$

The discussed assumption concerning the sum of the currents entering electrical elements and KCL results in the following conclusion.

Corollary (Generalized Kirchhoff's law)**.** *The sum of the currents entering any electric subcircuit equals the sum of the currents leaving this subcircuit.*

As an illustration of the above conclusion, let us consider the subcircuit surrounded with dashed line in Fig. 1.5.

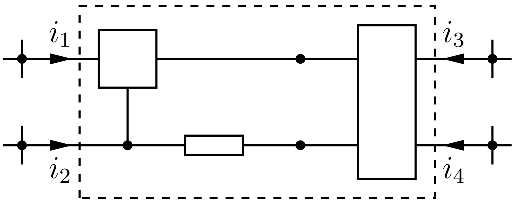


Figure 1.5: Currents entering the subcircuit

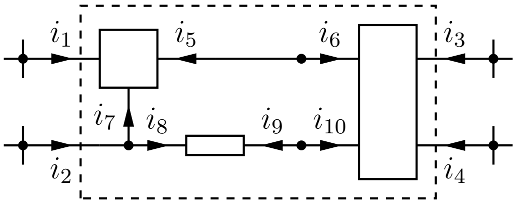


Figure 1.6: Currents related to the subcircuit

In this case, the above corollary states that

$$i_1(t) + i_2(t) + i_3(t) + i_4(t) = 0. \tag{1.1}$$

Let us show how the above statement results from KCL and the characteristics of electrical elements. For this purpose let us label each of the currents flowing between nodes and terminals of the elements as depicted in Fig. 1.6. Now we are able to write the equations stating that the sums of the currents entering each node and each element are zero (each current that appears in these equations is a function of time; for the clarity's sake we do not express this relationship explicitly):

$$\begin{aligned} -i_5 - i_6 &= 0, & i_1 + i_5 + i_7 &= 0,\\ i_2 - i_7 - i_8 &= 0, & i_8 + i_9 &= 0,\\ -i_9 - i_{10} &= 0, & i_3 + i_4 + i_6 + i_{10} &= 0. \end{aligned}$$

Summing these equation side by side we obtain equation (1.1). Indeed, all the "inner" currents of the subcircuit cancel out, because they appear exactly twice and with opposite sign.

1.1.2 Kirchhoff's voltage law (KVL)

Each electric circuit node can be assigned an electric potential with respect to a reference node. This property and the definition of the electric voltage as the difference of the potentials result in Kirchhoff's voltage law.

Kirchhoff Voltage Law (KVL). The sum of the voltage drops around any directed loop equals zero.

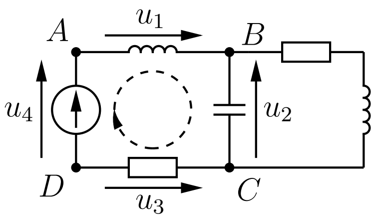


Figure 1.7

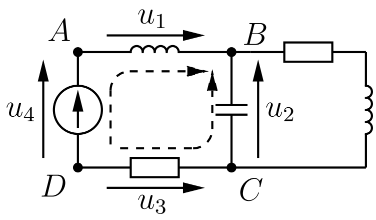


Figure 1.8

For example, in the circuit depicted in Fig. 1.7 the following equality holds:

$$u_1(t) - u_2(t) - u_3(t) + u_4(t) = 0. \tag{1.2}$$

It results from applying KVL to loop $ABCD$ (this loop is depicted in Fig. 1.7 with a dashed line). Voltages u_2 and u_3 in equation (1.2) are taken with "–" sign, because they are directed opposite to the orientation of the considered loop.

Notice that KVL can be equivalently formulated as follows:

KVL (alternative formulation). The voltage between any two nodes is the sum of voltage drops along any directed path connecting the considered nodes. In particular, this voltage does not depend on the choice of the path connecting the nodes.

In the example circuit depicted in Fig. 1.8 the voltage u_{BD} between node B and D can be expressed as (depending on the choice of one of the two paths depicted in the figure with dashed arrows):

$$u_{BD} = u_4 + u_1,$$
$$u_{BD} = u_3 + u_2.$$

Equating the right hand sides of the above equations we obtain:

$$u_1 + u_4 = u_2 + u_3,$$

which is equivalent to equation (1.2).

1.1.3 Equivalent resistance

Every linear two-terminal resistive subcircuit (a subcircuit consisting of linear resistors only) is equivalent to a single resistor. The resistance of such a resistor is called the equivalent resistance of the given two-terminal subcircuit.

Usually, one determines the equivalent resistance by the means of formulas for series and parallel connections. Recall that resistors R_1, R_2, $\ldots$, R_n are connected in series, if every subsequent two of them have precisely one common terminal and no other element is connected to this terminal.

The resistors R_1, R_2, $\ldots$, R_n are connected in parallel, if there exist two different nodes of the circuit, which are the terminals of each of the considered resistors.

The examples of series and parallel connections of resistors are depicted in Fig. 1.9 and Fig. 1.10, respectively.

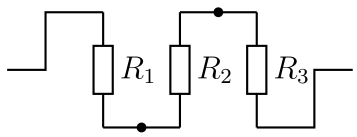


Figure 1.9: Series connection of resistors

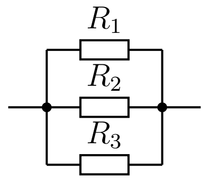


Figure 1.10: Parallel connection of resistors

A chain of resistors in series has the equivalent resistance equal to the sum of the resistances of the individual resistors. A set of resistors connected in parallel has the equivalent resistance equal to the reciprocal of the sum of the conductances of the respective resistors. Therefore, the equivalent resistance of the two-terminal subcircuit of Fig. 1.9 equals $R_1 + R_2 + R_3$, and the equivalent resistance of the two-terminal subcircuit of Fig. 1.10 equals $\left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}\right)^{-1}$.

If the considered two-terminal resistive subcircuit is more complicated, the formulas for resistors connected in series and in parallel may not be sufficient to determine the equivalent resistance. In such cases the following observation can be useful. If a two-terminal subcircuit D of equivalent resistance R_D is connected to a voltage source of electromotive force E (as shown in Fig. 1.11), then the current supplied by this source equals

$$I = \frac{E}{R_D},$$

as depicted in Fig. 1.12. Thus, the equivalent resistance may be determined as

$$R_D = \frac{E}{I},$$

where E is an arbitrary non-zero voltage and current I is determined with a help of KCL, KVL and Ohm's law.

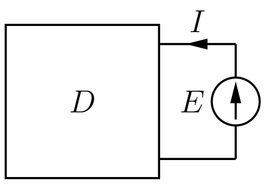


Figure 1.11

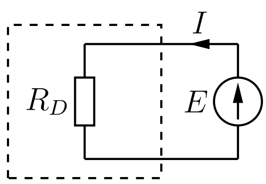


Figure 1.12

Sometimes, the following observation helps in finding the equivalent resistance of a linear two-terminal resistive subcircuit D. Assume that there are two nodes of D with the same electric potential (the voltage across these nodes is zero). Then, the addition of any resistor between these nodes does not alter the equivalent resistance of D. In particular, the equivalent resistance will not change, if the considered nodes are short-circuited (connected by a resistor of zero resistance). As an example, let us determine the equivalent resistance of the two-terminal subcircuit depicted in Fig. 1.13. At first, let us consider the two-terminal subcircuit of Fig. 1.14, whose equivalent resistance can be easily determined as $R_{\mathrm{eq}} = 2\ \mathrm{k}\Omega$ (a parallel connection of resistances $6\ \mathrm{k}\Omega$ and $3\ \mathrm{k}\Omega$). The voltage U_{BA} is zero. Indeed, voltage divider formula applied to each of the two parallel branches of the subcircuit gives

$$U_1 = U_2 = \frac{1}{3}U.$$

Therefore, by KVL we obtain

$$U_{BA} = U_2 - U_1 = 0,$$

and thus the two-terminal subcircuits of Fig. 1.13 and 1.14 have the same equivalent resistances 2 kΩ.

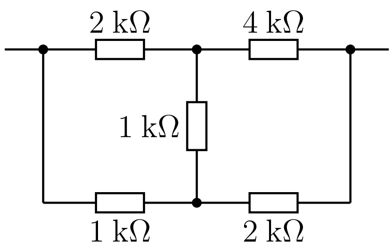


Figure 1.13

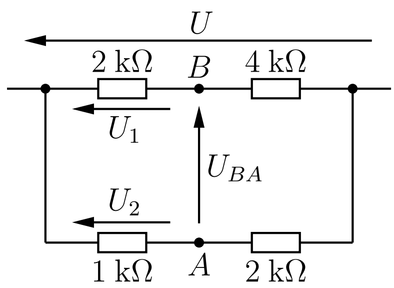


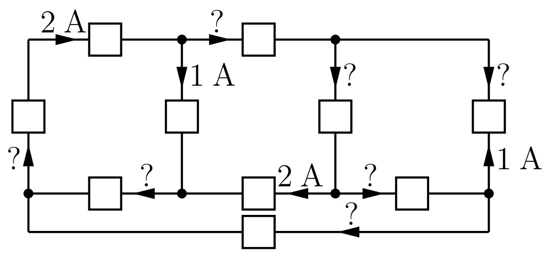
Figure 1.14

Once again, let us consider a general case of a linear two-terminal resistive subcircuit D. Let's analyse how its equivalent resistance will change, if we add a resistor between a pair of its nodes. We already know that if those nodes have equal electric potentials, then the equivalent resistance of D will not change. In all other cases, the equivalent resistance of D will be reduced. Let us emphasize this quite surprising fact: a resistor added to a linear two-terminal resistive subcircuit may only reduce its equivalent resistance. An obvious conclusion is that removal of a resistor being part of a linear two-terminal resistive subcircuit may only increase the equivalent resistance of this subcircuit. In particular, such a removal may change the equivalent resistance to $+\infty$, i.e., it may make the subcircuit equivalent to an open-circuit.

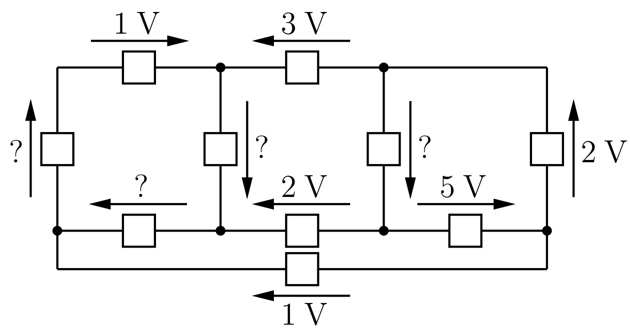
Problems for self study

Solutions to the problems are given on page 27.

1. Determine all currents labelled with question marks in the following circuit. In this exercise and the next one, squares represent some, but not necessarily same, two-terminal devices.

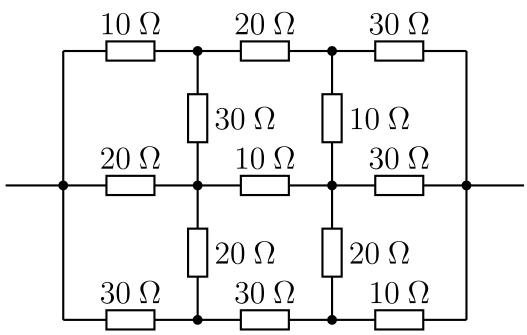


2. Determine all voltages labelled with question marks in the following circuit.



3. Consider a model of a cube built of 12 identical wire segments, each of 1 Ω resistance (every segment is an edge of the cube). Determine the equivalent resistance of the cube with its two vertices chosen as the terminals (consider all possible choices of the vertices).

4. Estimate the equivalent resistance of the following two-terminal subcircuit (find an equivalent resistance upper bound by removing some resistors; next, find a lower bound by short-circuiting some nodes of the subcircuit).



1.2 Homework

If a circuit's diagram can be drawn on a sheet of paper so that no elements overlap and no two segments representing conductors intersect, then we call such a circuit a planar circuit. Therefore, the circuit of Fig. 1.16 is planar, and the circuit of Fig. 1.15 is not (symbols $T_1, \ldots, T_6$ denote some three-terminal devices).

A part of a planar circuit diagram which forms a closed loop surrounding a blank area of the paper on which the diagram is drawn, is called an essential mesh or simply a mesh[1].

A mesh of a circuit is usually marked with a closed directed loop around the area surrounded with the mesh. As an example, each mesh of the circuit depicted in Fig. 1.16 is marked with a dashed line.

It turns out that for each planar circuit with N meshes, there are at most N linearly independent equations that can be derived directly from KVL.

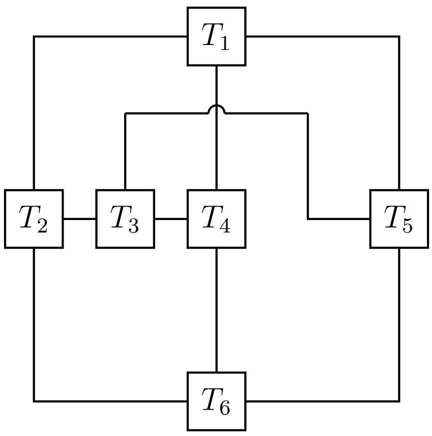


Figure 1.15: An example of a non-planar circuit.

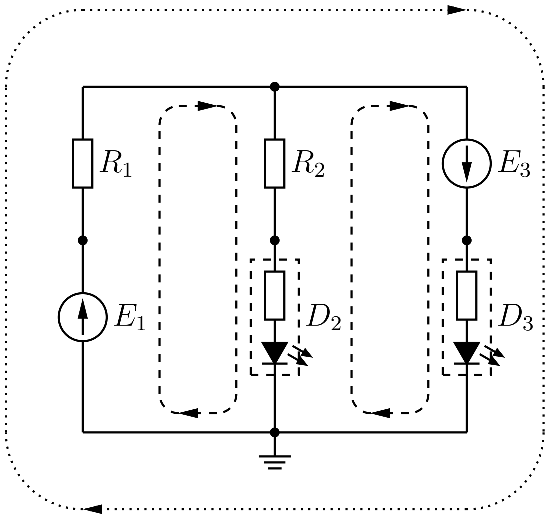


Figure 1.16: A planar circuit with essential meshes (marked with a dashed line) and the reference mesh (dotted line)

Homework 1.1. Consider the circuit of Fig. 1.16. Show that KVL equation for the reference mesh marked with a dotted line is a linear combination of the KVL equations for all the other meshes. In your solution use the voltage variables introduced in Fig. 1.22 only.

---

[1]Sometimes, it is convenient to treat the area outside the circuit as surround by the so called reference mesh. In Fig. 1.16 the reference mesh is marked with a dotted line.

1.3 Equivalent resistance

1.3.1 Verification of Ohm's law

In further course of this exercise, we'll be studying voltage-current characteristics (current as function of voltage) of linear and nonlinear resistors. The measuring devices used for this purpose (the stand-alone oscilloscope or the virtual digital oscilloscope) are capable of measuring *voltages* only. Thus, we must know how to measure the current indirectly, basing on Ohm's law and measurements of voltage across a reference resistor. In the introductory part of this exercise we will verify this fundamental law.

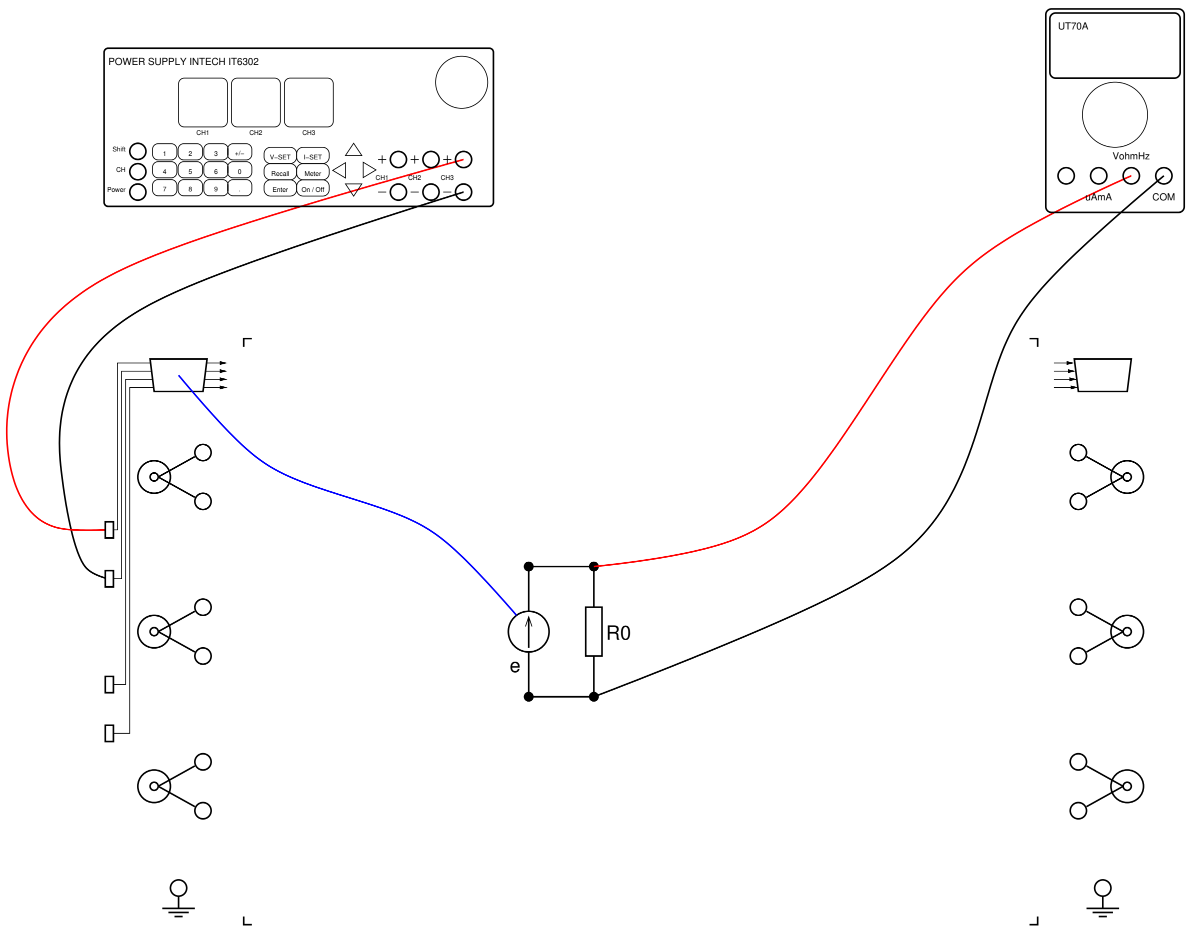


Figure 1.17: Measurement setup for verification of Ohm's law

1. **Before** assembling the measurement setup, switch off `CH3` output of the power supply (press `Shift | 3`) and set the voltage value for this channel to 0.1 V, and the current limit to 0.1 A. Next, assemble the setup as shown in Fig. 1.17, taking $R_0 = 100\ \Omega$.

2. The voltage source e has to be connected to the supply socket in the "tray", by means of which it is connected with two wires (with banana plugs) to the `CH3` channel of the power supply.

3. **Before powering on the instruments, the assembled setup is to be shown to the lab instructor for verification.** Switch on `CH3` channel of the power supply and activate its current measure function (press `Meter` button). The measured current is the current through R_0 (actually, with the voltmeter connected in parallel with this resistor, but the influence of this voltmeter on measurement of current is negligible — the internal resistance of the voltmeter is $10\ \mathrm{M\Omega}$).

4. The stand-alone digital multimeter UT70A is to be set to DC voltmeter in the range $20\ \mathrm{V}$. It will be used to measure voltage u across the examined resistor R_0 (the voltage across R_0 is smaller than the voltage supplied by the power supply because there are some voltage drops across the wires).

It is worth to verify proper polarization of the power supply, voltmeter and ampere meter.

5. Gently increase the power supply voltage to set the voltage u across resistor R_0 equal to *around* $0\ \mathrm{V}$, $1\ \mathrm{V}$, $2\ \mathrm{V}$, …, $5\ \mathrm{V}$. For each of these values read the value of current i flowing through the resistor. The measurement results are to be recorded in Tab. 1.2.

If it is impossible to set the required voltages, check if the power supply is not operating in "control current" mode.

6. We facilitate verification of Ohm's law by using the numerical computing environment MATLAB. After launching MATLAB, define in the command window, the vector of voltages expressed in volts **(Important! Below there are given only *sample* values of voltages and currents — make sure you've replaced them with your own measurements!)**:

```
u = [ 0.10 1.11 2.22 3.33 4.44 5.55 ]
```

and the corresponding vector of currents expressed in milliamperes:

```
i = [ 1.3 11.1 22.2 33.3 44.4 55.5 ]
```

Next, plot the obtained points of voltage-current characteristic of R_0:

In order to save time, you are encouraged to copy the commands from this PDF and paste them in MATLAB.

```
plot(u,i,'ob');
xlabel('u [V]'); ylabel('i [mA]');
grid;
```

Let's now fit a straight line (first order polynomial) to the obtained measurement points:

```
p = polyfit(u,i,1)
```

The first of the two thus obtained coefficients is the slope of the linear approximation of the characteristic in coordinates V- mA, i.e. the *conductance* of the resistor R_0 expressed in mS (nominally equal to $10\ \mathrm{mS}$). The other coefficient is the free term of the linear approximation of the characteristic, so we expect it to be roughly 0. Now let's plot the linear approximation of the characteristic on top of the measurement points, and add the value of the obtained resistance as the title of the plot:

```
hold on
plot(u,polyval(p,u),'b-')
title(sprintf('R=%d\\Omega',1e3/p(1)))
hold off
```

The distance between the measurement points and the obtained straight line is a measure of the quality of fulfillment of Ohm's law and the accuracy of the performed measurements. The obtained plot should be saved on disk, to be later printed out and attached to the report.

1.3.2 Series and parallel connections of resistors

Knowing that we can indirectly measure the current (by measurement of voltage across a resistor of fixed resistance), we can now determine the voltage-current characteristics of two different resistors and of their series and parallel connections. Beware of yet another difficulty. The measurement instruments that we will use in the exercise (stand-alone oscilloscope or virtual digital oscilloscope realized with `Maya44 USB` card) are capable of measuring only voltages *with respect to ground*. That is why the so called "floating" voltages will be measured as *differences* between two voltages with respect to ground. In the measurement block diagrams Fig. 1.18–Fig. 1.20, resistor $R_0 = 100\ \Omega$ is used for indirect measurement of current i flowing through the "horizontal" branch of the setup. The difference between the voltages on inputs `IN1` and `IN2` of the `Maya44 USB` card ($u_1 - u_2$) is equal to "floating" voltage on "horizontal" branch of the setup. The setup is excited with a sinusoidal voltage from output `OUT1` of the `Maya44 USB` card.

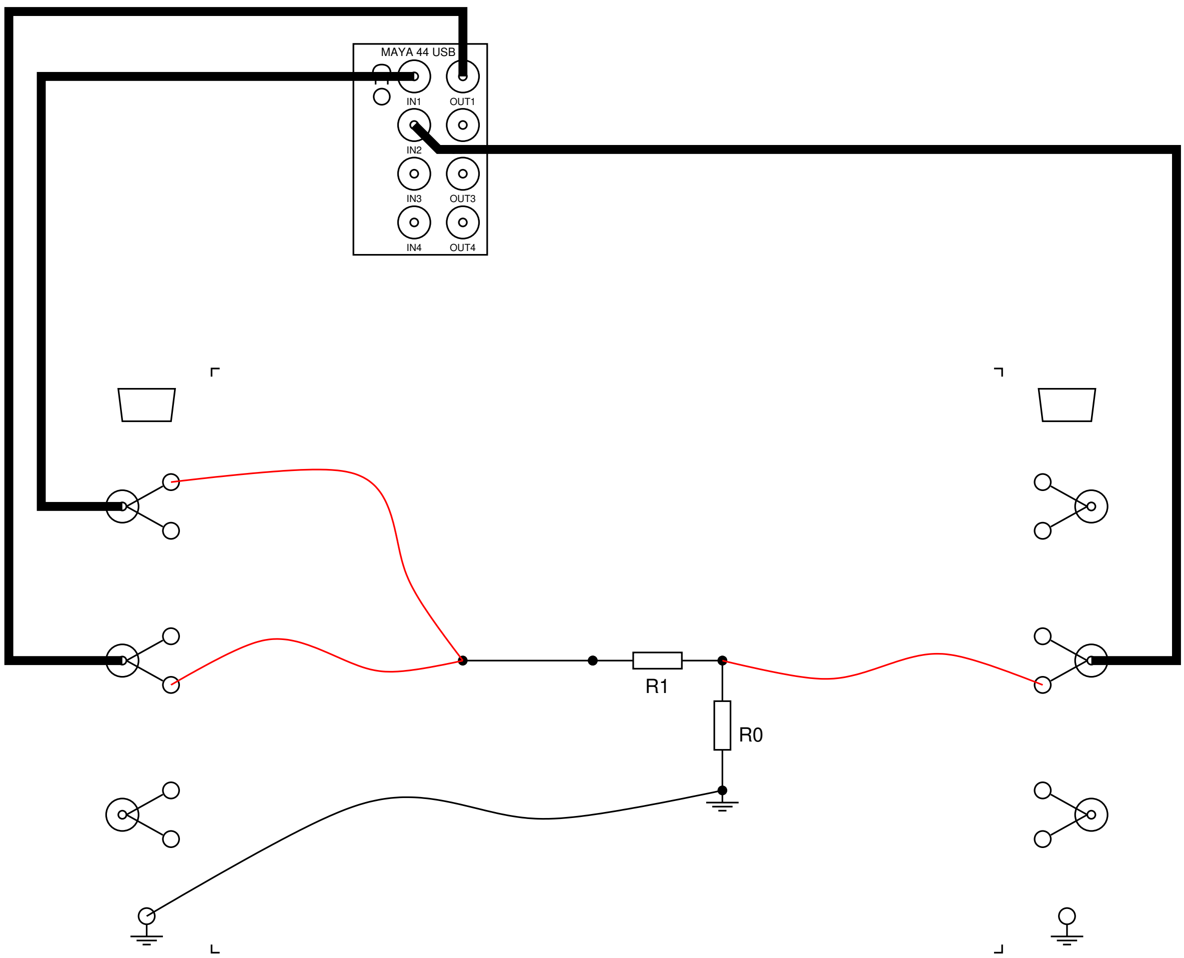


Figure 1.18: Setup for measuring voltage-current characteristic of the resistor R_1

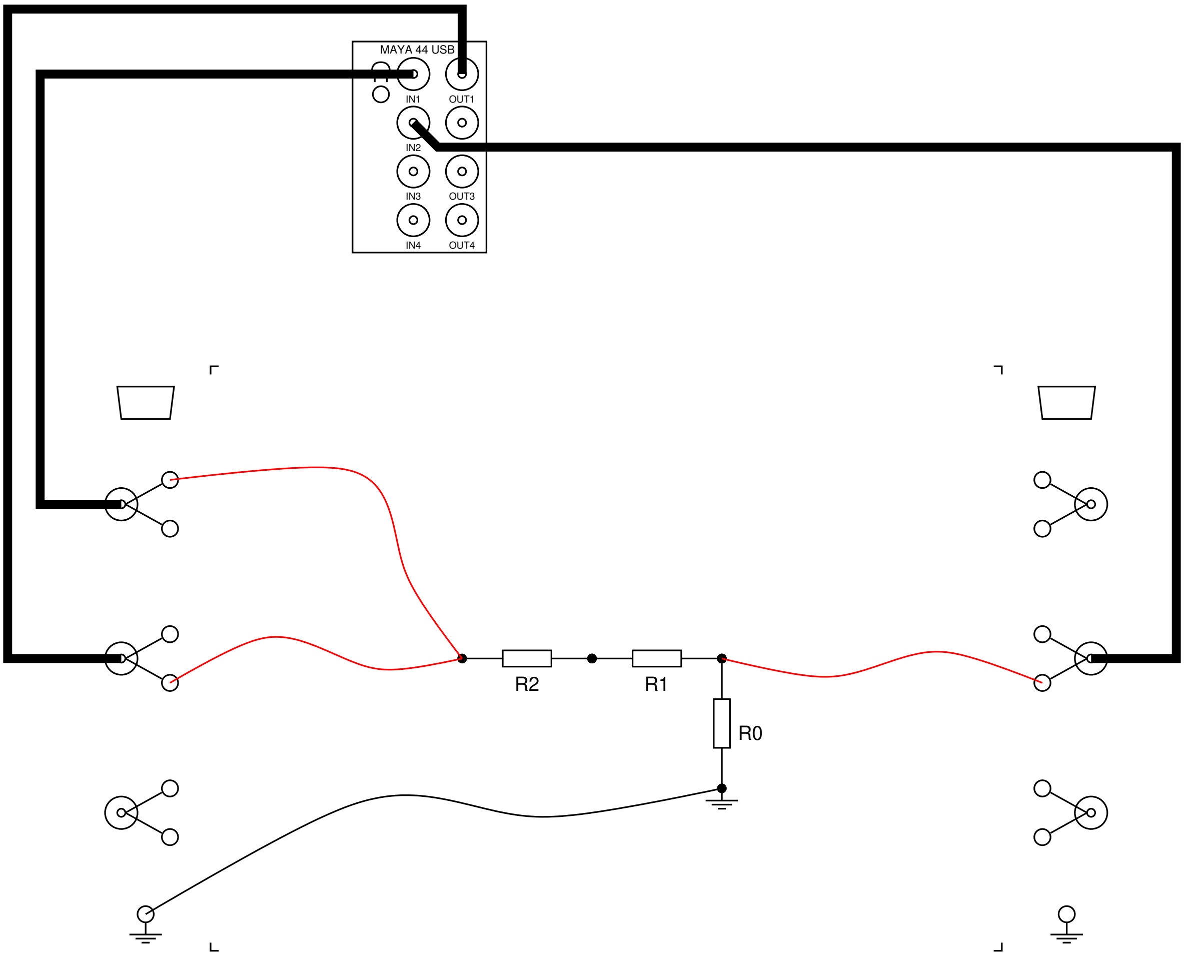


Figure 1.19: Setup for measuring voltage-current characteristic of the series connection of resistors R_1 and R_2

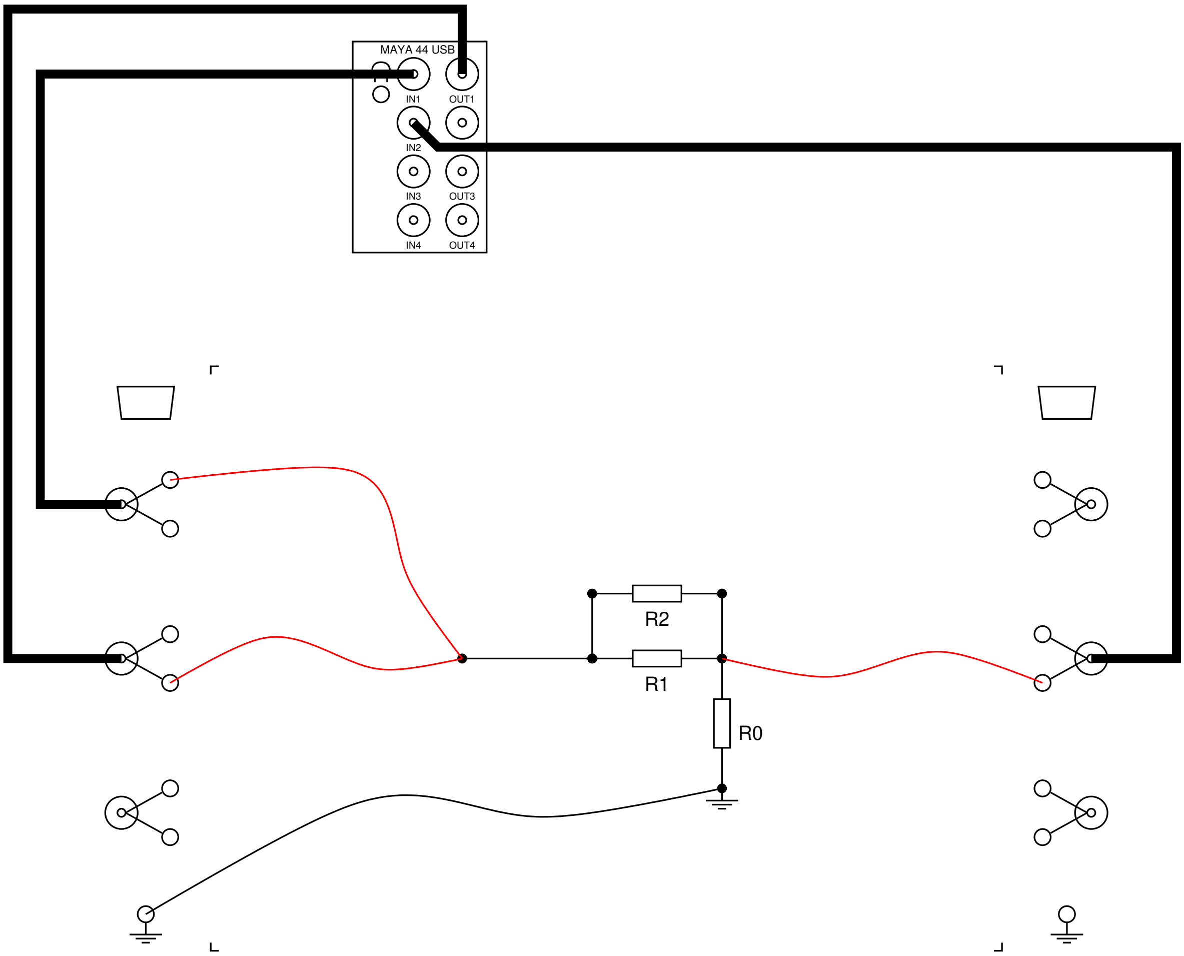
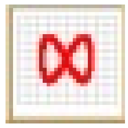


Figure 1.20: Setup for measuring voltage-current characteristic of the parallel connection of resistors R_1 and R_2



1. Power off the laboratory crate and assemble the measurement setup as shown in Fig. 1.18 for $R_1 = 1\ \mathrm{k\Omega}$, then power on the crate again.

2. Turn the computer on and launch the TOiS_Toy program. Activate two of the measuring instruments from the TOiS_Toy set:

   - sinusoidal waveform generator at the output `OUT1` of `Maya44 USB` card, with default presets regarding frequency (200 Hz) and amplitude ($E_{g,\max} = 0{,}5\ \mathrm{V}$),
   - oscilloscope operating in X-Y mode, the gain in channel X should be set to $100\ \mathrm{mV/div}$, and in channel Y to $10\ \mathrm{mV/div}$.

   Define the first signal in channel `X` as `Inp.1-Inp.2` and activate it with the left mouse button. In a similar way activate signal in channel `Y` as `Inp.2`.

3. Now measure the voltage-current characteristic of the resistor R_1, choosing single run mode (key `Run` in menu Run / Once or appropriate button from the toolbar in TOiS_-Toy's window). The measured characteristic should be saved by clicking button `store` in `Memory` panel in the oscilloscope window.

In order to avoid a problem with colors of the characteristics, one should (`clear`) the saved characteristics before each measurement of 4 characteristics.

4. Measure and save the characteristics again, with R_1 replaced by $R_2 = 500\ \Omega$ (measurement setup is identical to the one in Fig. 1.18, just R_1 should be replaced with R_2) without turning off the power of the laboratory crate.

5. With the laboratory crate power on, assemble the measurement setup including resistors R_1 and R_2 in series, as shown in Fig. 1.19, and then once again repeat the measurement of the characteristic of this connection and store it in the oscilloscope memory.

6. With the laboratory crate power still on, assemble the measurement setup including parallel connection of resistors R_1 i R_2, (Fig. 1.20), and store the characteristic measured in an analogical way as before.

1×

→PNG

Make sure the characteristics have indeed been saved, (check the saved file).

The obtained family of 4 voltage-current characteristics should be saved on disk, so that you can later print it, label it (at least with a legend showing which plot corresponds to which variant of connection of resistors R_1 and R_2), and attach it to the report. For each of the four plots determine the inverse of the slope of the corresponding line (i.e. resistance of a given resistor or equivalent resistance of a connection) and record the results in tab. 1.3. It can be done with TOiS_Toy by indicating a point with the mouse cursor in a given plot (as far from the origin of the coordinate system as possible) and pressing the left mouse button to see the values of voltages $u_x = u$ and $u_y = R_0 i$. Then the relevant resistance can be computed:

$$R = \frac{u}{i} = \frac{u_x}{u_y/R_0} = R_0\frac{u_x}{u_y} \tag{1.3}$$

1.3.3 General remarks regarding connections of resistors

At the end of this section, we will show some general rules regarding adding resistors to an existing configuration of resistors. Let's start with the following quite an illuminating experiment. Take a $1\ \mathrm{M\Omega}$ resistor from the set of elements. Set the digital multimeter UT70A to ohmmeter in the range $2\ \mathrm{M\Omega}$, and insert two cables ended with banana plugs into sockets `COM` and `V Ω Hz`. Then take the banana plugs at the loose ends of these cables and make them touch the terminals of the resistor, use your fingers to firmly press the banana endings against the terminals of the resistor to provide a good contact between them. Record the value of the resistance shown by the ohmmeter. Now put magnetic endings on the loose ends of the two cables, connect them to the terminals of the resistor using balls and without touching them with your fingers repeat the measurement of resistance. What is the value of resistance shown by the ohmometer now? It turns out that in the first case, we connected the equivalent resistance of our body in parallel with the resistor $1\ \mathrm{M}\Omega$. An insertion of a resistance to a one-port usually *decreases* the equivalent resistance of this one-port (but not always; as you should already know, in some cases it doesn't alter the resistance of the one-port at all!).

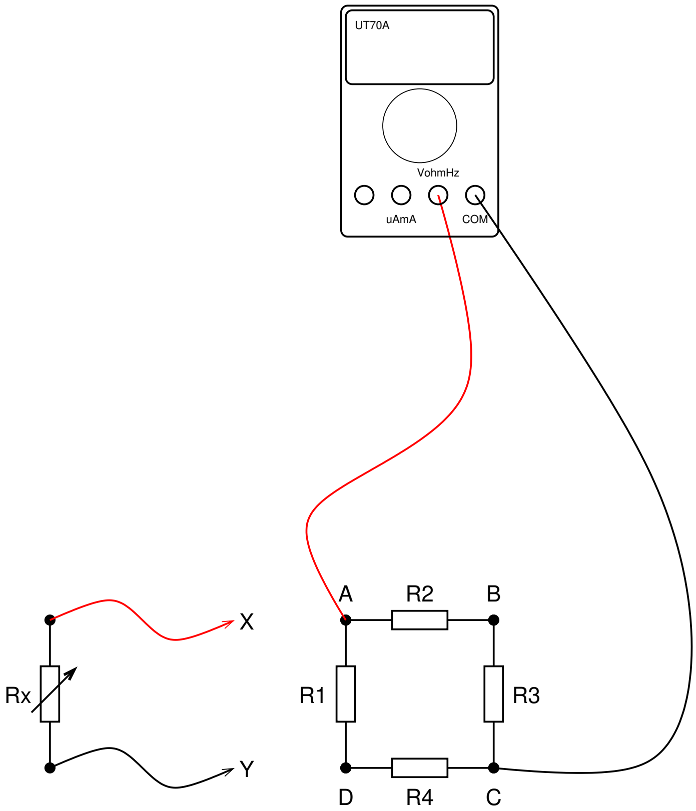


Figure 1.21: Setup for studying the influence of attaching/detaching resistors on the equivalent resistance of a one-port

In order to verify this principle, let's assemble the measurement setup shown in Fig. 1.21. Use the following values of elements: $R_1 = 1\ \mathrm{k}\Omega$, $R_2 = 2\ \mathrm{k}\Omega$, $R_3 = 10\ \mathrm{k}\Omega$, $R_4 = 5\ \mathrm{k}\Omega$. Set the digital multimeter UT70A to ohmmeter in the range $20\ \mathrm{k}\Omega$. Using the ohmmeter, measure the equivalent resistance $R_{\text{A-C}}$ of this configuration and record the result in Tab. 1.4. We will be attaching an additional resistor R_x to various points of the circuit. As the resistor R_x we'll use $10\ \mathrm{k}\Omega$ potentiometer, whose knob should be set to position close to the middle of its adjustment range. The preset value of potentiometer resistance should also be measured with the ohmmeter and recorded in the header of Tab. 1.4. Attach the two banana plug ended cables to the potentiometer, using magnetic adapters.

A potentiometer is just a resistor with adjustable resistance.

Next, connect terminals X and Y of these cables to terminals, subsequently, A-B, A-C, A-D, C-B, C-D i B-D of our one-port (see Fig. 1.21). Each time record the measured value of the equivalent resistance $R_{\text{A-C}}$ in Tab. 1.4. In the last case (potentiometer attached to terminals B-D) turn the knob of the potentiometer in the whole possible range. Why is the measured equivalent resistance not changing? (Recall the properties of the so-called balanced bridge.) As you see, there is no way to *increase* the equivalent resistance of a one-port by attaching an extra resistor to it.

Detach the potentiometer R_x from the one-port being examined, and then sequentially: first detach resistor R_1, then attach it back and detach R_2, attach R_2 and detach R_3, and, finally, attach R_3 back and detach R_4. **Important! Detaching a resistor means replacing it with an open-circuit!** Measure the value of equivalent resistance $R_{\text{A-C}}$ again using the ohmmeter and record it in tab. 1.4. Verify that by detaching resistors comprising a one-port we can not *decrease* its equivalent resistance.

1.4 Verification of Kirchhoff's laws

1.4.1 Verification of Kirchhoff's Laws in a DC circuit

In this exercise we will analyze the circuit shown in Fig. 1.22 (note the polarization of E_3 voltage source: $U_{E_3} = -E3!$).

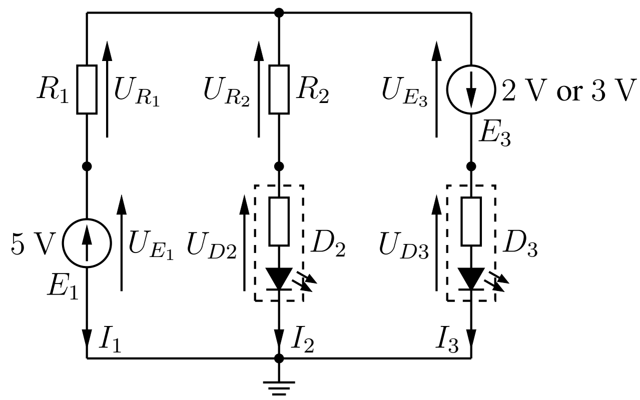


Figure 1.22

Each of the series connections of a resistor and a diode (marked in Fig. 1.22 with a dashed line) in this circuit will be considered as a single compound one-port. In the course of this exercise we will verify validity of KVL for the meshes depicted in Fig. 1.16 with a dashed line. We will also verify KCL for the ground node .

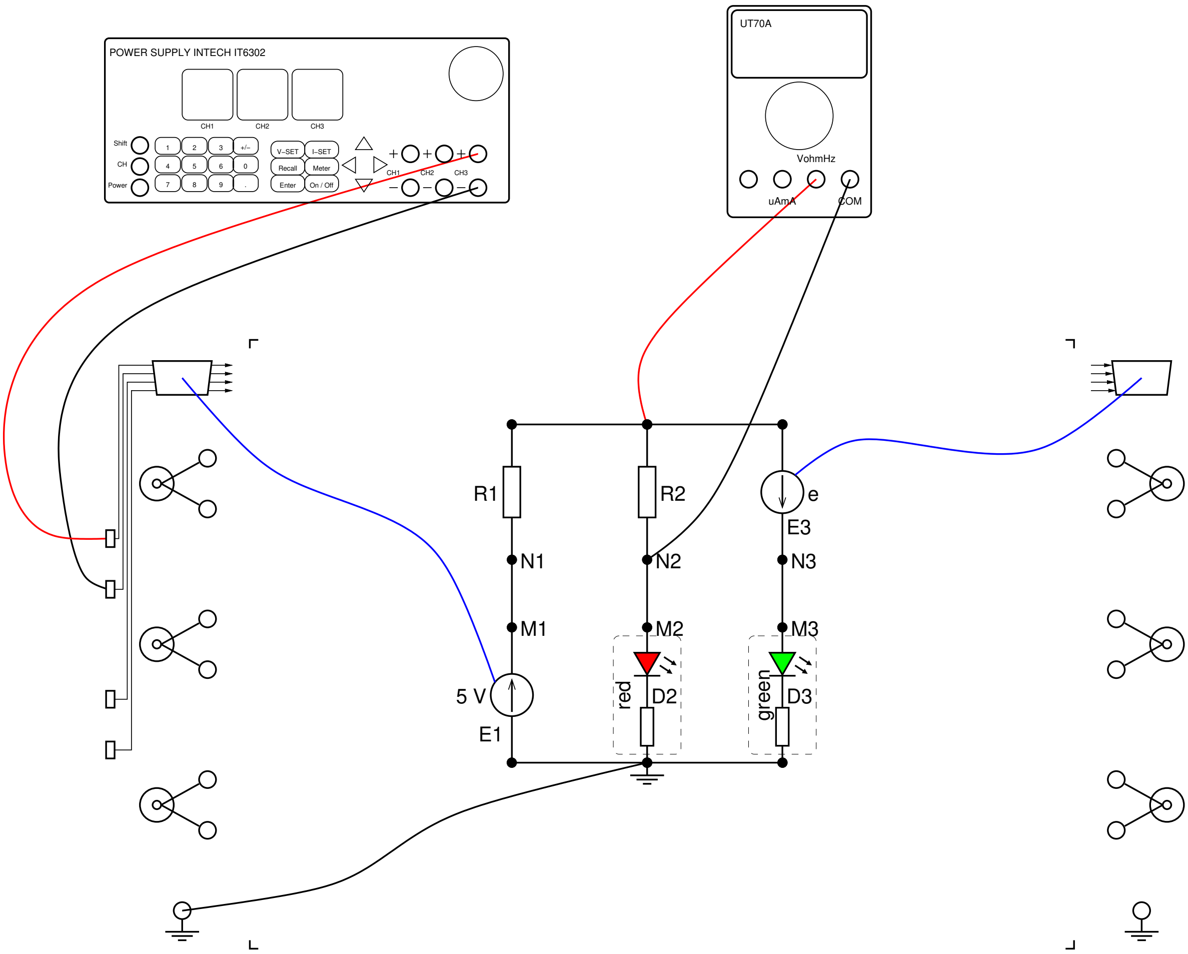


Figure 1.23: Setup for measuring voltages in task 1.4.1

1. Power off the measurement instruments and the crate. Switch off `CH3` output of the power supply (press `Shift | 3`) and set the voltage value for this channel to 2 V or 3 V (for even and odd stand numbers, respectively), and the current limit to 0.1 A.

Verify proper polarization of sources and diodes!

2. Assemble the measurement setup according to Fig. 1.23 using the following values of elements: $R_1 = 50\ \Omega$ i $R_2 = 50\ \Omega$. **Before turning the power on, show the assembled setup to the lab instructor for verification.** Connect the voltage source $E_1 = 5$ V to the supply socket in the tray which connects it with the voltage supply in the crate. The voltage source E_3 should be connected to the second supply socket in the tray, by means of which it is connected with two cables (with banana plugs) to the `CH3` channel of `IT6302` power supply. **Important! For E_3 source pick a 'universal' voltage source marked by *lower* case letter e.** Next, power on the instruments and the crate.

3. Using digital multimeter UT70A set to measure DC voltage in the range 20 V, measure voltages: U_{E_1}, U_{R_1}, U_{D_2}, U_{R_2}, U_{E_3} i U_{D_3} (see Fig. 1.22). Mind polarization of the voltmeter when connecting its 'hot' wire to the node corresponding to the end of the 'arrow' of a given voltage. Next, verify that the Kirchhoff's voltage law is fulfilled in the two meshes of the circuit. Results of measurements and calculations are to be

recorded in Tab. 1.5.

