CORELA

CORELA exercises

Handbook

Funded by the
Erasmus+ Programme
of the European Union

Contents

I Introduction 5
II Electrical circuit quantities 6
1. Theoretical tasks (all students) 6
2. Calculations of electrical quantities (student 1) 7
2.1 Resistors in a series electrical circuit 7
2.2 Resistors in a parallel electrical circuit 9
2.3 Resistors in a series-parallel combination electrical circuit 11
2.4 Comparation of the results 14
3. CORELA education platform simulation (student 2) 14
3.1 Simulation of a series electrical circuit 14
3.2 Simulation of a parallel electrical circuit 22
3.3 Simulation of a series-parallel combination electrical circuit 22
3.4 Comparation of the results 22
4. Measurements of electrical quantities (student 3) 22
4.1 Measurement of electric resistance 22
4.2 Resistors in a series electrical circuit 23
4.3 Resistors in a parallel electrical circuit 25
4.4 Resistors in a series-parallel combination electrical circuit 27
4.5 Comparation of the results 29
III Impedance measurement 30
1. Introduction 30
2. Theoretical tasks (all students) 31
3. Calculating impedance (student 1) 32
3.1 Calculations 32
3.2 Comparation of the results 34
4. Determination of impedance by simulation (student 2) 34
4.1 Activation of the exercise 34
4.2 Selecting the type of electrical circuit and reading the results 35
4.3 Comparation of the results 37
5. Impedance measurement (student 3) 37
5.1 Procedure 38
5.2 Comparation of the results 40
6. Procedure of writing and sending data to the educational platform 41

IV Static current-voltage characteristic of a rectifier diode 46
1. Introduction 46
2. Theoretical tasks (all students) 47
3. Calculation of 1N4001 diode current value depending on the biasing voltage (student 1) . 47
4. Simulation of measuring of diode static characteristic using the functions implemented in CORELA platform (student 2) 50
5. Practical realization and measurement (student 3) 55
6. Appendix 1: Catalogue values for diodes implemented in the platform 60
V Logic functions 61
1. Theoretical tasks (*student 1*) 61
1.1 Realisation of the logic function 61
1.2 Realisation of the logic function using NAND gates 64
1.3 Simplification of the logic function using Boolean algebra laws 65
1.4 Simplification of the logic function using Veitch diagram 67
2. CORELA education platform simulation (student 2) 69
2.1 Realisation of the logic function 69
2.2 Realisation of the logic function using NAND gates 72
2.3 Simplification of the logic function using Boolean algebra laws 74
2.4 Simplification of the logic function using Veitch diagram 76
3 Laboratory practise (student 3) 78
3.1 Realisation of the logic function 78
3.2 Realisation of the logic function using NAND gates 82
3.3 Simplification of the logic function using Boolean algebra laws 83
3.4 Simplification of the logic function using Veitch diagram 85
VI PWM DC motor control 88
1. Introduction 88
2. PWM control for DC motor (student 1) 89
2.1 Comparation of the results 90
3. Simulation of PWM control for DC motor (student 2) 91
3.1 Comparation of the results 93
4. Practical realization and measurement (student 3) 94
4.1 Comparation of the results 96
5. Procedure of writing and sending dana to the educational platform 96
VII Maximum Power Transfer 100

1. Introduction 100

2. Calculation of maximum power transfer (student 1) 101

2.1 Comparation of the results 102

3. Maximum power transfer simulation (student 2) 102

3.1 Comparation of the results 104

4. Practical realization and measurement (student 3) 105

4.1 Comparation of the results 107

5. Procedure of writing and sending dana to the educational platform 107

I Introduction

The developed platform is a product of the Erasmus+ CORELA project. The project aims to develop a common platform for technical Vocational Education and Training (VET). The CORELA platform has an extension with integrated RVL and is specially developed for electro technical higher educational institutions. The platform offers collaboration with other pupils or groups in terms to share knowledge, experience, or compare the obtained results from the different methodologies. The final or intermediate results from the platform can be provided to the Massive Open Online Course (MOOC) platform. The MOOC platform can present the results as a freely view assignment to all pupils, as forum thread for further discussion or private/final assignment results accessible only by the teacher.

The presented platform can operate in three different modes. In the first mode, the platform allows analytical computation. The analytical computation is oriented to theoretical assignments without the involvement of the real parameters and elements. The second mode introduces the simulation environment. The simulation involves testing different kinds of real scenarios, like a parameter deviation, parameter, and measurement uncertainty model mismatch, etc. The simulation is an intermediate step between ideal analytical computation and real experiments with real components and measurement tools. The third operation mode of the platform is a real-time experiment with real-time measuring and components. The main supported acquisition device is a DAQ Card. The DAQ can be used as a digital and analog input/output interface. The platform offers a variety of connections to different measurement tools. The platform also supports standard serial communication, which spread the functionality to the custom-designed experiments and measurement equipment. All the real-time data can be stored for later analysis and examination. All three operation modes of the platform offer a connection to the MOOC system, where all data and parameters of calculation, simulation, or real-time experiments can be presented. Regard to the discipline and the teaching methodology the presented results can be used for collaborative learning and interaction with different student groups or just automated data collection platforms for teacher supervision.

In continuation of the given text, six exercises from different fields in electro-technics are presented in three modes – analytical computation, simulation on the platform and experiment. Also, the procedure of presenting the results on the MOOC platform is included in every exercise. For better introduction to three working modes, video materials are included at the end of each exercise.

Dear pupils/students,

At the beginning of solving the theoretical tasks and before practical work read the instructions carefully. Follow the instructions and advices considering health and safety at work.

II Electrical circuit quantities

1. Theoretical tasks (all students)

1.1) Determine the values of the three resistors according to the *Resistor Color Code*.