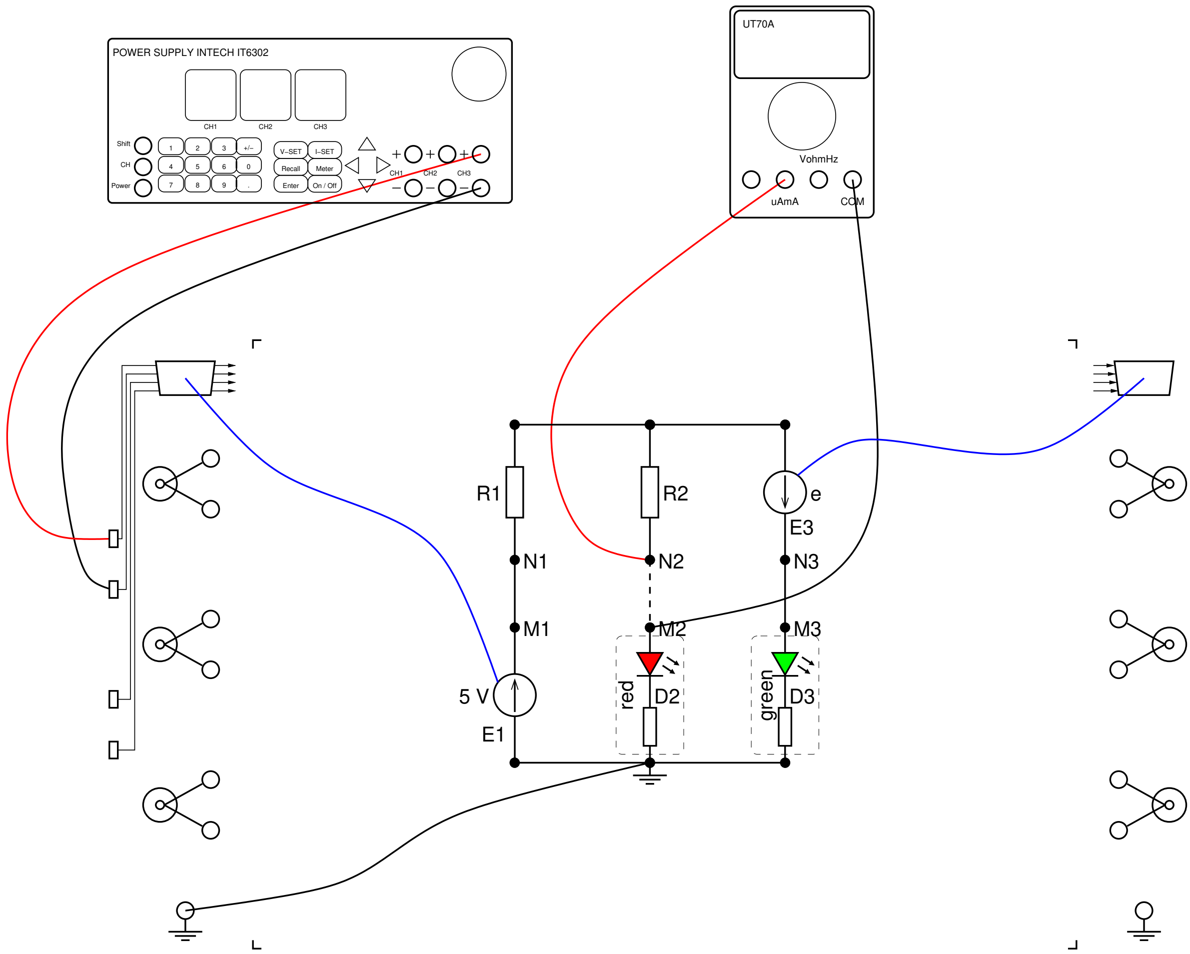


Figure 1.24: Setup for measuring currents in task 1.4.1

4. Using multimeter UT70A set to DC ammeter in the range $200\ \mathrm{mA}$ (see Fig. 1.24), measure currents I_1, I_2 and I_3 flowing toward the ground node in all three branches of the circuit. For example, in order to measure I_2, connect the 'cold' wire of the ammeter to point M_2, and the 'hot' one to point N_2 (replacing the rod connecting these points with the ammeter). Measure I_1 and I_3 in a similar way. Then verify if Kirchhoff's current law is fulfilled for the ground node.

The function of ammeter in multimeter UT70A is available through a different pair of sockets than the function of voltmeter.

1.4.2 Verification of Kirchhoff's laws for AC circuit

For the circuit shown in Fig. 1.25 we are going to verify that KCL holds also for currents varying in time. For this purpose, we will supply the circuit with a sine wave signal and measure the currents entering the ground node .

1. Power off the crate. Assemble the measurement setup according to Fig. 1.26. Take the following values of elements: $R = 100\ \Omega$, $R_L = R_C = 50\ \Omega$, $L = 10\ \mathrm{mH}$. The value of capacitance C depends on the number of the laboratory stand according to Tab.1.1.

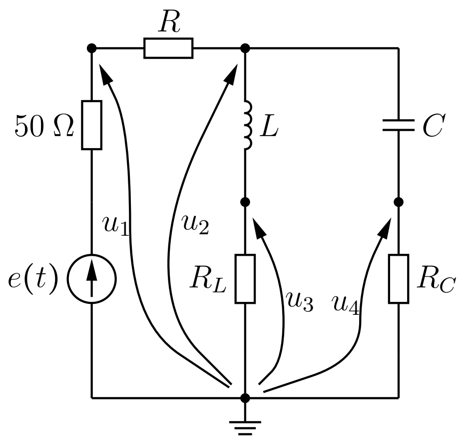


Figure 1.25

The voltages at the inputs `IN1` to `IN4` of the `Maya44 USB` card will correspond to nodal (with regard to ground) voltages u_1 to u_4 (see Fig. 1.25), respectively. Power on the crate.

Table 1.1: Conductances C from Fig. 1.26 for different laboratory stands

Stand	1, 5, 9, 13, 17	2, 6, 10, 14, 18	3, 7, 11, 15	4, 8, 12, 16
C [nF]	22	47	100	220

2. Set the signal gernator to generate sine wave with amplitude $1\ \mathrm{V}$ and frequency $5\ \mathrm{kHz}$.

3. In the examined circuit, the currents will be measured indirectly, via measurement of voltages on reference resistors. `Maya44 USB` card measures only voltages with respect to ground only. Therefore, the 'floating' voltage (on 'ungrounded' resistor R) will be determined as the difference between voltages u_1 and u_2. That is why in the digital oscilloscope in the TOiS_Toy program, we should define, and then activate, the first four channels as `Inp.1`, `Inp.1-Inp.2`, `Inp.3` and `Inp.4`. Set the gain of the oscilloscope to $200\ \mathrm{mV/div}$, and the time base to $100\ \mu\mathrm{s/div}$, then run the oscilloscope in the mode of repeated single run (`Run / Once: Repeat`). While changing generator frequency from $3\ \mathrm{kHz}$ to $8\ \mathrm{kHz}$ observe the waveforms of voltages (proportional to the considered currents).

4. For so chosen frequency, measure voltages u_1 to u_4. For this purpose, deactivate the oscilloscope in the TOiS_Toy program (`Run / Once: Stop`) and close its window. Then launch MATLAB and collect 64 samples of signals from each of the four channels in `Maya44 USB` card at the sampling frequency $48\ \mathrm{kHz}$, by typing the following instructions:

You are encouraged to copy MATLAB commands from this PDF.

```
u  = in(1000/48, 64, [1 2 3 4]);
u1 = u(:, 1);
u2 = u(:, 2);
u3 = u(:, 3);
u4 = u(:, 4);
```

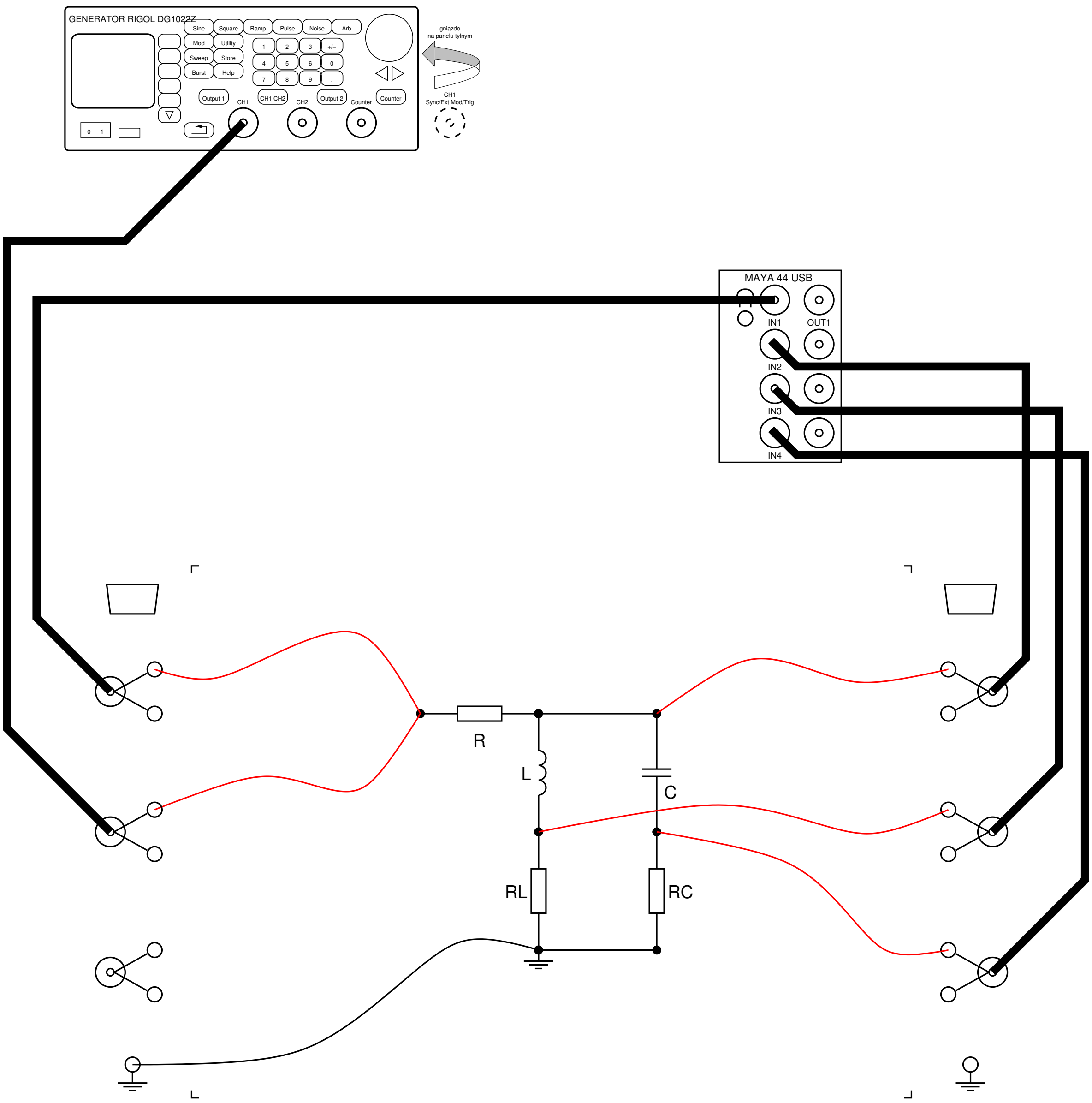


Figure 1.26: Measurement setup for task 1.4.2

The first of these commands (`in`) has three input arguments: sampling period expressed in microseconds, the number of samples to be collected and the vector of `Maya44 USB` input channel numbers that are to be sampled. Columns of the matrix returned by function `in` corresponding to the channel numbers can be 'extracted' from matrix `u` with the following list of commands.

Before typing (or pasting) further commands, it is worth to make sure that we have registered sensible voltages (not a noise). For this purpose, execute the following command:

```
plot(u);
```

and, if you have any doubts, ask the lab instructor for a comment.

Next, define the values of particular resistors in $\mathrm{k\Omega}$:

```
R  = 0.100;
RL = 0.050;
RC = 0.050;
```

and using Ohm's law compute the measured currents $[\mathrm{mA}]$:

```
iR = (u1 - u2) / R;
iL = u3 / RL;
iC = u4 / RC;
```

For plotting waveforms of the currents, we need to define vector of time instants in μs corresponding to subsequent samples of the signal

```
t = (0 : size(u, 1) - 1) * 1000/48;
```

now we can plot the current waveforms and their sum (which should be zero according to KCL):

```
plot(t,iR,  t,iL,  t,iC,  t,iR-iL-iC);
legend('i_R', 'i_L', 'i_C', 'sum');
grid;
xlabel('t [\mu{}s]');
ylabel('i [mA]');
```

1×

→PNG

Save the obtained plot on disk in PNG format, so that you can later print it and attach to the report. Comment on precision of fulfillment of Kirchhoff's current law (note that the precision is significantly affected by the 1% resistors tolerance and by similar order of accuracy of voltage measurements performed with `Maya44 USB` card).

Stand:	*Name and SURNAME*	*Grade*
	. . . .	. . . .
Ex. no.: **1**	. . . .	. . . .
	. . . .	. . . .
Topic: ***Circuit elements and circuit laws***		*Instructor's signature*
		Date

Equivalent resistance

Table 1.2: Resistance R_0 = Ω computed in MATLAB

u [V]						
i [mA]						

Table 1.3: Measured resistance

R_1 [kΩ]	R_2 [kΩ]	R_1 in series with R_2 [kΩ]	R_1 in parallel with R_2 [kΩ]

Table 1.4: Measured resistance $R_{\text{A-C}}$ [kΩ] for R_x = kΩ

R_x disconnected	$R_x \to$ A-B	$R_x \to$ A-C	$R_x \to$ A-D	$R_x \to$ C-B	$R_x \to$ C-D	$R_x \to$ B-D

R_1 disconnected	R_2 disconnected	R_3 disconnected	R_4 disconnected

Verification of Kirchhoff's laws

Table 1.5: Measured voltages [V]

U_{E_1}	U_{R_1}	U_{D_2}	U_{R_2}	U_{D_3}	U_{E_3}

$U_{E_1} + U_{R_1} - U_{R_2} - U_{D_2}$	$U_{D_2} + U_{R_2} - U_{E_3} - U_{D_3}$

Table 1.6: Measured currents $[\mathrm{mA}]$

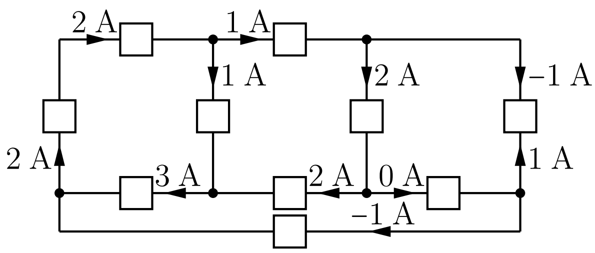
I_1	I_2	I_3

$I_1 + I_2 + I_3$

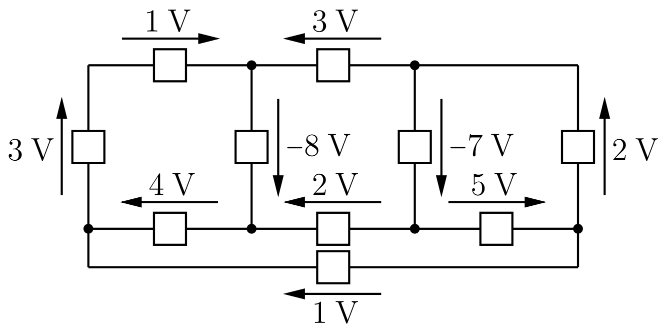
Bibliography

Solutions to self-study problems

1.



2.



3. The resistance "seen" from two adjacent terminals is $\frac{7}{12}$ Ω; from terminals on a diagonal of a face of the cube: $\frac{3}{4}$ Ω; and from terminals on a diagonal of the cube: $\frac{5}{6}$ Ω.

4. The validity and precision of the achieved estimation may be verified by comparing it with the exact value of the equivalent resistance R, which is

$$R = \frac{378955}{19689}\ \Omega \approx 19.25\ \Omega.$$

For example, by removing "vertical" resistors we get an inequality

$$R \leq 21\ \Omega,$$

whereas by short-circuiting the same resistors we get

$$R > 16.9\ \Omega.$$

Chapter 2

Thévenin and Norton equivalents

2.1 Introduction

2.1.1 Thévenin's and Norton's theorems

For the needs of this chapter let us define a linear DC (direct current) one-port as a two-terminal subcircuit consisting only of direct voltage sources, direct current sources and linear resistors.

Thévenin's theorem[1]**.** Every linear DC one-port which is not equivalent to a current source is equivalent to a resistor R_T and a constant voltage source E_T connected in series (see Fig. 2.1[2]). Such a connection is called a Thévenin equivalent. Resistance R_T is called the internal resistance or the output resistance of the considered one-port.

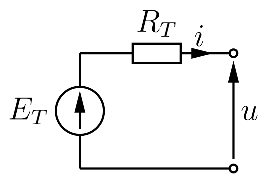


Figure 2.1: Thévenin equivalent source

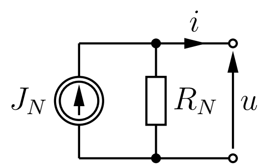


Figure 2.2: Norton equivalent source

Norton's theorem[3]**.** Every linear DC one-port which is not equivalent to a voltage source is equivalent to a resistor R_N and a direct current source J_N connected in parallel (see Fig. 2.2). Such a connection is called a Norton equivalent. Resistance R_N is called the internal resistance or the output resistance of the considered one-port.

[1]Thévenin's theorem was formulated by a French engineer, Léon Charles Thévenin in 1883. In fact, it was discovered even earlier (see [1]) by a German scientist, Hermann von Helmholtz, who gave and proved the theorem in 1853, i.e. four years before Thévenin was born. Thus, the same theorem is sometimes referred to as Helmholtz's or Thévenin-Helmholtz's theorem.

[2]In Fig. 2.1 voltage u and current i are labeled in lower case (not in upper case as we usually do in DC circuits) to stress that these are the variables in terms of which we define the given two-terminal subcircuit. For the same reason, when we draw current-voltage characteristic of a Thévenin equivalent (see Fig. 2.6), we label the axes of the coordinate system with letters u and i.

[3]Norton's theorem was formulated by an American engineer, Edward Lawry Norton in 1926. In the same year it was also stated and proved by a German physicist, Hans Ferdinand Mayer (see [1]). Thus, the same theorem is also referred to as Mayer-Norton's theorem.

Thévenin's and Norton's theorems may be adapted to other types of circuits (e.g. alternating or periodic current circuits) and extended to circuits including other linear components such as, for example, controlled sources or transformers.

Thévenin's and Norton's theorems imply that for every linear DC one-port which is neither equivalent to a voltage nor to a current source, there exist both its Thévenin and its Norton equivalents and they are related with the following formulas

$$R_T = R_N \qquad \text{and} \qquad E_T = J_N R_N. \tag{2.1}$$

2.1.2 Finding parameters of Norton and Thévenin equivalents

In this section we present a number of methods for the determination of the parameters of Thévenin and Norton equivalents. Notice that both Thévenin's and Norton's theorems imply that every linear DC one-port, schematically depicted in Fig. 2.3 that allows short-circuiting and open-circuiting (as depicted in Fig. 2.4 and 2.5) has its Thévenin and Norton equivalents with parameters given by the following formulas (see Fig. 2.1 and 2.2):

$$E_T = U_{\text{oc}}, \qquad J_N = I_{\text{sc}}, \qquad R_T = R_N = \frac{E_T}{J_N}.$$

In particular, the electromotive force of the source of the Thévenin equivalent is the open-circuit voltage across the one-port, and the current of the source of the Norton equivalent is the short-circuit current of the one-port.

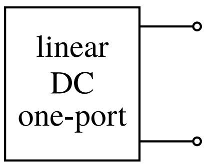


Figure 2.3

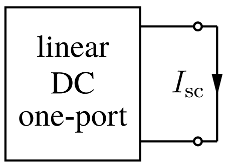


Figure 2.4: Short-circuit current

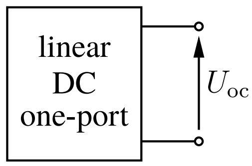


Figure 2.5: Open-circuit voltage

Many real life one-ports do not allow to be short-circuited or open-circuited because they can be thus damaged. Usually the problem concerns short-circuiting because it may result in a large short-circuit current causing high power dissipation. Thus, sometimes we need alternate methods of determining the parameters of Thévenin and Norton equivalents. All of them can be derived from the current-voltage characteristics of the equivalents. Such a characteristic, given in terms of variables introduced in Fig. 2.1 and 2.2, is presented in Fig. 2.6. It is a straight line crossing the voltage and current axes at E_T and J_N, respectively. Its slope is $-\frac{1}{R_T} = -\frac{1}{R_N}$.

On the basis of the current-voltage characteristic, we may find the parameters of equivalent subcircuits in a number of ways. One possibility has been already given: we take the open-circuit voltage for E_T, the short-circuit current for J_N, and then the internal resistance as the ratio E_T/J_N. Other possibilities include:

1. First, determine the open-circuit voltage $E_T = U_{\text{oc}}$ in the circuit of Fig. 2.5. Then, find voltage U_1 and current I_1 in the circuit of Fig. 2.7 which is obtained by connecting the considered one-port with another one which is not an open-circuit. We then have:

$$R_T = R_N = \frac{E_T - U_1}{I_1}, \qquad J_N = \frac{E_T}{R_T}. \tag{$\star$}$$

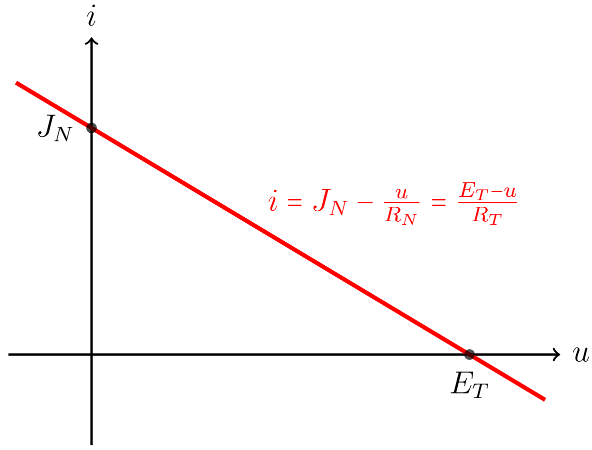


Figure 2.6: Current-voltage characteristic of Norton/Thévenin equivalent

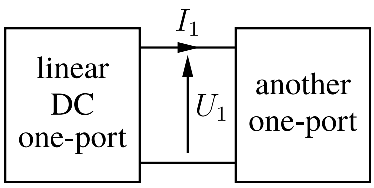


Figure 2.7

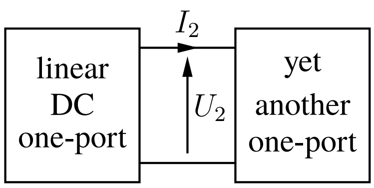


Figure 2.8

2. Determine two different voltage-current pairs (U_1, I_1) and (U_2, I_2) resulting from connecting the analyzed one-port subsequently with two other one-ports as depicted in Fig. 2.7 and 2.8. We then have

$$R_T = R_N = \frac{U_2 - U_1}{I_1 - I_2}, \quad E_T = U_1 + I_1 R_T = U_2 + I_2 R_T, \quad J_N = I_1 + \frac{U_1}{R_N} = I_2 + \frac{U_2}{R_N}. \quad (\star\star)$$

3. First, determine the open-circuit voltage E_T. Next, determine resistance R_0 of the resistor that when connected to the considered one-port as shown in Fig. 2.9 results in $U_1 = \frac{E_T}{2}$. Then

$$R_T = R_N = R_0, \qquad J_N = \frac{E_T}{R_0}.$$

In practice, we can determine the appropriate resistance of the resistor R_0 by connecting the considered one-port to a potentiometer (as in Fig. 2.10). The potentiometer is adjusted until equality $U_1 = \frac{E_T}{2}$ is achieved. Then, the resistance obtained by means of the potentiometer is measured with an ohmmeter.

Let us mention two other tools that can be useful in determining the parameters of the equivalents, at least in the case when we can modify the "internal structure" of the analyzed one-port. The first of these tools is the superposition rule. It allows to determine each of the parameters E_T and J_N as sums of voltages and currents accordingly. The summands of these sums result from letting one independent source (in a turn) act alone and reducing all other sources to zero. By reducing a current source to zero we mean replacing it with a gap

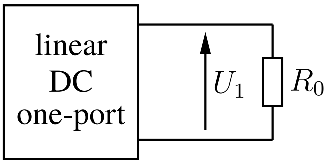


Figure 2.9

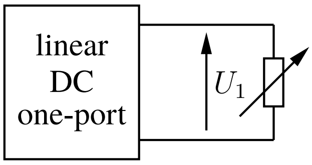


Figure 2.10

(open-circuit), and by reducing a voltage source to zero we mean replacing it with a wire (short-circuit). The second of the announced tools is the fact that the internal resistance of a linear DC one-port is the equivalent resistance (directly measurable with an ohmmeter) of the one-port obtained by reducing all independent sources of the original one-port to zero.

Problems for self study

Solutions to the problems are given on page 45.

1. Design two one-ports, each of which is built of $1\ \mathrm{V}$ voltage source and two $1\ \mathrm{k\Omega}$ resistors, such that their Thévenin equivalents have different and non-zero internal resistances and the same electromotive forces.

2. Assume that you have three resistors $2\ \mathrm{k\Omega}$ each, and three current sources: $1\ \mathrm{mA}$, $3\ \mathrm{mA}$, $6\ \mathrm{mA}$ at your disposal. Design a one-port whose Norton equivalent's parameters are $J_N = 5\ \mathrm{mA}$, $R_N = 3\ \mathrm{k\Omega}$.

3. Prove formulas ($\star$) and ($\star\star$).

2.2 Homework

In the laboratory exercise you will analyze the one-port of Fig. 2.11. Values of parameters:

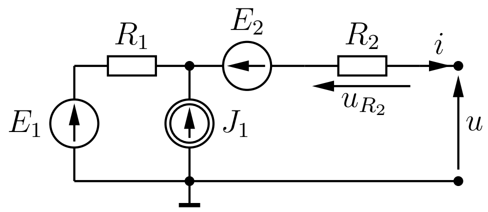


Figure 2.11: The linear DC one-port used in the exercise

E_1, E_2, J_1, R_1, R_2 are given in Tab. 2.1 (they depend on the number of the stand at which you do the exercise).

Table 2.1

Stand no.	E_1 [V]	E_2 [V]	J_1 [mA]	R_1 [kΩ]	R_2 [kΩ]
1	15	5	-5	0,5	2
2	15	5	-5	0,5	1
3	15	5	-4	1	5
4	15	5	-4	1	2
5	15	5	-4	1	0,5
6	15	5	-4	1	0,2
7	15	5	-2	2	1
8	5	-5	-5	0,5	2
9	5	-5	-5	0,5	1
10	5	-5	-4	1	5
11	5	-5	-4	1	2
12	5	-5	-4	1	0,5
13	5	-5	-4	1	0,2
14	5	-5	-2	2	1
15	5	-5	-1	1	2
16	5	-5	1	2	1

Homework 2.1. Taking appropriate parameters from Tab. 2.1, find the parameters of Thévenin and Norton equivalents (E_T, R_T, J_N, R_N) for the one-port of Fig. 2.11.

Homework 2.2. Determine and draw the relationship between voltage u_{R_2} and voltage u in the circuit of Fig. 2.12. Assume the parameters of the circuit have the same values as in the previous task.
Tip: We already know that the one-port to the left of the indicated terminals has the Thévenin equivalent with parameters E_T and R_T and thus $i = (E_T - u)/R_T$.

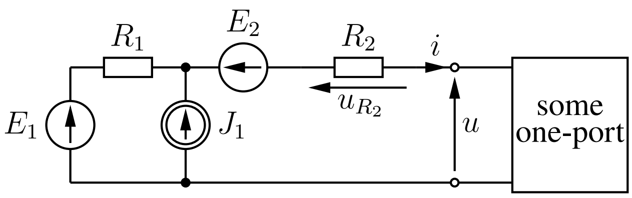


Figure 2.12

2.3 Parameters of Thévenin and Norton equivalents

In this exercise we will examine one-port shown in Fig. 2.11. The values of particular elements of this one-port depend on the lab stand number according to Tab. 2.1. We will use various methods to measure parameters of its Thévenin and Norton equivalents. We will also determine voltage-current characteristic of this one-port (which at the same time is the characteristic of the equivalents).

2.3.1 Measurement setup

Power off the crate and switch off `CH3` channel of the power supply. Assemble measurement setup according to Fig. 2.13, taking values of elements according to Tab. 2.1. For this purpose:

1. Connect $15\ \mathrm{V}$ or $5\ \mathrm{V}$ grounded voltage source E_1 element to a supply socket splitter in the tray, by means of which it is connected to the voltage supply in the crate.

2. Connect the current source card to the tray using a BNC cable and a BNC-banana plug converter, minding proper polarization: plug the 'cold' wire into green banana socket, and the 'hot' one into the red socket. Set the appropriate value *and sign* of current on the front panel of the current source card. Connect the current source element of the El-Go set to the supply socket splitter in the tray, by means of which it is connected to the current source card.

3. Set the voltage value for `CH3` channel of the power supply to $5\ \mathrm{V}$, and its current limit to $0.1\ \mathrm{A}$. Connect the output of `CH3` channel to the tray by means of two cables (with banana plugs), minding polarization of the voltage[4]. Connect the voltage source element $E_2 = 5\ \mathrm{V}$ of the El-Go set, to the second supply socket in the tray, by means of which it is connected with `CH3` channel of the power supply. **Important! Out of the available elements pick the 'universal' voltage source labelled with *lower* case letter e, *not* the $5\ \mathrm{V}$ source connected to ground.**

Before turning the power on, show the assembled setup to the lab instructor for verification. Power on the crate and the devices.

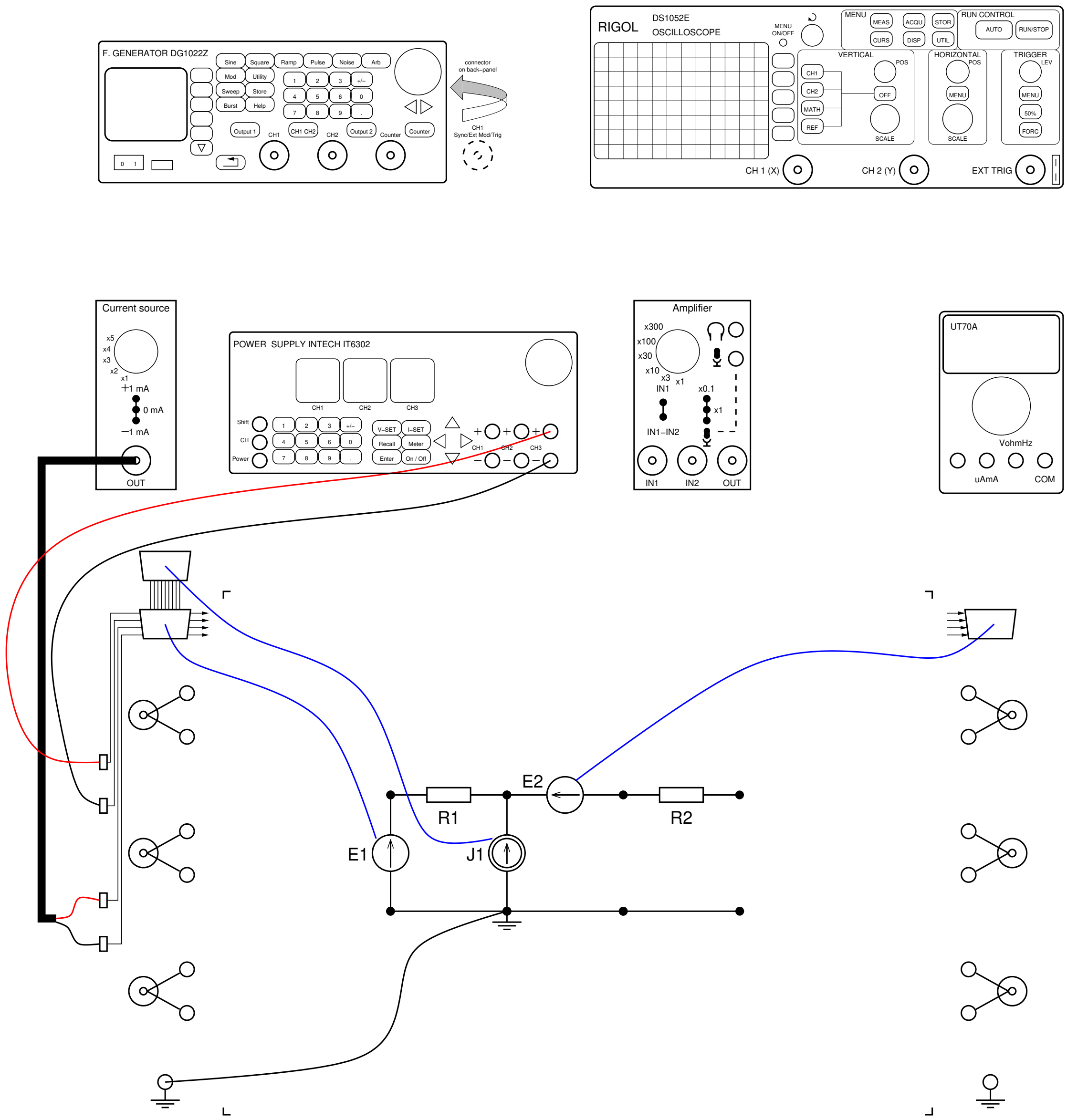


Figure 2.13: Initial measurement setup

2.3.2 Measurement of Thévenin equivalent's electromotive force

The Thévenin equivalent's electromotive force can be measured as the voltage across opened terminals of a one-port. For this purpose, we will use digital multimeter UT70A set to DC voltmeter in the range $20\ \mathrm{V}$. Internal resistance of this voltmeter (of the order of $10\ \mathrm{M\Omega}$) is by five orders larger than the value R_T calculated in homework, so we can consider the voltmeter an open circuit. Connect the multimeter to the setup shown in Fig. 2.14 (red *solid* line connected to the multimeter).

[4] If for a given lab stand the value of electromotive force E_2 given in Tab. 2.1 is negative, the cables should be *crossed* compared to Fig. 2.13, i.e. the red and the black cable on the voltage supply side should be swapped, while polarization on the side of the investigated setup should remain unchanged.

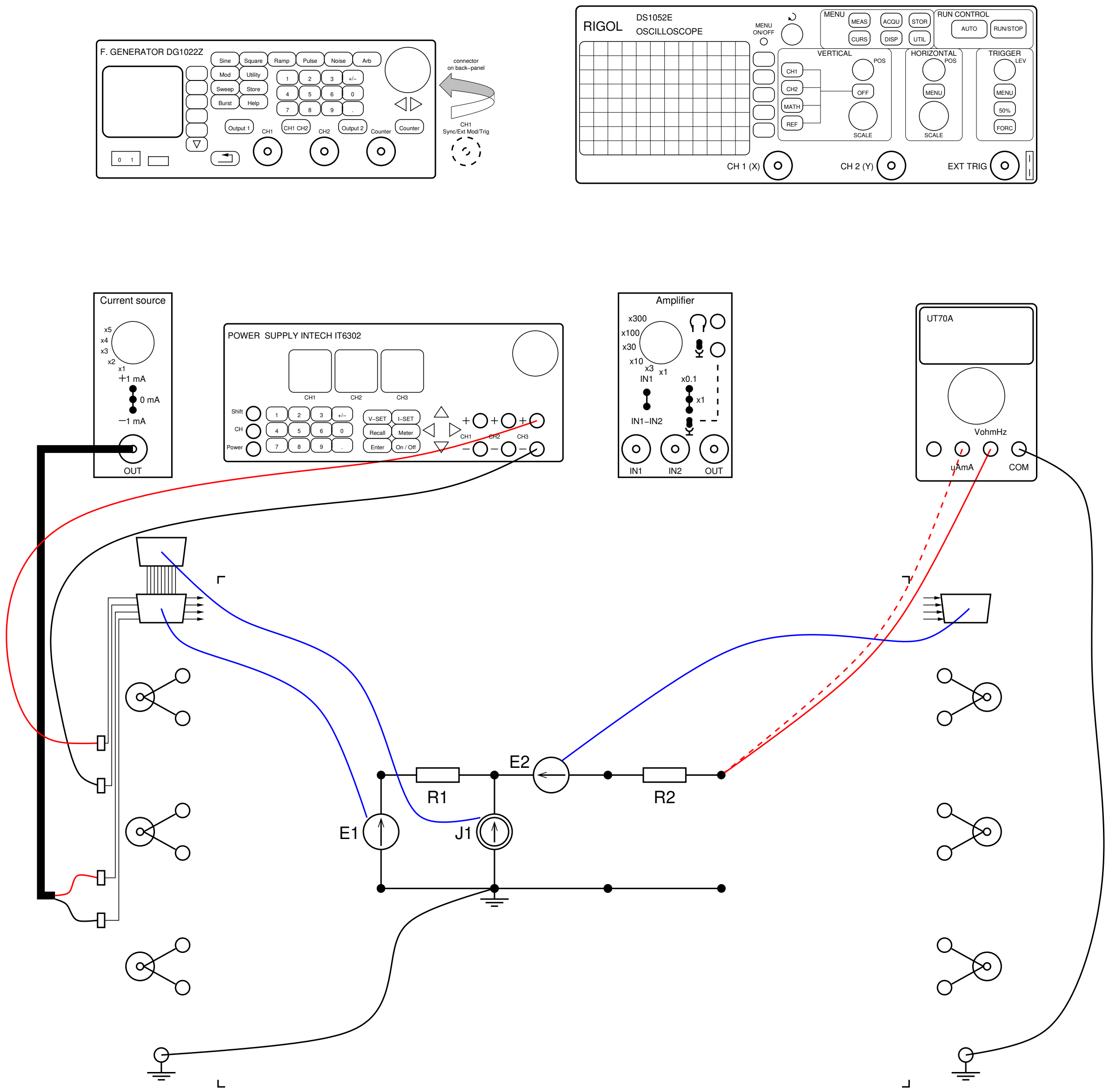


Figure 2.14: Setup for the measurement of the Thévenin equivalent's electromotive force, Norton equivalent's current (red dashed line) and their internal resistance

2.3.2.1 Direct measurement of Thévenin equivalent's electromotive force

Measure the value E_T with the voltmeter, record it in Tab. 2.2 and compare with the value obtained in homework. The latter value should also be recorded in this table, in the field $E_{T\ \text{homework}}$.

2.3.2.2 Measurement of Thévenin equivalent's electromotive force using superposition rule

We can determine parameters E_T and J_N using superposition rule. According to this rule, we sum up effects from each of the independent sources acting alone. In the analyzed circuit, measure open-circuit voltages

$$\begin{aligned} E'_T &= E_T|_{E_2=0,J_1=0} \\ E''_T &= E_T|_{E_1=0,J_1=0} \\ E'''_T &= E_T|_{E_1=0,E_2=0} \end{aligned}$$

Recall that reduction of a *voltage* source to zero means replacing it with a *wire*, in our case it means removal of the voltage source element from the circuit and *replacing* it with a long magnetic rod from the El-Go set[5]. Reducing a *current* source to zero means replacing it with a *gap*, i.e. disconnecting the current source element from the circuit, *without* replacing it with a rod. The same effect can be achieved (although it is not as 'educative' as the first way) by either disconnecting the BNC cable form the current source card or by switching the polarization knob in this card to position $0\ \mathrm{mA}$. Record the measured values of voltages and their sum in Tab. 2.2. Compare the sum with the value E_T measured directly.

2.3.3 Measurement of Norton equivalent's current

Norton equivalent's current can be measured as the current through the short-circuited terminals of a one-port. For that purpose, we will use digital multimeter UT70A set to DC ammeter in the range $200\ \mathrm{mA}$. We can assume that the internal resistance of the ammeter is much smaller than the value R_T determined in homework, so we are justified to consider it a short circuit. Connect the multimeter to the measurement setup as shown in Fig. 2.14 (red *dashed* line connected to the multimeter).

2.3.3.1 Direct measurement of Norton equivalent's current

Measure the value J_N with the ammeter, record it in Tab. 2.3, and compare with the value obtained in homework (the value obtained in homework should also be recorded in the same table in the field $J_{N\ \text{homework}}$) .

2.3.3.2 Measurement of the Norton equivalent's current using superposition rule

In a similar way as before, the Norton equivalent's current can be determined using superposition rule. For this purpose, in the analyzed circuit measure the short-circuit currents:

$$\begin{aligned} J'_N &= J_N|_{E_2=0,J_1=0} \\ J''_N &= J_N|_{E_1=0,J_1=0} \\ J'''_N &= J_N|_{E_1=0,E_2=0}\,. \end{aligned}$$

The reduction to zero can be realized in the same way as in the measurement of Thévenin source electromotive force. Record values measured with the ammeter and their sum in Tab. 2.3. Compare the sum with J_N value measured directly.

[5]Important! If you disconnect the source element only from the supply socket DB-9, it will become an *open circuit*, not a short circuit. That is why it is necessary to insert a rod in place of the switched-off *voltage* source.

2.3.4 Measurement of the internal resistance

2.3.4.1 Indirect measurement of the internal resistance based on measurement of E_T and J_N

When knowing Thévenin equivalent's EMF and Norton equivalent's current, one can determine the internal resistance of these equivalents (which is equal to the internal resistance of the original one-port):

$$R_T = R_N = \frac{E_T}{J_N}$$

By using the results of direct measurements of E_T and J_N, compute the value R_T and record it in Tab. 2.4. Compare it with the value determined in homework.

2.3.4.2 Direct measurement of the internal resistance with an ohmmeter

When having access to the 'internal structure' of the analyzed one-port, we can measure its internal resistance as the equivalent resistance of a sourceless one-port obtained by reducing to zero all of the independent sources in the original one-port. For this purpose, we will use digital multimeter UT70A set to measure resistance. Assemble the measurement setup as shown in Fig. 2.14 with the red *solid* line connected to the multimeter. Reduce to zero *all* of the current and voltage sources (current sources should be removed from the circuit, voltage sources must be replaced with short-circuits). Record the measured value of resistance in Tab. 2.4 and compare it with the value obtained in the homework. When the measurement is completed, connect all of the three sources E_1, E_2 and J_1 back to the analyzed circuit.

2.3.4.3 Measurement of the internal resistance using 'half-voltage' method

Assemble the measurement setup as shown in Fig. 2.15, using digital multimeter UT70A as a DC voltmeter in the range 20 V. By manipulating the knob of the adjustable resistor R_0 (potentiometer adjustable in the range $0 \ldots 10$ kΩ), make the voltmeter show half of the value E_T measured directly. Then, detach the potentiometer from the circuit and measure its resistance with the multimeter which operates as an ohmmeter. Record the measured value of resistance in Tab. 2.4 as $R_{T\ \text{'1/2' met.'}}$ and compare it with the values obtained by using other methods.

2.4 Measurement of Thévenin and Norton equivalents characteristic

In this section, we will use a setup shown in Fig. 2.16. The characteristic of the analyzed one-port (Fig. 2.11) can be captured with an oscilloscope which operates in X-Y mode. Oscilloscopes measure *voltages* only hence the need for a voltage which is proportional to current i through the one-port. In the analyzed circuit, the voltage across resistor R_2 is the most convenient choice for such an indirect measurement of current i (see homework 2.2). However, it is a 'floating' voltage, i.e., none of the terminals of this resistor is grounded and oscilloscopes measure electrical potentials with respect to ground only, i.e. voltages between arbitrary nodes

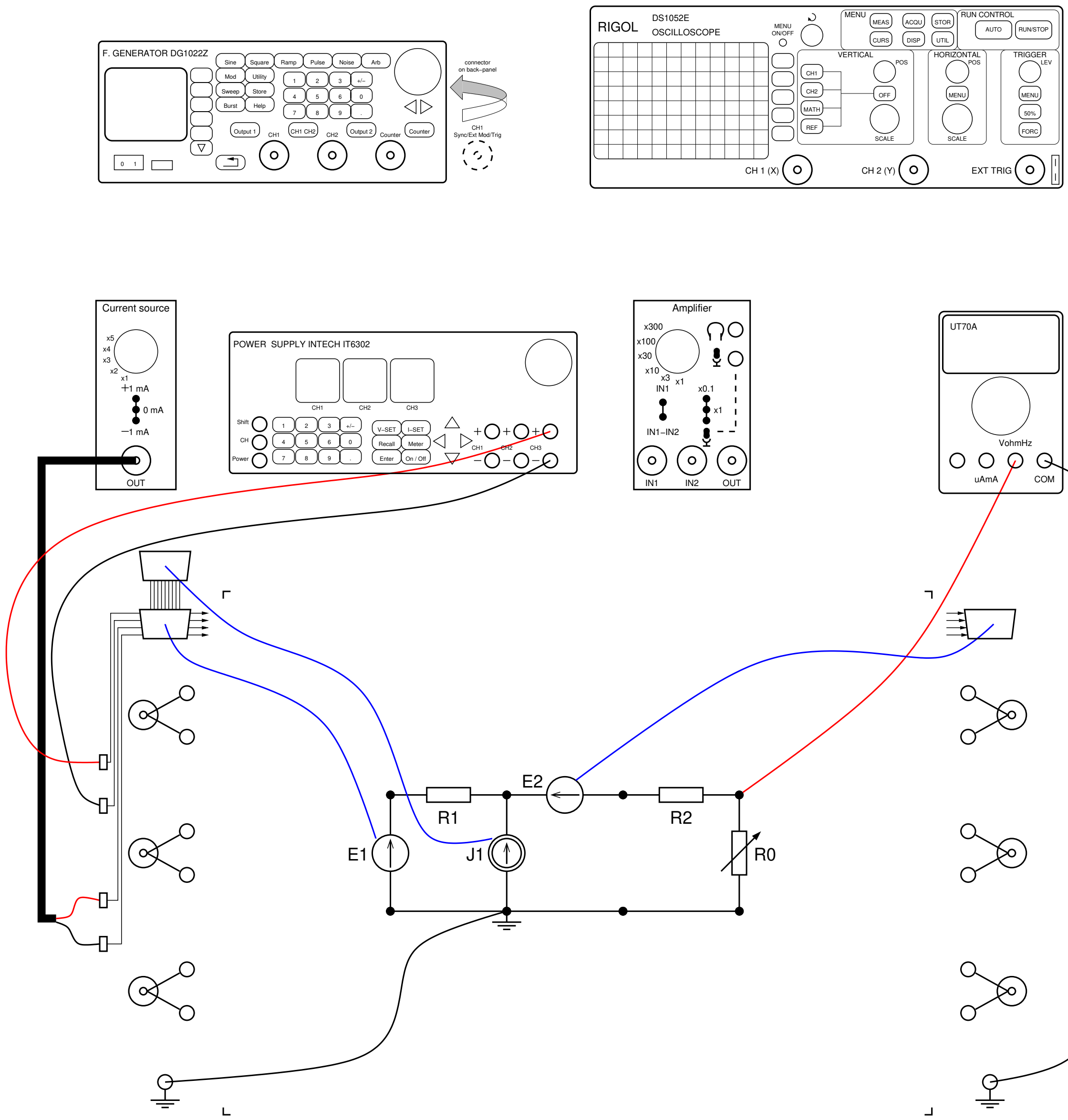


Figure 2.15: Setup for the measurement of the internal resistance by using 'half-voltage' method

and the ground node. For this reason we will measure current i using amplifier card set to subtract the electrical potential of the right terminal of R_2, which is voltage u across the analyzed one-port, and that of the left terminal of R_2. As a result it will give voltage $u_{R_2} = R_2 i$.

To observe the characteristic on the oscilloscope, one should enforce various values of voltage across the terminals of the analyzed one-port. For this purpose, we will use the sawtooth waveform generator connected to the terminals of the one-port. The function generator is not an ideal voltage source as its internal resistance equals $50\ \Omega$. Therefore, the range of its output voltages might need some adjustments after it is connected to the analyzed one-port.

1. Power off the crate and the devices. Assemble measurement setup according to Fig. 2.16. **Before turning the power on, show the assembled setup to the lab instructor for verification.** Power on the crate and the devices.

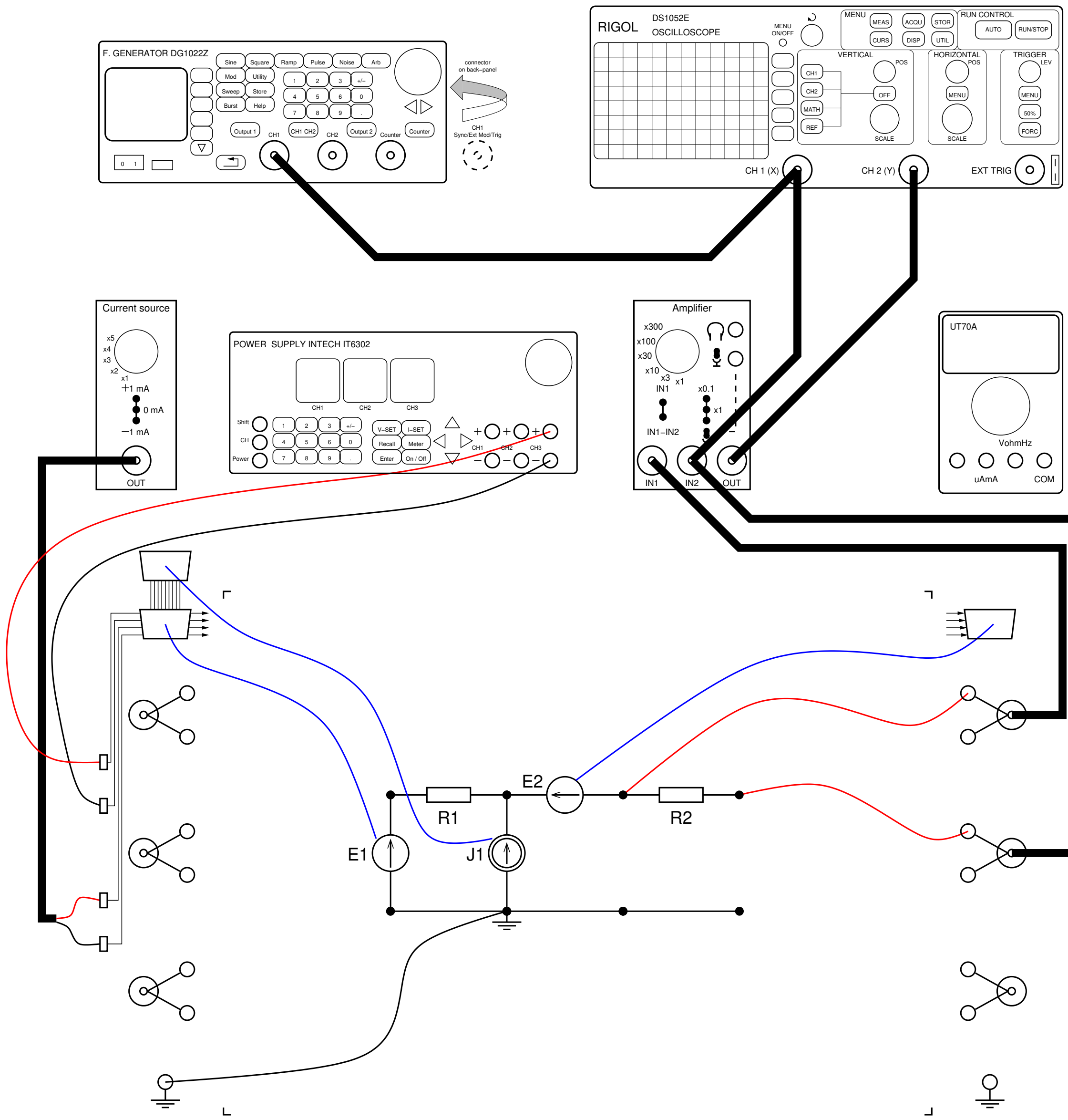


Figure 2.16: Setup for the measurement of the Thévenin/Norton equivalent voltage-current characteristic

2. Set DC coupling in both oscilloscope channels, their sensitivity to $1\ \mathrm{V/div}$, and the time base to $1\ \mathrm{ms/div}$. At first, only activate channel X and set it as the trigger source.

3. Set the function generator to generate sawtooth wave (`Ramp`) with frequency $100\ \mathrm{Hz}$, symmetry set to 50%, minimum value (`LoLevel`) set to $-0.5\ \mathrm{V}$, and maximum value set to $E_T + 0.5\ \mathrm{V}$. Switch on the output of the function generator and verify the parameters of the generated signal with the oscilloscope.

4. Switch the oscilloscope to X-Y mode. Both channels should be DC coupled. By using `POSITION` knob for one channel and then for the other, set the origin of coordinates two divs to the right and one div up from the bottom left corner of the screen. (The origin is indicated by colour 'arrows' in the left and upper display margins.) If needed, adjust the amplitude and DC offset of the signal from the generator, and sensitivity and

shift of the two oscilloscope channels.

5. Based on the captured oscillogram, draw voltage-current characteristic of the investigated one-port. Find values E_T, J_N and R_T on the characteristic and record them in Tab. 2.2, 2.3 and 2.4. Compare these results with the ones obtained with other methods.

2.5 Verification of the theorems on equivalent sources

In this exercise we will try to verify that the Norton and Thévenin equivalents with parameters determined in the previous exercise are indeed equivalent to the analyzed one-port. For this purpose, consider the analyzed one-port loaded with a series connection of a blue LED and a protective resistor $R_{\text{LED}} = 100\ \Omega$ (see Fig. 2.17). We will verify that the voltage U and the current I shown in Fig. 2.17 are the same regardless of whether the one-port surrounded with a dashed line is the original one-port or whether it is realized as its Thévenin/Norton equivalent.

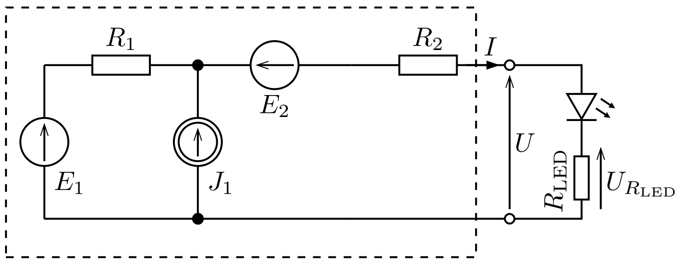


Figure 2.17: The analyzed linear DC one-port loaded with a LED and a resistor

2.5.1 Determination of the operating point of the analyzed one-port

Assemble the measurement setup according to Fig. 2.18. When it is assembled, the diode should glow. Use digital multimeter UT70A set to measure DC voltages in the range 20 V, to measure the voltage U across the load (red *solid* line connected to the multimeter in Fig. 2.18) and to indirectly measure the current through the load $I = U_{R_{\text{LED}}}/R_{\text{LED}}$ by measuring voltage across resistor R_{LED} (red *dashed* line connected to the multimeter in Fig. 2.18). Measure both voltages U and $U_{R_{\text{LED}}}$. Calculate current I using measured valued of $U_{R_{\text{LED}}}$. Mark the measured operating point (U, I) on the graph of the analyzed one-port voltage-current characteristic obtained in the previous exercise.

If time permits, you may check if superposition rule holds for the voltage across the analyzed *nonlinear* one-port as it did for the determined earlier open-circuit voltage and short-circuit current.

2.5.2 Replacement of the analyzed one-port with its Thévenin or Norton equivalent

Check whether the operating point of the LED load will not change when the analyzed one-port is replaced with its Thévenin or Norton equivalent, For this purpose, assemble the setup according to Fig. 2.19 or according to Fig. 2.20. In the case of Thévenin equivalent, set the voltage of `CH1` channel of the power supply to appropriate voltage value (equal to E_T). **Do not use `CH3` channel at this stage of experiment, as `CH3` voltage is limited to** 5 V**.** In the case of Norton equivalent, set the current source card to appropriate current value that is equal

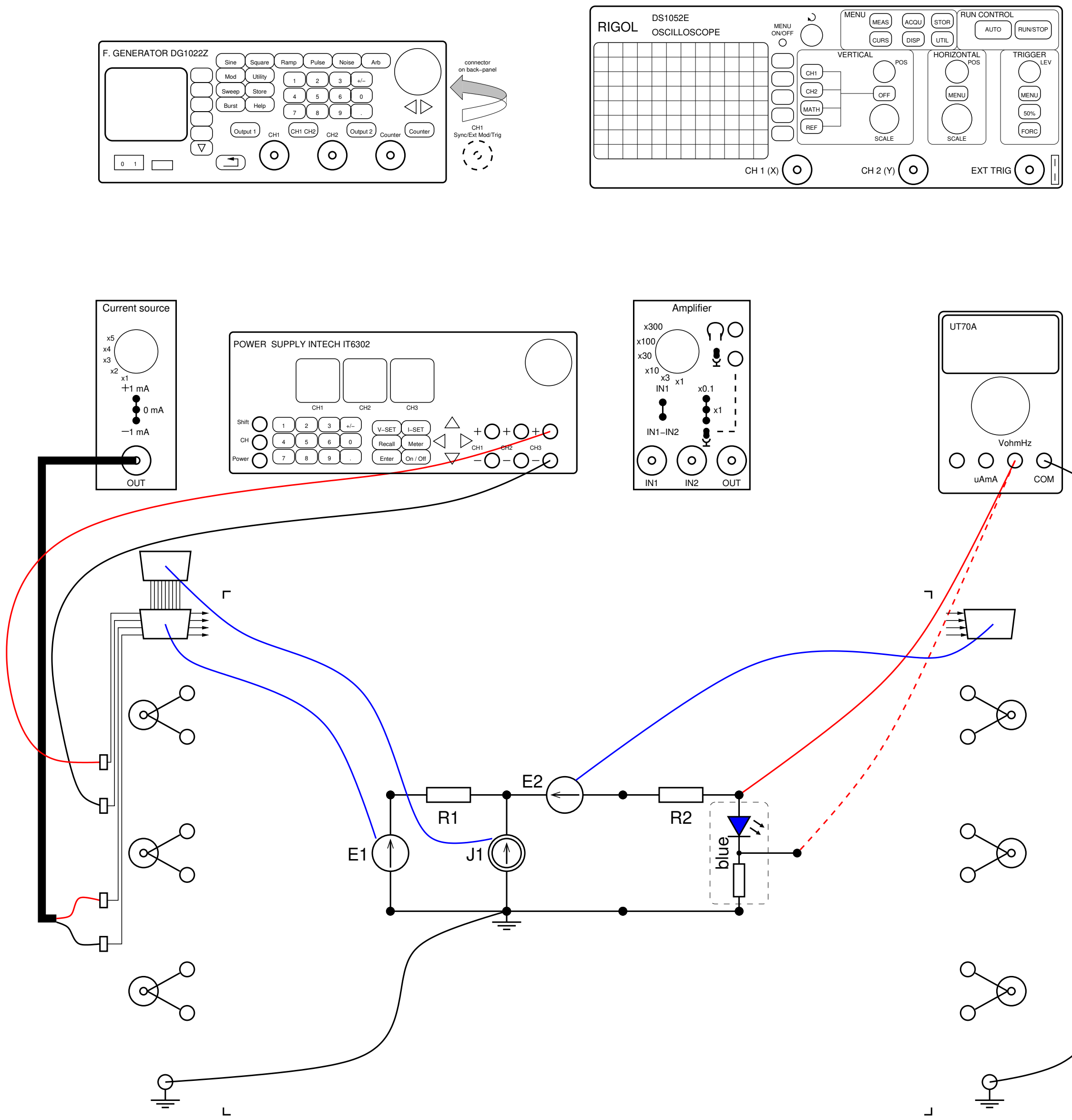


Figure 2.18: Setup for the measurement of the operating point for loaded analyzed one-port

or at least close to J_N. In both casses, you may use a series connection of resistors R_1 and R_2 from the original circuit as resistor $R_T = R_N$. Comment why it is a valid approach in the lab report. Measure the operating point $(U_{\text{eq}}, I_{\text{eq}})$ of the load once again. Compare the measured operating point for the Thévenin and Norton equivalent with the operating point (U, I) of the original circuit.

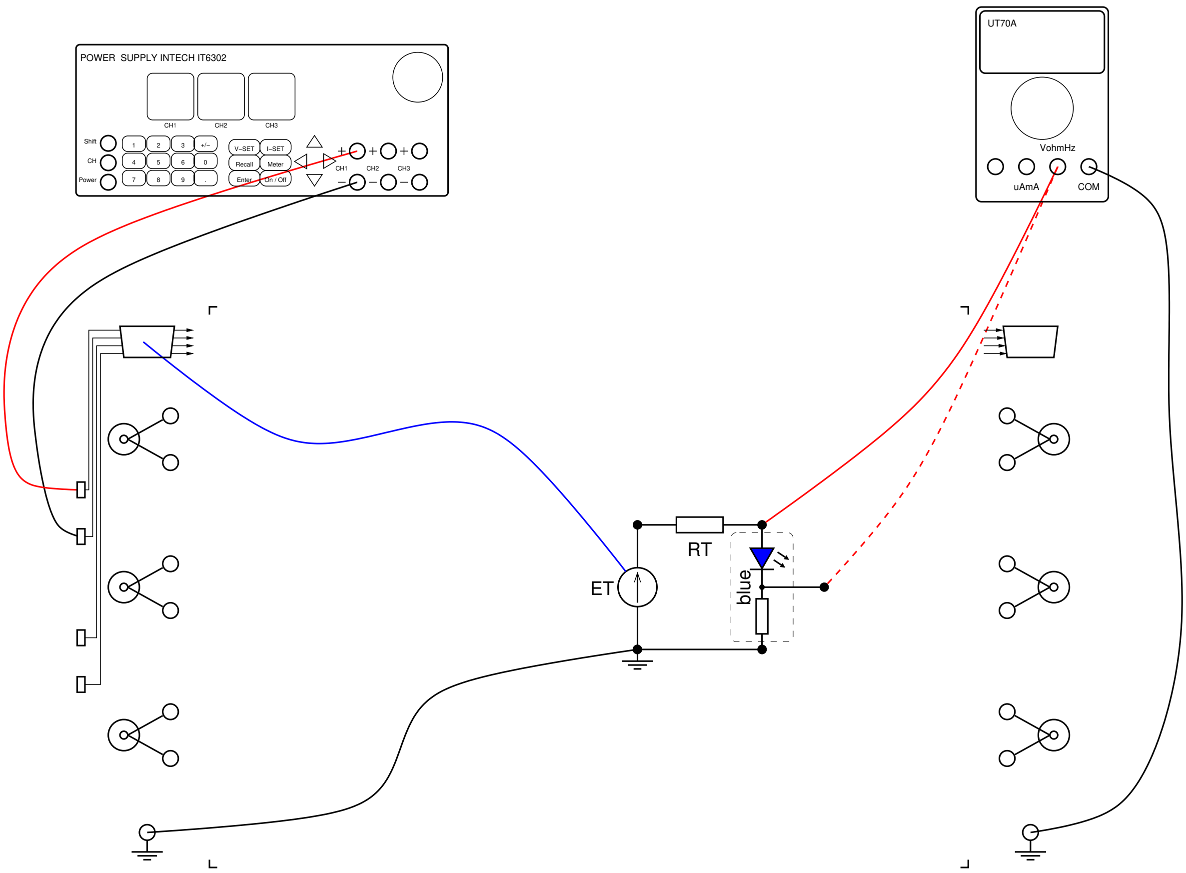


Figure 2.19: Setup for the measurement of the operating point of the loaded Thévenin equivalent

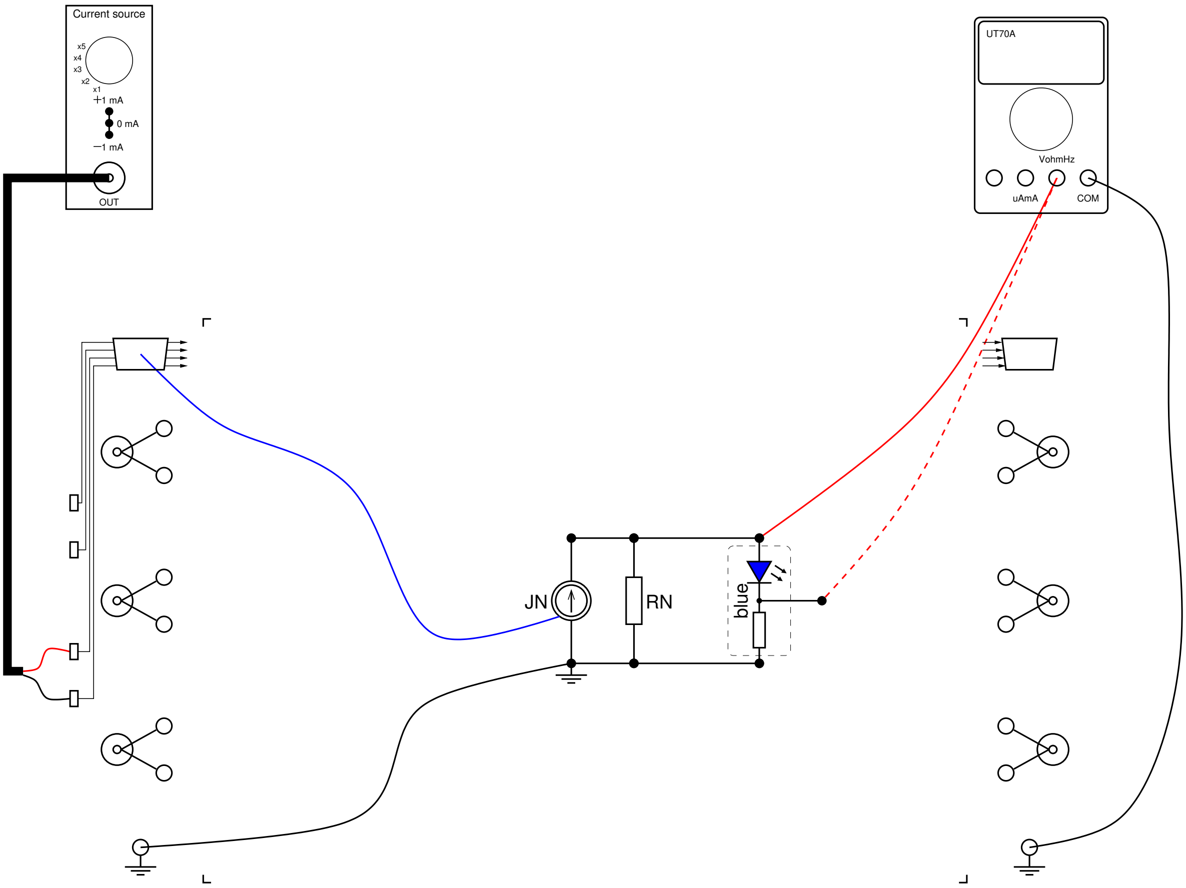


Figure 2.20: Setup for the measurement of the operating point of the loaded Norton equivalent

Stand:	*Name and SURNAME*	*Grade*
Ex. no.: **2**		
Topic: ***Thévenin and Norton equivalents***		*Instructor's signature*
		Date

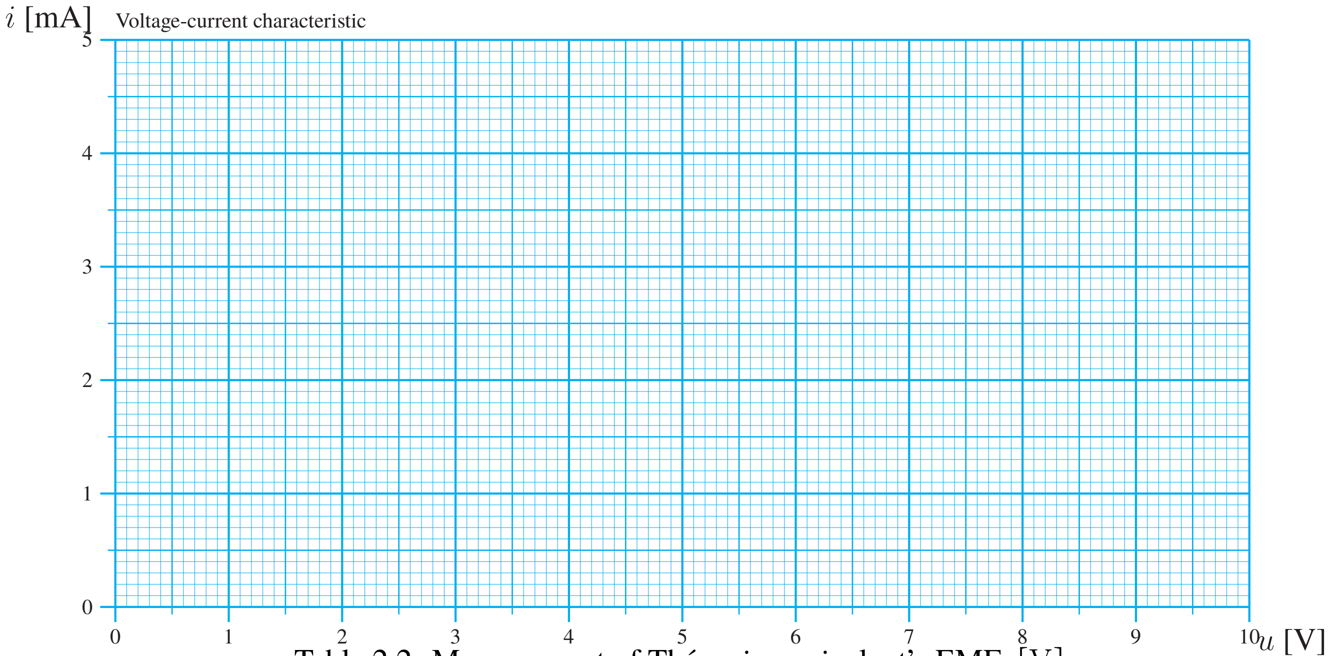


Table 2.2: Measurement of Thévenin equivalent's EMF [V]

E_T homework	E_T	E'_T	E''_T	E'''_T	$E'_T + E''_T + E'''_T$	E_T char u-i

Table 2.3: Measurement of Norton equivalent's current [mA]

J_N homework	J_N	J'_N	J''_N	J'''_N	$J'_N + J''_N + J'''_N$	J_N char u-i

Table 2.4: Measurement of Thévenin/Norton equivalent's internal resistance [kΩ]

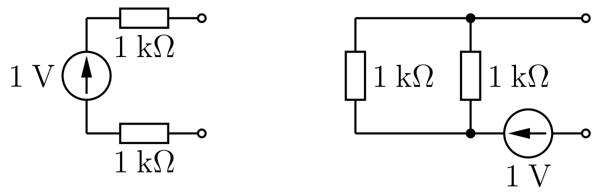
R_T homework	R_T	R_T ohmmeter	R_T '1/2' met.	R_T char u-i

Bibliography

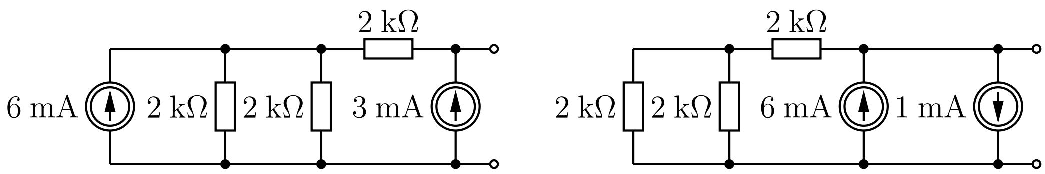
[1] Don Johnson. Origins of the equivalent circuit concept: The voltage-source equivalent. In *Proceedings of the IEEE*, volume 91, pages 636–640, 2003.

Solutions to self-study problems

1. Below, there is presented a sample solution to the problem, i.e. two one-ports which have Thévenin equivalents with same electromotive forces (1 V) and different internal resistances (2 kΩ and $\frac{1}{2}$ kΩ, respectively).



2. Here is an example of two one-ports that meet the problem requirements:



3. Points with coordinates

$$(i, u) = (\text{the current through this one-port, the voltage across the one-port terminals})$$

form the current-voltage characteristic of the given one-port. In the case of linear DC one-ports which are not ideal (voltage or current) sources, this characteristic is given by equation (see Fig. 2.6)

$$i = J_N - \frac{u}{R_N} = \frac{E_T - u}{R_T}. \tag{2.2}$$

Substituting pairs

$$(u, i) = (U_r, 0\ \mathrm{A}),$$
$$(u, i) = (U_1, I_1), \quad I_1 \neq 0,$$

into Equation (2.2) we get the following set of equations

$$0\ \mathrm{A} = \frac{E_T - U_r}{R_T},$$
$$I_1 = \frac{E_T - U_1}{R_T},$$

by which we get (taking into account (2.1)) Equations ($\star$).

Similarly, substituting pairs

$$
\begin{aligned}
(u,i) &= (U_1, I_1),\\
(u,i) &= (U_2, I_2) \neq (U_1, I_1)
\end{aligned}
$$

into Equation (2.2) we get

$$
\begin{aligned}
I_1 &= \frac{E_T - U_1}{R_T},\\
I_2 &= \frac{E_T - U_2}{R_T}.
\end{aligned}
$$

These equations result in

$$E_T = U_1 + I_1 R_T = U_2 + I_2 R_T,$$

and then also

$$U_1 - U_2 = R_T (I_2 - I_1).$$

Equations ($\star\star$) are obtained from the above equations by supplementing them with Relations (2.1).

Chapter 3

Alternating Current circuits

3.1 Introduction

3.1.1 Phasors

Any signal $x(t)$ that can be written in the form

$$x(t) = X_m \cos(\omega t + \varphi_x), \quad X_m \geq 0, \quad \omega > 0 \tag{3.1}$$

is called an alternating signal. X_m is called amplitude, ω — pulsation, and φ_x — the initial phase of the signal.

A circuit in which all the currents and voltages are alternating and of the same pulsation is called an alternating current circuit, or AC circuit, for brevity.

It is convenient to describe alternating signals using phasors. A phasor of an alternating signal (3.1) is a complex number X[1] defined as

$$X = X_m \mathrm{e}^{\mathrm{j}\varphi_x} = X_m \cos\varphi_x + \mathrm{j}X_m \sin\varphi_x. \tag{3.2}$$

The mapping that assign phasors to alternating signals of a certain pulsation is a linear isomorphism.The linearity means that phasor of an alternating signal z that is a linear combination

$$z = ax + by, \quad a, b \in \mathbb{R}$$

of alternating signals x and y is a linear combination of the phasors

$$Z = aX + bY.$$

The isomorphity implies that having phasor X and pulsation ω one can uniquely reconstruct an alternating signal $x(t)$ having X as its phasor. The signal x can be defined as[2]

$$x(t) = |X| \cos(\omega t + \arg X) = \mathfrak{Re}(X\mathrm{e}^{\mathrm{j}\omega t}),$$

[1]More precisely, phasors of alternating voltages of a certain pulsation form a space isomorphic to the space of complex numbers: a phasor of an alternating voltage is a complex number with unit V. Similarly, the phasors of alternating currents of a certain pulsation form a space isomorphic to the space of complex numbers (a phasor of an alternating current is a complex number with unit A).

[2]Henceforth $\mathfrak{Re}\, z$ and $\mathfrak{Im}\, z$ denote the real and the imaginary parts of a complex number z, respectively.

where $|X|$ is the magnitude, and $\arg X$ is the argument of the phasor X.
If phasor X is given in polar form

$$X = X_m e^{j\varphi}, \quad X_m > 0,$$

then $|X| = X_m$ and $\arg X = \varphi$. If phasor X is given in algebraic form

$$X = a + jb,$$

then

$$|X| = \sqrt{a^2 + b^2}, \qquad \arg X = \begin{cases} \operatorname{arc\,tg} \frac{b}{a} & \text{if } a > 0, \\ \operatorname{arc\,tg} \frac{b}{a} + \pi & \text{if } a < 0, \\ \frac{\pi}{2} & \text{if } a = 0 \text{ and } b > 0, \\ -\frac{\pi}{2} & \text{if } a = 0 \text{ and } b < 0. \end{cases}$$

Due to linearity of the mapping which assigns phasors to alternating signals, we can apply Kirchhoff's current and voltage laws to phasors: the sum of the phasors of all the currents which enter a node (or a subcircuit) of a circuit is zero and the sum of the phasors of all the voltage drops along any oriented loop in a circuit is zero as well.

3.1.2 Resonance

Every sourceless linear one-port which is an element of an AC circuit can be described with its impedance Z, which usually depends on pulsation ω.

According to Ohm's law, there is a relationship

$$U = ZI,$$

where U is a voltage phasor and I is a current phasor depicted in Fig. 3.1.

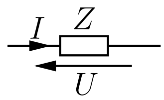


Figure 3.1

In AC circuits we may observe an interesting phenomenon called resonance. Actually, there are two definitions of resonance. The first defines a resonant pulsation of a one-port, which has impedance Z, as a pulsation ω_r, for which the magnitude of impedance $|Z|$ attains (as a function of pulsation) a strict local extremum (minimum or maximum). According to the other definition, a resonant pulsation of a one-port is a pulsation ω_r, at which the impedance Z becomes purely real (i.e. it has non-zero imaginary part for pulsations belonging to a neighbourhood of ω_r except for ω_r, at which the imaginary part vanishes). A resonant frequency f_r is related to resonant pulsation with equation

$$\omega_r = 2\pi f_r.$$

The two definitions of resonant pulsation are not equivalent in general. Despite this, we do not introduce different symbols for each of them, because they do coincide for one-ports studied in this exercise[3].

[3]For one-port in Fig. 3.2 the two definitions of resonant pulsations are equivalent, provided that $R_L = R_C$.

The connection of a resonant one-port and a source of alternating current or voltage is called a resonant circuit. In the exercise we will consider a resonant circuit presented in Fig. 3.2.

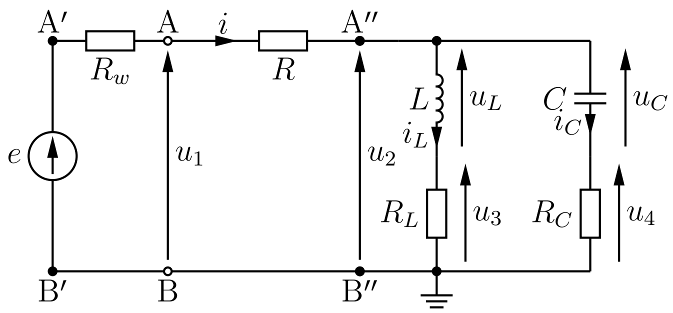


Figure 3.2

One item of the homework is to verify that the one-port to the right of terminals A-B (and also the one-port to the right of A′-B′ and the one-port to the right of A″-B″) has resonant pulsation (according to the definition "$\mathfrak{Im}(Z) = 0$") equal to

$$\omega_r = \frac{1}{\sqrt{LC}}, \tag{3.3}$$

provided that

$$R_L = R_C \neq \rho = \sqrt{\frac{L}{C}} \tag{3.4}$$

In particular, this pulsation does not depend neither on particular value of resistance R, nor on the value of R_w[4].

3.1.3 Maximum power transfer theorem

Instantaneous power delivered to a one-port shown in Fig. 3.3 can be expressed as

$$p(t) = i(t)u(t). \tag{3.5}$$

If such a one-port is an element of an AC circuit of pulsation ω, then

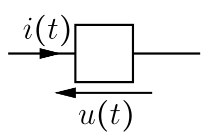


Figure 3.3

$$\begin{aligned} i(t) &= I_m \cos(\omega t + \varphi_i), & I_m &\geq 0, \\ u(t) &= U_m \cos(\omega t + \varphi_u), & U_m &\geq 0. \end{aligned}$$

[4]As a problem for self study one may verify that pulsation ω_r given with formula (3.3) is also a resonant pulsation according to the first of the given definitions of resonant pulsations, i.e., one may verify that the magnitude of the impedance of the one-port to the right of terminals A-B of Fig. 3.2 attains a local maximum at ω_r.

Thus, the instantaneous power given by (3.5) takes the following form

$$p(t) = I_m U_m \cos(\omega t + \varphi_i) \cos(\omega t + \varphi_u) = \underbrace{\frac{1}{2} I_m U_m \cos(\varphi_i - \varphi_u)}_{\text{constant term}} + \underbrace{\frac{1}{2} I_m U_m \cos(2\omega t + \varphi_i + \varphi_u)}_{\text{alternating term}}. \tag{3.6}$$

Notice that pulsation of the alternating term is twice as big as the pulsation of signals i and u.

Mean power, which, in the context of AC circuits, is also called *active power*, is the mean value P of the instantaneous power

$$P = \frac{1}{T} \int_{t=t_0}^{t_0+T} p(t) \mathrm{d}t, \quad T = \frac{2\pi}{\omega}.$$

Hence, the active power supplied to a one-port of Fig. 3.3 equals

$$P = \frac{1}{2} I_m U_m \cos(\varphi_i - \varphi_u)$$

(the mean value of the second component in formula (3.6) is zero). Mean power can be expressed in terms of phasors U (of voltage u) and I (phasor of current i) as

$$P = \frac{1}{2} \mathfrak{Re}(U\overline{I}).$$

If the one-port is described with an impedance Z, then the following equalities hold

$$P = \frac{1}{2} \mathfrak{Re}(U\overline{I}) = \frac{1}{2} |I|^2 \mathfrak{Re}(Z) = \frac{1}{2} |U|^2 \mathfrak{Re}\left(\frac{1}{Z}\right).$$

An important conclusion resulting from these equalities is that the active power supplied to reactive devices like inductors and capacitors is zero. In particular, the active power delivered to one-ports consisting of resistors and reactive devices is dissipated solely in the resistors.

Consider an AC circuit presented in Fig. 3.4. The following theorem provides an answer to

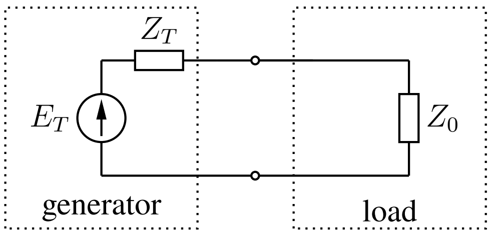


Figure 3.4

an important question about maximum power that can be delivered to the load in the considered circuit.

Maximum Power Transfer theorem. In the alternating current circuit of Fig. 3.4[5] the mean power transferred to the load does not exceed the *available power* P_{avail} of the generator [6]

$$P_{\text{avail}} = \frac{|E_T|^2}{8\,\mathfrak{Re}(Z_T)}. \tag{3.7}$$

[5] More precisely, for the theorem to hold, we need to assume $\mathfrak{Re}(Z_T) > 0$, which in real circuits is always true.

[6] The mean power transferred to the load can not exceed P_{avail} given by formula (3.7) even if we allow the load to be constructed with arbitrary devices (e.g. sources).

Moreover, the power transferred to the load equals P_{avail} if and only if the impedance Z_0 of the load is conjugated to the internal impedance Z_T of the generator, i.e.,

$$Z_0 = \overline{Z_T}.$$

3.1.4 Operational amplifier

The third part of the exercise concerns operational amplifiers (op-amps for short).

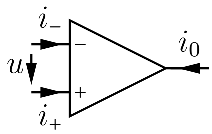


Figure 3.5

In the circuit theory, an operational amplifier is often considered a three-terminal device described by equations (see Fig. 3.5):

$$u = 0, \quad i_+ = i_- = 0. \tag{3.8}$$

The above equations enable to study most of linear circuits incorporating op-amps as their parts. In spite of this, you have to be aware of a subtle discrepancy between the above model of the op-amp and the general model of a lumped device. Namely, for the above model, one cannot assume that currents i_+, i_- and i_0 sum up to zero. The explanation to this phenomenon lays in the fact that real op-amps have at least five terminals (two of which are used for voltage supply). The existence of these extra terminals will be schematically depicted by one extra terminal which will usually be grounded. A description of real op-amps is out of scope of this manual. Let us just stress the fact that it should not be assumed that current i_0 is zero.

3.1.4.1 Voltage follower

One of the main applications of op-amps is called a *voltage follower*. This is a two-port shown in Fig. 3.6.

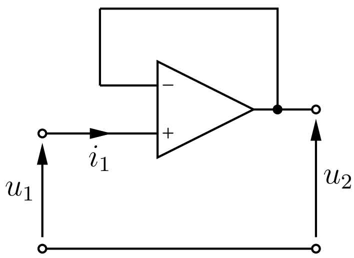


Figure 3.6: Voltage follower

Let us derive two equations that govern this two-port. One of these equations is

$$i_1 = 0 \tag{3.9}$$

which is implied by $i_1 = i_+$. In order to find the second equation we will analyze voltages. The voltage between "+" and "−" terminals is zero and the "−" terminal is short-circuited with the output of the op-amp. Thus we have

$$u_2 = u_1. \tag{3.10}$$

Equations (3.9) and (3.10) constitute a set that describes the voltage follower completely.

Voltage followers enable to reduce internal (output) impedances of real signal sources. Let us demonstrate this impedance reduction with a circuit presented in Fig. 3.7. To the left

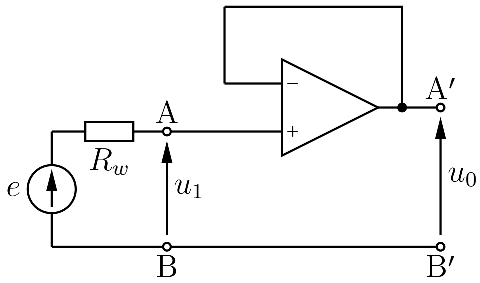


Figure 3.7: Voltage follower reduces internal resistance of a generator

of terminals A-B we have a voltage generator with internal resistance R_w. After attaching the generator to a voltage follower we get a new one-port with terminals A′-B′. According to Equation (3.9), the current flowing through R_w is zero and there is no voltage drop across this resistor. Using (3.10), we thus get

$$u_0 = u_1 = e.$$

The open-circuit voltage across terminals A′-B′ of our "new" one-port equals e. Please note that if one attaches any other one-port (for example, a resistor) to these terminals, then the above considerations will stay valid and we will still have

$$u_0 = e.$$

Since the voltage across terminals A′-B′ stays constant independently of the one-port attached to the analyzed one-port, the one-port (to the left of specified terminals) is equivalent to an ideal (!) voltage source. We thus reduced the internal resistance of the generator (R_w) to zero. Of course, in real circuits the output resistance of the one-port shown in Fig. 3.7 is very small (how small depends on a particular op-amp) but grater than zero.

3.1.4.2 Inverting amplifier and voltage adder

Another typical application of op-amps is called an *inverting amplifier* and is presented in Fig. 3.8. Its name originates from the following equation

$$u_0 = -\frac{R}{R_1}u_1. \tag{3.11}$$

The inverting amplifier is described completely by the above equation supplemented with

$$u_1 = i_1 R_1. \tag{3.12}$$

One may interpret the above equations as follows. At the "input" port, the inverting amplifier behaves as a resistor with resistance R_1 and at the output port it acts as ideal voltage source of electromotive force u_0 given by (3.11).

Let us derive Equations (3.11) and (3.12). Voltage u_{R_1} equals the input voltage u_1 because, according to (3.8), voltage across terminals "+" and "−" of the op-amp is zero. Ohm's law gives (3.12). Since the current flowing into terminal "−" is zero (see (3.8)), then the following must hold

$$i = i_1 = \frac{u_1}{R_1}.$$

The voltage drop u_R across R equals $-u_0$ (we use (3.8) again). Hence,

$$u_0 = -u_R = -iR = -\frac{R}{R_1}u_1.$$

A Similar analysis may be performed for *voltage adder*, also known as*summing amplifier*, which is presented in Fig. 3.9. The analysis would then give

$$\begin{aligned} u_0 &= -\frac{R}{R_1}u_1 - \frac{R}{R_2}u_2 = -R\left(\frac{u_1}{R_1} + \frac{u_2}{R_2}\right), \qquad (3.13)\\ u_1 &= i_1 R_1,\\ u_2 &= i_2 R_2. \end{aligned}$$

From the input side, the presented voltage adder behaves like two resistors: R_1 at the port across which voltage u_1 is depicted, and R_2 at the other input port, where voltage is u_2. At the output side, the voltage adder acts as an ideal voltage source of electromotive force given by (3.13).

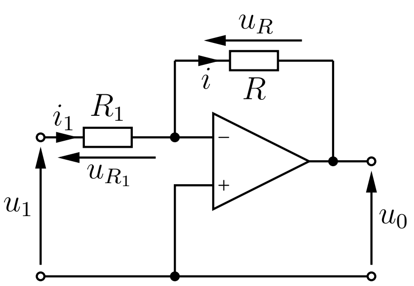


Figure 3.8: Inverting amplifier

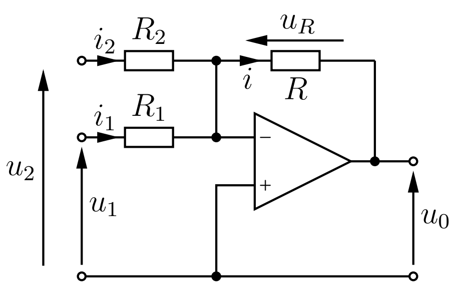


Figure 3.9: Voltage adder

3.1.4.3 Integrator

The last of op-amp applications considered in the exercise is called *inverting integrator* or *integrator* for short. It is presented in Fig. 3.10 and described by equations

$$u_1 = i_1 R_1$$

and

$$u_0 = -u_C = u_0(t_0) - \frac{1}{C}\int_{t_0}^{t} i_1(t)\mathrm{d}t = u_0(t_0) - \frac{1}{R_1 C}\int_{t_0}^{t} u_1(t)\mathrm{d}t.$$

The latter equation can be simplified to

$$u_0 = -\frac{1}{R_1 C}\int_{t_0}^{t} u_1(t)\mathrm{d}t,$$

provided that t_0 is an instant of time at which voltage u_0 is zero.

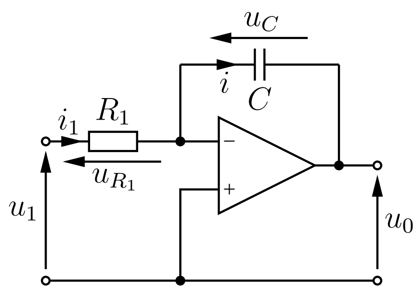


Figure 3.10: Inverting integrator

3.1.5 Fourier series

A signal $x(t)$ is called periodic if there exists a number $T > 0$, called a period of the signal, such that for every time instant t the following equality holds

$$x(t+T) = x(t).$$

In the remaining part of this chapter, we assume that T is a fundamental period, i.e., it is the smallest period of the given signal. If a series

$$\underbrace{X_0}_{\text{constant term}} + \underbrace{X_{1m}\cos(\omega t + \varphi_1)}_{\text{first harmonic}} + \underbrace{X_{2m}\cos(2\omega t + \varphi_2)}_{\text{second harmonic}} + \dots \tag{3.14}$$

is pointwise convergent to $x(t)$ at every continuity point of x, then it is called a Fourier series of signal x.

If series (3.14) is the Fourier series of signal x, then its coefficients are given by the following formulas

$$X_0 = \frac{1}{T}\int_0^T x(t)\mathrm{d}t, \tag{3.15}$$

$$X_{km} = 2\left|\frac{1}{T}\int_0^T x(t)\mathrm{e}^{-\mathrm{j}k\omega t}\mathrm{d}t\right|, \tag{3.16}$$

$$\varphi_{km} = \arg\left(\frac{1}{T}\int_0^T x(t)\mathrm{e}^{-\mathrm{j}k\omega t}\mathrm{d}t\right), \tag{3.17}$$

where pulsation ω and period T satisfy the following relation

$$\omega = \frac{2\pi}{T}. \tag{3.18}$$

In the exercise we will approximate voltage signal $u(t)$, presented in Fig. 3.11, with part of its Fourier series. The first three non-zero harmonics will then be taken. The Fourier series of $u(t)$ is

$$u(t) = \frac{4U}{\pi}\cos\left(\omega t - \frac{\pi}{2}\right) + \frac{4U}{3\pi}\cos\left(3\omega t - \frac{\pi}{2}\right) + \frac{4U}{5\pi}\cos\left(5\omega t - \frac{\pi}{2}\right) + \dots =$$
$$= \frac{4U}{\pi}\left(\frac{\sin(\omega t)}{1} + \frac{\sin(3\omega t)}{3} + \frac{\sin(5\omega t)}{5} + \dots\right), \quad (3.19)$$

where pulsation ω is given by (3.18).

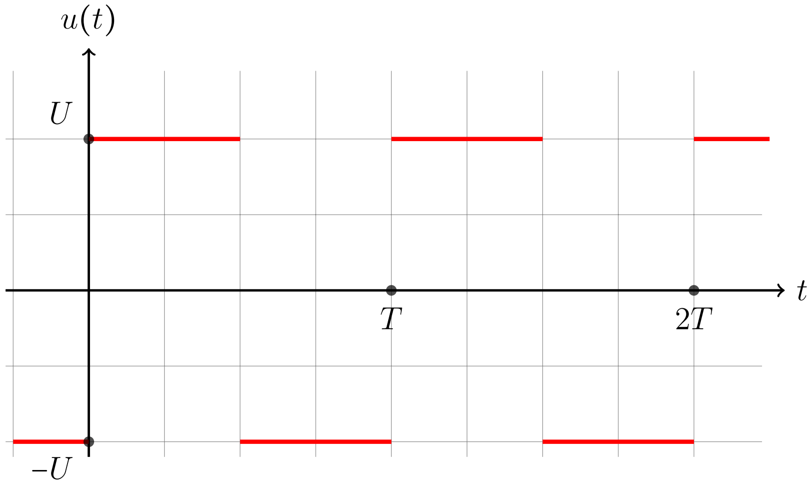
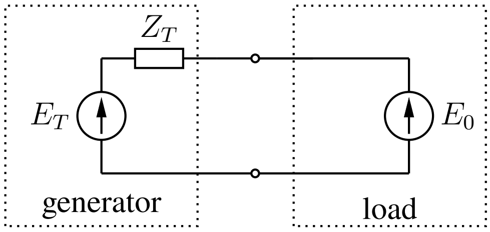


Figure 3.11: Pulse wave

Problems for self study

Solutions to the problems are given on page 69.

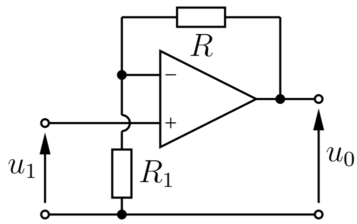
1. Check that if $\Re\mathfrak{e}(Z_T) > 0$, then the mean power delivered to the voltage source E_0 in the following AC circuit, drawn in the phasor domain, does not exceed the generator's available power P_{avail} given by formula (3.7) (see footnote [6]).



2. Derive equation

$$u_0 = u_1\left(1 + \frac{R}{R_1}\right)$$

that describes the following two-port known as a non-inverting amplifier.



3. Derive the Leibniz formula for π:

$$\frac{\pi}{4} = 1 - \frac{1}{3} + \frac{1}{5} - \frac{1}{7} + \frac{1}{9} - \dots$$

with a help of Equation (3.19). Note, that the right-hand side of (3.19) converges pointwise to $u(t)$ at times t at which signal u is continuous.

3.2 Homework

In the exercise you will study a circuit shown in Fig. 3.2. It consists of an ideal voltage source connected to a one-port presented in Fig. 3.12.

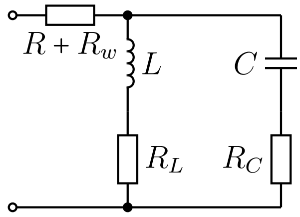


Figure 3.12: Resonant one-port

Homework 3.1. Show that the resonance pulsation for the one-port of Fig. 3.12 (the pulsation for which the imaginary part of the impedance vanishes) equals

$$\omega_r = \frac{1}{\sqrt{LC}}, \tag{3.20}$$

provided that condition (3.4) holds. Pick parameters from Tab. 3.1 and compute resonant frequency of the considered one-port.

Homework 3.2. If $R_L = R_C = R$, then for the resonant pulsation ω_r given by Formula (3.20), phasors I, I_L and I_C of currents i, i_L and i_C depicted in Fig. 3.2 equal

$$I = \frac{2ERC}{L + RC(3R + 2R_w)}, \tag{3.21}$$

$$I_L = \frac{E(RC - \mathrm{j}\sqrt{LC})}{L + RC(3R + 2R_w)}, \tag{3.22}$$

$$I_C = \frac{E(RC + \mathrm{j}\sqrt{LC})}{L + RC(3R + 2R_w)}, \tag{3.23}$$

where E is the phasor of electromotive force e. Draw the phasor diagram for the above current phasors. Assume $E = 1\mathrm{e}^{\mathrm{j}0}$ V, $R = 50\,\Omega$, $R_w = 600\,\Omega$, $L = 10$ mH and pick C value from Tab. 3.1,

---

Homework 3.3. Determine the available power and the internal (output) impedance of the generator depicted in Fig. 3.13. Find values $R_{0,\mathrm{match}}$ and $C_{0,\mathrm{match}}$ for which the mean power transferred to the load is maximum. Assume $R_w = R_1 = 50\,\Omega$, $L_1 = 10$ mH and

$$e(t) = E_m \cos 2\pi f t, \qquad E_m = 1\,\mathrm{V},$$

and pick frequency f from Tab. 3.2.

Homework 3.4. Derive Equation (3.13) for the voltage adder presented in Fig. 3.9.

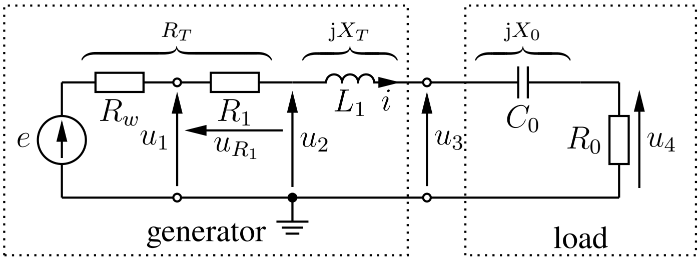


Figure 3.13: Circuit analyzed in the exercise module on Maximum Power Transfer theorem

3.3 Analysis of waveforms and phasors in a resonant circuit

In this exercise module, we will analyze a parallel resonant one-port LC located to the right of terminals A–B of the circuit presented in Fig. 3.2). In such a one-port there occurs a resonance of *currents*. If the Q-factor of the one-port is big, the amplitudes of currents i_L and i_C are much larger than the amplitude of current i. In the exercise we will observe these currents with an oscilloscope. Since oscilloscopes are capable of showing waveforms of voltages only, we will measure the currents indirectly as voltages on reference resistors R, R_L and R_C. These resistors are of relatively low resistance $R = R_L = R_C = 50\ \Omega$.

Notice that the voltage across resistor R is a „floating" voltage, i.e., none of the terminals of R is grounded. Therefore, it is impossible to measure the voltage directly with the oscilloscope or `Maya44 USB` card. In order to measure current i through the LC one-port voltages u_2 and u_1 (see Fig. 3.2) need to be subtracted.

Note that the *parallel* resonant circuit should be supplied by a source with as the largest possible internal resistance (an ideal current source would be the best option). That is why there will be an additional resistor $R_s = 500\ \Omega$ inserted into the analyzed circuit. This resistor connected in series with the reference resistor $R = 50\ \Omega$, and the output resistance of the `Maya44 USB` card ($R_{\mathrm{out}} \approx 100\ \Omega$) simulates internal resistance of the source equal to $R_w = R_{\mathrm{out}} + R_s \approx 600\ \Omega$.

In this exercise module we will measure waveforms and draw phasor diagrams of currents in the circuit for resonant frequency f_0 as well as for frequencies f_L and f_C at which the amplitude of the current i_L or current i_C, respectively, is equal to the amplitude of the total current i.

3.3.1 Measurement of frequencies f_0, f_L and f_C of the resonant circuit

Power off the instruments and the crate, and assemble the measurement setup according to Fig. 3.14. Use the following parameter values: $R = R_L = R_C = 50\ \Omega$, $R_s = 500\ \Omega$, $L = 10$ mH, as well as the values assigned to your particular stand in Tab. 3.1.

Turn the computer on and run the virtual wobbuloscope program from the TOiS_Toy set. Define the following signals in the first three channels of the instrument:

1. `Inp.1-Inp.2` — indirect measurement of current i,

2. `Inp.3` — indirect measurement of current i_L,

3. `Inp.4` — indirect measurement of current i_C

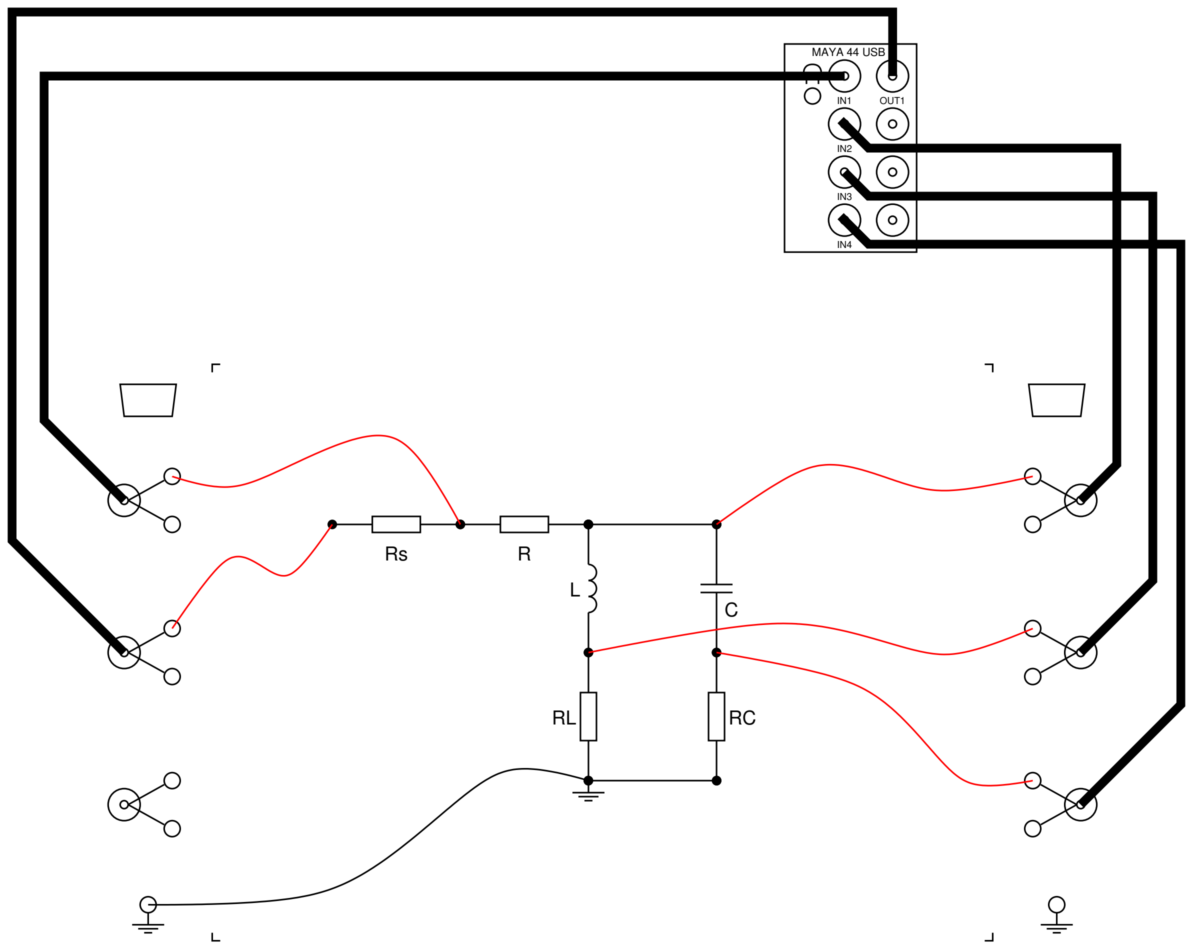


Figure 3.14: Measurement setup for analysis of the resonant circuit.

and activate these three channels by checking appropriate boxes. Set the gain of the wobbuloscope to initial value $20\ \mathrm{mV/div}$, the range of pulsations to $2\ \mathrm{kHz}$ to $20\ \mathrm{kHz}$ and generator output voltage amplitude to $1\ \mathrm{V}$.

Acquire and plot in one graph the three amplitude transfer functions (using linear scale), by choosing single run mode (key `Run` in menu Run / Once or the relevant push-button from the toolbar). From the graph, determine the resonant pulsation f_0, for which amplitude of the current i attains a *minimum*, and frequencies f_C and f_L, for which the graphs of the amplitudes of currents i_C and current i_L, respectively, cross the graph of the amplitude of current i. Record the obtained values in Tab. 3.3. Compare the resonant pulsation with the value calculated in

Table 3.1

Stand	C [nF]
1, 5, 9, 13, 17	22
2, 6, 10, 14, 18	47
3, 7, 11, 15	100
4, 8, 12, 16	220

homework.

3.3.2 Phasor diagram for the waveforms in the analyzed resonant circuit

Run the sinusoidal signal generator program from the TOiS_Toy set to generate signal at output `OUT1` of the `Maya44 USB` card. Then, run the vector voltmeter program and the oscilloscope program. In these two programs (voltmeter and oscilloscope), define the following six channels to be shown:

1. `Inp.1-Inp.2` — indirect measurement of current i, also used as a reference for initial phases of the other signals (we thus assume that the current signal has a zero initial phase),
2. `Inp.2` — measurement of voltage u_2,
3. `Inp.2-Inp.3` — measurement of voltage u_L,
4. `Inp.2-Inp.4` — measurement of voltage u_C,
5. `Inp.3` — measurement of voltage u_3 (and indirectly of current i_L),
6. `Inp.4` — measurement of voltage u_4 (and indirectly of current i_C).

Activate only the first channel (`Inp.1-Inp.2`) of the vector voltmeter.

Set the generator signal amplitude to $1\ \mathrm{V}$ and its frequency to the measured value of f_0. Both in the voltmeter and the oscilloscope, set the slider `Pos.` to position 0.5 and the gain to initial value $200\ \mathrm{mV/div}$. Then, set the time base in the oscilloscope to $100\ \mathrm{\mu s/div}$.

Choose a sine-wave interpolation between samples. For this purpose, in the menu Options / Oscilloscope check the option `Sinus` and uncheck all the other options. You can also use the relevant push-buttons from the toolbar in the TOiS_Toy window.

Run the TOiS_Toy program in repeated single run mode (key `Repeat` in menu Run / Once or the relevant push-button from the toolbar in the TOiS_Toy window).

3.3.2.1 Phasor diagram for currents

2×

→PNG

In the vector voltmeter and the oscilloscope, activate only the channels used for current measurement (channels 5–6), and in the voltmeter also the reference channel `Inp.1-Inp.2`. Increase the gain in the two instruments to $50\ \mathrm{mV/div}$. Change the generator frequency in the range f_C to f_L and observe the waveforms of the currents in the oscilloscope and the corresponding phasor diagrams. The phasor diagram of currents for resonant frequency and a chosen (most interesting!) phasor diagram for a different frequency are to be saved on disk so that they can be later printed and attached to the report. *Important! The graphs should be saved only in the single run mode* Run / Once. In the phase diagrams for the currents attached to the report, verify graphically that the Kirchhoff's current law: $i = i_L + i_C$, is fulfilled. Compare the measured phasor diagram for the resonant frequency with the diagram determined in homework. How can you tell from the phasor diagram of currents whether the frequency is higher or lower than the resonant frequency?

3.4 Matching load for maximal power transfer

3.4.1 Setup

In this exercise module, we will analyze the circuit shown in Fig. 3.13. We will use the virtual sinusoidal waveform generator from the TOiS_Toy set to generate the supply waveform. However, the internal resistance of the `Maya44 USB` card is too large for our purposes (it is equal to ca. $100\ \Omega$). That is why the output of the `Maya44 USB` card will be connected to the amplifier card set to gain 1. The amplifier (a voltage follower) allows to obtain sufficiently low value of the internal resistance $R_w \approx 50\ \Omega$.

Power measurements will be performed with powermeter from the set of the virtual instruments TOiS_Toy. This meter computes and shows the product of two voltage waveforms (this product is proportional to instantaneous power delivered to a one-port):

- voltage $u(t)$ on the one-port, for which the power is measured,
- voltage on resistor R_i used for indirect measurement of current $i(t)$ flowing through the analyzed one-port,

i.e. the following product

$$u(t) \cdot R_i i(t) = p(t) R_i. \tag{3.24}$$

Remember that the instruments equipped with BNC sockets measure voltages with respect to ground. It concerns e.g the `Maya44 USB` card used in the virtual powermeter. In our measurement setup, the resistor R_1 for measurement of current is „floating" (none of its terminals is connected to ground) and that is why the voltage across R_1 will be determined as the difference between two nodal voltages: $u_{R_1} = u_1 - u_2$ (see Fig. 3.13). The voltage on the load in the analyzed setup is a voltage with respect to ground (u_3) and hence can be measured directly.

In the exercise we will use the *instantaneous* power meter from the TOiS_Toy set to solve the maximum power transfer problem, in which we need to measure the *active* power delivered to the load. The active power is the mean value of instantaneous power and can be easily determined as the arithmetic average of the maximum and the minimum of the instantaneous power:

$$P = \frac{\max_t p(t) + \min_t p(t)}{2} = \frac{\max_t(u_{R_1}(t) u_3(t)) + \min_t(u_{R_1}(t) u_3(t))}{2R_1}. \tag{3.25}$$

However, performing such calculation in the course of matching the load to the source after each change of the load impedance would be rather inconvenient. It is worth to seek another way to measure the active power in the load, such that it wouldn't require any calculations.

For this purpose, we can use the fact that active power is dissipated solely in the *resistance* of the load. In the analyzed circuit the load has the simple form of a *series* connection of a resistor R_0 and a capacitor C_0, so its resistance is equal to R_0. The active power dissipated in the whole load is thus equal to the mean power dissipated in R_0:

$$P = P_{R_0} = \frac{1}{T} \int_{t_0}^{t_0+T} p_{R_0}(t) \mathrm{d}t, \tag{3.26}$$

where the instantaneous power $p_{R_0}(t)$ is determined on the basis of the same current i as previously, and voltage u_4 across resistor R_0:

$$p_{R_0}(t) = i \cdot u_4 = \frac{u_1 - u_2}{R_1} u_4. \tag{3.27}$$

What is the benefit from replacing the straight-forward active power measurement method with measurement of power $p_{R_0}(t)$ in resistor R_0? The latter power is always non-negative and its minimal value is always zero:

$$p_{R_0}(t) = i \cdot u_4 = i \cdot iR_0 = i^2 R_0. \tag{3.28}$$

Thus, the active power P_{R_0}, determined as the average of the maximum and the minimum of the instantaneous power, will be simply equal to half of the maximal instantaneous power:

$$P_{R_0} = \frac{\max_t p_{R_0}(t) + \min_t p_{R_0}(t)}{2} = \frac{\max_t p_{R_0}(t)}{2} = \frac{\max_t (u_{R_1}(t) u_4(t))}{2R_1}. \tag{3.29}$$

The process of „tuning" the load to maximize power P is thus equivalent to „tuning" it to the maximum of the instantaneous power $\max_t p_{R_0}(t)$. It means we should adjust the load so that the waveform of the instantaneous power $p_{R_0}(t)$ is „the largest".

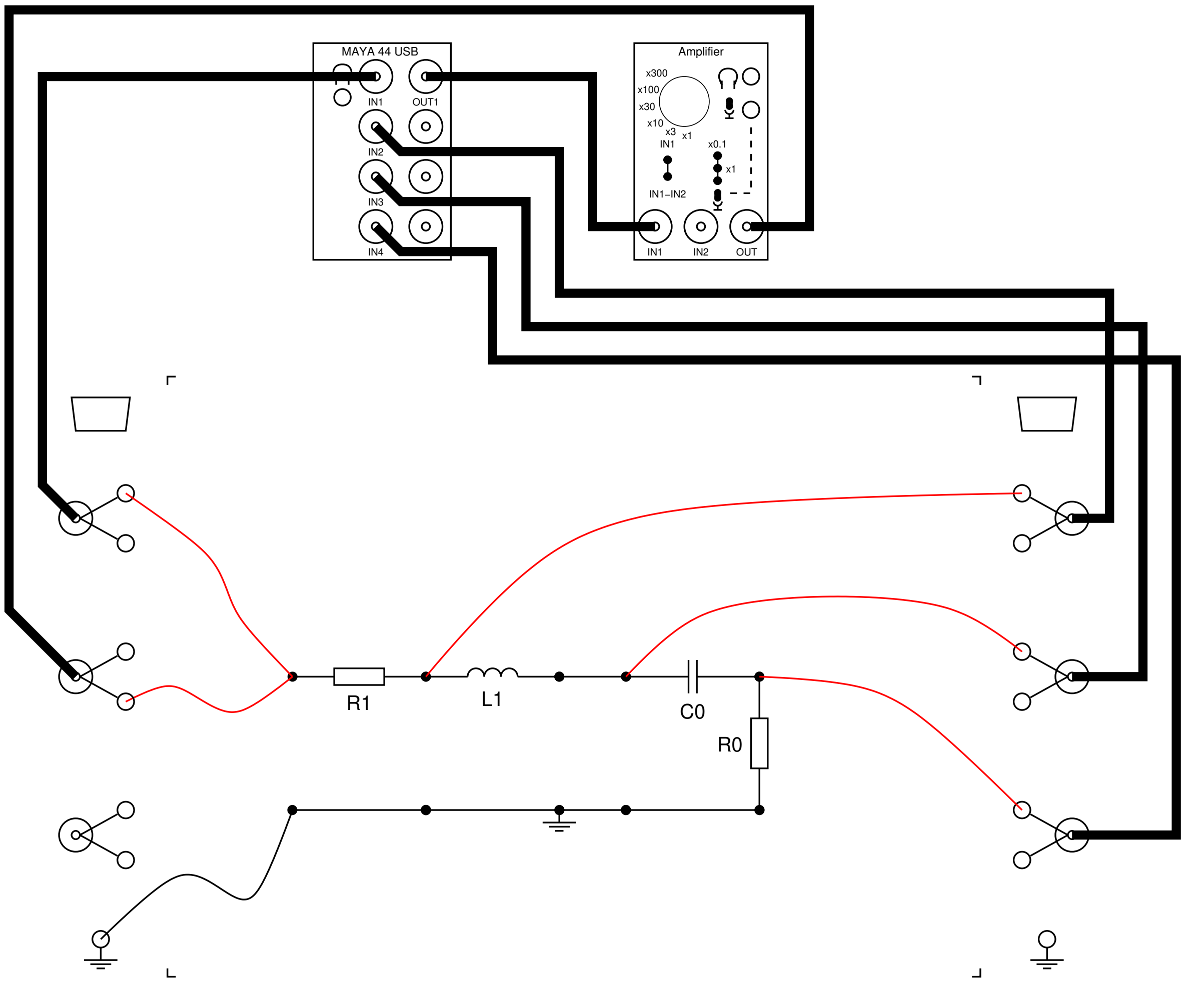


Figure 3.15: Measurement setup for realization of power match.

Power off the crate and assemble the measurement setup according to Fig. 3.15. Use the following parameter values: $R_0 = R_1 = 50\ \Omega$, $L = 10$ mH and the nominal capacitance $C_0 = C_{0,\mathrm{nom.}}$ assigned to your particular stand in Tab. 3.2. Then, power on the crate.

Run the TOiS_Toy program. Activate the sinusoidal waveform generator on output 1 (menu Sources). Set the frequency f of the generated signal according to your stand number, as given in Tab. 3.2 and its amplitude to 1 V. Next, choose the powermeter from menu Measure and set the slider `Pos` to position 0.5. Set gain to $0.05\ \mathrm{V}^2/\mathrm{div}$ and time base to $100\ \mu\mathrm{s}/\mathrm{div}$. Choose sine-wave interpolation between samples.

By pressing left mouse button in the right panel of the powermeter (titled $V_1 * V_2$) define the following signals to be visualized in the first two channels:

1. `(Inp.1-Inp.2)*(Inp.3-0)` — a waveform proportional to instantaneous power p in the load, or more precisely the following product

$$(u_1 - u_2)u_3 = pR_1, \tag{3.30}$$

2. `(Inp.1-Inp.2)*(Inp.4-0)` — a waveform proportional to instantaneous power p_{R_0} in the resistance of the load, or more precisely:

$$(u_1 - u_2)u_4 = p_{R_0}R_1. \tag{3.31}$$

Activate the second waveform only (check the relevant option in the right panel of the powermeter).

Run the TOiS_Toy program in repeated single run mode (key `Repeat` in menu Run / Once or the relevant push-button from the toolbar in the TOiS_Toy window).

3.4.2 Choice of the optimal resistance of the load

It can be easily verified that for an arbitrary fixed resistance R_0 of the load, maximal active power transfer to the load occurs when reactance of the load is opposite to the internal reactance of the source:

$$X_0 = -X_T \qquad \Leftrightarrow \qquad \frac{-1}{2\pi f C_0} = -2\pi f L_1. \tag{3.32}$$

Connect capacitor $C_0 = C_{0,\mathrm{nom.}}$ (according to Tab. 3.2) to the measurement setup presented in Fig. 3.15. This time, we will use a parallel connection of a potentiometer $1\ \mathrm{k}\Omega$ and a resistor $200\ \Omega$ as resistance R_0. This allows a finer adjustment of the load resistance in the range of interest.

Gently turning the potentiometer knob and simultaneously observing the waveform $p_{R_0}(t)R_1$ in the powermeter, adjust the resistance R_0 until amplitude of this waveform attains a maximum. Read the value $\max_t(u_1 - u_2)u_4$ from the powermeter and record it in Tab. 3.4. Then, activate the first channel in the right panel of the powermeter that is defined as `(Inp.1-Inp.2)*Inp.3`, and record in Tab. 3.4 the extreme values $\max_t(u_1 - u_2)u_3$ and $\min_t(u_1 -$

Table 3.2

Stand	f	$C_{0,\mathrm{nom.}}$
1, 4, 7, 10, 13, 16	3,39 kHz	220 nF
2, 5, 8, 11, 14, 17	2,32 kHz	470 nF
3, 6, 9, 12, 15, 18	1,59 kHz	1 µF

$u_2)u_3$. Filling in Tab. 3.4, verify that the two methods of active power calculation, described with formulas:

$$P_{R_0} = \frac{\max_t(u_1 - u_2)u_4}{2R_1} \tag{3.33}$$

and

$$P = \frac{\max_t(u_1 - u_2)u_3 + \min_t(u_1 - u_2)u_3}{2R_1}, \tag{3.34}$$

give (approximately) equal results and that these results agree with the available power P_{avail} of the source computed in homework.

1×

→PNG

Save the oscillogram of both waveforms $(u_1 - u_2)u_4$ and $(u_1 - u_2)u_3$ for the load matched to the source, and later print it and attach to the report.

Detach resistor R_0 (realized as a parallel connection of a potentiometer $1\ \mathrm{k\Omega}$ and a resistor $200\ \Omega$) from the circuit and measure its resistance $R_{0,\mathrm{match}}$ with the ohmmeter of the multimeter UT70A. Record the measurement result in the report template. Compare this resistance with the expected value $R_T = R_w + R_1$.

3.5 Realization of basic op-amp circuits

3.5.1 Inverting amplifier

One of the fundamental op-amp circuits is the so-called inverting amplifier. The term ”inverting” refers to the fact that for alternating signals, the amplifier ’inverts” the output signal (with respect to the input signal), i.e. it changes its phase by $180°$. Power off the instruments and the crate, and assemble the measurement setup according to Fig. 3.16 (only the elements and connections marked with solid line). Use the following elements paramaters: $R_1 = 200\ \Omega$ and $R = 2\ \mathrm{k\Omega}$. Connect the supply cable of the op-amp element to the matching socket in the tray. To protect the generator output from accidental short-circuiting, we again connect it to the circuit indirectly, by means of the amplifier card set to amplification of input `IN1` with gain 1 (voltage follower configuration). **Before turning the power on, show the assembled setup to the lab instructor for verification.** After powering on the crate and the instruments, start the measurements.

Set the oscilloscope to external triggering and DC coupling in both channels. Preset the gain in the first channel to $0.5\ \mathrm{V/div}$, and in the second channel to one tenth of it (because the gain of the analyzed amplifier is 10), i.e. to $5\ \mathrm{V/div}$. Since the amplifier inverts the phase of the signal (changes its sign), to facilitate comparison of the signals at the input and output of the amplifier, activate inversion of the signal in the second channel of the oscilloscope. Set the time base to $0.2\ \mathrm{ms/div}$.
Set the function generator to generate a sine wave (no offset) of frequency $1\ \mathrm{kHz}$ and amplitude $1\ \mathrm{V}$ i.e., $2\ \mathrm{Vpp}$ peak-to-peak value.

With the generator set to generate sine and then sawtooth (`RAMP`) waves, verify that the amplitude of the output voltage is indeed 10 times larger then that of the input voltage and that the output is inverted. To do so, place the waveforms from both channels of the oscilloscope one on top of the other. Increase the amplitude of the signal from the generator until characteristic nonlinear distortions (saturation) occur in the signal at amplifier output. At what levels of voltages (positive and negative) do these limiting effects appear? How do they relate to amplifier supply voltages ($\pm 15\ \mathrm{V}$)?

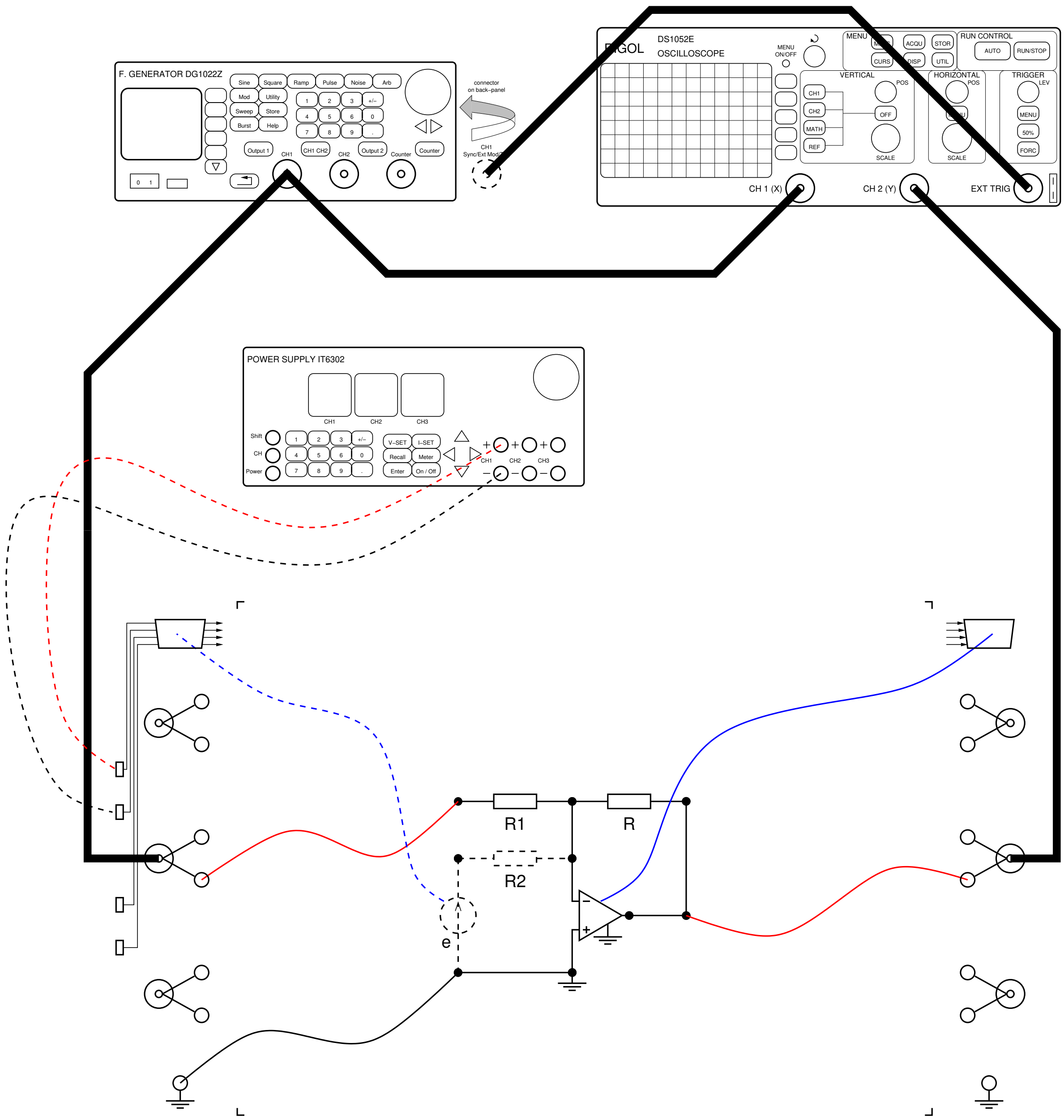


Figure 3.16: Measurement setup for analysis of inverting (connections marked with solid lines) and adder (additional connections marked with dashed lines).

3.5.2 Adder

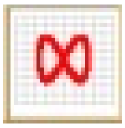
A particular feature of the op-amp operating in the circuit from the previous section is that the *whole* current flowing to its input is redirected to the resistor R connecting the input with the output. It is this current that determines the output voltage of the amplifier. And what will happen if there are *two* currents flowing to the input of the amplifier? The output voltage will be proportional to their *sum* — we will obtain an *adder* (or a summing amplifier).

Switch off the first channel of the power supply and set its voltage to zero. We will extend the measurement setup from the previous section, by connecting an additional resistor R_2 =

5 kΩ to it, a voltage source e and wires marked in Fig. 3.16 with a dashed line. Configure the oscilloscope in the same way as in the previous section, i.e. set external triggering, DC coupling in both channels, ten times lesser gain and sign inversion in the output channel. Choose the time base 0.2 ms/div. Set the function generator to generate a sine wave (no offset) of frequency 1 kHz and 2 Vpp peak-to-peak value. Switch on the output of the `CH1` channel of the power supply.

Gently changing the voltage across resistor R_2, observe the changes of the offset of the signal at the output of the amplifier. In a similar way a DC offset is added to signals in function generators. By swapping the cables with banana plugs connected to the voltage supply, one can obtain negative DC component in the output signal. For larger voltages, there may again appear distortions caused by limiting of the output voltages in a real amplifier.

Lets build a circuit that sums harmonic signals in order to approximate a rectangular wave. It follows from the Fourier analysis that for this purpose one should sum up subsequent *odd* harmonics (i.e. the first, third, fifth, etc.), with the weights inverse-proportional to the harmonic number (i.e. 1, $\frac{1}{3}$, $\frac{1}{5}$, etc.) and with identical initial phases.



Power off the instruments and the crate, and assemble the measurement setup according to Fig. 3.17 (an adder with three inputs). Assume the following parameters: R_1 = 1 kΩ, R_2 = 3 kΩ (use the 10 kΩ potentiometer set with a help of ohmmeter), R_3 = 5 kΩ and R = 2 kΩ. Particular outputs of the `Maya44 USB` card are to be delivered to relevant resistors, whereas the output of the amplifier is to be connected to the input `IN1` of this card. Wires used for supplying the op-amp are to be connected to the matching socket of the tray. Power on the crate.

Launch the environment TOiS_Toy, and run three sinusoidal signal generator programs, which will generate signals at outputs `OUT1`, `OUT2` and `OUT3` of the `Maya44 USB` card. Then run the oscilloscope program, and the spectrum analyzer program. Activate the first channel (`Inp.1`) in both the oscilloscope and the spectrum analyzer.

Leave the default amplitude values of signals from all generators, equal to 0.5 V. Set the frequencies of generators in 1, 2 and 3 to 1 kHz, 3 kHz and 5 kHz, respectively. In both instruments (oscilloscope and spectrum analyzer) set the slider `Pos.` to position 0.5 and preset their gain to 200 mV/div. Set the time base in the oscilloscope to 200 µs/div, and in the spectrum analyzer set the resolution of the frequency axis to 500 Hz/div.

Choose linear interpolation between samples. For this purpose, make sure that the option `Piecewise` is checked in the menu Options / Oscilloscope, and uncheck all the other options. You can also use the relevant push-buttons from the toolbar in the TOiS_Toy window.

Run the TOiS_Toy program in repeated single run mode (key `Repeat` in menu Run / Once or the relevant push-button from the toolbar in the TOiS_Toy window).

1×

→PNG

Observe (with a help of the oscilloscope) the waveform that is an approximation of a rectangular wave with its three first odd harmonics. With a help of the spectrum analyzer, verify that proportions of the amplitudes of particular bars agree with theoretically required (i.e. $1 : \frac{1}{3} : \frac{1}{5}$) and that initial phases are identical. Save the oscillogram to print it out later (don't forget to attach it to the report).

Important! The graphs should be saved only in the single run mode Run / Once.

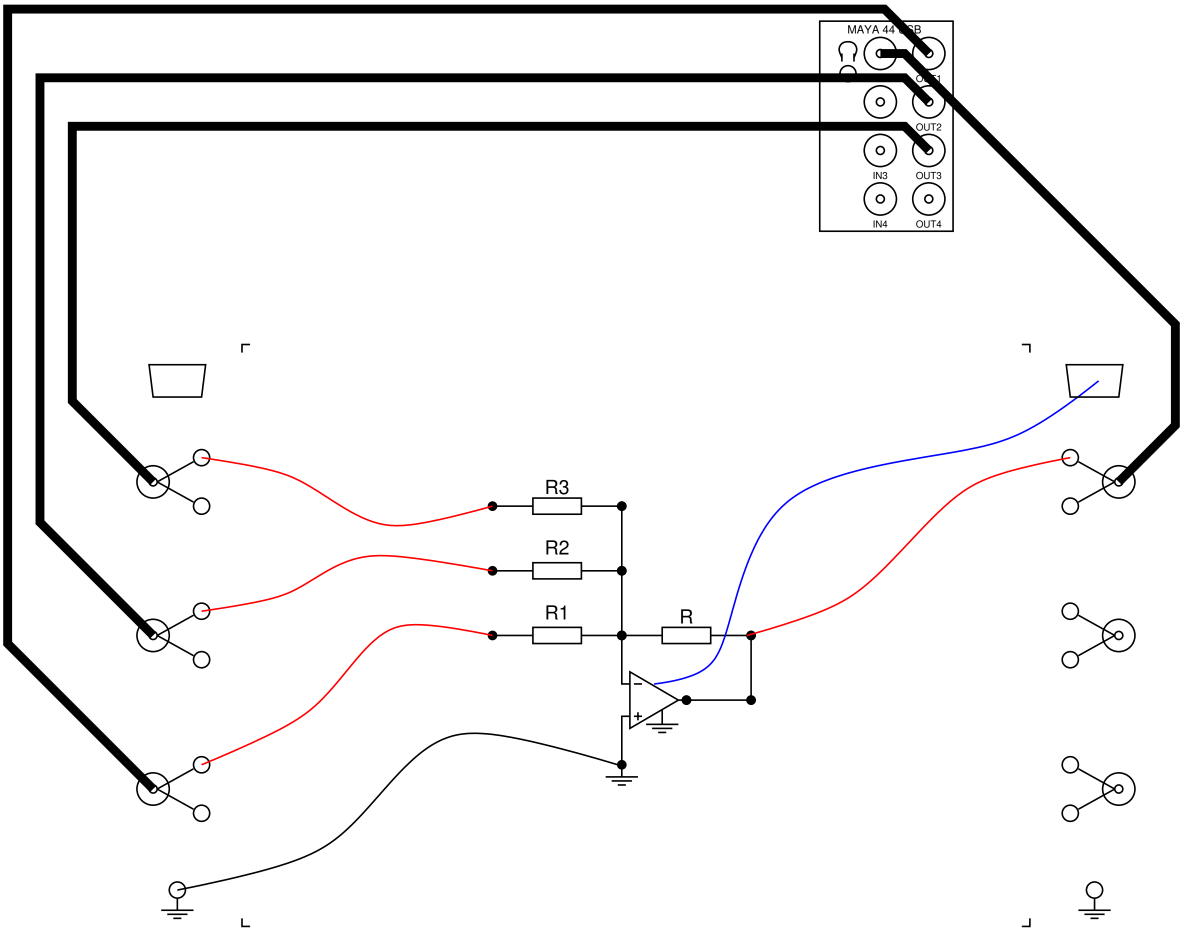


Figure 3.17: Measurement setup for analysis of the adder and the integrator (after replacing the resistor R with a capacitor C).

3.5.3 Integrator

Realization of an integrator is very simple: it is enough to replace the resistor R that connects the input with the output of the op-amp, with a capacitor $C = 1$ μF. Observe (with a help of the oscilloscope) the waveform that is the integral of the approximated rectangular wave. The integral of an ideal rectangular wave is a triangular signal: linearly increasing whenever the rectangular signal is positive and decreasing whenever the rectangular signal is negative. (In our case we should take into account that we deal with *inverting* integrator).

Stand:	*Name and SURNAME*	*Grade*
Ex. no.: **3**	. . . .	. . . .
	. . . .	. . . .
	. . . .	. . . .
Topic: ***Alternating current circuits***		*Instructor's signature*
		Date

Analysis of waveforms and phasors in a resonant circuit

$C =$ nF

Table 3.3

$f_{0\text{ homework}}$ [kHz]	f_C [kHz]	f_0 [kHz]	f_L [kHz]

Matching load to source for maximal active power transfer

$C_{0,\text{nom.}} =$ μF

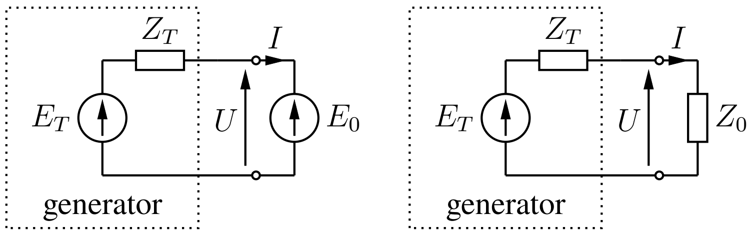
$R_0 =$ Ω

Table 3.4

$\max_t(u_1 - u_2)u_4$	P_{R_0}	$\max_t(u_1 - u_2)u_3$	$\min_t(u_1 - u_2)u_3$	P	$P_{\text{avail.}}$
[V^2]	[mW]	[V^2]	[V^2]	[mW]	[mW]

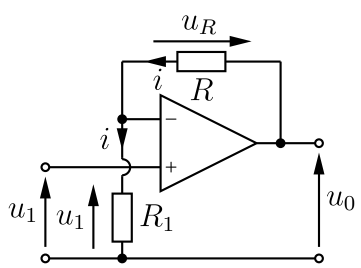
Solutions to self-study problems

1. Mean power delivered to source E_0 equals $P = \frac{1}{2}\,\mathfrak{Re}(U\overline{I})$. For the considered circuit we have $U = E_0$. If $I = 0$ A, then the mean power delivered to the load is zero and thus it does not exceed the generator's available power. On the other hand, if $I \neq 0$ A, then replacing the source E_0 with impedance $Z_0 = \frac{E_0}{I}$ does not alter neither the voltage U nor the current I.



Mean power delivered to the source E_0 in the original circuit is thus equal to the power delivered to the impedance Z_0 in the "new" circuit. The latter power does not exceed the generator's available power by Maximum Power Transfer theorem.

2. By equations (3.8), the voltage across resistor R_1 equals the input voltage u_1.



By Ohm's law, the current through resistor R_1 is $i = \frac{u_1}{R_1}$. Using (3.8) once more, we get that the current through R is the same and thus

$$u_R = Ri = u_1 \frac{R}{R_1}.$$

Eventually, using Kirchhoff's voltage law, we get

$$u_0 = u_1 + u_R = u_1 \left(1 + \frac{R}{R_1}\right).$$

3. It is enough to substitute $t = \frac{\pi}{2\omega} = \frac{T}{4}$ into formula (3.19) and then to multiply both sides of the resulting equation by $\frac{\pi}{4U}$.

Chapter 4

Non-linear circuits and small-signal analysis

4.1 Introduction

4.1.1 Composing voltage-current characteristics

4.1.1.1 Basic elements

In this section we will study one-ports that are described completely with their voltage-current characteristics. The voltage-current characteristic of a device comprises of all pairs (u, i) that define admissible, instantaneous, voltage across and current through the one-port. Voltage and current sources are examples of one-ports described with voltage-current characteristics. Their characteristics are presented in Fig. 4.1 and 4.2, respectively.

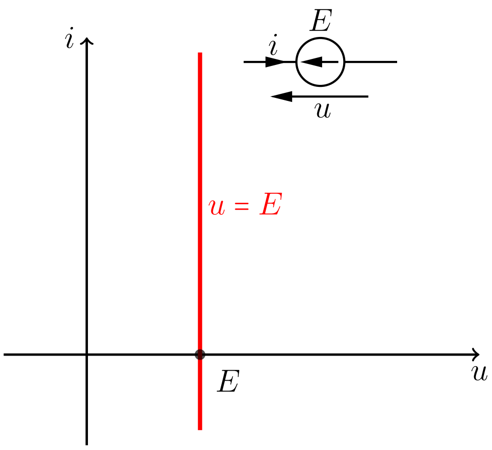


Figure 4.1: Voltage-current characteristic of a voltage source

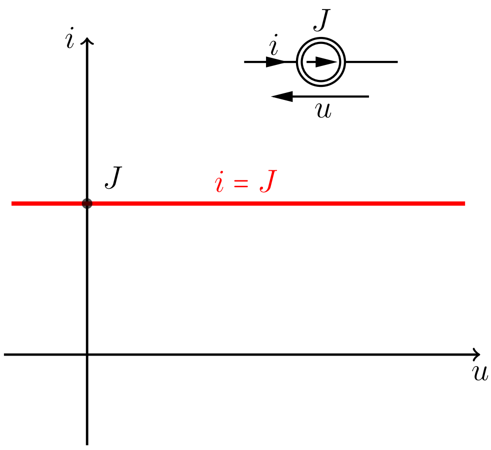


Figure 4.2: Voltage-current characteristic of a current source

Other devices that fall into the considered class are linear (see Fig. 4.3) and non-linear resistors. An example of the latter is a diode. Several models of diode are used. Each of them is related to a particular voltage-current characteristic. One of the simplest model is called the ideal diode. The ideal diode acts a short-circuit, if current i is positive (see Fig. 4.4), and as an open-circuit, if the voltage u is negative. Every diode described by such a model will be denoted with symbol $D_{\lrcorner}$).

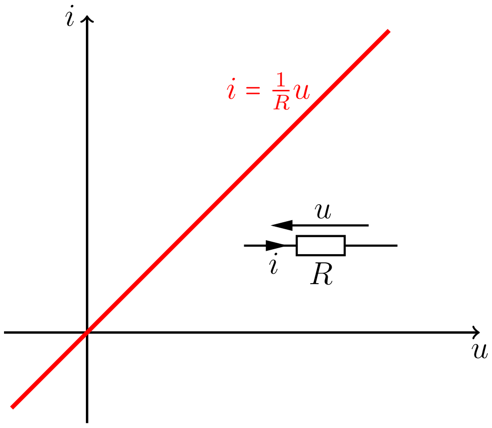


Figure 4.3: Voltage-current characteristic of a (linear) resistor

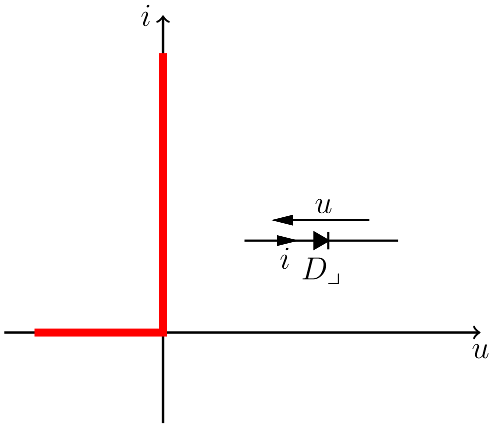


Figure 4.4: Characteristic of an ideal diode

4.1.1.2 Series connections

Let us now show how to determine the voltage-current characteristic of a one port comprising two one-ports X_1 and X_2 connected in series as shown in Fig. 4.5.

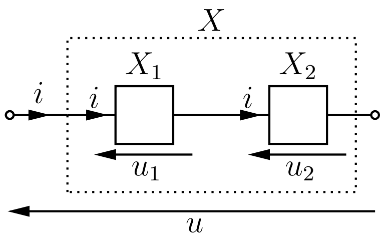


Figure 4.5: One-ports X_1 and X_2 connected in series

Assume that the characteristic of the first one-port is a curve χ_1. It consists of points $(u_1, i) \in \chi_1$. Also, assume that characteristic of the second one-port is a curve χ_2. According to Kirchhoff's laws, the current through each of the one-ports equals i, and the voltage across the whole device is the sum of voltages u_1 and u_2 across X_1 and X_2, respectively. Thus, the characteristic of X consists of all the points (u, i) for which there exist points

$$(u_1, i) \in \chi_1 \quad \text{and} \quad (u_2, i) \in \chi_2,$$

such that

$$u = u_1 + u_2.$$

In other words, the characteristic of a one-port comprising two one-ports connected in series is the result of "summation of the characteristics of individual one-ports along voltage axis" (i.e., summation of the characteristics treated as graphs of functions mapping currents to voltages). The procedure is illustrated in Fig. 4.7.

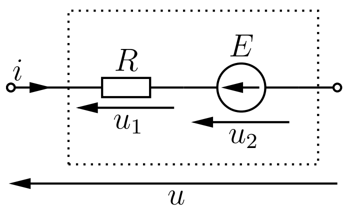


Figure 4.6: Series connection of a resistor and a voltage source

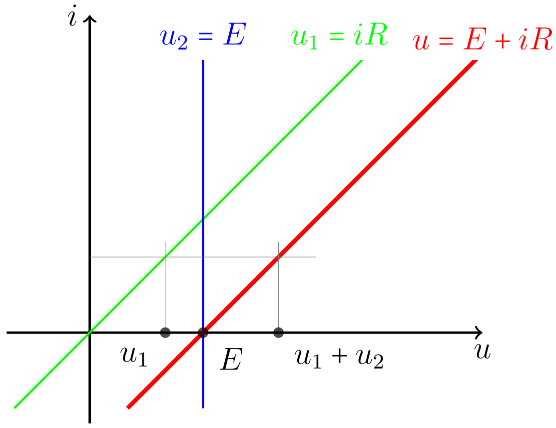


Figure 4.7: Construction of the characteristic of the one port presented in Fig. 4.6

4.1.1.3 D_{U_p,R_d} diode model

In order to construct voltage-current characteristic of a device comprising more than two one-ports connected in series we can either "sum all the individual characteristics along voltage axis" or perform such summation step by step (summing just two characteristics at a time). In this way we may obtain characteristic of the one-port of Fig. 4.8. The characteristic (it can be obtained by summation of characteristics shown in Fig. 4.4 and Fig. 4.7) is presented in Fig. 4.9. A one-port having such characteristic represents another model of a diode. This model will be further denoted with symbol D_{U_p,R_d}. Parameter U_p is called a threshold voltage, and R_d is called a dynamic resistance of the diode.

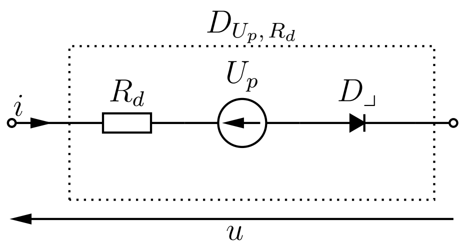


Figure 4.8: D_{U_p,R_d} diode model

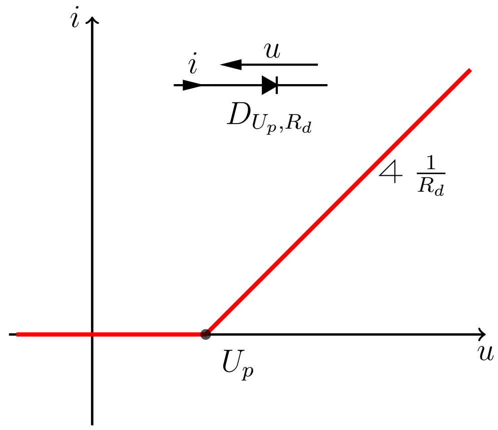


Figure 4.9: Characteristic of a D_{U_p,R_d} diode

4.1.1.4 Parallel connections

Now, let us examine a parallel connection of one-ports presented in Fig. 4.10. According to Kirchhoff's laws, the voltages across these one-ports are equal, and the currents through the one-ports sum up to the total current, i.e., $i = i_1 + i_2$. Therefore, the voltage-current characteristic of the one-port shown in Fig. 4.10 consists of all the points (u, i) for which

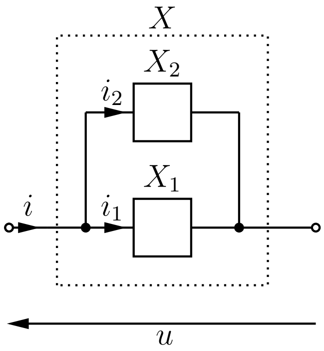


Figure 4.10: Parallel connection of one-ports X_1 and X_2

there exist points (u, i_1) and (u, i_2) lying on characteristics of X_1 and X_2, respectively, and such that

$$i = i_1 + i_2.$$

In other words, the characteristic of a one-port comprising two one-ports connected in parallel is the result of "summation of the characteristics of individual one-ports along current axis" (i.e., summation of the characteristics treated as the graphs of functions mapping voltages to currents). The procedure is illustrated in Fig. 4.12.

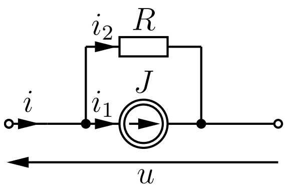


Figure 4.11: A resistor and a current source in series

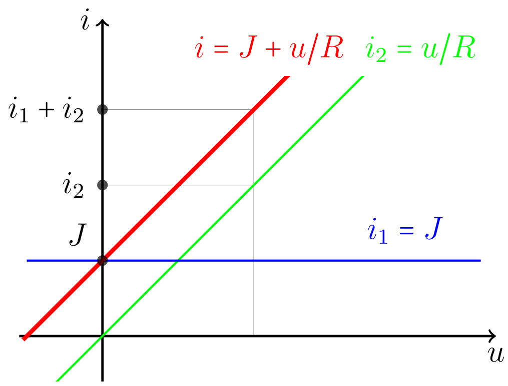


Figure 4.12: Construction of the characteristic of the one port presented in Fig. 4.11

4.1.2 Small-signal analysis

Usually, it is a hard problem to determine an analytic solution to a non-linear circuit. For a single non-linear one-port and a given voltage waveform across its terminals, one may find the current through the one-port in a graphical way, by "projecting the voltage through the voltage-current characteristic". Such a procedure is illustrated in Fig. 4.13. It shows that even though the voltage waveform is alternating (harmonic), the current waveform is no longer harmonic (although it is still periodic).

Let us study the signals shown in Fig. 4.14. The voltage across the non-linear one port has its DC component U_0 and a "small" harmonic (alternating) component. The current through the one-port also has its direct component and a periodic rest. This periodic residuum may

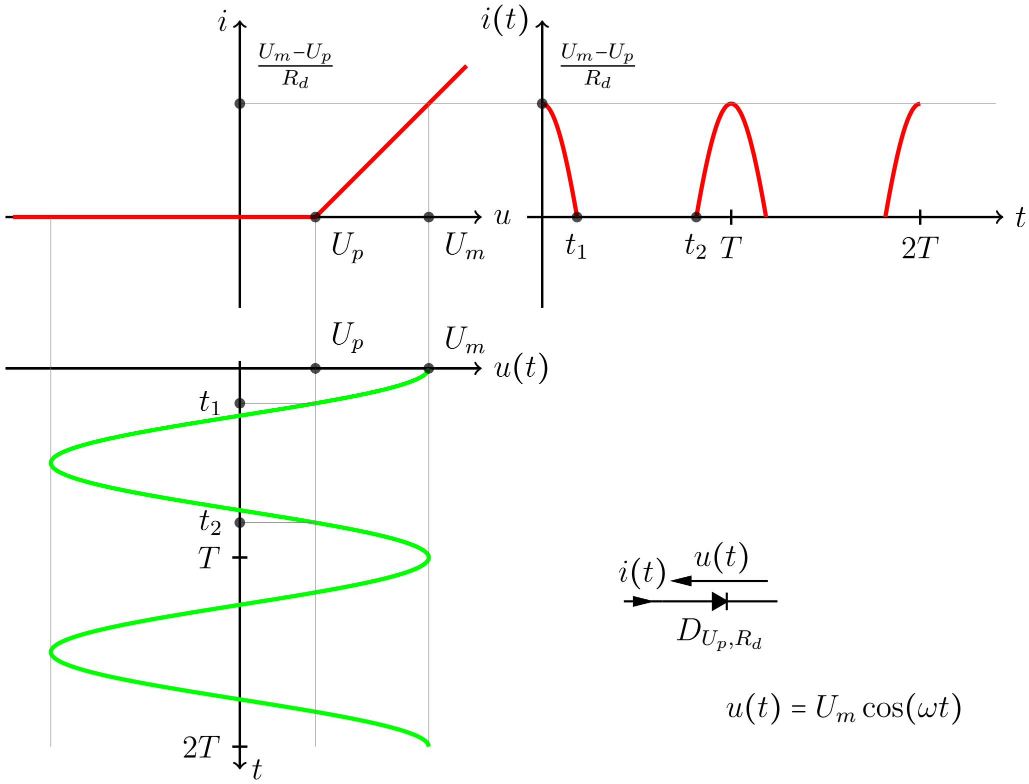


Figure 4.13: Projection of a voltage waveform through the voltage-current characteristic of a diode

be approximated with its first harmonic. Indeed, if we replaced the non-linear characteristic with its tangent line at point (U_0, I_0) (in Fig. 4.15 this linearization is shown as a black line segment), then the voltage waveform would project to a current waveform that is marked in Fig. 4.15 with a black line. The result of such a projection would consists of a constant and one harmonic only and, as you may observe in Fig. 4.15, does not differ from the original accurate current waveform (projection through the non-linear characteristic) significantly. Thus one may consider the result of "linear" projection a good approximation to the exact current. Such approximations constitute a base of the so-called *small-signal analysis*.

Two parameters play an important role in the small-signal analysis of non-linear resistive one-ports. These are dynamic resistance R_d and dynamic conductance G_d. If a considered non-linear resistive one-port has voltage-current characteristic having a differentiable parametrization

$$u = f(i),$$

in a neighbourhood of a point $(U_0,\ I_0)$ (called an operating point of the one-port), then the dynamic resistance and conductance are defined as, respectively,

$$R_d|_{(U_0,I_0)} = f'(I_0), \quad G_d|_{(U_0,I_0)} = \left(R_d|_{(U_0,I_0)}\right)^{-1}. \tag{4.1}$$

If the characteristic around the considered operating point have a differentiable parametrization

$$i = h(u),$$

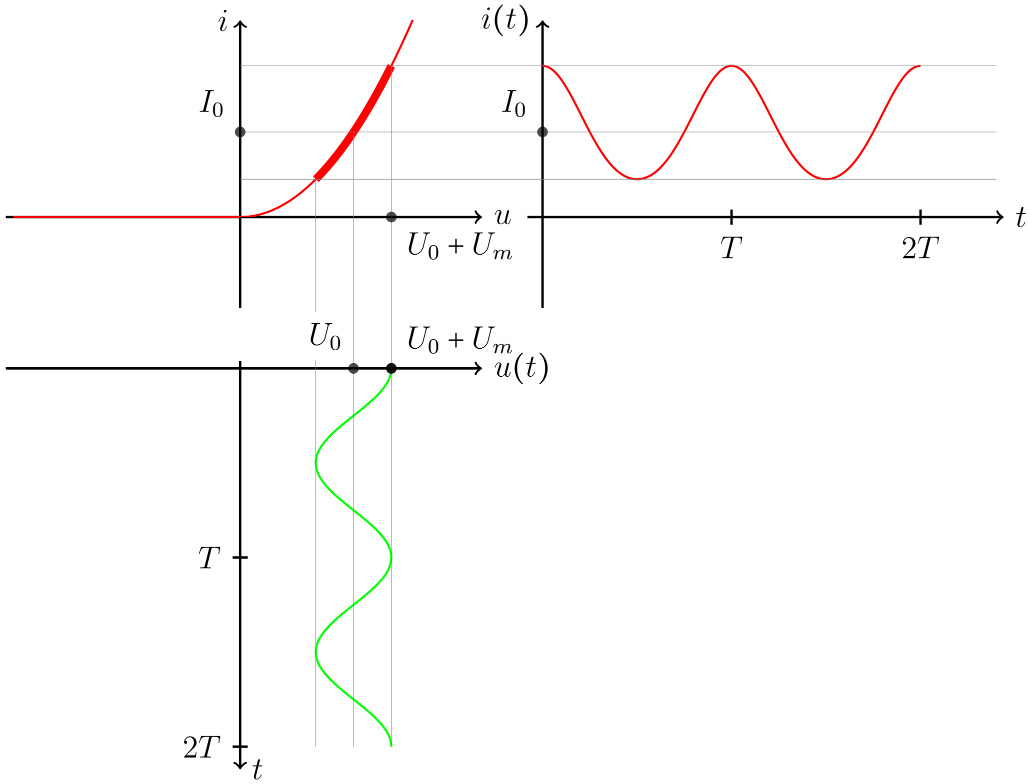


Figure 4.14: Projection of a voltage waveform through the voltage-current characteristic of a non-linear one-port

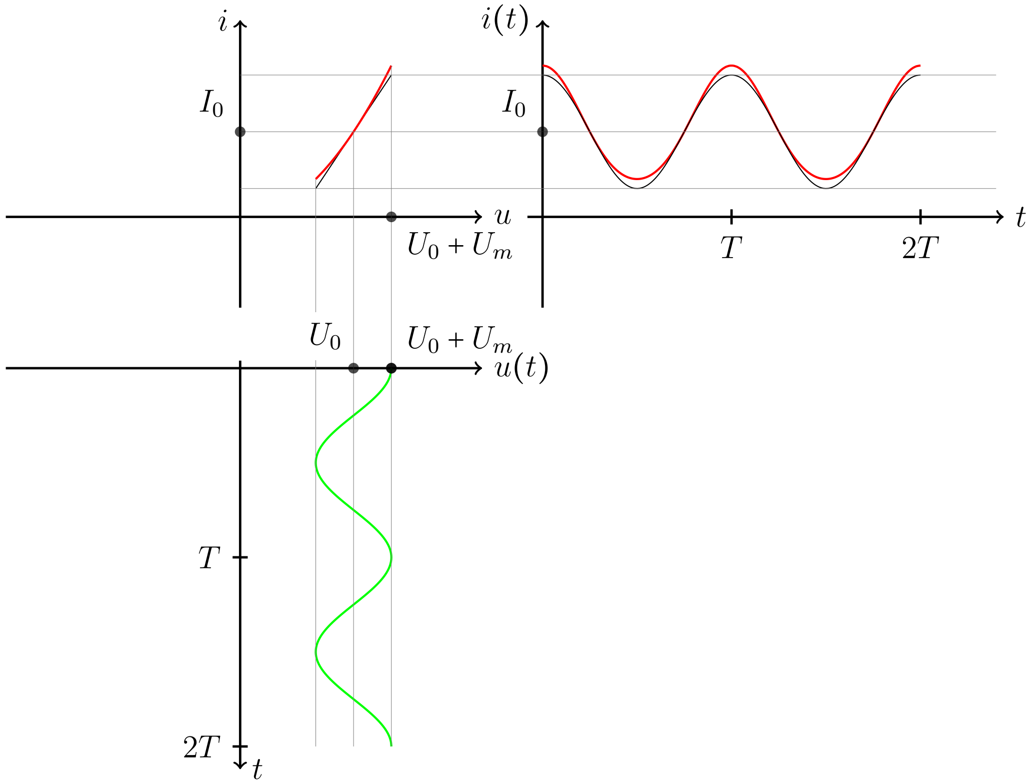


Figure 4.15: Projection of a voltage waveform through a linearized characteristic of a non-linear one-port (the result is drawn with black line; the exact current, see Fig. 4.14, is drawn with red line)

then the above dynamic parameters equal

$$G_d|_{(U_0,I_0)} = h'(U_0), \quad R_d|_{(U_0,I_0)} = \left(G_d|_{(U_0,I_0)}\right)^{-1}. \tag{4.2}$$

Let us stress that both the value of dynamic resistance and of dynamic conductance depend on the operating point, that is, they depend on the particular direct voltage U_0 across the one-port and direct current I_0 through this port.

Performing small-signal analysis we find "small" periodic signals after linearizing all the non-linear devices of a circuit. The term "small" refers to the fact that the smaller are the periodic excitations (e.g. voltages across one-ports) the more exact approximations we get using linearization of the non-linear devices.

To shed more light on the above considerations, let us study the following example. By means of a small-signal method, we will find current $i(t)$ flowing in the circuit shown in Fig. 4.16. Assume

$$e(t) = 3\text{ V} + e_1(t), \qquad U_p = 1\text{ V},$$
$$e_1(t) = 10\cos(\omega t)\text{ mV} \qquad R_d = 1\text{ k}\Omega.$$

Firstly, let us note that the one-port being a serial connection of a resistor R and a diode D_{U_p,R_d}

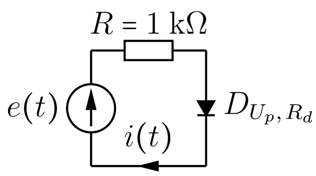


Figure 4.16

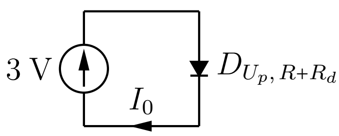


Figure 4.17

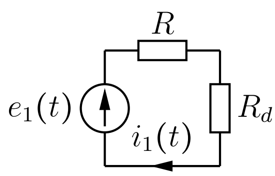


Figure 4.18

is equivalent to a diode $D_{U_p,R+R_d}$ (as shown in Fig. 4.17). The voltage across diode $D_{U_p,R+R_d}$ oscillates around the value $U_0 = 3$ V, and the current through this diode oscillates around the value $I_0 = 1$ mA. The operating point of the non-liner device is thus $(U_0, I_0) = (3\text{ V}, 1\text{mA})$. In a small neighbourhood of this point the voltage-current characteristic of the diode is given by equation

$$i = \frac{u - 1\text{ V}}{2\text{ k}\Omega}$$

and equivalently

$$u = 1\text{ V} + i \cdot 2\text{ k}\Omega.$$

The dynamic resistance of the diode at the operating point (U_0, I_0) is thus equal to

$$R_d + R = 2\text{ k}\Omega.$$

Therefore, the pulsating component $i_1(t) = i(t) - I_0$ of the current in the considered circuit may be approximated as the current in circuit presented in Fig. 4.18. Please note that the same circuit might have been obtained directly from the original circuit trough linearization of diode $D_{1\text{ V},R_d}$ (we would get a resistor R_d) around its operating point $(2\text{ V}, 1\text{ mA})$. Next, we find the solution to the linearized circuit

$$i_1(t) = \frac{10\cos(\omega t)\text{ mV}}{R + R_d} = 5\cos(\omega t)\ \mu\text{A}.$$

The result of the small-signal analysis is thus

$$i(t) \approx I_0 + i_1(t) = 1\text{ mA} + 5\cos(\omega t)\ \mu\text{A}. \tag{4.3}$$

Lets take another look at the characteristic of $D_{1\text{ V},2\text{ k}\Omega}$ diode (see Fig. 4.9). This characteristic is a straight line for voltages belonging to the segment centered at $U_0 = 3$ V and of a radius $E_m = 10$ mV. Therefore not only is current $i(t)$ well approximated with the right hand side of the equation (4.3), but it is in fact exactly equal to it.

4.1.3 Rectifiers

Rectifiers are non-linear two-ports that convert alternating current (or voltage) to direct current (or voltage). It is desired that for an alternating voltage of fixed amplitude supplying the input port of a rectifier, the voltage across the output port has possibly large direct component and possibly small pulsating residuum.

First, lets consider a *half-wave rectifier* presented in Fig. 4.19. Assume that such a two-port is loaded with a resistor R and supplied with alternating voltage $e(t) = E_m \cos\omega t$ (we will assume $E_m > U_p$), as shown in Fig. 4.20.

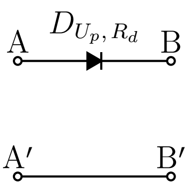


Figure 4.19: Half-wave rectifier

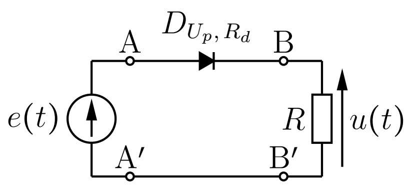


Figure 4.20: A circuit containing a half-wave rectifier

The one port to the right of terminals A-A′ is equivalent to a diode $D_{U_p,R+R_d}$. "Projecting waveform $e(t)$ through the characteristic" of this diode (as shown in Fig. 4.13), we get the waveform of the current through resistor R. Multiplying this current by R we eventually get voltage u at the output port of the rectifier

$$u(t) = \begin{cases} \left(e(t) - U_p\right)\frac{R}{R+R_d} & \text{if } e(t) \geq U_p, \\ 0 & \text{if } e(t) < U_p. \end{cases} \tag{4.4}$$

Clearly, this voltage has a non-zero direct component. If $R \gg R_d$, then voltage u has approximately the shape of "positive half-waves" of voltage e that are trimmed from bottom by U_p (dashed line in Fig. 4.22).

We can improve the rectifying function of the half-wave rectifier (i.e. we can enlarge the direct component and reduce the pulsating residue at the output port) by adding a capacitor C in parallel with resistor R, as depicted in Fig. 4.21.

The exact analysis of the signals in the circuit presented in Fig. 4.21 is out of the scope of this manual. Hence, let us just sketch the result of such an analysis. If at some instant of time t_0 (see Fig. 4.22) voltage $e(t)$ becomes greater than $u(t)$ by more than U_p, then, starting from that instant and till another instant $t_1 > t_0$ (see Fig. 4.22) at which $e(t_1) = u(t_1) + U_p$, voltage u is descried with the formula

$$u(t) = A\mathrm{e}^{-\frac{t-t_0}{\tau_0}} + B\cos(\omega t + \beta),$$

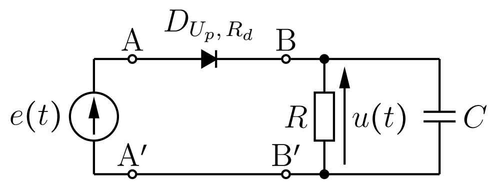


Figure 4.21: Half-wave rectifier loaded with a parallel RC one-port

where

$$B = E_m \frac{R}{R+R_d}, \quad A = e(t_0) - U_p - B\cos(\omega t_0 + \beta), \quad \beta = -\mathrm{arc\,tg}(\omega\tau_0), \quad \tau_0 = \frac{RR_dC}{R+R_d}.$$

In particular, if

$$R \gg R_d, \quad \text{and } \omega\tau_0 \ll 1, \tag{4.5}$$

then for $t \in (t_0, t_1)$ voltage $u(t)$ may be approximated as

$$u(t) \approx e(t) - U_p,$$

i.e. voltage u follows the electromotive force e decreased by U_p.
If at some instant of time t_1 electromotive force $e(t)$ becomes smaller than $u(t) + U_p$, then, till t_2 at which

$$e(t_2) > U_p + u(t_2),$$

voltage u is given by the following formula

$$u(t) = (e(t_1) - U_p)\mathrm{e}^{-\frac{t-t_1}{RC}}.$$

If conditions (4.5) are satisfied, then for $t \in (t_1, t_2)$ voltage u may be approximated as linearly decreasing function

$$u(t) \approx \big(e(t_1) - U_p\big)\left(1 - \frac{t-t_1}{RC}\right).$$

Thus, voltage u in circuit shown in Fig. 4.21 has the shape presented in Fig. 4.22.

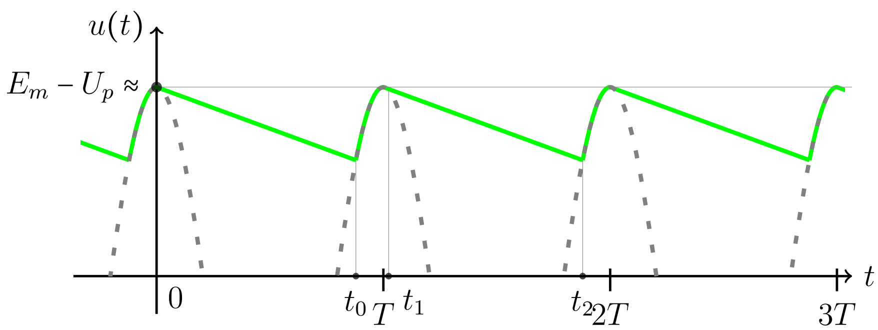


Figure 4.22: Voltages u of Fig. 4.21 (solid line) and of Fig. 4.20 (dashed line).

In the rectifier shown in Fig. 4.23 there are two diodes. The one-port to the right of terminals A-A′ is equivalent to resistor R and (one!) diode $D_{2U_p,2R_d}$ in series. Thus, the rectifier of Fig. 4.23 is just another half-wave rectifier, which differs from the previously considered

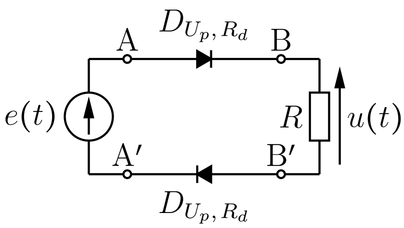


Figure 4.23: Half-wave rectifier comprising two diodes

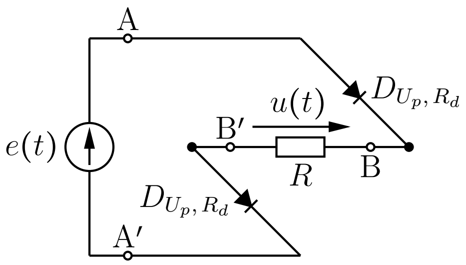


Figure 4.24: Half-wave rectifier as a part of a Graetz bridge

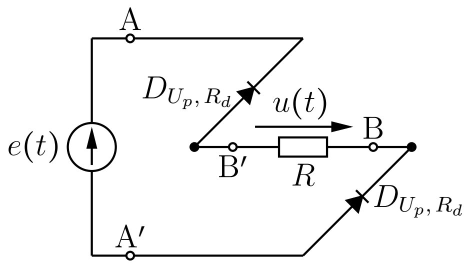


Figure 4.25: Half-wave rectifier as a part of a Graetz bridge

in the threshold voltage ($2U_p$ instead of U_p) and the dynamic resistance of the diode ($2R_d$ in place of R_d). Therefore, voltage u from Fig. 4.23 has a waveform presented in Fig. 4.26.

The circuit diagram from Fig. 4.23 can be also drawn as in Fig. 4.24. A similar circuit is presented in Fig. 4.25. The reversed polarity of the diodes results in a waveform u shown in Fig. 4.26 with green line. In a sense, in the circuit from Fig. 4.24 "positive half-waves of electromotive force e are exploited" and in the circuit from Fig. 4.25 — "the negative half-waves are used". One may pose a question whether it is possible to exploit "full wave" in one rectifier. It occurs possible in a structure called a *diode bridge* or a *Graetz bridge*[1] shown

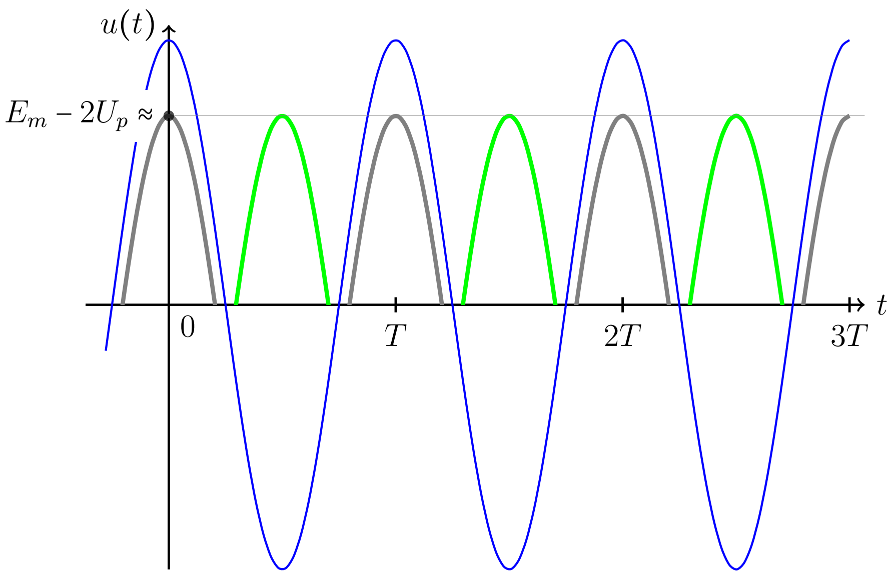


Figure 4.26: Waveform of voltage u of Fig. 4.23 and 4.24 (bold gray line) and of Fig. 4.25 (green line). The electromotive force e is shown with blue line

in Fig. 4.27 (the diode bridge is a two-port consisting of four diodes in configuration shown in the Figure).

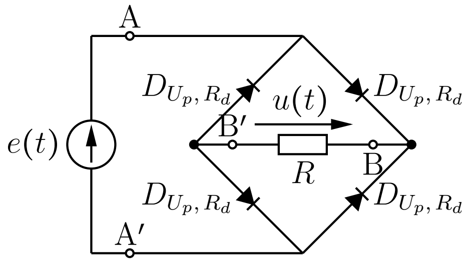


Figure 4.27: Diode bridge (Graetz bridge)

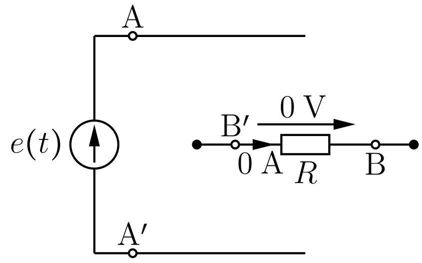


Figure 4.28

Let's recall that every D_{U_p,R_d} diode acts as an open-circuit when supplied with a voltage smaller than U_p. Depending on a time instant (or on the instantaneous value $e(t)$, to be precise), the circuit of Fig. 4.27 behaves like

- the circuit of Fig. 4.24, if $e(t) > 2U_p$,
- the circuit of Fig. 4.25, if $e(t) < -2U_p$,
- the circuit of Fig. 4.28, if $|e(t)| \leq 2U_p$.

The waveform of voltage u from Fig. 4.27 is marked with bold gray line in Fig. 4.29 (see also Fig. 4.26). In the same Figure, the green line depicts the waveform of a voltage that would result from attaching a capacitor in parallel with resistor R (similarly as in the case of half-wave rectifier).

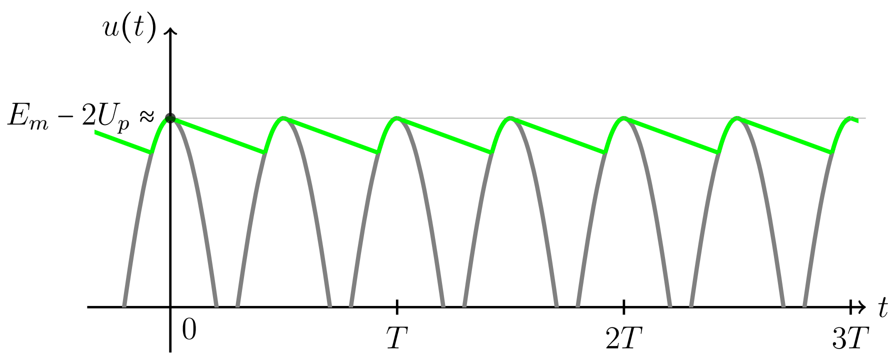


Figure 4.29: The waveform of voltage across output port of a Graetz bridge of Fig. 4.27 without (gray line) and with a capacitor connected in parallel with resistor R (green line)

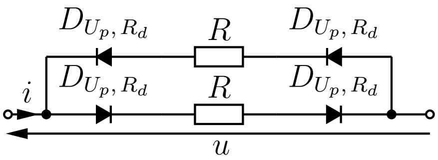
Let us note that the waveform presented in Fig. 4.29 compared to the voltage across the output of half-wave rectifier (see. Fig. 4.22), has two times smaller period and a smaller pulsating component.

[1]The name of the bridge is a tribute to Leo Graetz (1856 – 1941), a German physicist who studied such circuits. It is worth to mention that the true inventor of the bridge was a Polish electrotechnician Karol Pollak (1859 – 1928), who even patented his invention in Germany (patent number DRP 96564)! By the way, Pollak was also the inventor of electrolytic capacitors.

Problems for self study

Solutions to the problems are given on page 96.

1. Sketch the voltage-current characteristic of the following one-port. Assume $U_p = 1$ V, $R_d = 0{,}5$ kΩ, $R = 1$ kΩ.



2. A non-linear resistor is described with the following formula:

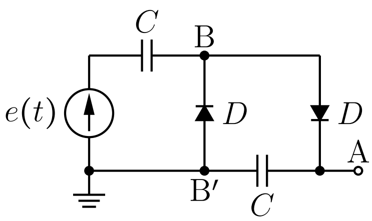
$$i = \alpha u^3, \qquad \alpha = 1\ \mathrm{mA/V^3}$$

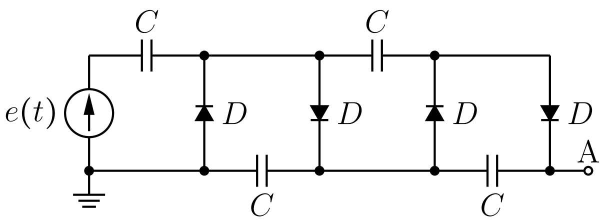
Find the exact and an approximated (by the means of small-signal method) waveforms of the current through this resistor provided that the voltage across the resistor equals

$$u(t) = 1\ \mathrm{V} + 5 \cos \omega t\ \mathrm{mV}.$$

(The following formula may help you in the task $\cos x = \frac{e^{jx}+e^{-jx}}{2}$).

3. Sketch the waveforms of the electric potentials of A terminals of the following circuits[2], provided that $e(t)$ is an alternating voltage, and each of the diodes is of model $D = D_{0\,\mathrm{V},R}$.



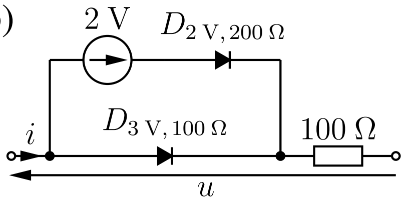


[2]The presented circuits have a structure of the so-called Cockroft-Walton generator. The name is to pay tribute to Irish (John Cockroft) and British (Ernest Walton) physicists, who used such circuits in the accelerator they constructed, and in which the first artificial nuclear disintegration was performed. They won the Nobel Prize in Physics for their research. As it often happens, the name of the circuit does not pay tribute to its true inventor, a Swiss physicist Heinrich Greinacher, who invented the circuit in 1919.

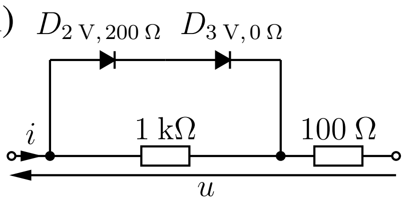
4.2 Homework

Homework 4.1. Determine and draw the voltage-current characteristic of the one-port marked below with “(p)” (even stand numbers) or “(n)” (odd stand numbers).

(p)



(n)



Homework 4.2. Determine the current through the one-port considered in problem 4.1, provided that the voltage across this one-port equals

$$u(t) = U_0 + 0{,}5\cos\omega t\ \mathrm{V}.$$

Consider two cases: $U_0 = 1$ V and $U_0 = 6$ V.

4.3 Composing voltage-current characteristics

In this section we will measure voltage-current characteristics of nonlinear one-ports that include a blue LED (in series with resistor $R_{\mathrm{blue}} = 100\,\Omega$), to which we will connect other elements in series or/and in parallel. As in Exercise 2, we will obtain graphs of the characteristics of the analyzed one-ports using the oscilloscope operating in the X-Y mode, with channel X of the oscilloscope fed with voltage u across terminals of the analyzed one-port, and channel Y driven by voltage proportional to current i through this one-port. In the analyzed circuit it will be most convenient to use the voltage across the resistor R_{blue} for such indirect measurement of current i . In order to enforce various values of voltage across the analyzed one-port, which is needed to measure its characteristic, we will use the triangular wave generator from the multifunctional instrument connected to the terminals of the one-port. In order to protect the generator output against accidental short circuits, we will connect it to the circuit by means of a follower, that is by the amplifier card set to amplify the signal from input `IN1` with gain 1. In the course of this exercise we will use the oscilloscope to determine the gradient of the voltage-current characteristic of a nonlinear one-port at a given operating point. This task will be relatively easy because the characteristics of the one-ports analyzed in the exercise can be regarded as piecewise linear. Now, we will present the procedure of the measurement of dynamic resistance on the basis of an oscillogram.

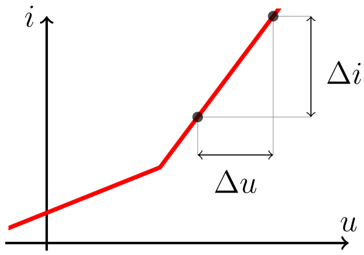


Figure 4.30: Definition of the slope of the voltage-current characteristic…

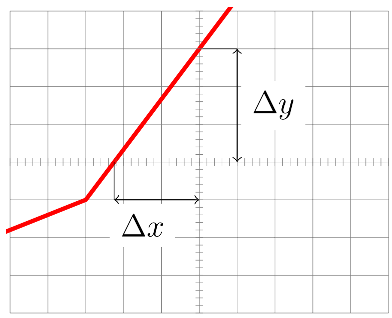


Figure 4.31: …and a way to measure it using an oscilloscope

In the case of the characteristic shown in Fig. 4.30, the dynamic resistance corresponding to a particular segment of the characteristic is equal to the *inverse* of the gradient of the graph:

$$R_d = \frac{1}{G_d} = \frac{\Delta u}{\Delta i}.$$

If the dynamic resistance is to be determined using the oscilloscope, first read (at the bottom of its screen) the gains of both channels, i.e. values S_x and S_y expressed in $[\mathrm{V/div}]$. Then, using the knobs for shifting the oscillogram along the horizontal (X) and vertical (Y) axis, move the graph of the characteristic in such a way that the segment for which we measure the slope, fits completely in the upper left quarter of the screen as shown in Fig. 4.31. Try to place it as much to the left and to the top of the screen as possible (i.e. so that segments Δx and Δy are as long as possible) — this improves accuracy of measurement. Read values Δx and Δy, using the fine division (0.2 div) imprinted on the main grid lines in the screen of the oscilloscope. You can also use the oscilloscope cursors. When computing dynamic resistance, remember that the oscilloscope measures *voltages*, so current i from Fig. 4.30 has to be measured indirectly, by measuring voltage across the resistor $R_{\text{blue}} = 100\ \Omega$. Eventually, we obtain the following formula:

$$R_d = \frac{\Delta u}{\Delta i} = \frac{\Delta x \cdot S_x}{\Delta y \cdot S_y / R_{\text{blue}}} = R_{\text{blue}} \frac{\Delta x}{\Delta y} \cdot \frac{S_x}{S_y}. \tag{4.6}$$

4.3.1 Characteristic of the blue LED (with a resistor)

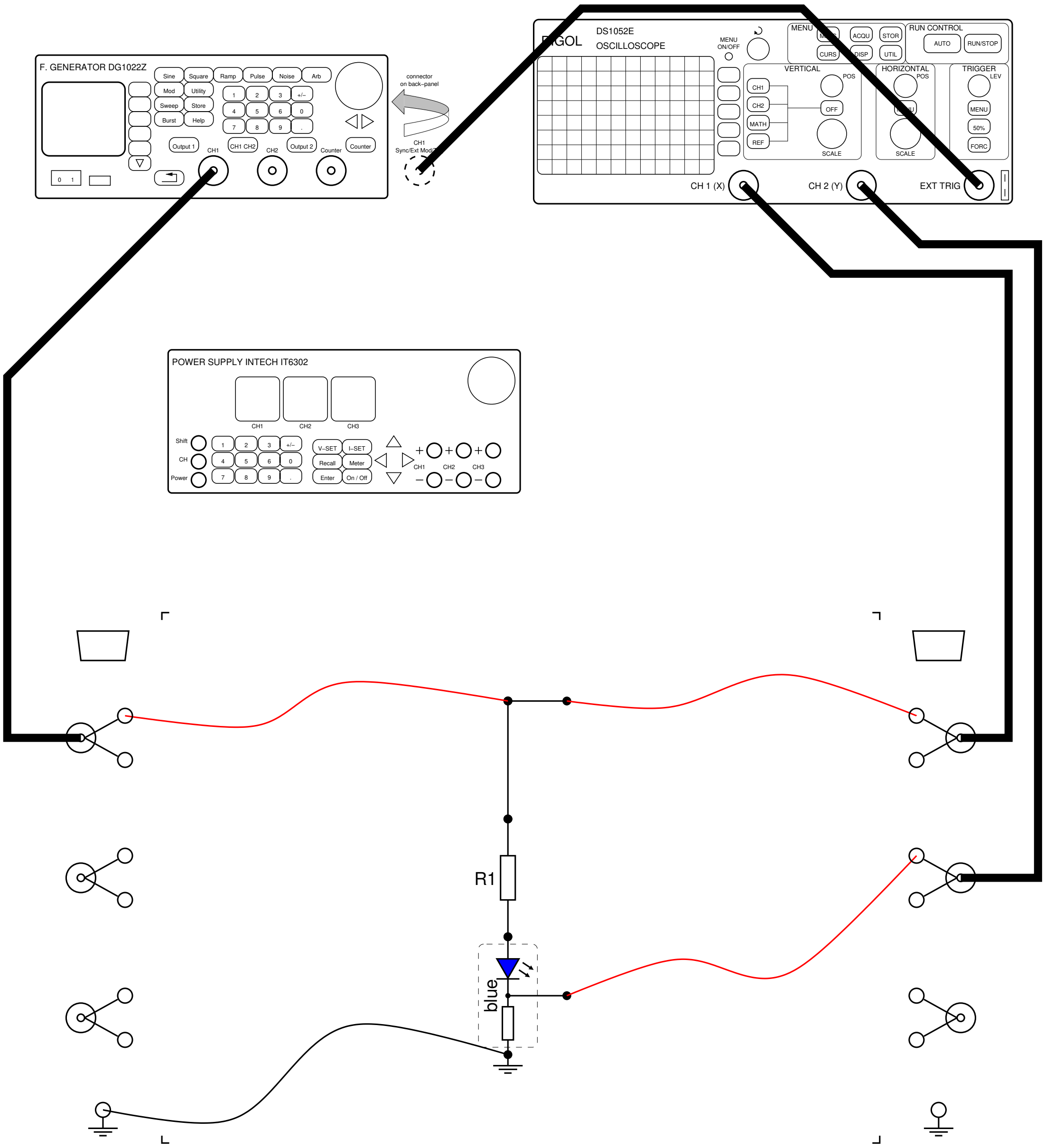


Figure 4.32: Setup for measuring voltage-current characteristic of a blue LED with a resistor

1. Deactivate outputs of function generator. Assemble measurement setup as shown in Fig. 4.32, using resistor $R_1 = 500\ \Omega$.

2. Set the function generator to generate triangular wave (`Ramp`) of frequency $1\ \mathrm{kHz}$, peak-to-peak value $20\ \mathrm{Vpp}$, mean value $0\ \mathrm{V}$ and symmetry 50%. Activate `CH1` output of the generator.

3. Initially, preset the gains in the oscilloscope to $2\ \mathrm{V/div}$ in channel X and to $0.2\ \mathrm{V/div}$ in channel Y. Set the oscilloscope to X-Y mode of operation. Set both of its channels to DC coupling (DC). Using the shift knob `POSITION` for one channel and then for the other, set the origin of coordinates (indicated by colour „arrows” on the left and top margin of the screen) near to the lower boundary of the screen (e.g. one div above the first grid line) and center it horizontally.

4. Observe the voltage-current characteristic in the screen. It can be approximated with a piecewise linear characteristic consisting of two line segments: the left segment is horizontal and corresponds to zero current, and the right one has some slope. When approximated in this way, the characteristic is fully characterized by two numbers: threshold voltage U_p and slope G_d (see characteristic of the diode D_{U_p,R_d} shown in Fig. 4.9). Both values are to be measured using the oscilloscope, then the threshold voltage and the dynamic resistance $R_d = G_d^{-1}$ are to be recorded into appropriate spaces provided in Tab. 4.1. The graph of the characteristic (approximated with two line segments) is to be drawn in the report. Check if the dynamic resistance $R_d = G_d^{-1}$ of the right segment is somewhat greater than the value of resistance $R_{\mathrm{blue}} + R_1$ (by the value of resistance of the LED *itself*).

5. Change the generated signal frequency from $1\ \mathrm{kHz}$ to $1\ \mathrm{Hz}$ and observe simultaneously the dot in the screen of the oscilloscope and the LED. On which segment of the characteristic does the LED emit light (conduct current)? Set the frequency range in the function generator back to $\times 1\ \mathrm{k}$.

4.3.2 Changing slope of a segment of characteristic by adding a resistor in parallel

For the purpose of small-signal analysis to be performed in the next part of this excersise, we will assemble a one-port, whose characteristic will have a non-zero gradient in the whole range of input voltages. It can be obtained by connecting a resistor R_2 in parallel with a diode (or diodes).

This time, the characteristic of the one-port will be treated as a voltage-voltage characteristic. The input voltage is the voltage u across the analyzed one-port, and the output voltage — voltage $u_{R_{\mathrm{blue}}}$ across the resistor $R_{\mathrm{blue}} = 100\ \Omega$, used so far for indirect measurement of the one-port's current.

1. Deactivate outputs of function generator. Assemble measurement setup as shown in Fig. 4.33, using the following parameters of elements: $R_1 = 500\ \Omega$, $R_2 = 1\ \mathrm{k}\Omega$. Activate `CH1` output of the generator.

2. Make sure that the oscilloscope is running in X-Y mode, as in section 4.3.1. Set the gain in channel Y to $0.2\ \mathrm{V/div}$ and shift the origin of coordinates a bit further from the lower boundary of the screen (e.g. to two divs above the first grid line), and center it horizontally.

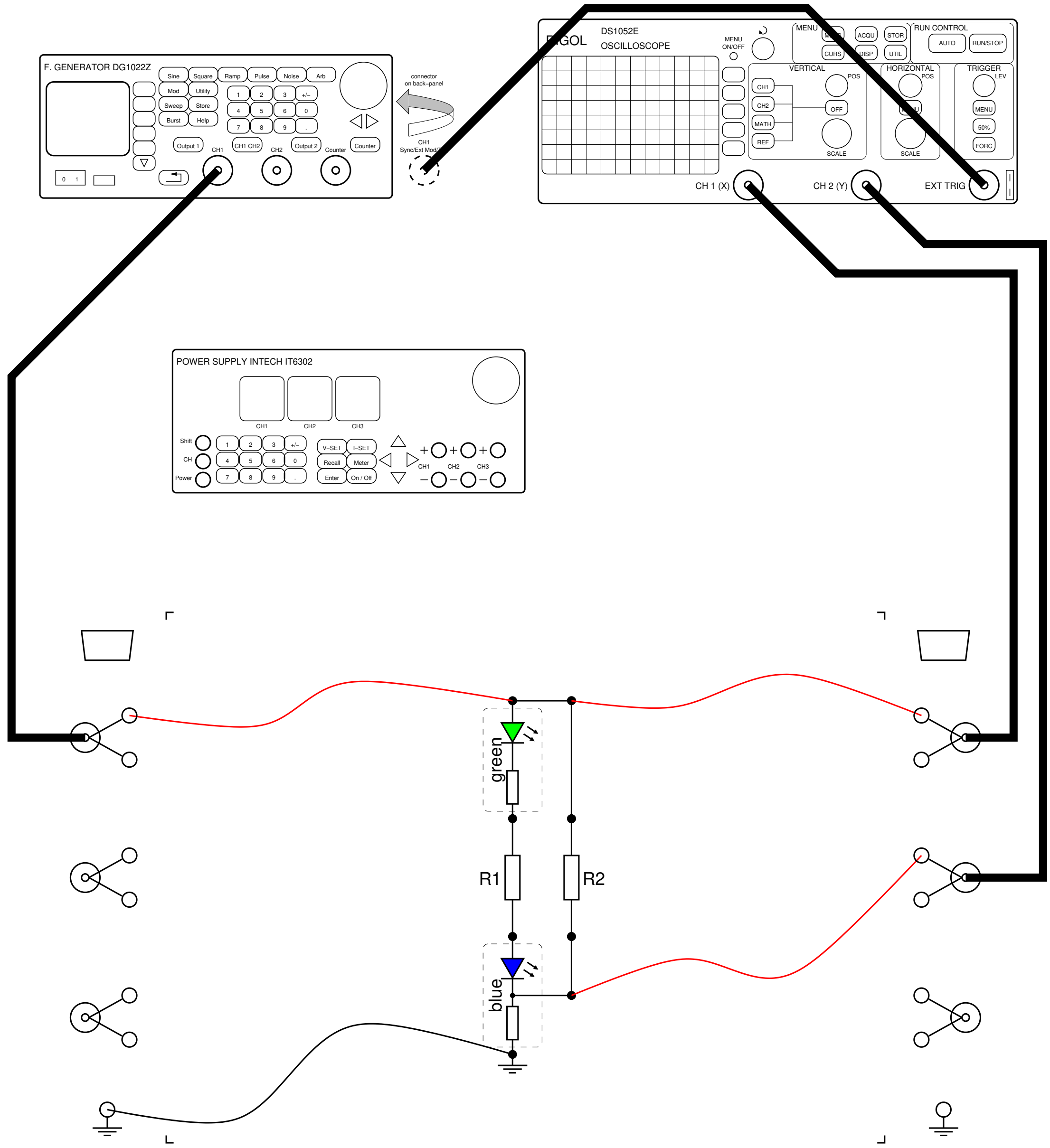


Figure 4.33: Setup for measuring voltage-current characteristic of a one-port used for small-signal analysis in the next section

3. Change the frequency of the triangular signal from the function generator back to $1\ \mathrm{kHz}$.

4. Observe in the screen the characteristic $u_{R_{\text{blue}}}(u)$. It consists of two line segments with non-zero slopes $\frac{\Delta u_{R_{\text{blue}}}}{\Delta u}$ The slope of each of the segments can be considered the „gain" of the circuit for (small) alternating component when the (DC) operating point lies on a given segment. For voltage u below the threshold voltage (this includes negative voltages, when both — green and blue — diodes are reverse-polarized), our circuit reduces to a voltage divider with the division ratio:

$$\frac{u_{R_{\text{blue}}}}{u} = \frac{\Delta u_{R_{\text{blue}}}}{\Delta u} = \frac{R_{\text{blue}}}{R_{\text{blue}} + R_2}. \tag{4.7}$$

For voltage u above the threshold voltage, both diodes are forward-polarized, and our circuit *for the alternating component* reduces to a voltage divider with the division ratio:

$$\frac{\Delta u_{R_\text{blue}}}{\Delta u} = \frac{R_\text{blue}}{R_\text{blue} + (R_2 \parallel R_d)}, \tag{4.8}$$

where R_d is a series connection of resistor R_1, the dynamic resistance of the green diode, resistance $R_\text{green} = 200\ \Omega$ placed on the board with the green diode and the dynamic resistance of the blue diode (*without* resistance R_blue placed on its board):

$$R_d = R_1 + R_{d,\text{green}} + R_\text{green} + R_{d,\text{blue}}. \tag{4.9}$$

As a first approximation, you can assume that $R_{d,\text{green}} = R_{d,\text{blue}} = 20\ \Omega$.

The graph of the characteristic (approximated with two line segments) is to be drawn as a separate figure in the report. Take into account the gain of each of the channels of the oscilloscope. The voltage corresponding to the vertex of the characteristic and the measured gradients $\frac{\Delta u_{R_\text{blue}}}{\Delta u}$ of the segments of the characteristic are to be recorded in appropriate cells provided in Tab. 4.1. Compare the measured gradients of the characteristic segments with the values computed theoretically using formulas (4.7)–(4.9).

5. For a while, change frequency of the signal from 1 kHz to 1 Hz and observe simultaneously the dot in the screen of the oscilloscope and whether *both* diodes emit light (or not). Note the segment to which belongs the operating point of the circuit when the diodes emit light.

4.4 Small-signal analysis

4.4.1 General description of the measurement setup

In the small-signal analysis we will be exciting the nonlinear circuit from Fig. 4.34 (identical as in the setup in Fig. 4.33) with a signal that has an alternating component e_g (sinusoidal signal from the generator of the TOiS_Toy program) on top of an adjustable DC component E_0 (voltage from the adjustable voltage supply in the multifunctional device). To obtain the sum of these signals (alternating and direct) we will use the amplifier card — actually, the alternating signal will be *subtracted* from the DC component, but it makes no difference (it only means the change of the initial phase of the sine wave by 180 degrees). The slopes measured in the section 4.3.2 can be used for purposes of this section.

The TOiS_Toy program uses `Maya44 USB` card, which *does not transfer the DC component* (it is just as if we set the coupling in the oscilloscope channel to `AC`). That is why in this part of the exercise we will observe with measurement instruments the alternating component of the waveform u_{R_blue} only. It may happen that the spectrum analyzer of the TOiS_Toy set will show rudimentary value of the DC spectrum bar, (it is the *first* value in column `A [mV]` in the analyzer window), but it is only a negligible inaccuracy resulting from transient states.

The characteristic of the analyzed nonlinear circuit measured in section 4.3.2 can be regarded as consisting of two linear segments, joined at the gradient change point. This means that the alternating component will fulfill the assumptions of the small-signal method, if only the range of the input voltages (DC component plus the alternating one) does not include the gradient change point. In particular, if the DC component is equal to the voltage at the gradient change point, *none* alternating current will fulfill these assumptions: lower and upper halves of the sine wave will act on segments of different slopes.

4.4.2 Assembly and calibration of the measurement setup

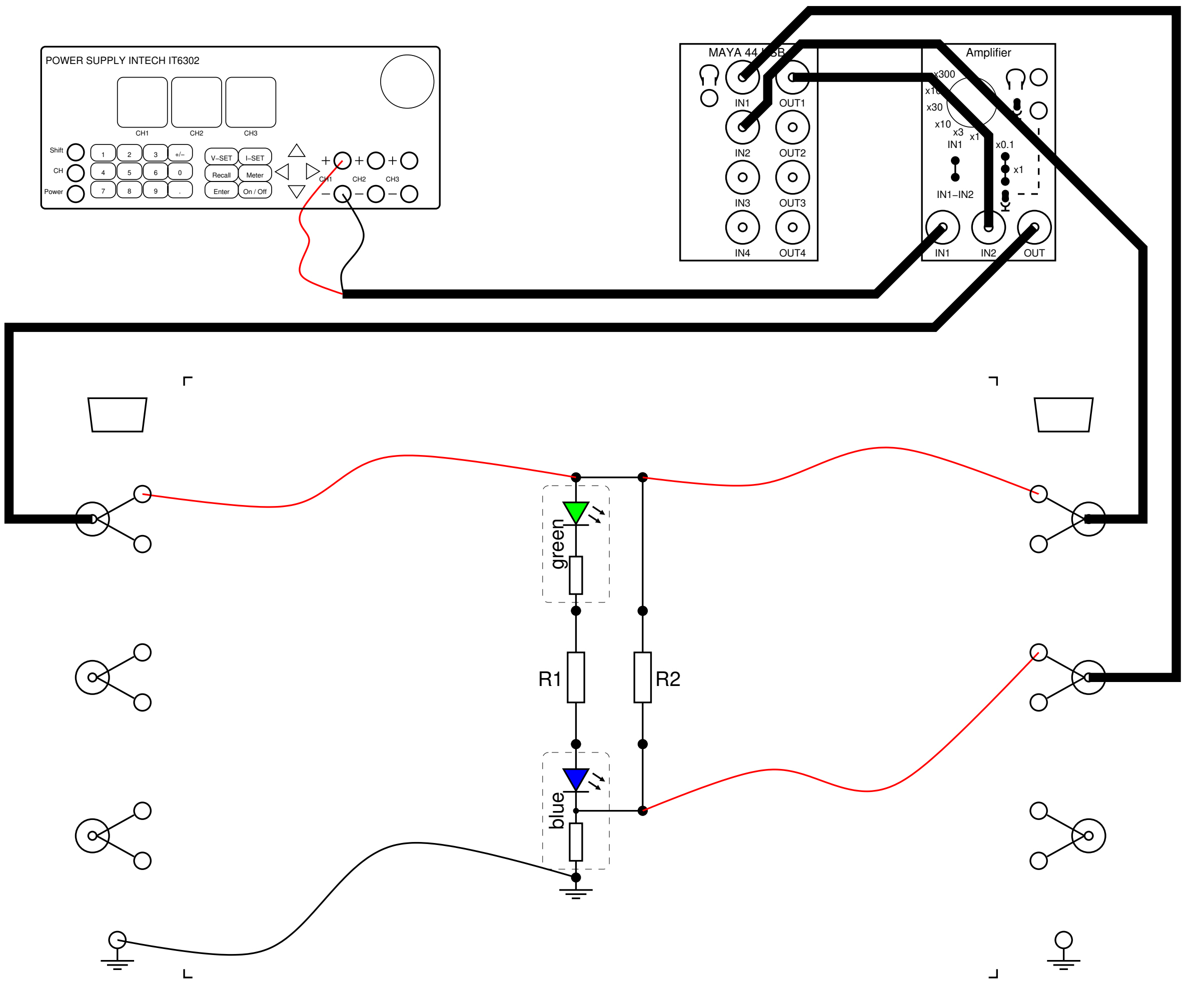


Figure 4.34: Setup for small-signal analysis

Power off the crate and deactivate `CH1` channel of the power supply. Set `CH1` channel voltage to a small value e.g., 0.5 V and its current limit to 0.1 A. Assemble the measurement setup according to Fig. 4.34 (as in section 4.3.2, using resistors $R_1 = 500\ \Omega$ and $R_2 = 1\ \mathrm{k}\Omega$). Set the amplifier to perform subtraction of input signals and its gain to 1. Next, power on the crate and activate `CH1` channel of the power supply. Turn on the computer and launch the TOiS_Toy environment. Run the following three virtual instruments from the TOiS_Toy set:

- the sine wave generator at output `OUT1` of `Maya44 USB` card, leave the default frequency (200 Hz) and amplitude ($E_{g,\max} = 0{,}5$ V) settings of the generator,
- the oscilloscope, in which activate channel `Inp.1`, set the slider `Pos.` to value 0.5 and gain to 20 mV/div, use the default time base setting 1 ms/div,

- the spectrum analyzer (operating in bar mode), in which set: `Pos.` to value 0.5, horizontal scale to $100\ \mathrm{Hz/div}$ (these are default values), and gain to $5\ \mathrm{mV/div}$.

Next, initiate operation of the TOiS_Toy program in repeated single run mode (key `Repeat` in menu Run / Once or appropriate button from the TOiS_Toy window toolbar).

To compensate possible gain inaccuracies of the amplifier, the virtual instruments in TOiS_Toy need to be calibrated. For this purpose, in the spectrum analyzer temporarily activate channel `Inp.2`, from menu Options choose Input/Output, and in the opened window `Input/Output Options`, with the slider `Input gain` gently adjust the amplitude of the first harmonic (shown by the spectrum analyzer as the *second* value in column `A [mV]`) as close as possible to value 250 (it should be *half* of the sine wave amplitude). Then, close the `Input/Output Options` window and activate channel `Inp.1` in the spectrum analyzer.

4.4.3 Measurements of the DC component for various DC operating points

1. For supply voltage of $E_0 = 0$, measure the amplitude $U_{R_{\mathrm{blue,max}}}$ of signal $u_{R_{\mathrm{blue}}}$ with the oscilloscope of the TOiS_Toy set, by pointing with the cursor, the maximum of the waveform and pressing the left mouse button. Compute the voltage gain for the alternating component at this operating point:

$$K_u = \frac{U_{R_{\mathrm{blue,max}}}}{E_{g,\mathrm{max}}}$$

   and compare it with the slope of the characteristic at the origin (measured in section 4.3.2).

   Record the computed values of the amplitude of the alternating component of voltage $u_{R_{\mathrm{blue}}}$ and of the voltage gain in appropriate cells provided in Tab. 4.2.

2. Gently increase voltage E_0 (with a step not greater than $0.1\ \mathrm{V}$) of the voltage supply until the LEDs[3] barely start to emit light (it should help if you shield the diodes from the outside sources of light and reduce the distance between your eye and the diodes). Record thus determined value E_0 in Tab. 4.2, and then also the values of amplitude of the alternating component of voltage $u_{R_{\mathrm{blue}}}$ and of voltage gain at this operating point E_0.

3. In an even more gentle way (**without exceeding the value** $10\ \mathrm{V}$**!**), increase voltage E_0 of the power supply until maximal value of amplitude of the *second* harmonic of waveform $u_{R_{\mathrm{blue}}}$ is obtained. It is shown by the spectrum analyzer as the *third* value in column `A [mV]`). Record in Tab. 4.2 this „critical" value $E_{0,\mathrm{crit.}}$[4]. Observe in the oscilloscope large differences between the upper and lower „halves" of the output sine wave. Save the plots of the waveform and of the amplitude spectrum of voltage on disk, and later print them and attach to the report. *Important! The plots need to be saved in the single run mode (*Run / Once*) only.*

2×

→PNG

[3] The diodes flicker instead of emitting light in a continuous way because the measurements are performed in the mode of repeating single runs of TOiS_Toy's instruments.

[4] Search for the maximum of the second harmonic is similar (it is a very rough approximation of it) of measurement of total harmonic distortion. This maximum should be located *near* the measured characteristic slope change point (but it can be shown that *not* exactly at this point).

4. Increase voltage E_0 of the voltage supply to $E_{0,\text{crit.}} + 1$ V. Record this value in Tab. 4.2, and then also the values of amplitude of the alternating component of voltage $u_{R_{\text{blue}}}$ and of voltage gain at this operating point E_0. Compare the determined gain with the slope of the right segment of the characteristic measured in section 4.3.2[5].

5. Repeat the same measurement procedure for $E_0 = E_{0,\text{crit.}} + 2$ V.

On the horizontal axis of the analyzed one-port characteristic measured in section 4.3.2, mark the intervals corresponding to the changes of value of the input voltage in each of the above cases 1–5.

4.5 Full- and half-wave rectifier

4.5.1 General description of the measurement setup

In this part of the exercise, you will examine the principle of a full-wave rectifier (in the form of Graetz bridge) and a half-wave rectifier (as „half" of this bridge), and also a way of smoothing output voltage with a capacitor. It is a drawback of Graetz bridge that if excited with a voltage with respect to ground (and with this kind of voltage we deal at the output of the generator used in the lab), the bridge output voltage (obtained across its „diagonal") is a floating voltage. To observe it with the oscilloscope, we will have to use the amplifier card, which will subtract nodal voltages (with respect to ground) at both nodes of the output diagonal of the bridge. To illustrate the principle of the bridge, we will build it with LEDs, which is never done in practice because of their large threshold voltages.

4.5.2 Assembly of the measurement setup

Power off the crate and deactivate output of the function generator.

1. Assemble measurement setup according to Fig. 4.35 using resistor $R = 5$ kΩ (*without* capacitor C marked with dashed line). This resistor simulates the load of the bridge, through which the rectified current is to flow. In the upper branches of the bridge, we will use red LEDs, and in the lower - green LEDs. Then, power on the crate.

2. In the stand-alone oscilloscope, activate both its channels with gains 2 V/div and DC coupling (DC). Use external triggering and set the time base to 2 ms/div.

3. Set the function generator to generate sinusoidal signal with frequency 100 Hz, peak-to-peak value 16 Vpp and offset 0 V.

4. Set the amplifier used for measurement of the „floating" voltage across the diagonal of the bridge to subtraction of input signals with gain 1.

[5]Notice that this time the diodes do not flicker but emit light in a continuous way, this is because even though the measurements are still performed with TOiS_Toy's instruments operating in the repeated single run mode, the DC component of the voltage supply provides *continuous* forward polarization of the diodes.

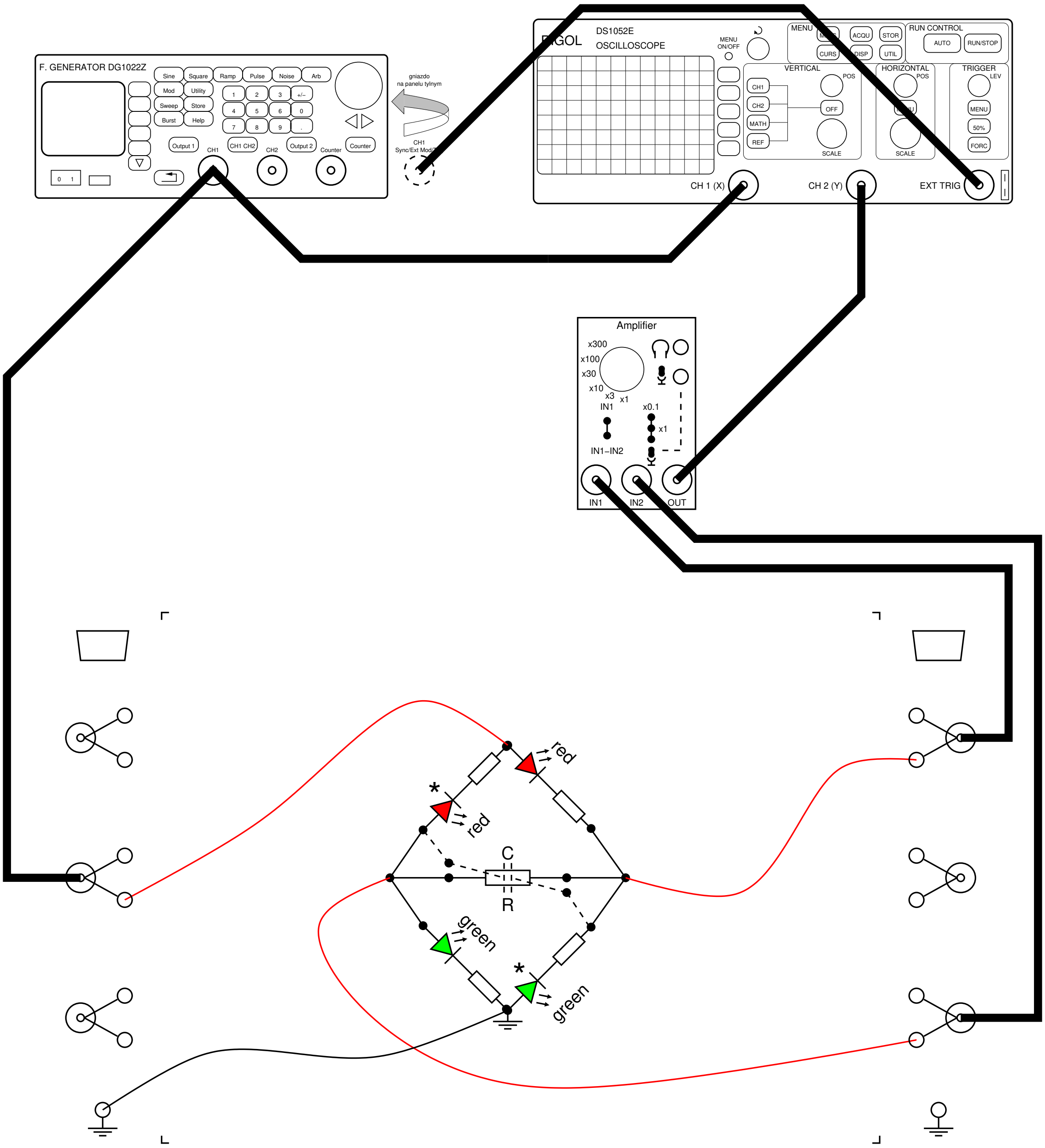


Figure 4.35: The setup for analysis of the Graetz bridge

4.5.3 Analysis of various rectifiers

1. Observe in the oscilloscope, the input voltage (from the generator) and the output voltage (across the resistor inserted into the diagonal of the bridge). It can be seen that the full-wave rectifier does not compute the absolute value of the input signal. By what value are the peaks of the rectified „halves” of the sine wave smaller than the amplitude of the input signal? In the oscilloscope, shift the input waveform upwards trying to make it coincide with the upper fragments of the input sine wave. Are the rectified „halves” indeed shifted fragments of peaks of the input sine wave? Reverse the polarization of

the second channel of the oscilloscope by activating sign inversion mode, and then try to make the inverted output waveform coincide with the lower fragments of the input sine wave. Sketch the oscillogram of the input and output(the latter *without* shift or inversion) waveforms in the report. Since it may be difficult to draw the oscillogram manually, it is recommended to save it on disk, and later print it and attach to the report.

1×

→BMP

2. Without powering off the devices, disconnect one terminal of the red LED marked in Fig. 4.35 with a star and observe the work of a half-wave rectifier. Connect it back and disconnect one terminal of the other red LED, *not* marked with a star in Fig. 4.35. Observe the work of the second half-wave rectifier (that uses other halves of the sine wave than the previous one).

3. For a moment, change the frequency of sinusoidal signal to 1 Hz. Observe how opposite LEDs conduct in pairs the positive and negative halves of the input sine wave. Simultaneously observe (in the oscilloscope) how input signal is „pulled up” by both large enough positive and large enough negative output voltages. Repeat these observations for both half-wave rectifiers and then set the generator frequency again to 100 Hz. Write down the conclusions.

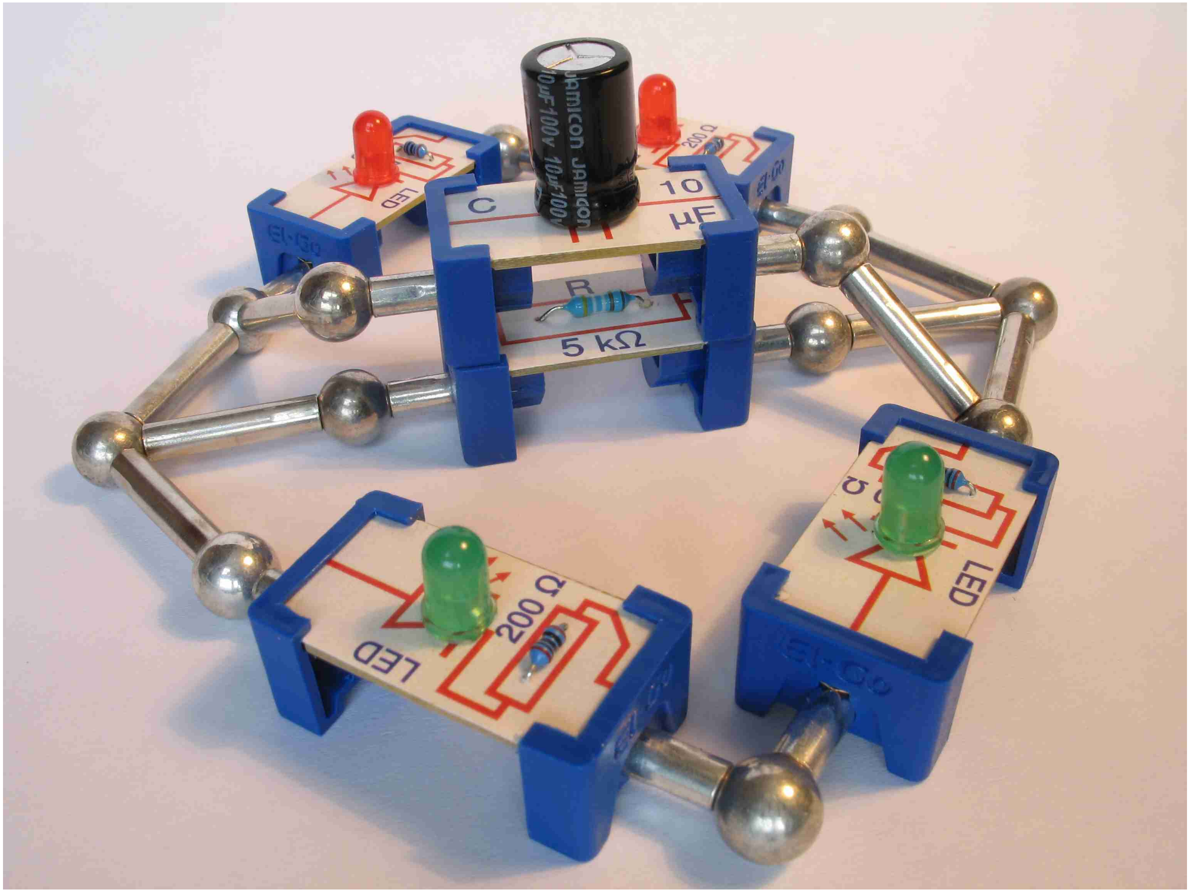


Figure 4.36: The way of connecting a smoothing capacitor (C = 10 μF) to the Graetz bridge

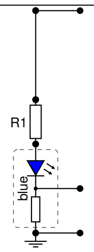
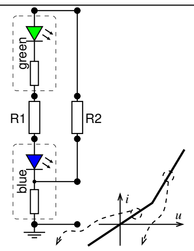
4. Connect capacitor C = 1 μF marked in Fig. 4.35 with a dash line, in parallel with resistor R. This capacitor is to be placed „on top” of resistor R, as shown in Fig. 4.36. Observe

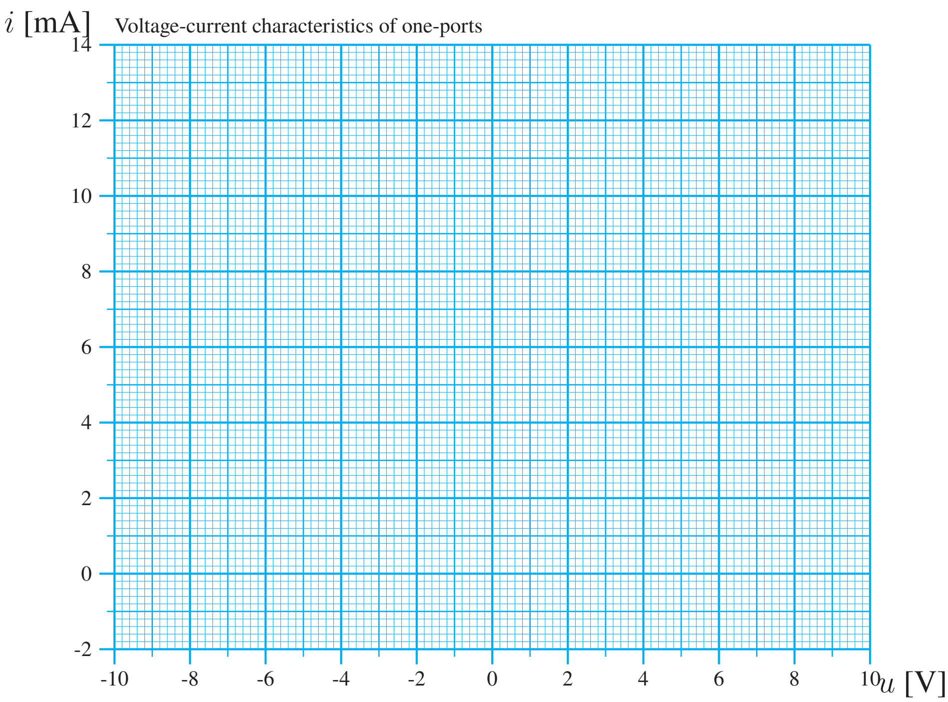
in the oscilloscope how connecting a capacitor reduced the ripples in the output voltage (smoothed the voltage). Check how so smoothed waveform looks for a half-wave rectifier. Repeat similar observations for $C = 10\ \mu\mathrm{F}$ and $C = 100\ \mu\mathrm{F}$. Once again write down the conclusions.

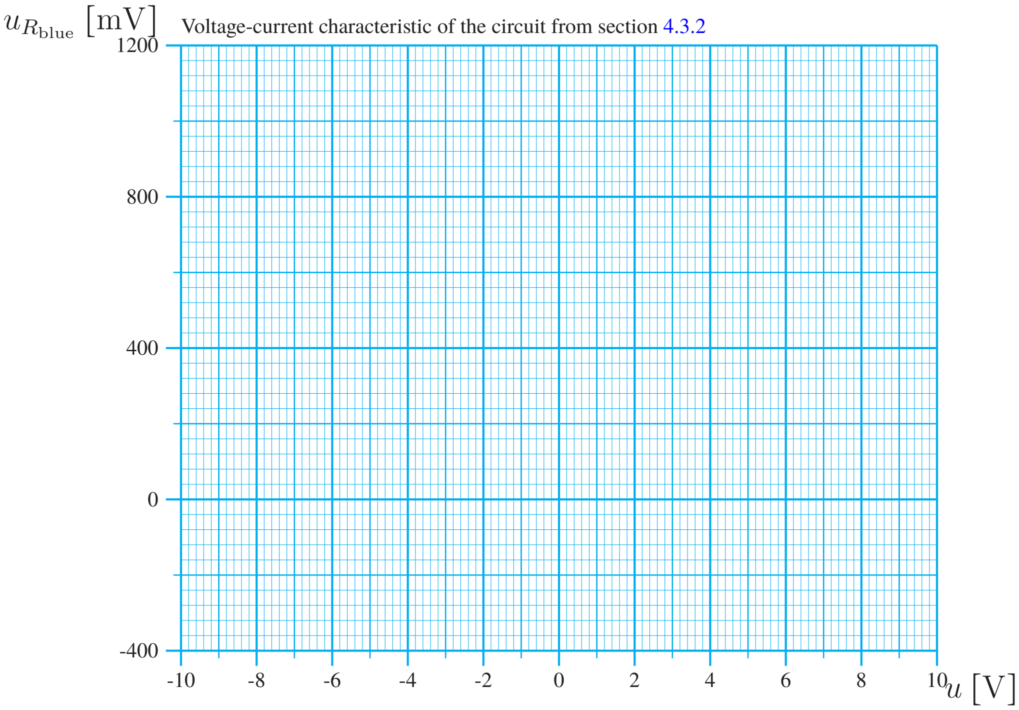
Stand:	*Name and SURNAME*	*Grade*
Ex. no.: ***4***		
Topic: ***Nonlinear circuits and small-signal analysis***		*Instructor's signature*
		Date

Composing voltage-current characteristics

Table 4.1: Parameters of characteristics of nonlinear one-ports

			
U_p [V]		———	
R_d [Ω]		———	———
$\frac{\Delta u_{R_{\text{blue}}}}{\Delta u}$ [V/V]	———		



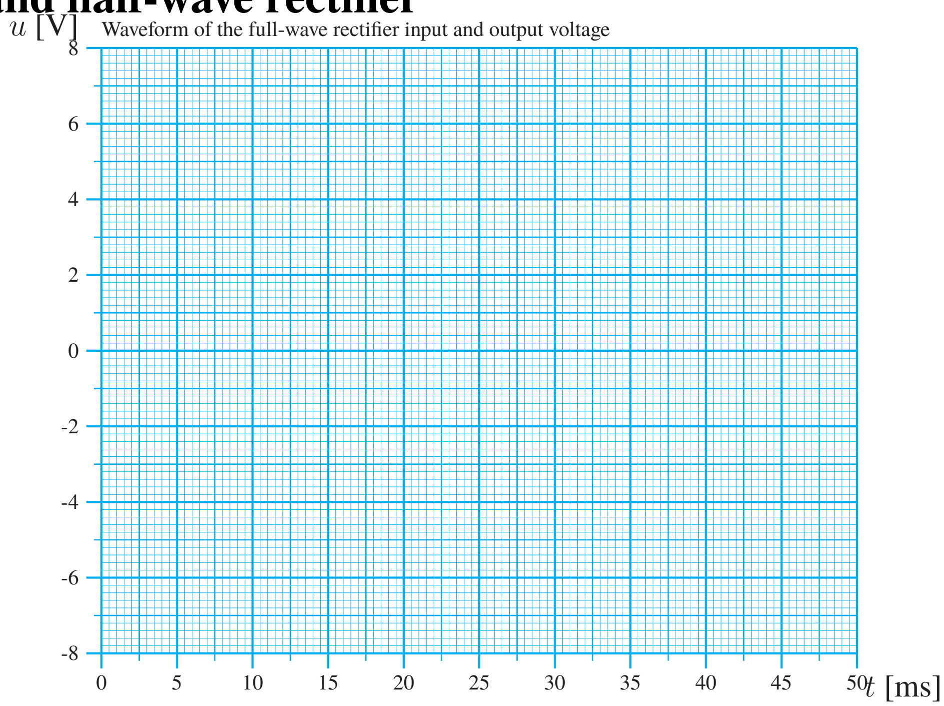


Small-signal analysis

Table 4.2: Analysis of nonlinear one-ports at various operating points

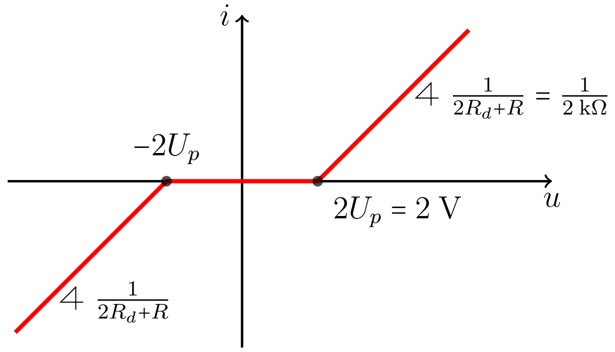
	E_0 [V]	$U_{R_{\text{blue}},\max}$ [V]	K_u [V/V]	Measurement conditions
1	0			supply voltage = 0
2				diodes barely emit light
3	$E_{0,\text{crit.}}$ =			max. 2-nd harm.
4				$E_{0,\text{crit.}}$ + 1 V
5				$E_{0,\text{crit.}}$ + 2 V

Full- and half-wave rectifier



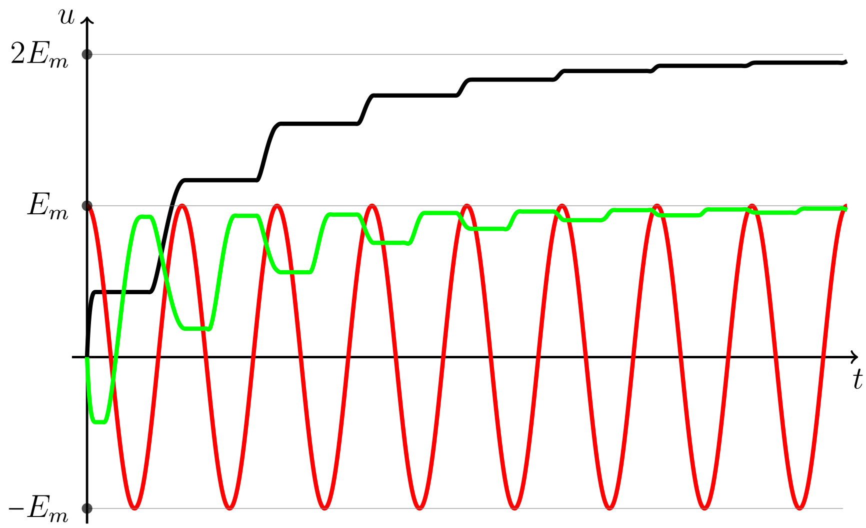
Solutions to self-study problems

1.



2. $$i(t) = (1 + 0,005\cos\omega t)^3 \text{ mA} =$$
$$= \left(1 + \frac{75}{2\cdot 10^6}\right) \text{mA} + 15\left(1 + \frac{25}{4\cdot 10^6}\right)\cos\omega t\ \mu\text{A} + \frac{75}{2}\cos 2\omega t \text{ nA} + \frac{125}{4}\cos 3\omega t \text{ pA}$$
$$\approx 1 \text{ mA} + 15\cos\omega t\ \mu\text{A}.$$

3. In the following figure, the waveform of the electric potential for the node A of the first circuit is drawn with black, the electromotive force is marked with red line and the voltage across capacitor that is connected directly to the voltage source is marked with green. The graph was constructed for $\tau = RC = T/16 = 2\pi/(16\omega)$, where $e(t) = E_m\cos\omega t$.



For the second of the considered circuits, the signals look analogously and potential of node A tends to $4E_m$.

Chapter 5

Filtering and transient states

5.1 Introduction

5.1.1 Filters

In the whole subsection 5.1.1 we will be dealing with steady states only. Consider a linear circuit depicted in Fig. 5.1. A particular case of such a circuit is presented in Fig. 5.2 and is analyzed in this subsection.

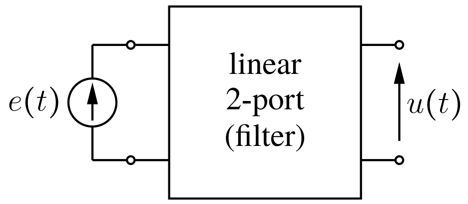


Figure 5.1: A filter supplied with an ideal voltage source

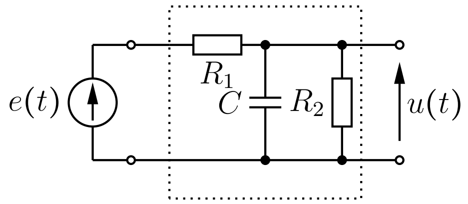


Figure 5.2: An example of a filter

The source's EMF $e(t)$ in the above circuits may be treated as an input signal, and the voltage $u(t)$ — as an output signal. If $e(t) = E = \mathrm{const}$, then the circuits become DC circuits. The linearity of the circuit of Fig. 5.1 implies that the voltage $u(t)$ is proportional to EMF E, i.e.

$$u(t) = U = H_0 E,$$

where constant H_0 depends on the structure of the one-port only. For the circuit presented in Fig. 5.2, the DC analysis reduces to the analysis of the circuit of Fig. 5.3. Therefore, using

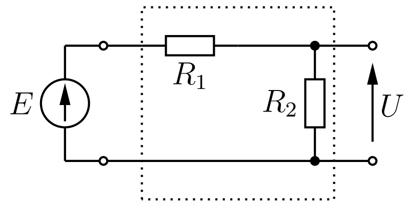


Figure 5.3: Circuit of Fig. 5.2 with constant EMF $e(t) = E$

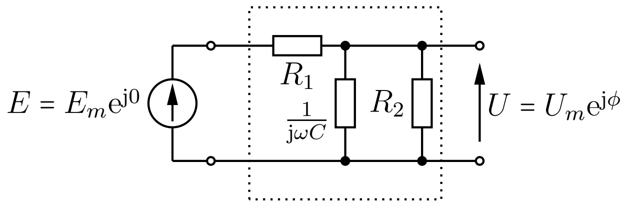


Figure 5.4: Circuit of Fig. 5.2 with alternating EMF $e(t) = E_m \cos \omega t$

the voltage divider formula we get

$$u(t) = U = \frac{R_2}{R_1 + R_2} E.$$

Accordingly, the constant H_0 equals

$$H_0 = \frac{R_2}{R_1 + R_2}.$$

If a harmonic voltage

$$e(t) = E_m \cos \omega t \tag{5.1}$$

is applied to the input of the circuit of Fig. 5.1, then the circuit becomes an alternating current circuit. The output voltage waveform

$$u(t) = U_m \cos(\omega t + \phi)$$

may be determined with the phasor method. Thus we get a phasor

$$U = U_m \mathrm{e}^{\mathrm{j}\phi},$$

which can be represented as a product of phasor $E = E_m \mathrm{e}^{\mathrm{j}0}$ and a complex number H_ω

$$U = H_\omega E. \tag{5.2}$$

The circuit of Fig. 5.2, with EMF of form (5.1), can be represented in phasor domain as in Fig. 5.4. Using the voltage divider formula in its admittance form we get

$$U = \frac{\frac{1}{R_1}}{\mathrm{j}\omega C + \frac{1}{R_1} + \frac{1}{R_2}} E = \frac{R_2}{R_1 + R_2 + \mathrm{j}\omega R_1 R_2 C} E,$$

and therefore

$$H_\omega = \frac{R_2}{R_1 + R_2 + \mathrm{j}\omega R_1 R_2 C}.$$

Notice that the obtained complex number H_ω depends on pulsation ω, which is why we put ω in the subscript of symbol H_ω. In particular, the amplitude of voltage $u(t)$ can be much smaller for certain pulsations than for others, despite the fact that the amplitude E_m does not change. In this way the two-port affects the input voltages and that is why the two-port of Fig. 5.1 is called a *filter*.

Making use of the linearity of the circuit depicted in Fig. 5.1 and the superposition principle, we may deduce that for EMF $e(t)$ of the form

$$e(t) = E_0 + E_{1m} \cos(\omega t + \varphi_{e_1}),$$

the output voltage of the circuit is

$$u(t) = \underbrace{H_0 E_0}_{U_0} + \underbrace{|H_\omega| E_{1m}}_{U_{1m}} \cos(\omega t + \underbrace{\varphi_{e_1} + \arg H_\omega}_{\varphi_{u_1}}).$$

For a similar reason, if we have a T-periodic voltage $e(t)$ at the input,

$$e(t) = E_0 + \sum_{k=1}^{+\infty} E_{km} \cos(k\omega t + \varphi_{e_k}), \tag{5.3}$$

then the voltage $u(t)$ will have the following form

$$u(t) = \underbrace{H_0 E_0}_{U_0} + \sum_{k=1}^{+\infty} \underbrace{|H_{k\omega}| E_{km}}_{U_{km}} \cos(k\omega t + \underbrace{\varphi_{e_k} + \arg H_{k\omega}}_{\varphi_{u_k}}), \quad \omega = \frac{2\pi}{T}.$$

To conclude the above considerations: a complex function H_ω of variable ω, determines the output voltage $u(t)$ for a given periodic input $e(t)$. Therefore it is called a *frequency response*. The magnitude of a frequency response $|H_\omega|$ is called an *amplitude frequency response* of a two-port, and the argument of the frequency response $\arg H_\omega$ is called a *phase frequency response*.

The fact that an amplitude response is not constant results in the fact that various input signal harmonics are attenuated to a various degree. For a given filter, i.e., for a given frequency response, pulsations are commonly divided into three bands. A *pass-band* consists of pulsations for which the amplitude response is relatively big, e.g. larger than 90% of its maximum value. A *stop-band* consists of pulsations for which the amplitude response is relatively small, e.g. not larger than 10% of its maximum value. The remaining pulsations form a *transition band*. Depending on locations of the bands, various filter classes are defined (see Fig. 5.5):

LP *low-pass filters*, for which the pass-band is an interval containing zero pulsation,

HP *high-pass filters*, for which the pass-band is made up of all pulsations bigger than some non-zero pulsation,

BP *band-pass filters*, for which the pass-band is an interval not containing zero pulsation,

BS *band-stop filters*, for which the stop-band is an interval not containing zero pulsation.

The graph of the amplitude response of the filter of Fig. 5.2 is shown in Fig. 5.6. From this graph we may conclude that in this case we are dealing with a low-pass filter.

5.1.1.1 The operating state of a two-port

The frequency response of a linear two-port is defined as the ratio of the output and the input voltage phasors provided that the output terminals are open-circuited. In practice, we usually deal with a situation in which a filter is located between a source with a non-zero internal impedance Z_w and some load, which is represented with a certain impedance Z_0. Such a situation, called an operating state of a two-port, is shown in Fig. 5.7.

In such cases we are interested in how the output voltage $u(t)$ depends on EMF $e(t)$. The relation between $u(t)$ and $e(t)$ is no longer described by the frequency response of the filter from Fig. 5.7. However, if the impedances Z_w and Z_0 are fixed, then we can treat the part of a circuit of Fig. 5.7 as a new filter, as in Fig. 5.8, whose frequency response determines the desired relation.

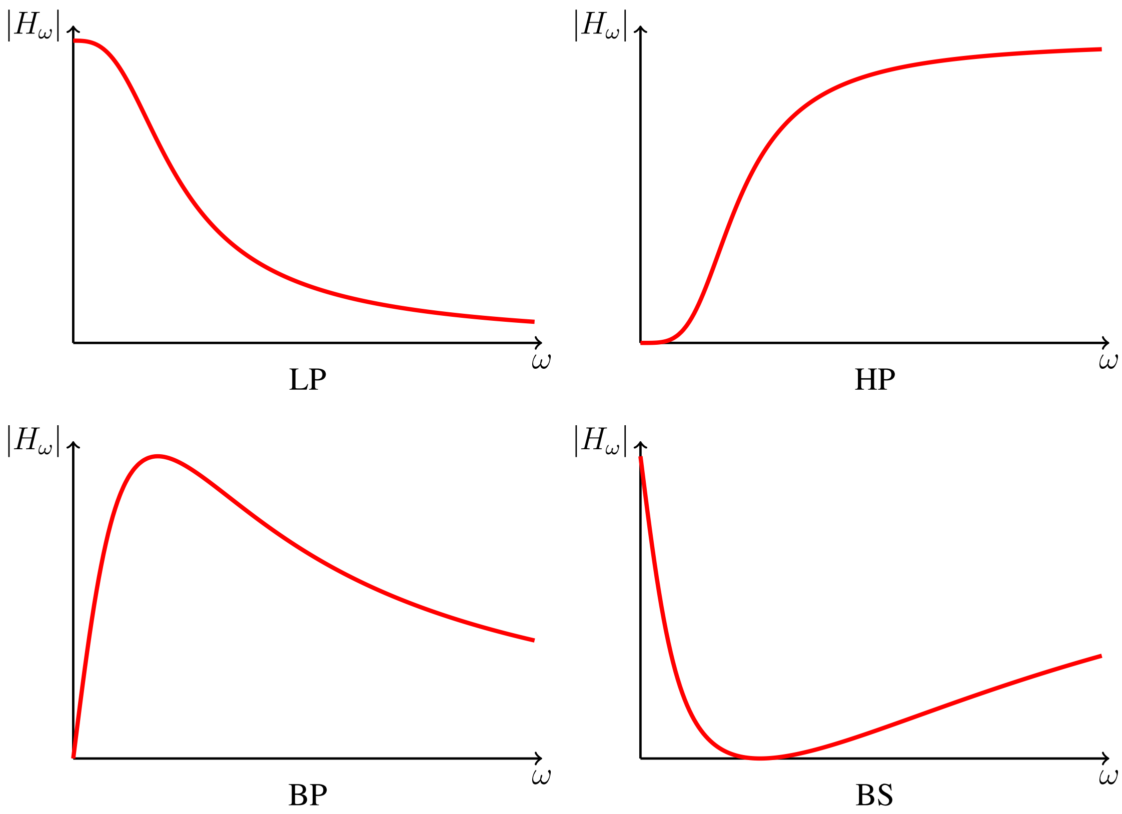


Figure 5.5: Examples of amplitude responses: low-pass (LP) filter, high-pass (HP) filter, band-pass (BP) filter and band-stop (BS) filter

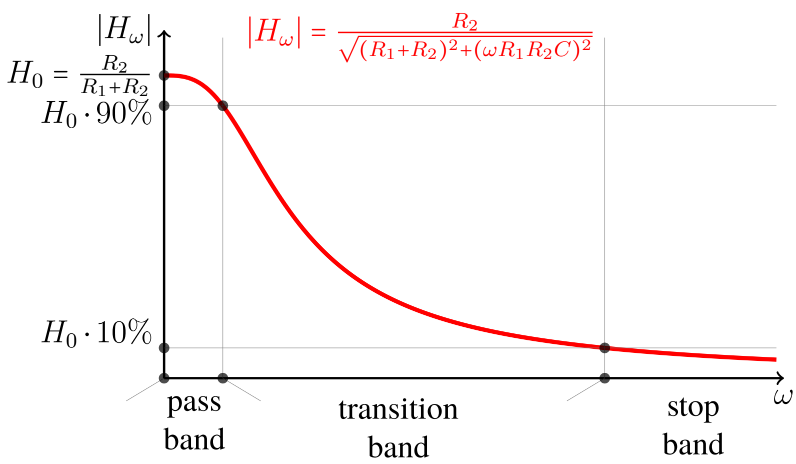


Figure 5.6: The amplitude response of the filter of Fig. 5.2

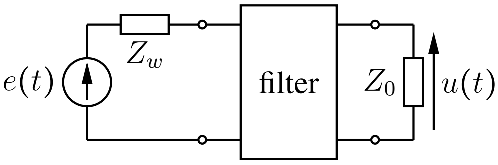


Figure 5.7: A loaded filter supplied with a real source

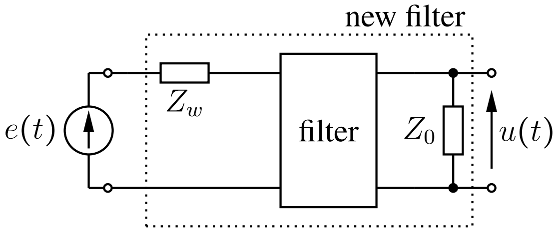


Figure 5.8: Reduction of the circuit of Fig. 5.7 to the situation of Fig. 5.1

5.1.2 Transient states

One of the basic principles that govern electric circuits is the following law:

Commutation law. The voltages across capacitors and the currents through inductors are continuous functions of time. Therefore energies stored in these devices are continuous as well[1].

The commutation law helps to determine voltages and currents in circuits, in which *transient states* appear. A transient state is any solution to a circuit which converges to a periodic solution but is not periodic itself[2].

In this lab exercise we will deal with transient states in circuits which satisfy the following assumption:

Assumption 1. There exist time instants:

$$t_0 < t_1 < t_2 < \cdots < t_N$$

such that for each interval of the form

$$(t_k, t_{k+1}), \quad k = 0, 1 \ldots, N - 1$$

the circuit comprises of resistors, capacitors, inductors and constant sources only.

For a circuit satisfying condition (1) there seems to be a DC solution for each time interval (t_k, t_{k+1}). In fact it is rarely the case because such solutions would violate the commutation law when considered altogether.

5.1.2.1 First order circuits

Let us study the circuit presented in Fig. 5.9 for time instants $t \in (t_0, t_1)$, $t_0 < t_1$. Assume that voltage u_C across the capacitor equals U_0 at t_0.

Current $i(t)$ through resistor R equals

$$i(t) = \frac{E - u_C(t)}{R} = Cu_C'(t),$$

[1]Let us recall that the energy stored in a capacitor C, across which the voltage is u_C, equals $w_C = \frac{1}{2}u_C^2$; and the energy stored in an inductor L, through which the current is i_L equals $w_L = \frac{1}{2}i_L^2$.

[2]Note that steady states in DC circuits and steady states in AC circuits are just particular cases of periodic solutions.

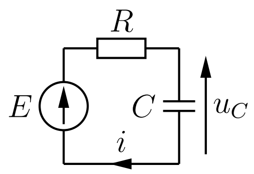


Figure 5.9

where $'$ denotes a derivative with respect to to time. We thus get an ordinary differential equation for function u_C:

$$\underbrace{RC}_{\tau}\, u_C' + u_C = E, \tag{5.4}$$

where τ is called a time constant.

The solution to (5.4) may be written as

$$u_C(t) = A\mathrm{e}^{-\frac{t}{\tau}} + E. \tag{5.5}$$

Commutation law requires $u_C(t_0) = U_0$ and thus

$$A = (U_0 - E)\mathrm{e}^{\frac{t_0}{\tau}}.$$

Eventually, the solution has the following form

$$u_C(t) = E - (E - U_0)\mathrm{e}^{-\frac{t-t_0}{\tau}}, \quad t \geq t_0. \tag{5.6}$$

Function $u_C(t)$ is plotted in Fig. 5.10.

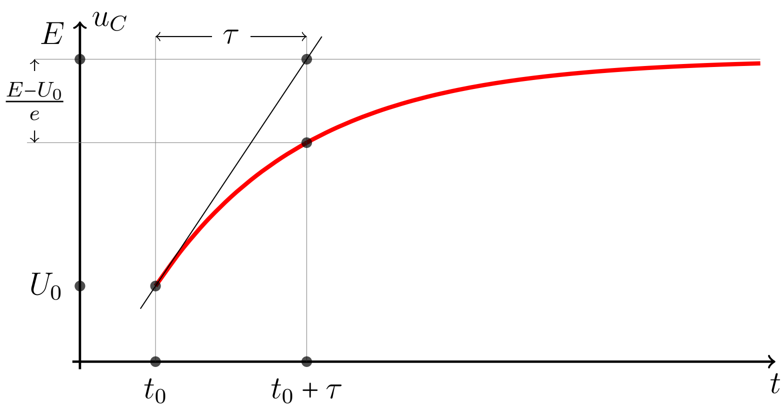


Figure 5.10: $u_C(t) = E - (E - U_0)\mathrm{e}^{-\frac{t-t_0}{\tau}}$, $t \geq t_0$

Note that voltage E in formula (5.6) is the asymptotic value of function u_C ($u_C(t) \xrightarrow{t\to+\infty} E$)[3]. The speed of convergence of $u_C(t)$ to the asymptotic value depends on time constant τ. According to (5.6), the difference

$$E - u_C = (E - U_0)\mathrm{e}^{-\frac{t-t_0}{\tau}} \tag{5.7}$$

[3] Voltage E is the voltage we would obtain in the DC analysis of the circuit of Fig. 5.9.

decreases e-fold[4] during each time interval of length τ. We can also characterize the time constant in another way. Namely, τ constitutes the length of the interval whose one edge is $(t_\star, E)$, and the other edge is at the intersection of the asymptote (given with formula $u = E$) and the line tangent at $t_\star$ to the graph of function u_C. In the above statement, the choice of time instant $t_\star$ is arbitrary. In Fig. 5.10 the corresponding construction for $t_\star = t_0$ is presented.

First order circuits excited with square wave. Let us consider the circuit presented in Fig. 5.11. The circuit satisfies Assumption 1). Assume that switch K is being closed at time instants

$$t_0 = 0,\ t_2 = T,\ t_4 = 2T,\ \dots,$$

and is being opened again at times

$$t_1 = \frac{T}{2},\ t_3 = 3\frac{T}{2},\ t_5 = 5\frac{T}{2},\ \dots$$

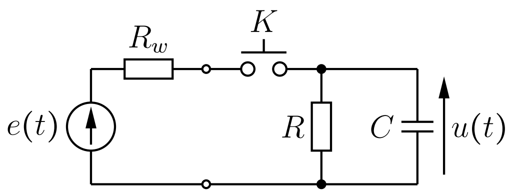


Figure 5.11

For each time interval of the form (t_k, t_{k+1}) the capacitor is attached to a linear one-port that has its Thévenin equivalent (see subsection 2.1.1) of EMF E_k and internal resistance R_k. For the considered circuit we have

$$\begin{aligned} &E_k = E_T = E\frac{R}{R+R_w}, && R_k = R \parallel R_w, && \text{if } k \text{ is even},\\ &E_k = 0, && R_k = R, && \text{if } k \text{ is odd}. \end{aligned}$$

Using the above parameters and the results of the analysis of the circuit presented in Fig. 5.9, we may write voltage u depicted in Fig. 5.11 as

$$u(t) = E_k - (E_k - U_k)\mathrm{e}^{-\frac{t-t_k}{\tau_k}}, \qquad \text{for } t \in (t_k, t_{k+1}),$$

where $\tau_k = R_k C$, and U_k is the voltage across the capacitor at t_k. According to the commutation law, voltage u at t_1 equals

$$U_1 = u(t_1) = E_0 - (E_0 - U_0)\mathrm{e}^{-\frac{t_1-t_0}{\tau_0}},$$

at t_2 it is equal to

$$U_2 = u(t_2) = E_1 - (E_1 - U_1)\mathrm{e}^{-\frac{t_2-t_1}{\tau_1}},$$

[4] $e = \lim_{n\to\infty}\left(1 + \frac{1}{n}\right)^n$ ☺

and so on.

Assume that

$$(R \parallel R_w)C \ll T, \quad RC \ll T.$$

We then have

$$U_1 \approx E_T, \quad U_2 \approx 0, \quad U_3 \approx E_T, \quad \dots$$

and assuming $u(0) = 0$, we thus get the voltage u as shown in Fig. 5.12.

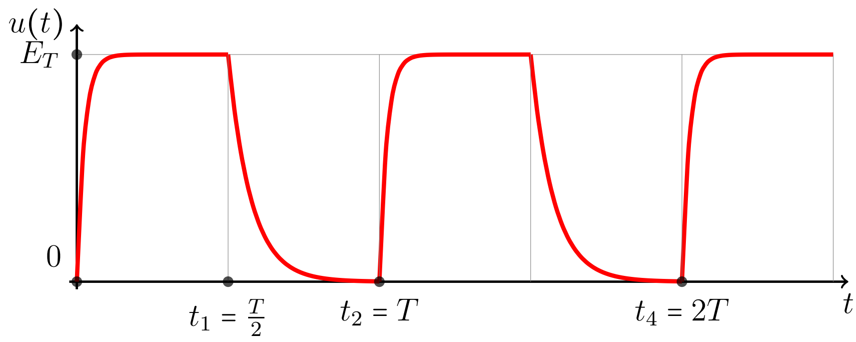


Figure 5.12

5.1.2.2 Second order circuits

Consider the circuit presented in Fig. 5.13 for which

$$u_C(t_0) = U_0, \quad i_L(t_0) = I_0. \tag{5.8}$$

It is a second order circuit because, as we show below, it is described by a second order differential equation.

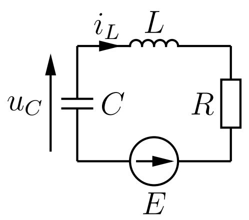


Figure 5.13: An RLC circuit (a second-order circuit)

According to Kirchhoff's voltage law

$$u_C - Li_L' - Ri_L = E. \tag{5.9}$$

Capacitor equation gives

$$i_L = -Cu_C' \tag{5.10}$$

and thus

$$i_L' = -Cu_C''. \tag{5.11}$$

Substituting (5.10) and (5.11) into (5.9), we get

$$(Q\tau)^2 u_C'' + \tau u_C' + u_C = E, \tag{5.12}$$

where

$$\tau = RC, \quad Q = \frac{\sqrt{L/C}}{R}. \tag{5.13}$$

Parameter $Q > 0$ is the quality factor of the series resonant circuit made of RLC elements. The qualitative nature of the solutions to equation (5.12) depends on location of the roots of its characteristic equation

$$(Q\tau)^2\omega^2 + \tau\omega + 1 = 0. \tag{5.14}$$

We distinguish three classes of solutions to (5.12) depending on these roots.

Overdamped case. If $Q < \frac{1}{2}$, then characteristic equation (5.14) has two different negative roots:

$$\omega_1 = \frac{-1 - \sqrt{1 - 4Q^2}}{2Q^2\tau}, \quad \omega_2 = \frac{-1 + \sqrt{1 - 4Q^2}}{2Q^2\tau}.$$

In this case, the general solutions to equation (5.12) takes the following form

$$u_C(t) = E + A\mathrm{e}^{\omega_1(t-t_0)} + B\mathrm{e}^{\omega_2(t-t_0)},$$

where A and B are constants determined by initial conditions (5.8).

Graphs of voltage u_C and current i_L for the overdamped case are shown in Fig. 5.14.

Critically damped case. If $Q = \frac{1}{2}$, then characteristic equation (5.14) has a double negative root

$$\omega_1 = \omega_2 = -\frac{2}{\tau}.$$

In this case, the general solutions to equation (5.12) takes the following form

$$u_C(t) = E + A\mathrm{e}^{-\frac{t-t_0}{\tau/2}} + B(t - t_0)\mathrm{e}^{-\frac{t-t_0}{\tau/2}},$$

where A and B are constants determined by initial conditions (5.8).

Graphs of voltage u_C and current i_L for the critically damped case are shown in Fig. 5.15.

Underdamped case. If $Q > \frac{1}{2}$, then characteristic equation (5.14) has two conjugated roots

$$\omega_1 = -\frac{\omega_r}{2Q} + \mathrm{j}\omega_s, \quad \omega_2 = -\frac{\omega_r}{2Q} - \mathrm{j}\omega_s,$$

where

$$\omega_s = \frac{\sqrt{1 - \frac{1}{4Q^2}}}{Q\tau} = \omega_r\sqrt{1 - \frac{1}{4Q^2}}, \quad \omega_r = \frac{1}{\sqrt{LC}}. \tag{5.15}$$

Pulsation ω_r is the resonant pulsation of the series resonant circuit made of RLC elements.

In this case, the general solutions to equation (5.12) takes the following form

$$u_C(t) = E + A\mathrm{e}^{-\omega_r(t-t_0)/2Q}\cos\big(\omega_s(t - t_0) + \varphi\big),$$

where A and φ are constants determined by initial conditions (5.8).

Graphs of voltage u_C and current i_L for the critically damped case are shown in Fig. 5.16.

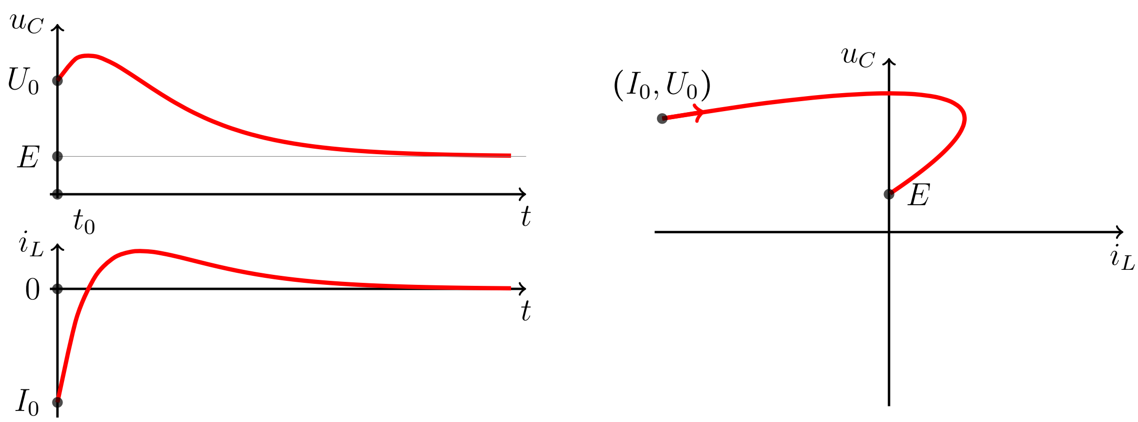


Figure 5.14: Overdamped response of an RLC circuit (the right subfigure shows the evolution of the signals in the so called phase space, i.e., it shows the set of points $(i_L(t), u_C(t))$ for $t \geq t_0$)

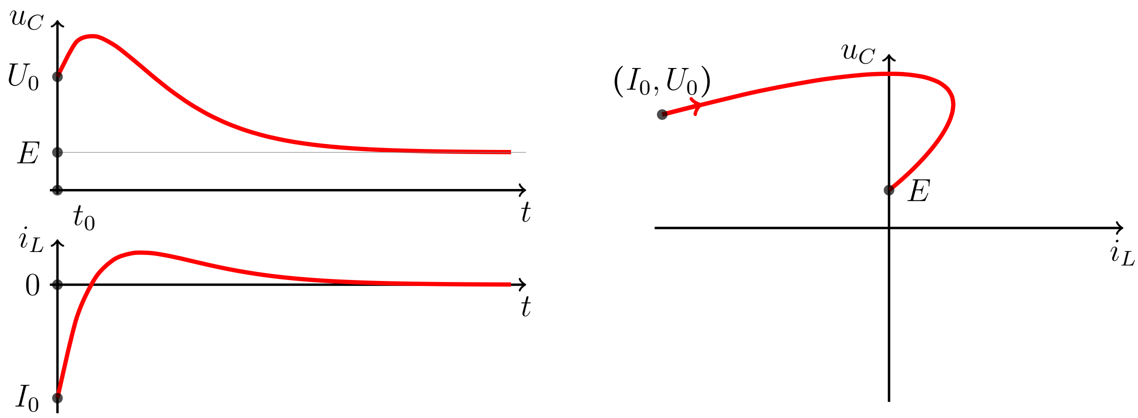


Figure 5.15: Critically damped response of an RLC circuit

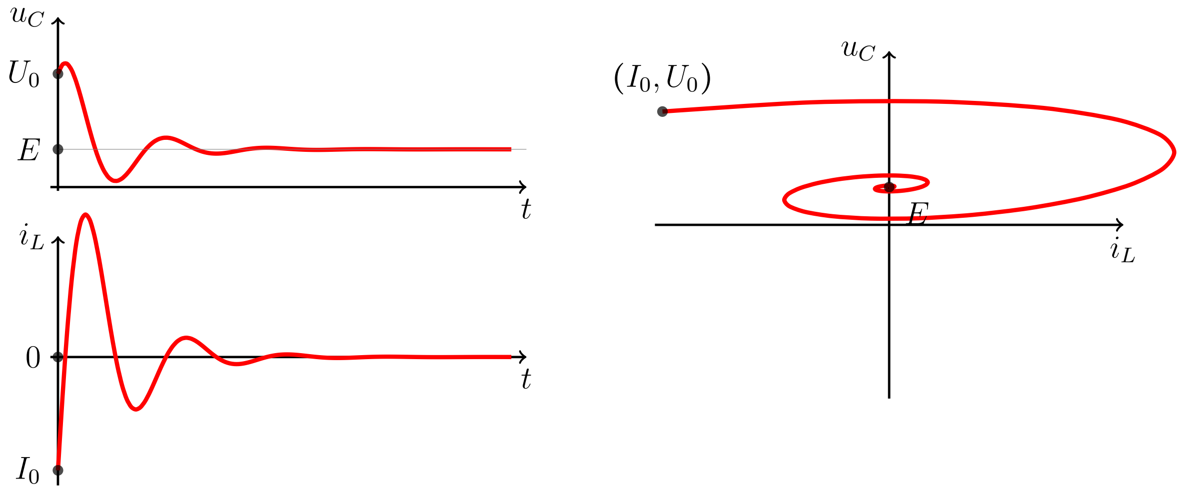


Figure 5.16: Underdamped response of an RLC circuit

Another example of a second order circuit. The circuit presented in Fig. 5.17 will also be studied during the lab exercise. This circuit is described by the following set of equations

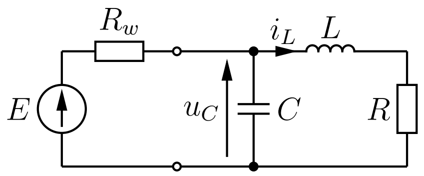


Figure 5.17

$$\frac{E - u_C}{R_w} - i_L = C u_C', \tag{5.16}$$
$$u_C - L i_L' - R i_L = 0. \tag{5.17}$$

Solving the first equation for current i_L and then substituting it to (5.17) we obtain

$$\underbrace{\frac{R_w LC}{R_w + R}}_{Q_\star^2 \tau_\star^2} u_C'' + \underbrace{\left(\frac{L}{R + R_w} + (R \parallel R_w)C\right)}_{\tau_\star} u_C' + u_C = \underbrace{E\frac{R}{R + R_w}}_{E_\star}. \tag{5.18}$$

Thus, by introducing new parameters $Q_\star$, $\tau_\star$ and $E_\star$, we have reduced the above equation to (5.12).

5.1.2.3 MOSFET as a switch

In the lab exercise we will be using a MOSFET (metal-oxide semiconductor field-effect transistor)[5] as a switch. The symbol of such a device is shown Fig. 5.18. The terminals of MOSFET are called: gate (G), source (S) and drain (D).

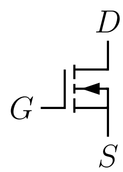


Figure 5.18: MOSFET

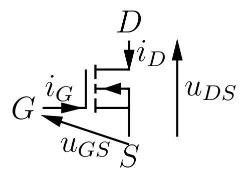


Figure 5.19

For the ranges of inter-terminal voltages used in the exercise[6], the MOSFET may be described with equations:

$$i_G = 0, \tag{5.19}$$
$$i_D = 0, \text{ if } u_{GS} < U_T, \tag{5.20}$$
$$u_{DS} = 0, \text{ if } u_{GS} \geq U_T, \tag{5.21}$$

where U_T is a positive voltage called a threshold voltage. In other words, if $u_{DS} < U_T$, then the MOSFET is equivalent to the 3-terminal device shown in Fig. 5.20 (terminals D and S are open-circuited), and if $u_{DS} \geq U_T$, then the MOSFET is equivalent to the device shown in Fig. 5.21 (terminals D and S are short-circuited).

[5]To be precise, we will be using an enhancement-mode n-channel MOSFET.

[6]In the lab exercise the MOSFET will operate in two modes: the cut-off mode and the triode mode.

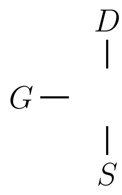


Figure 5.20: MOSFET for $u_{GS} < U_T$

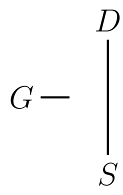


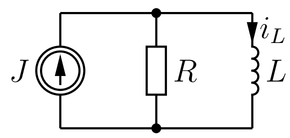
Figure 5.21: MOSFET for $u_{GS} \geq U_T$

Problems for self study

Solutions to the problems are given on page 123.

1. Construct a filter whose pass-band is an interval containing zero and whose stop-band is an interval containing a given pulsation ω_* (hint: recall a series resonant circuit).

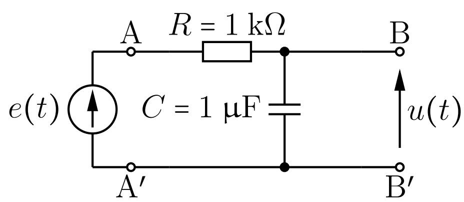
2. Find and sketch current i_L waveform for $t \geq t_0$ provided that $i_L(t_0) = I_0$. What should the value of I_0 be for the transient state not to appear?



3. After what multiple of the time constant τ will the difference between voltage u_C given by (5.6) and the electromotive force E decrease 10-fold? (and 100-fold or 1000-fold?)

5.2 Homework

Homework 5.1. Sketch the amplitude response of the following 2-port (with ports A-A′ and B-B′). Find voltage $u(t)$ provided that $e(t) = 1\ \mathrm{V} + 2\cos\omega t\ \mathrm{V}$, where $\omega = 1\ \mathrm{krad/s}$.



Homework 5.2. In Subsection 5.5 you will be analyzing the circuit presented in Fig. 5.13, where

$$C = 1\ \mu\mathrm{F},\ L = 10\ \mathrm{mH},\ R = 50\ \Omega.$$

Check that for the above parameters there will be an underdamped transient state in the circuit. Compute parameters τ, Q (see formulas (5.13)) and ω_s (see formula (5.15)).

5.3 Measurement of frequency response of filters

In the exercise, we will analyze the acoustic effects of filtration, so you are encouraged to bring to the lab a pair of headphones with a typical „mini-jack" plug.

Let us recall a fundamental fact that since impedances of an inductor and a capacitor depend on frequency, then properties of any circuit comprising these elements will, generally, depend on frequency. By measuring the amplitude of the output voltage (or its phase shift with regard to the input voltage) for many various frequencies, one could obtain the amplitude (or phase) frequency response of the analyzed circuit. However, such a procedure would be very time-consuming and inconvenient. As we already know from section 3.3, it is more convenient to measure such a response using a wobbuloscope. So, power off the multifunctional device and the oscilloscope, assemble the measurement setup as in Fig. 5.22 (for now, instead of building the complete two-port to be analyzed — connect only the two long magnetic rodes and five balls shown in the figure) and power on the crate. `Maya44 USB` card's output resistance is quite large (equal to $100\ \Omega$). Therefore, we will connect it to the setup using an amplifier (with output impedance $Z_w = 50\ \Omega$, the same as output resistance of the generator), operating as a two-port (with unitary gain).

Turn on the computer and run the virtual wobbuloscope program from the TOiS_Toyset. The wobbuloscope should be set to generate a signal with amplitude $1\ \mathrm{V}$ at output `OUT1` of the `Maya44 USB` card. The frequency range should be set to $200\ \mathrm{Hz} - 20\ \mathrm{kHz}$. Now you are ready to take the measurements.

5.3.1 Measurement of frequency response

We will now try to construct a low-pass filter, i.e. a two-port that transfers low frequency signals without change, and strongly attenuates high frequency signals. Let us notice that if in the setup of Fig. 5.7 the filter is not present, then for $Z_w = Z_0 = R$ the internal impedances of the signal source and the load constitute a voltage divider with ratio $\frac{1}{2}$, independent of frequency. Let us insert between the source and the load R a series inductor L, as shown in Fig. 5.23. Then, for low frequencies, for which the inductor's reactance is small (for „zero frequency" it is actually a short circuit), the effective voltage gain of the two-port (i.e. the ratio of the phasor $U_{\text{B-B}'}$ of the output voltage to the phasor E of EMF of the source) will remain practically the same. For high frequencies on the other hand, the reactance of the inductor may become theoretically arbitrarily large, which will prevent signals at such frequencies from being transferred to the two-port's output.

1. Without powering off the crate, assemble the analyzed two-port in the form of a simple filter shown in Fig. 5.23, comprising elements $R = 50\ \Omega$ and $L = 10\ \mathrm{mH}$. Set the wobbuloscope gain to $50\ \mathrm{mV/div}$. Measure the amplitude frequency response of the two-port (in a logarithmic scale), by choosing the single run mode (key `Run` in menu Run / Once or the relevant push-button from the TOiS_Toy's window toolbar). The measured response is to be saved using the push-button `store` in the panel `Memory` of the wobbuloscope window.

As you see, the obtained amplitude response is far from being perfect: even for the upper part of the acoustic band, our filter attenuates the signals merely 10-fold. How can we improve it? Aside hampering „transfer" through the two-port of signals at high frequencies thanks to

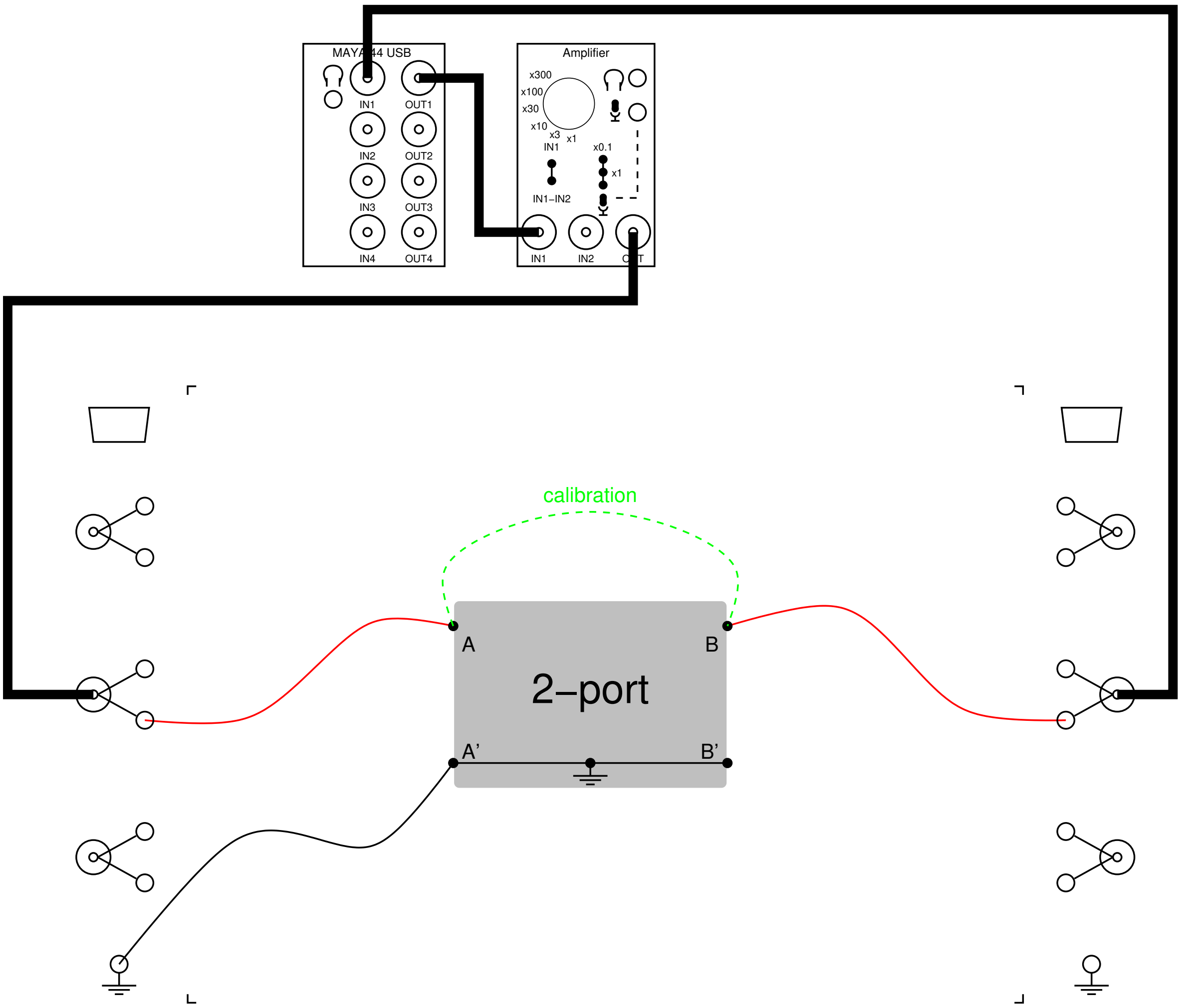


Figure 5.22: Setup for measuring frequency response with a wobbuloscope

the series inductor, will now try to „short-circuit" the signal to ground at high frequencies by using a *parallel capacitor*.

2. Without powering off the crate, connect a capacitor C = 1 μF to the analyzed two-port, as shown in Fig. 5.24. Measure the amplitude response of the two-port (in a logarithmic scale), by choosing again the single run mode (Run / Once). This time also save the measured response in the wobbuloscope memory.

The response of the new filter should be much better than the previous one. Now, what would be the effect of „short-circuiting" the signal to ground at a different location? For example, at the input instead of at the output of the analyzed two-port?

3. Do not power off the crate, move the capacitor C = 1 μF in the analyzed two-port from the output to the input as shown in Fig. 5.25. Measure the amplitude frequency response of the two-port (in a logarithmic scale) in the single run mode (Run / Once).

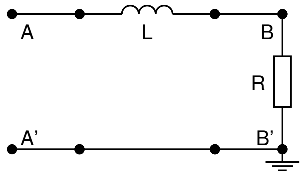


Figure 5.23: A simple filter with one inductor

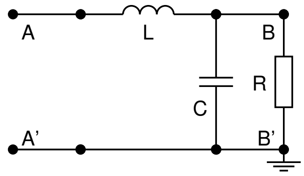


Figure 5.24: An improved LC filter

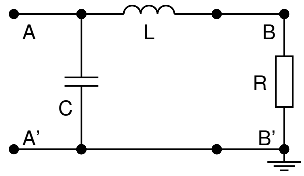


Figure 5.25: Another variant of the improved LC filter

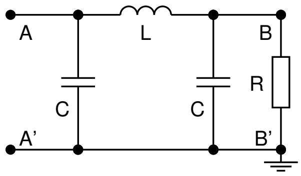


Figure 5.26: „Combination" of LC filters proposals of Fig. 5.24 and Fig. 5.25

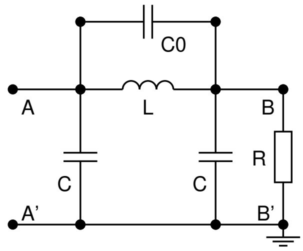


Figure 5.27: An LC filter with a frequency response zero

The measured response should be practically identical to the response for the two-port of Fig. 5.24[7], so there is no use in saving it. Have we then accomplished nothing? Quite the contrary! Let us combine the two ideas of Fig. 5.24 and Fig. 5.25 — two „short circuits" to ground for high frequencies (one at the input and one at the output of the filter — see Fig. 5.26) are better than one short circuit (either at the input, as in Fig. 5.25, or at the output, as in Fig. 5.24).

4. Without powering off the crate, connect another capacitor $C = 1\ \mu\mathrm{F}$ to the analyzed two-port, as shown in Fig. 5.26. Measure the amplitude frequency response of the two-port (in a logarithmic scale), by choosing the single run mode (key `Run`. Save the measured response in the wobbuloscope memory.

The obtained response should be better than that of the previous filters. Is there any way we can further improve it? What if for some frequency the two-port did not transfer the signal from the input to the output *at all*? Let us recall from the resonant circuits theory, that a *parallel* resonant one-port becomes an *open circuit* when in resonance. So, if we replaced the series inductor L with a parallel resonant one-port $L \parallel C_0$, then we will introduce a so-called frequency response zero, that is a frequency, for which the signal at output is zero.

5. Without powering off the crate, attach capacitor $C_0 = 100\ \mathrm{nF}$ to the analyzed two-port, as shown in Fig. 5.27. Measure the amplitude frequency response of the two-port (in

[7]We leave it to the curious Reader to verify that the effective voltage gain of the filter of Fig. 5.24 and of Fig. 5.25, driven by a harmonic voltage source with internal impedance $Z_w = R$, are *exactly* the same and equal

$$H_\omega^{\mathrm{ef}} = \frac{1}{2 - \omega^2 LC + \mathrm{j}\omega(RC + L/R)}.$$

a logarithmic scale), by choosing the single run mode (key `Run`). Save the measured response in the wobbuloscope memory.

1× →PNG

The four responses obtained in one graph are to be saved to disk, so that you can later add a description and labels, and then print them and attach to the report. For all four responses determine the lower and upper frequency of the transition band. Let us define this band (quite arbitrarily, for the purpose of this exercise) as the range of frequencies, for which the amplitude of the signal at the filter output is between 90% and 10% of the amplitude of the input signal (note that it is easy to identify transitions of the amplitude responses through the first and last horizontal grid lines in the wobbuloscope window). Values of these frequencies can be read from the wobbuloscope by indicating a relevant point on the response with a cursor and pressing the left mouse button. The results of the measurements are to be recorded in Tab. 5.1. Once the measurements are completed, close the wobbuloscope program and turn off the computer.

5.3.2 Analysis of acoustic effects caused by various filters

The pass bands of all of the filters analyzed in the previous section have been so chosen that if an acoustic signal from the computer (from a sound file or from internet radio) is delivered to their inputs, the differences in their functioning can be heard. Let us modify the measurement setup according to Fig. 5.28. Connect the output of the analyzed two-port to the second amplifier, with the gain also set to unity. This amplifier will be used as the headphones amplifier, so if needed you can gently increase its gain, **minding the possibility of losing your hearing if you do too much!**

For each of the five filters shown in Fig. 5.23–Fig. 5.27, listen to the signal from the radio, every dozen or more seconds „switching off” the filter by disconnecting it from ground and short-circuiting terminals A-B — this connection is shown in Fig. 5.22 with a green dash line. Performing such a switching allows to hear the difference in timber (tone color) in the original and the filtered signal. Describe in brief your acoustic impressions in the report.

Let us notice that similarly constructed filters are used in the crossovers of professional loudspeakers so that for example high frequency signals do not reach the woofer. Moreover, the type of coil that is used in the lab is in fact intended for crossovers used in audiophiliac loudspeakers (and that is why it is so large and beautiful).

5.4 Analysis of transient states in first order circuits

5.4.1 Analysis of transient states with the use of a mechanical key

Our adventure with transient states begins with examination of a simple first order circuit, in which after pressing (short-circuiting) the key, the capacitor C charges to the value equal to the supply voltage, and after releasing (open-circuiting) the key — exponentially discharges to zero through resistor R. The process of charging is very fast (it occurs with a very small time constant), because the internal resistance of the voltage supply (connected to the voltage source e board) is very small (it is a very good approximation of an ideal voltage source), and the resistance of the closed key is also negligibly small. The time of discharge through resistor R is by several orders of magnitude longer.

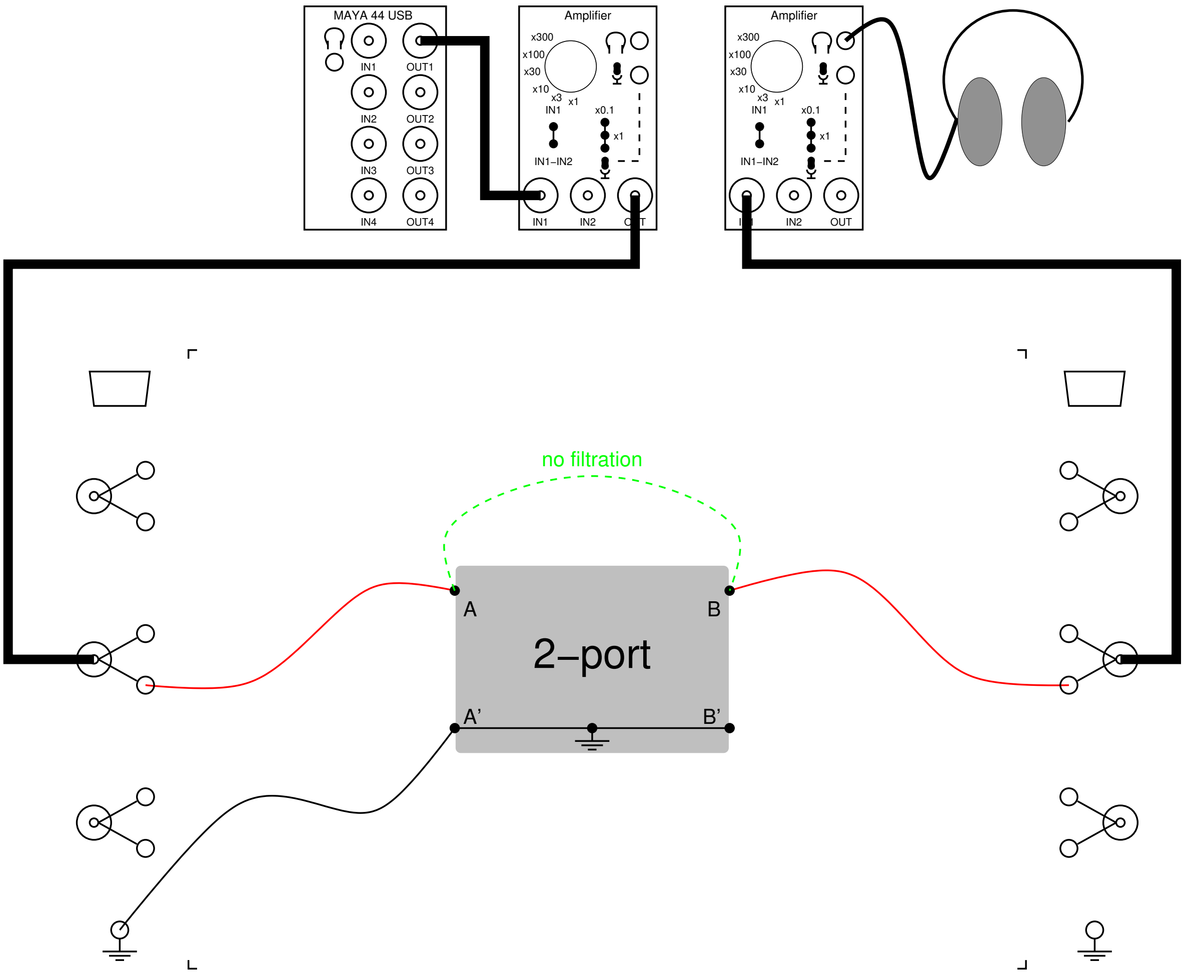


Figure 5.28: The setup for analysis of acoustic effects introduced by various filters

1. First, disconnect power supply from the circuit and set its `CH3` channel voltage to $5\ \mathrm{V}$ and its current limit to $0.1\ \mathrm{A}$. Next, keeping `CH3` channel disconnected (or switching off its output), assemble measurement setup shown in Fig. 5.29, using the following values of elements: $C = 10\ \mu\mathrm{F}$ and $R = 100\ \mathrm{k}\Omega$. Connect the voltage source e to the supply socket in the tray by means of which it is connected with two cables (with banana endings) to `CH3` channel of the power supply. Switch on `CH3` channel output and power on oscilloscope. Activate the first channel of the oscilloscope and set its gain to $1\ \mathrm{V/div}$, choose DC coupling, automatic triggering, and set time base to $0.5\ \mathrm{ms/div}$. Shift the horizontal line displayed in the oscilloscope toward the bottom of the screen (e.g. two divs from the bottom).

2. Press and hold the key to charge the capacitor to $5\ \mathrm{V}$, and then release the key and observe in the screen the process of discharging the capacitor: migration of the horizontal line toward zero. Repeat the observation several times. Every time when the line reaches about one div from „zero", stepwise increase the gain of the oscilloscope, up to its maximal value[8].

[8]For large gains, the line in the oscilloscope starts to appear „thick", which is the result of noise and other

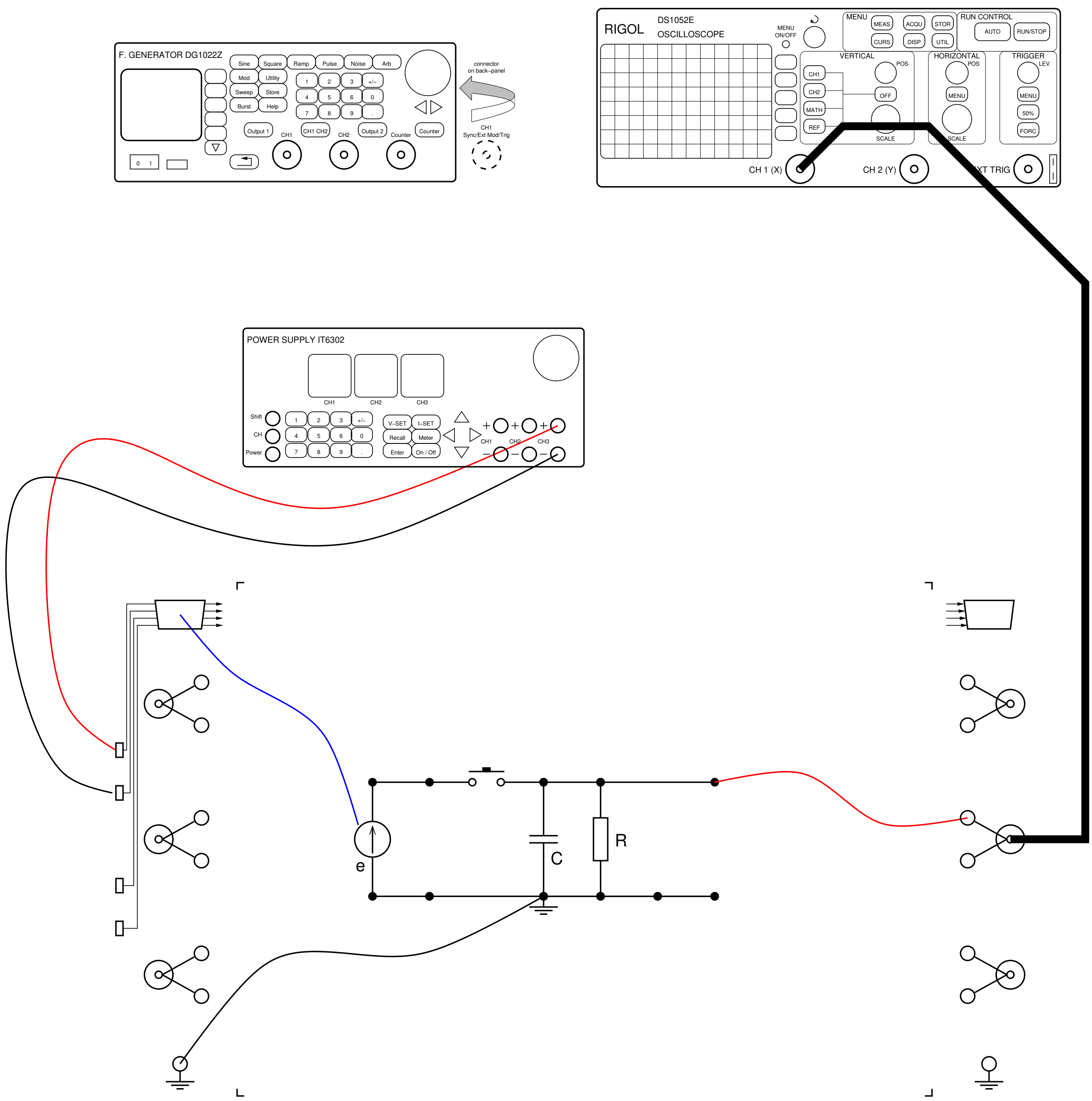


Figure 5.29: Measurement setup with a mechanical key, for observation of transient states in first order circuits

The processes which we observe are a bit too fast to measure the time constant, so let us try to increase it by an order of magnitude.

3. Without turning off the power, replace resistor R with a $1\ \mathrm{M}\Omega$ one, press and hold the key and again observe the process (now much slower!) of discharging the capacitor. Using a watch with a second hand[9] measure the time constant of the circuit as the time

disturbances, which are inevitable in every electronic device, and for larger gains become apparent.

[9] In case you haven't one, you can turn on the computer and click the current time shown in the right bottom corner of the screen.

of the e-fold[10] of the initial voltage. Repeat the measurement several times to make sure the result is repeatable. Compare the measured time constant τ' with the theoretical value $\tau = RC$.

The discrepancy between the theory and the measurement is too large to accept it and proceed. What is the cause of it?

4. Without turning off the power, remove resistor R from the measurement setup and repeat the experiment, observing the process of discharging the capacitor. Which way is the capacitor discharged this time? Through the input resistance of the oscilloscope $R_{\text{osc.}} = 1\ \text{M}\Omega$! Its value is imprinted on the front panel close to the input socket. Measure the time constant once again, as the time of the e-fold decline of voltage and compare the measured value τ'' with the theoretical value $\tau = R_{\text{osc.}}C$.

It is worth to remember the extent to which the measurement instrument can influence the result of measurement. But is it solely the measurement instrument that has an influence? What about the experimenter?

5. In the setup with resistor R removed, once again press and release the key. Observe slow migration of the horizontal line in the oscilloscope toward zero. After a short while, touch the terminals of the capacitor with two fingers of one hand. The process of discharging the capacitor should now become much faster[11].

We have indeed been observing the process of discharging the capacitor, but is it in fact occurring as an exponential waveform? To obtain an ideal exponential curve in the screen, we have to speed up the transient processes in our circuit, which we can do by decreasing the time constant by several orders of magnitude.

6. Without turning off the power, connect resistor $R = 1\ \text{k}\Omega$ and replace the capacitor with $C = 1\ \mu\text{F}$. Set the oscilloscope to the normal trigger mode (the horizontal line in the screen will disappear), and the trigger source to *descending* slope of the signal from the first channel. Press and hold the key, and after a while release it. In the oscilloscope you should observe a single curve exponentially decaying to zero. If you do not see anything, try to set a relevant trigger level.

The proposed method is somewhat inconvenient. Let us try to automate it.

5.4.2 Analysis of transient states with the use of an electronic key

In this section, the mechanical key will be replaced with an electronic key — with an MOSFET operating as a switch controlled with the generator. To make a „good" switch of the transistor, both in the short-circuit and open-circuit mode, one should deliver significant negative and positive voltages to its gate. For this purpose, we will use the amplifier with gain set to a very large value. In this case, the amplifier will act as a comparator, for all negative voltage at its input, it will produce the voltage of approximately -14 V (close to its negative supply voltage)

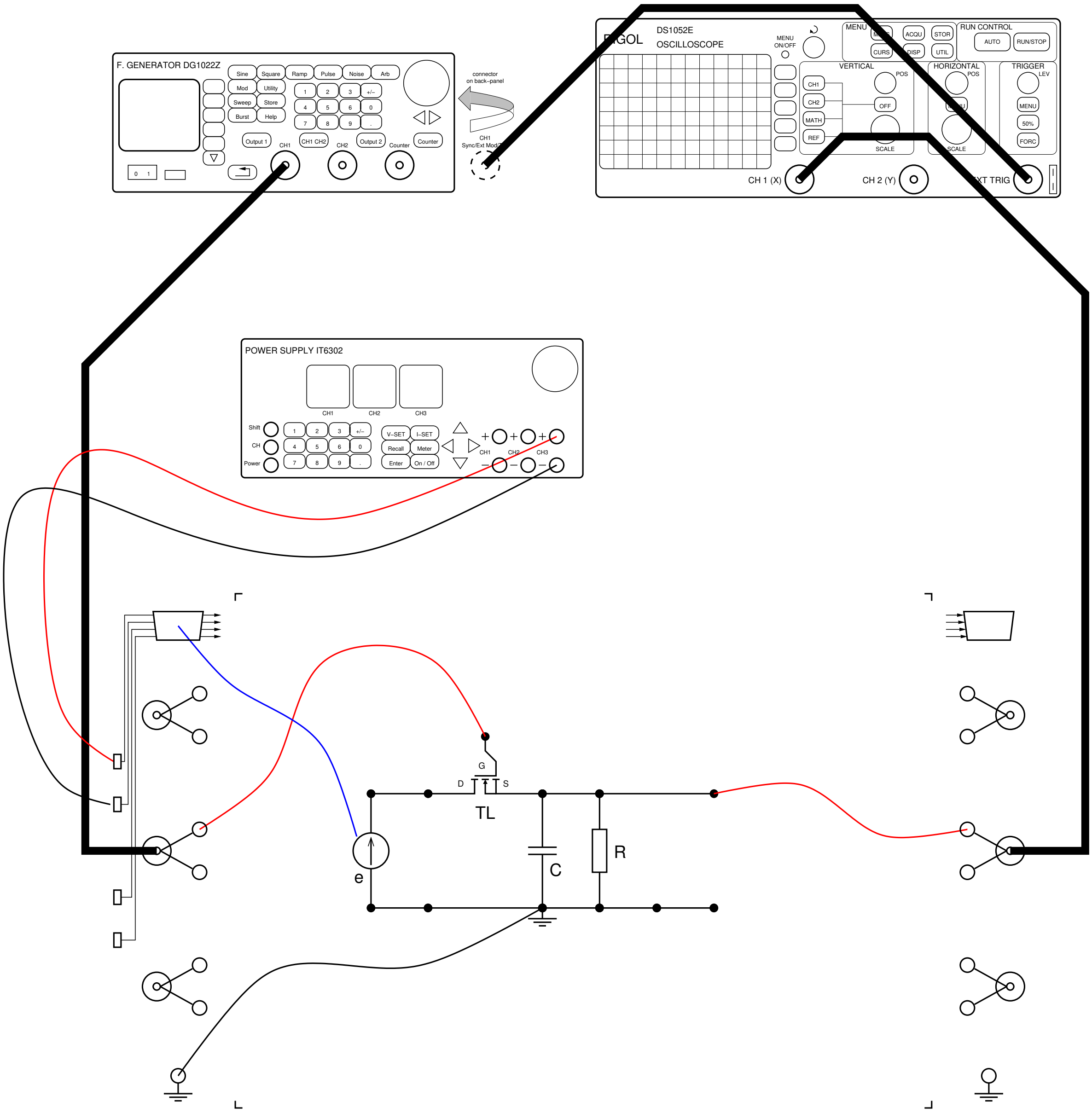


Figure 5.30: Measurement setup with an electronic key, for observation of transient states in a first order circuit

at the output, and for all positive input values — it will produce the voltage of approximately $+14$ V at the output.

1. Power off the crate and the power supply. Next, replace the mechanical key with a MOSFET (**it must be marked with** $\mathrm{T_L}$ **symbol, not** $\mathrm{T_R}$), connected in the way shown in Fig. 5.30. In the oscilloscope, set automatic trigger from external source (descending

[10] Since $e \approx 2{,}71828182845904523536028747135266249775724709369995\ldots$, the e-fold decline means decline to about 37% of the initial value. If the initial voltage is 5 V, the e-fold declined voltage is approximately equal to $1{,}8$ V.

[11] If it is not the case, try to press the capacitor with your fingers more firmly, or even moisten them a bit.

slope). Configure `CH1` channel of the function generator to generate pulse (square) wave with amplitude 20 Vpp, duty cycle 1%, 0 offset and freqency 100 Hz.

2. Observe in the oscilloscope, a waveform exponentially decaying to zero[12]. Draw the oscillogram of this waveform in a suitable grid area provided in the report template, pay attention to precision of the drawing. You can also save the oscillogram to disk, and later print it and attach to the report. Next, choose an arbitrary point close to the beginning of the obtained graph and draw a straight line tangent to the graph at this point[13]. The point where the tangent intersects with the asymptote of the exponential waveform determines an instant of time that is exactly one time constant ahead of the moment corresponding to the point of tangency. Compare the time constant measured this way with the theoretical value RC.

1× →BMP

3. Without turning off the power, replace resistor $R = 1\ \mathrm{k\Omega}$ with a $10\ \mathrm{k\Omega}$ potentiometer. Adjust its resistance and observe in the oscilloscope the shape of the exponential curve for various values of time constant. Briefly comment on the observations in the report.

5.5 Analysis of transient states in second order circuits

Analysis of transient states in first order circuits is quite simple, but the phenomena that occur in second order circuits are much more interesting. The second order circuits include reactive elements of both types: capacitors and inductors. There may occur in them a transfer of energy of the electric field (capacitor) to the magnetic field (inductor), and conversely. If the energy losses during this process are small, then we observe oscillations of voltages and currents in the circuit.

5.5.1 Analysis of transient states with the use of an electronic key

In the circuit shown in Fig. 5.31 (solid line only), after short-circuiting the transistor key, the capacitor is charged to the value of voltage equal to EMF of voltage source $e = E_0$, and the current through the inductor is equal to E_0/R. The initial conditions for both of the reactive elements are then non-zero in the moment of open-circuiting the transistor key. When the capacitor begins to discharge, it will increase the current through the inductor. Even when it's fully discharged, if only the energy losses in the circuit are sufficiently small, (i.e. the quality factor of the RLC one-port is large), then the continuity of the current through the inductor will cause the inductor to „pull" the voltage across capacitor onto the negative voltages side, so the inductor will „recharge" the capacitor in the opposite direction, at the same time decreasing its own current to zero. Then the situation will repeat with inverted signs of the waveforms

[12] You may speed up the time base for a while, for example to $1\ \mathrm{ms/div}$, to observe a complete cycle of discharging *and charging* the capacitor.

[13] An interesting way of drawing a tangent with everyday object is to use a mirror (or a well-polished knife blade) and a triangular ruler (a set square). First, place the mirror at a given point on the curve, more or less „perpendicular" to that curve, and then rotate it around the vertical axis that crosses this point, so that two fragments of the curve: the one seen directly and the mirrored one, transform one into the other „smoothly", i.e. without a "sawtooth". Keeping the mirror in the so established orientation, attach a set square to it, to determine a straight line perpendicular to the mirror. This line will be the tangent to the graph that we want to determine. It is a so-called „normal line" method of determining a tangent line.

and it will continue to cyclically reoccur. The amplitude of oscillations will however decrease with time, as in every cycle some amount of energy will be dissipated in the resistor in the form of a heat.

1. With disconnected power supply set its `CH3` channel voltage to $5\ \mathrm{V}$. Then, connect the supply to the analyzed circuit.

2. Assemble the setup shown in Fig. 5.31 (solid line only). Use the following parameters of elements: $C = 1\ \mu\mathrm{F}$, $L = 10\ \mathrm{mH}$ and $R = 50\ \Omega$. Power on the multifunctional device and the oscilloscope. Activate the two channels of the oscilloscope, both with DC coupling (DC), set the gain in channel X to $0.5\ \mathrm{V/div}$, and in channel Y to $2\ \mathrm{V/div}$. Choose automatic trigger mode and set time base to $0.2\ \mathrm{ms/div}$. Position both horizontal lines displayed in the oscilloscope at half height of the screen.

3. Witch `CH1` channel of the function generator, set its `CH1` channel to generate (Similarly as in section 5.4.2) pulse wave of amplitude $20\ \mathrm{Vpp}$, duty cycle 1%, 0 offset and frequency $100\ \mathrm{Hz}$ (one may increase the frequency a bit to obtain better oscillogram but do not exceed $500\ \mathrm{Hz}$). Then activate relevant channels and connect to the analyzed circuit both the power supply and the generator.

4. Activate both channels X and Y of the oscilloscope. Observe in the screen two oscillations decaying to zero, with a phase shift between them. How many periods (count for example the maxima of the oscillation of voltage across capacitor C) are there in the oscillations before the circuit attains a steady state? Compare this number with the quality factor Q_{RLC} of a series RLC resonant circuit. What is the frequency of oscillations (you can compute it for example as the inverse of the time interval between the first and the second maximum in the waveform of the voltage across capacitor C, which you can measure with the oscilloscope using its cursors)? Compare this value with the resonant frequency f_r of a series RLC resonant circuit.

5. Connect another resistor $R = 50\ \Omega$ to the circuit in parallel, as shown in Fig. 5.31 with a dash line, thus reducing the loss resistance to $\frac{R}{2} = 25\ \Omega$. Change the time base to $0.5\ \mathrm{ms/div}$. Again observe the oscillations in the screen. How many periods are there this time before the circuit attains a steady state? Compare this number with the quality factor $Q_{\frac{R}{2}LC}$ of a series $\frac{R}{2}LC$ resonant circuit. What is the frequency of oscillations this time? Compare this value with the resonant frequency f_r of a series $\frac{R}{2}LC$ resonant circuit. Sketch the oscillograms of the two waveforms in appropriate grid area provided in the report template. You can also save the oscillogram to disk, and later print it and attach to the report.

1× →BMP

6. Set the oscilloscope to the X-Y mode of operation, and DC coupling (DC) in both of its channels. Using the shift knob `POSITION` for one channel and then for the other, set the origin of coordinates (indicated by colour „arrows” in the left and top margin of the screen) in the center of the screen. In the mode of continuous gain adjustment for a given channel, turn the gain knob until the contour of the „snail” observed in the screen takes the shape of a circle rather than an ellipse. (will be flattened neither horizontally, nor vertically). This oscillogram should also be saved to disk, and later printed and attached to the report.

1× →BMP

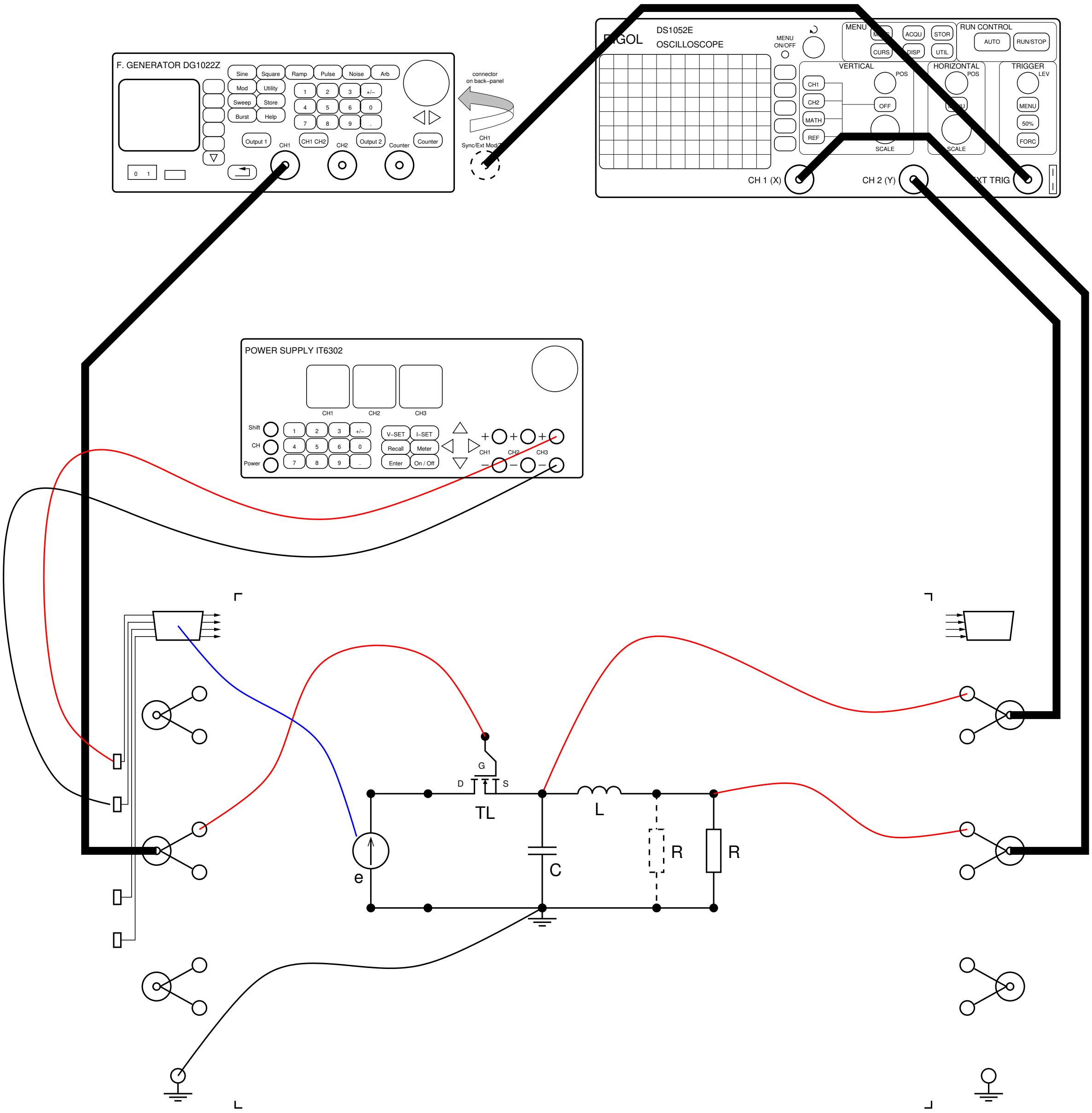


Figure 5.31: The setup with an electronic key, for observation of transient states in a second order circuit

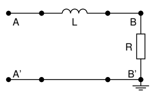
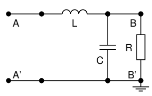
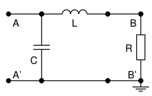
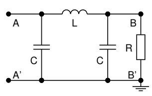
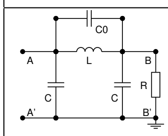
The „snail" (a logarithmic spiral) observed in the screen in the phase space, illustrates mutual transfers of energy, of the electric field (capacitor) to the magnetic field (inductor), and conversely. The *horizontal* axis represents the voltage across resistor $\frac{R}{2}$, which can be interpreted as a quantity proportional the the *current through inductor* L. The *vertical* axis on the other hand, represents the *voltage across capacitor* C. The „snail" starts at the point determined by the initial conditions: the voltage across capacitor in the moment of commutation (open-circuiting the key) is equal to EMF of the voltage source $e = E_0$, and the current through inductor in the moment of commutation is such that the voltage across resistor $\frac{R}{2}$ is also E_0.

The „snail" graph in the first and third quadrants of the phase space is such that the square of the voltage across the capacitor diminishes to zero, and the square of the current through the inductor grows starting from zero. In these quadrants we observe a transfer of energy from the electric field to the magnetic field. In turn, in the second and fourth quadrants of the phase space, one can read from the „snail" graph that square of the current through the inductor diminishes to zero, and the square of the voltage across the capacitor grows starting from zero. In these quadrants we observe a transfer of energy in the opposite direction: from the magnetic filed to the electric field. *Square* of the distance of a given point of the „snail" graph from the origin of coordinates can be interpreted as a quantity proportional to the total energy accumulated in the circuit at a given time instant, which is the sum of energies accumulated in the capacitor and in the inductor. The fact that this point gets closer and closer to the origin of coordinates indicates that there are energy losses in the circuit related to the presence of resistor $\frac{R}{2}$.

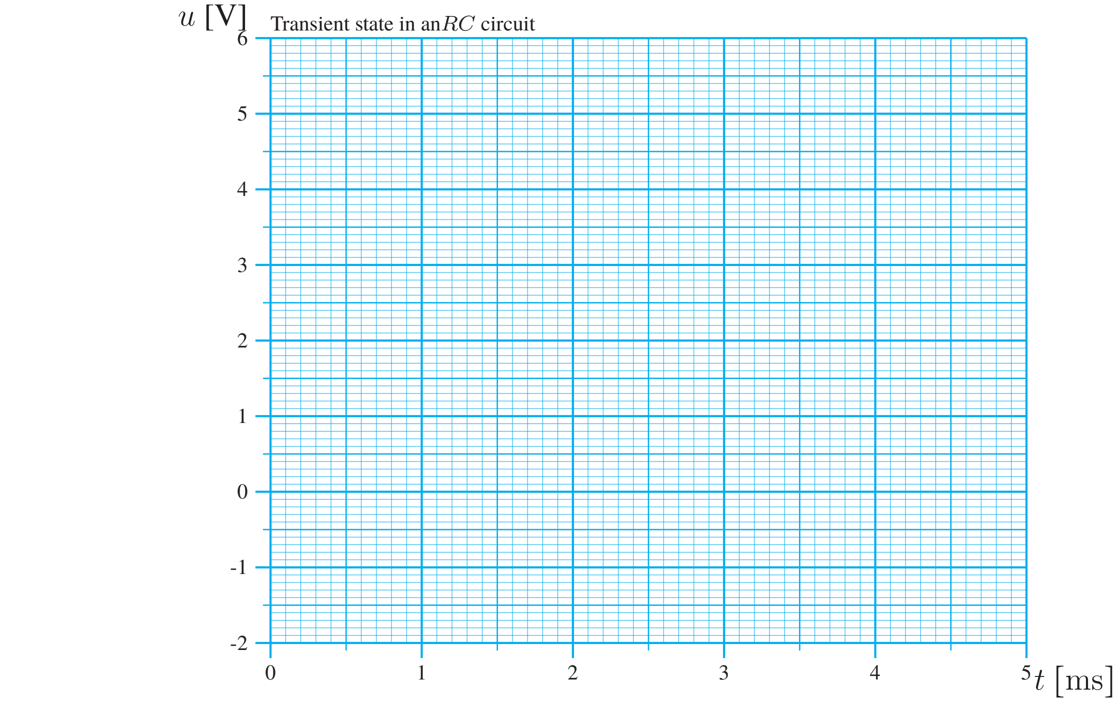
Stand:	*Name and SURNAME*	*Grade*
	. . . .	. . . .
Ex. no.: **5**	. . . .	. . . .
	. . . .	. . . .
Topic: ***Filters.*** ***Transient states.***		*Instructor's signature*
		Date

Measurement of frequency responses of filters

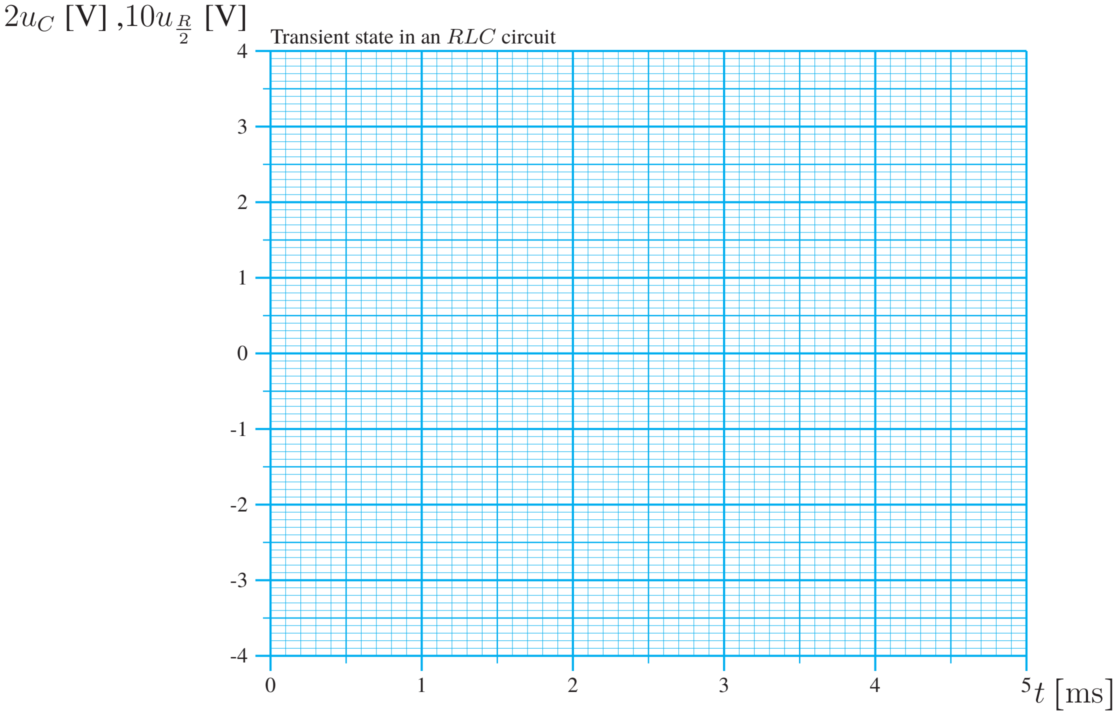
Table 5.1: Transition bands of various filters

Examined filter	f_1 [kHz]	f_2 [kHz]
		
		
		
		
		

Analysis of transient states in first order circuits

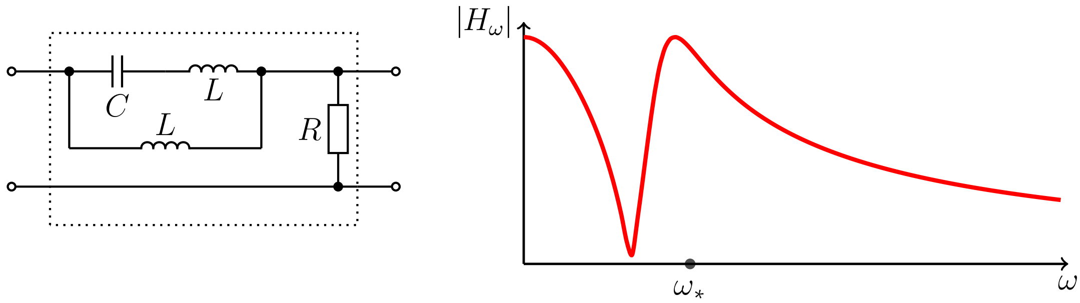


Analysis of transient states in second order circuits



Solutions to self-study problems

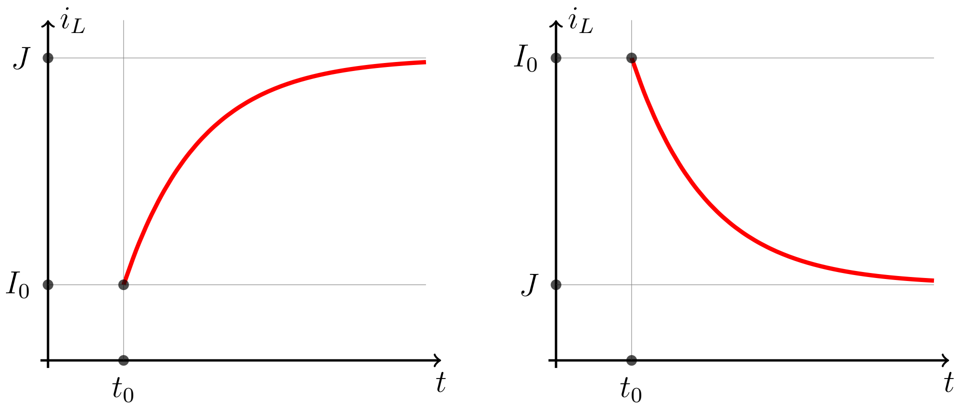
1. An example of a 2-port having the desired properties and its amplitude characteristic are given below.



2. For $t \geq t_0$, current i_L equals

$$i_L(t) = J + (I_0 - J)\mathrm{e}^{-\frac{t-t_0}{\tau}}, \quad \text{where } \tau = \frac{L}{R}.$$

In the considered circuit there will be no transient state if $I_0 = J$. The waveforms of current i_L for $I_0 < J$ and $I_0 > J$ are presented in the following pictures, respectively.

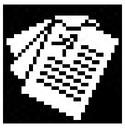


3. The difference changes 10-fold after $\ln 10\tau \approx 2.3\tau$. A 100-fold and a 1000-fold decrease is obtained after $\tau \ln 100 \approx 4.6\tau$ and $\tau \ln 1000 \approx 6.9\tau$, respectively.

Appendix A

Figure printing

A.1 Introduction



During each laboratory session, a team carrying out the experiments is allowed to print up to two two-sided A4 sheets, two figures per side each. The figures should be carefully prepared according to the instructions in the manual. A TOiS2PDF software is provided to facilitate printing of experiment results. To invoke this software one should click the relevant icon located on the desktop.

The printing with TOiS2PDF is a four-stage process:

1. During measurements one has to export or save figures to files of prescribed formats only. For the oscillograms obtained with `RIGOL DS1052E` digital oscilloscope the correct format is `BMP`. For the figures obtained with MATLAB and TOiS_Toy software — it has to be `PNG`. The preferred file location is in the `C:tmp` folder or the desktop.

2. It is only when all necessary measurements are completed and necessary figures saved to disk, that one should run TOiS2PDF software (see Fig. A.1). The interface of the program is hopefully quite intuitive. Four big push buttons invoke dialogs allowing inclusions of graphics files. In the corresponding four text boxes one should enter the titles of the figures. After including the files and specifying the titles one should press the `Generate PDF` and then `Show PDF` push buttons, to see the resulting `PDF` document. In particular, the `Show PDF` button invokes the default `PDF` viewer, which is PDF-XChange Viewer.

3. If the generated `PDF` document is correct (e.g. the titles are not too long, which prevents figures from being moved to next pages), one may enhance it by adding extra comments, remarks, labels or other graphical elements using the rich features of PDF-XChange Viewer.

4. Once the `PDF` document has its final form, one may print it choosing a printer corresponding to the laboratory stand number (one printer serves stands 1–7, the other serves stands 8–16).

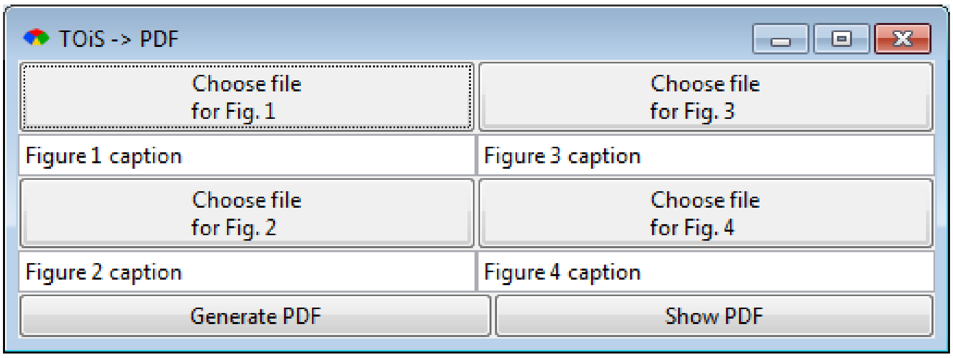


Figure A.1: TOiS2PDF's GUI

A.2 Exporting BMP from UltraScope

To save an oscillogram obtained with RIGOL DS1052E oscilloscope to a BMP file one should perform the following steps:

1. Run UltraScope software. You can find the icon of the UltraScope software on the desktop.

2. Click the forth button (labeled "Connect to Oscilloscope") of the Ultrascope's toolbar.

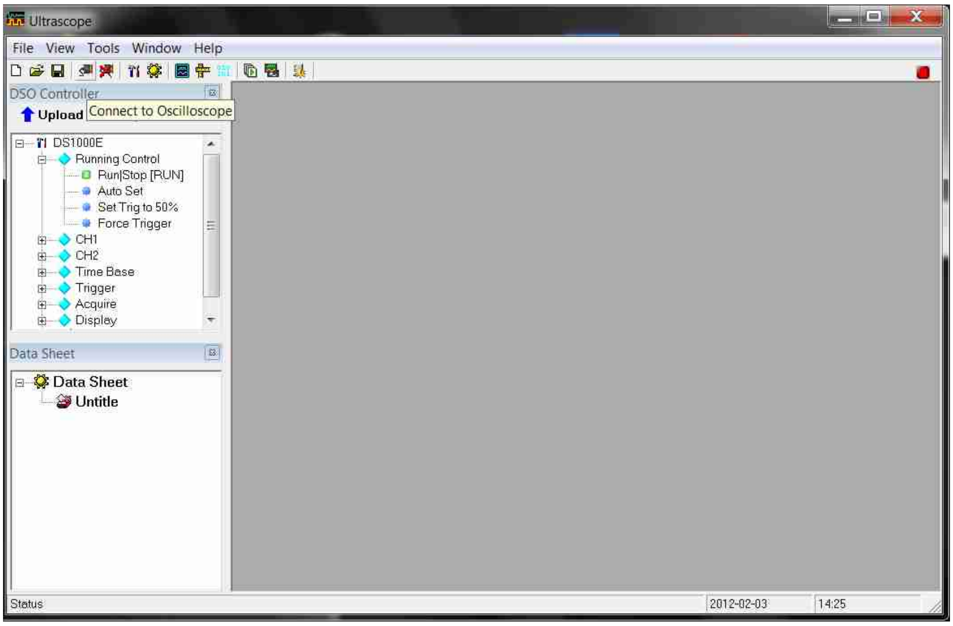


Figure A.2: Connecting UltraScope software to oscilloscope

3. In the invoked dialog window (entitled Resource, see Fig. A.3) choose the USB port the oscilloscope is connected to.

4. After establishing successful connection with the oscilloscope, a small box located to the right of the toolbar changes its color from red to blue (compare Fig. A.2 and Fig. A.4). Click the eighth button (labelled `Add New Waveform`) of the `UltraScope`'s toolbar.

5. If the second channel waveform is to be saved as well, in the settings tree of `DSO Controller` set `CH2:Display` to `ON`.

6. Refresh the waveform in `Wave...` window either by pressing `Refresh` button or by hitting F5 key (see Fig. A.5). **To see in `Ultrascope` exactly the same waveform as seen on the oscilloscope, one should stop the oscilloscope with `RUN/STOP` button prior to refreshing the waveform.**

7. Resize the `Wave...` window so that the `Export` push-button of the section `Output` becomes visible and click it (see Fig. A.7).

8. Choose the name of the `BMP` file in the invoked export-file dialog window (see. A.8). The preferred file location is in the `C:tmp` folder or the desktop.

9. After exporting the waveform, disconnect `UltraScope` from the oscilloscope either by pressing `Disconnect` button (as shown in Fig. A.9) or by pressing the `FORCE Local` oscilloscope button located in `TRIGGER` section.

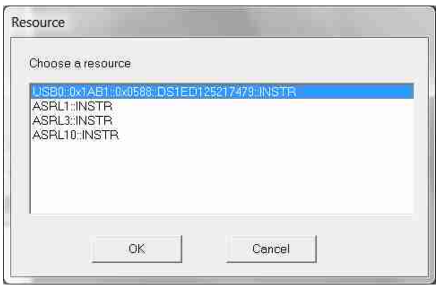


Figure A.3: Choosing USB port

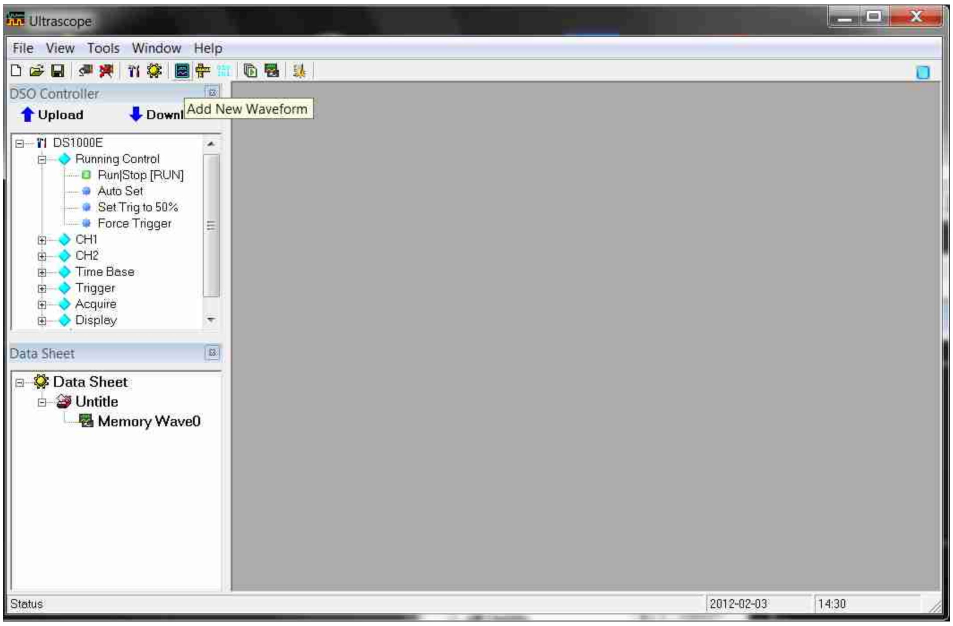


Figure A.4: Adding new waveform

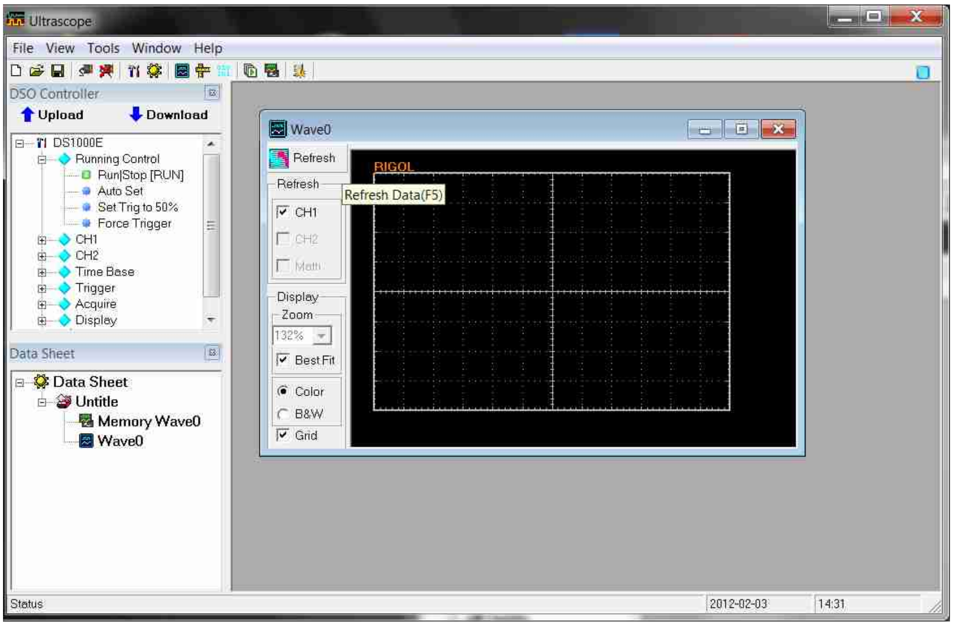


Figure A.5: Waveform refreshing

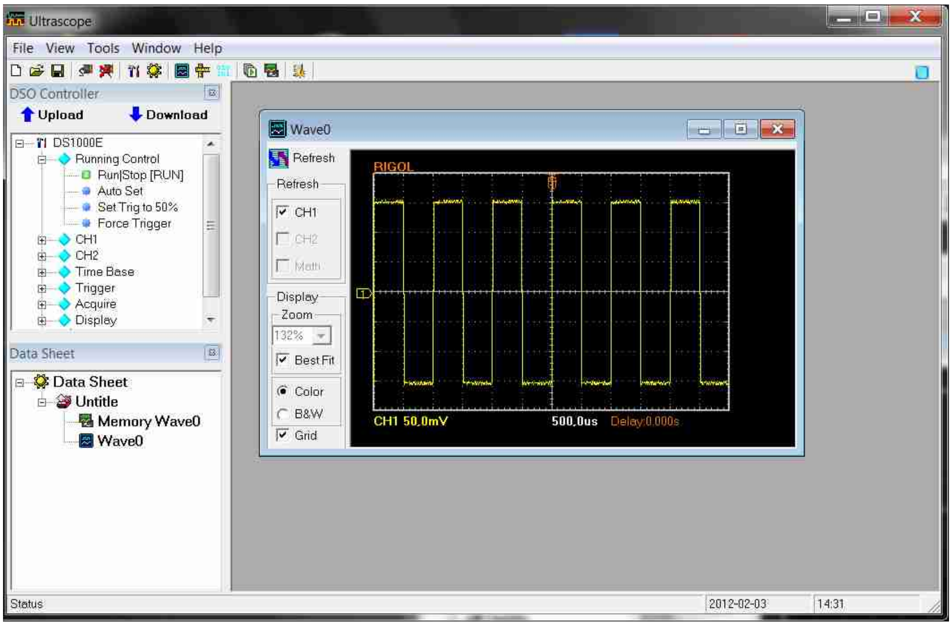


Figure A.6: Example of a waveform

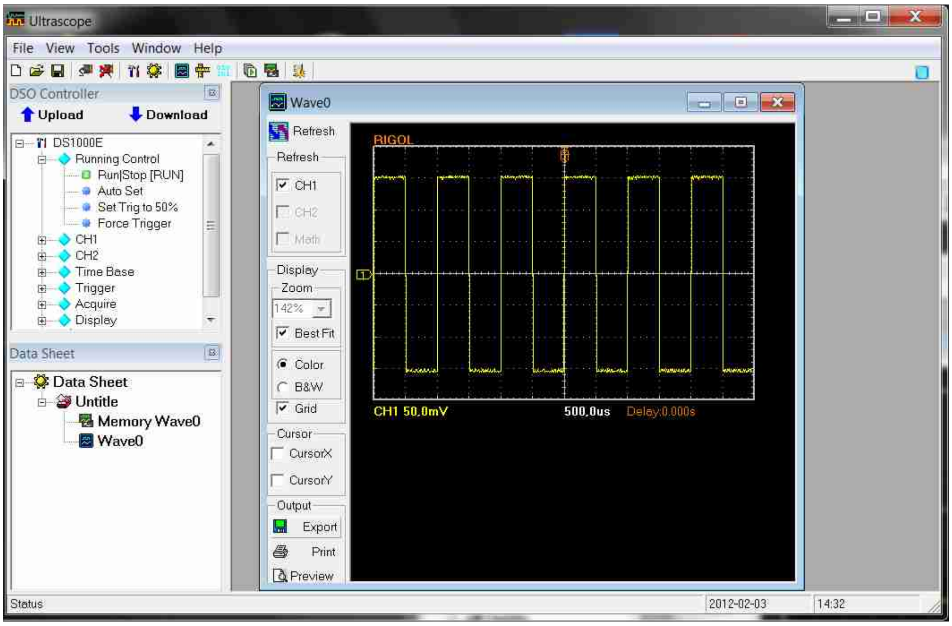


Figure A.7: Exporting of a waveform to a file

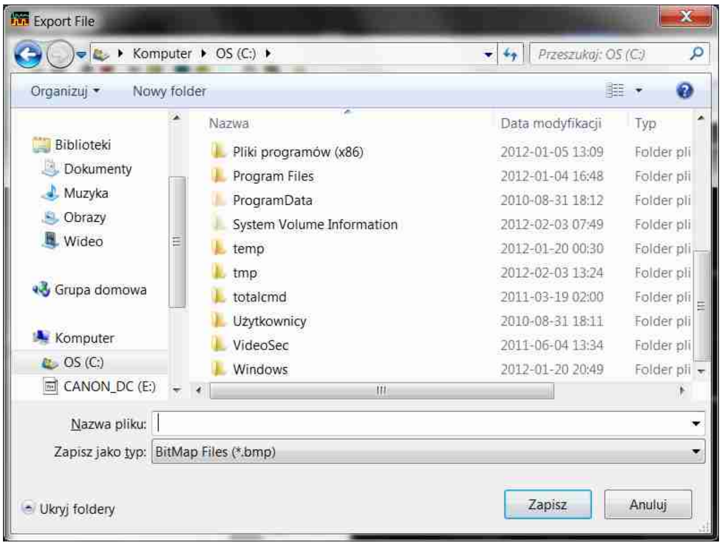


Figure A.8: Export File dialog window

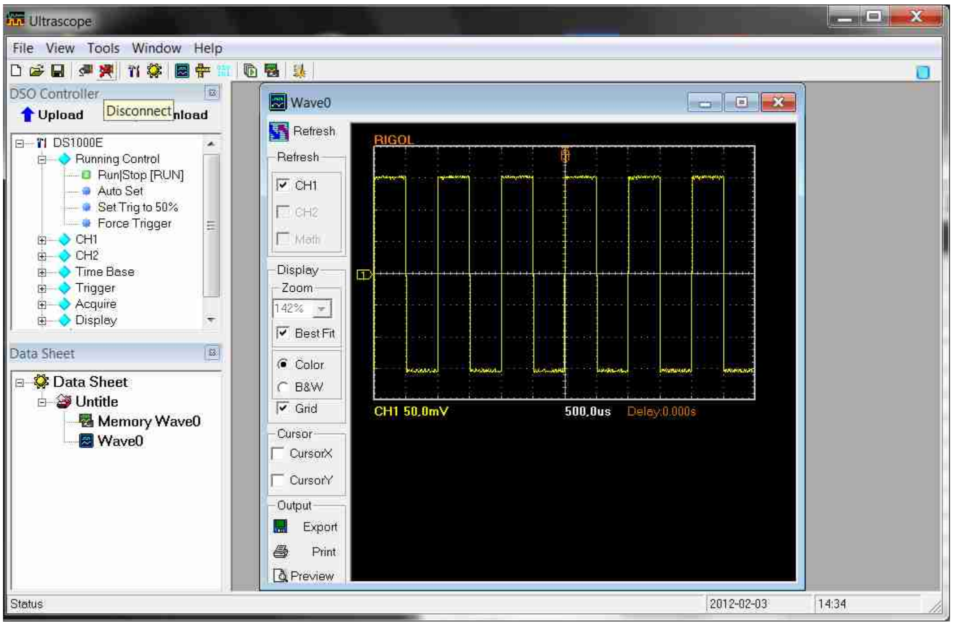


Figure A.9: Disconnecting `UltraScope` from the oscilloscope

A.3 Exporting graphics from TOiS_Toy software

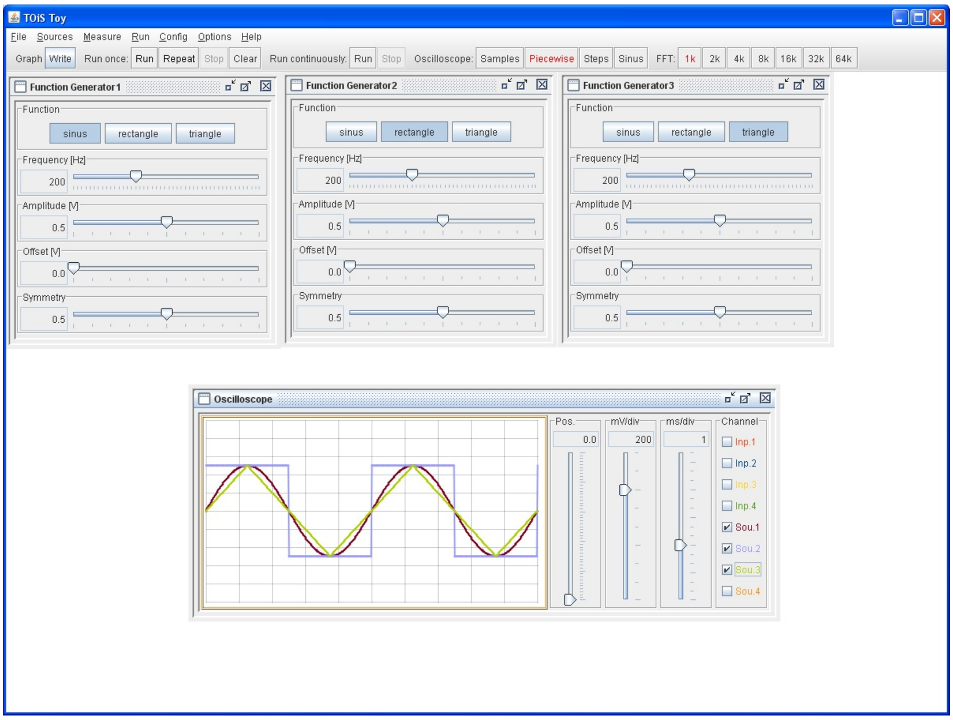


Figure A.10: TOiS_Toy's GUI

Figures made with TOiS_Toy software should be saved in (Run / Once) mode only. In order to export `PNG` graphics from TOiS_Toy one should perform the following steps:

1. Click `Graph Write` button located at the left side of the toolbar (see Fig. A.10).
2. In the invoked dialog window (see Fig. A.11) choose the appropriate virtual device in `Graph` section and set the size of the graph (it is strongly recommended to leave the default size: 1400×1000). Then, click `Write` button.
3. In the opened save-file dialog window (see Fig. A.12) choose the file name of the `PNG` file. The preferred file location is in the `C:tmp` folder or the desktop.

A.4 Exporting PNG graphics from MATLAB

In order to export a MATLAB figure to a `PNG` file one should perform the following steps:

1. Chose `Export Setup` from the `File` menu of the current figure (see Fig. A.13).
2. In the invoked dialog window click `Export` button (see Fig. A.14).
3. In the opened save-as dialog window choose a `PNG` file name for the figure (see Fig. A.15). The preferred file location is in the `C:tmp` folder or the desktop.

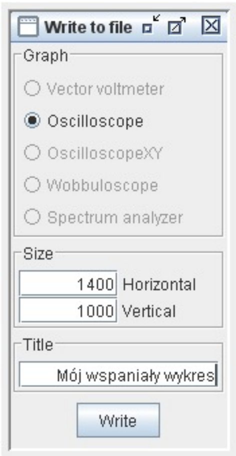


Figure A.11: Image parameters dialog window of TOiS_Toy

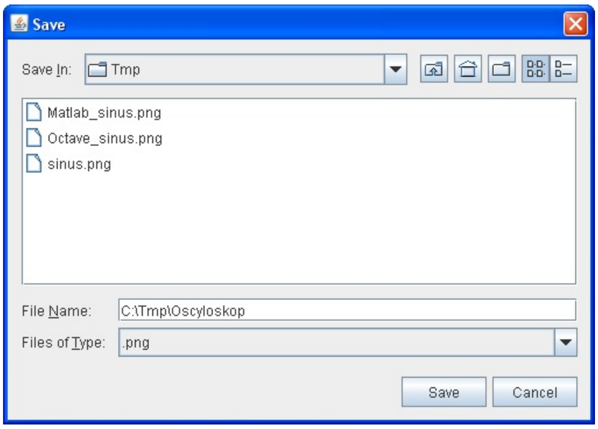


Figure A.12: Save file dialog window of TOiS_Toy

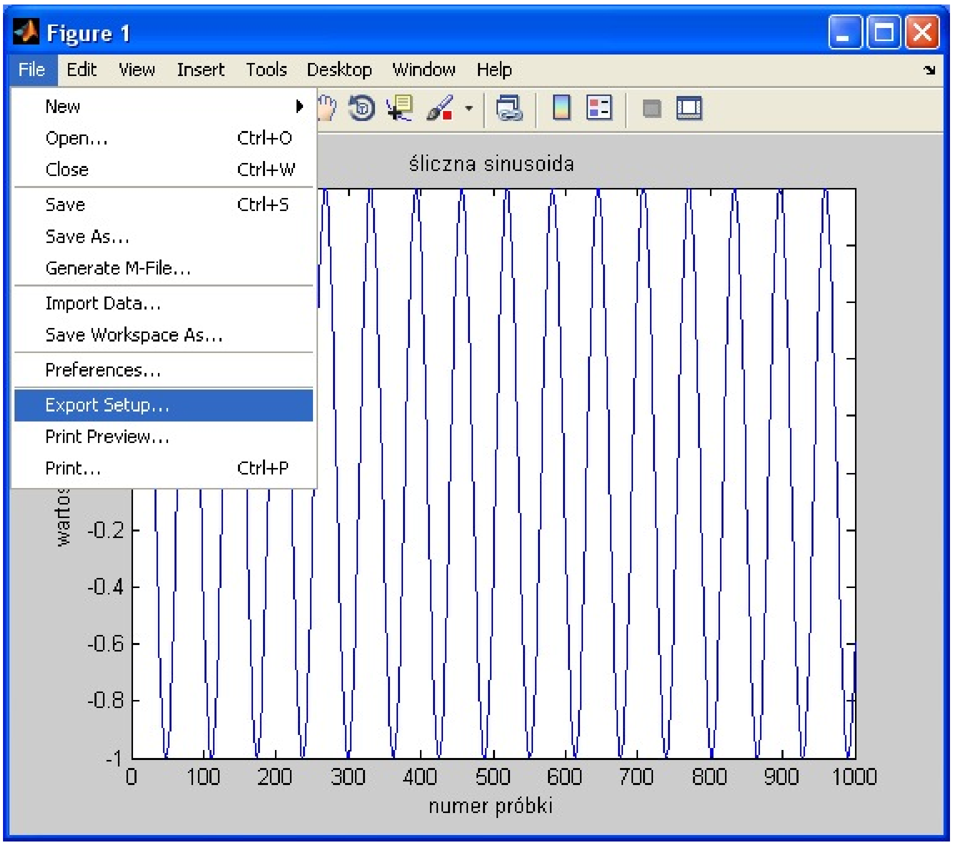


Figure A.13: MATLAB figure GUI

The above procedure may be also replaced with the following MATLAB command:

```
print c:\tmp\file_name.png -dpng
```

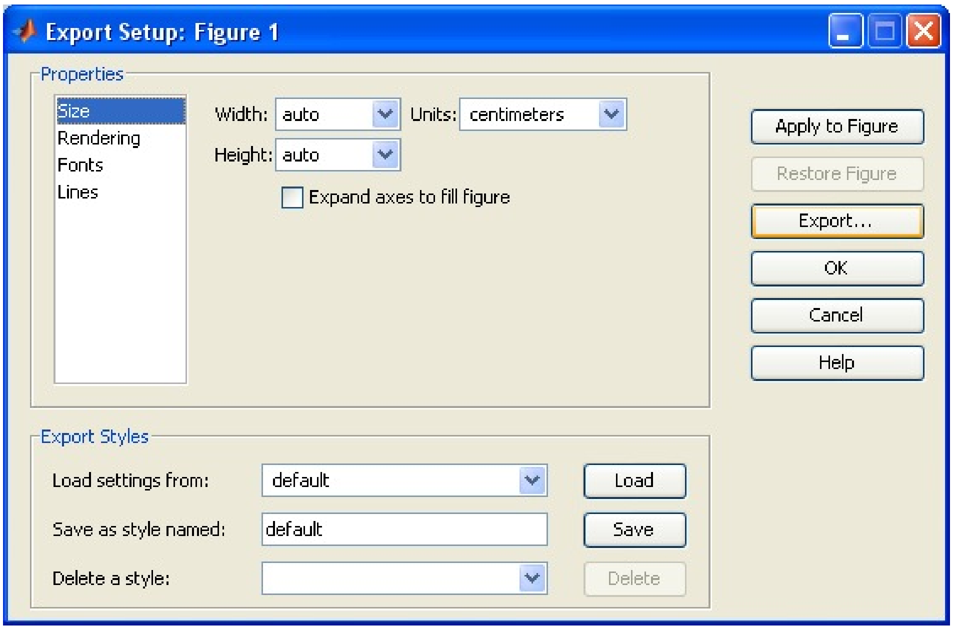


Figure A.14: Export Setup dialog window

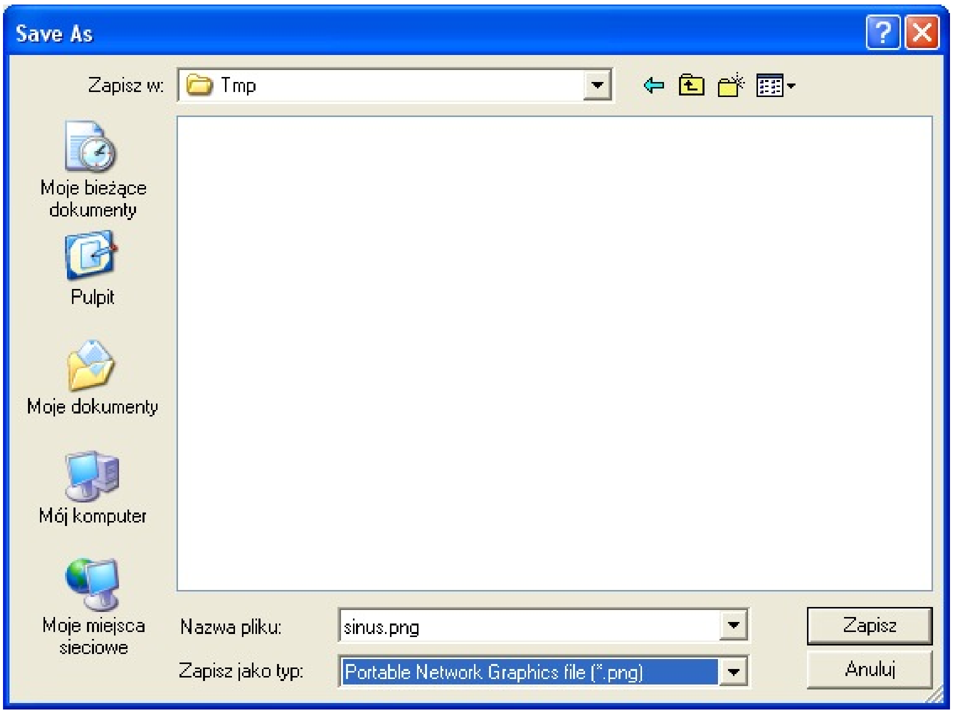


Figure A.15: Save As dialog window