Resistor 1 (R_1): brown, green, black, brown, red

Resistor 2 (R_2): brown, black, black, brown, red

Resistor 3 (R_3): green, black, brown, red

Table 1: Electric resistance values according to the Resistor Color Code

resistor	numerical value (record) *	electrical resistance R [Ω]	absolute error p [Ω]	minimum rersistance R_{min} [Ω]	maximum rersistance R_{max} [Ω]
R_1	1 5 0 x 10 Ω +/- 2 %	1500 Ω	30 Ω	1470 Ω	1530 Ω
R_2	1 0 0 x 10 Ω +/- 2 %	1000 Ω	20 Ω	980 Ω	1020 Ω
R_3	5 0 x 10 Ω +/- 2 %	500 Ω	10 Ω	490 Ω	510 Ω

* numerical value:
figure_figure_figure x decade [Ω] +/- tolerance [%] or figure_figure x decade [Ω] +/- tolerance [%]

1.2) Describe Ohm's law (definition) and write an equation (formula) that mathematically illustrates Ohm's law.

The electric current flown through the electrical resistor (element) is directly proportional to the electric voltage across the resistor (between the terminals of the resistor).

$$U = R \cdot I \qquad I = \frac{U}{R} \qquad R = \frac{U}{I}$$

1.3) Describe the characteristics of a series electrical circuit. Write down the mathematical equations which illustrate described characteristics.

1.4) Describe the characteristics of a parallel electrical circuit. Write down the mathematical equations which illustrate described characteristics.

1.5) What is a voltage loop regarding the electrical circuit? Describe the Kirchhoff's 2nd Law and illustrate it with mathematical equation.

1.6) What is a current node / junction regarding the electrical circuit? Describe the Kirchhoff's 1st Law and illustrate it with mathematical equation.

1.7) How do you measure the electric voltage?
a) Name the measuring instrument.
b) Describe the insertion of the measuring instrument into the electrical circuit or connection to the measured element.
c) Draw the scheme of the measured element and the instrument.

1.8) How do you measure the electric current?
a) Name the measuring instrument.
b) Describe the insertion of the measuring instrument into the electrical circuit or connection to the measured element.
c) Draw the scheme of the measured element and the instrument.

1.9) How do you measure the electric resistance?
a) Name the measuring instrument.
b) Describe the insertion of the measuring instrument into the electrical circuit or connection to the measured element.
c) Draw the scheme of the measured element and the instrument.

2. Calculations of electrical quantities (student 1)

2.1 Resistors in a series electrical circuit

2.1.1) The three resistors are connected in series to a direct voltage source. The series electrical circuit scheme/diagram is shown in Figure 2.1.

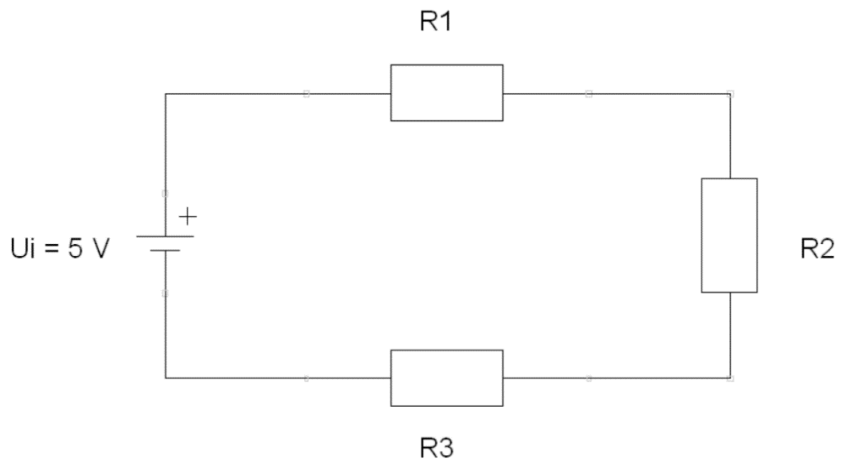


Figure 2.1: Resistors in series electrical circuit

2.1.2) Calculate total resistance R_N, minimum total resistance R_{Nmin} and maximum total resistance R_{Nmax} of the series electrical circuit based on individual resistance values determined by the resistance color code.

a) Total resistance R_N:

$$R_N = R_1 + R_2 + R_3 = 1500\,\Omega + 1000\,\Omega + 500\,\Omega = 3000\,\Omega$$

b) Minimum total resistance R_{Nmin}:

$$R_{Nmin} = R_{1min} + R_{2min} + R_{3min} = 1470\,\Omega + 980\,\Omega + 490\,\Omega = 2940\,\Omega$$

c) Maximum total resistance R_{Nmax}:

$$R_{Nmax} = R_{1max} + R_{2max} + R_{3max} = 1530\,\Omega + 1020\,\Omega + 510\,\Omega = 3060\,\Omega$$

2.1.3) Direct voltage supply or voltage source U_i is 5 V DC. Calculate the value of current supply or current source I_i.

U_i = 5 V DC

$$I_i = \frac{U_i}{R_N} = \frac{5\,V}{3000\,\Omega} = 0{,}00167\,A = 1{,}67\,mA$$

2.1.4) Calculate the values of electrical quantities in the electrical circuit. Fill in the Table 2.1; write down the calculated values of each resistor.
Note: When calculating electrical quantities, write down suitable equations (formulas).

Table 2.1: Calculated values of electrical quantities in series electrical circuit
** Copy the determined values from Table 1*

element	electric current I [mA]	electric voltage U [V]	electric resistance R_m [Ω] *
Resistor 1			
Resistor 2			
Resistor 3			

$$I_1 = I_i = 0{,}001\,A \;\rightarrow\; U_1 = R_1 \cdot I_1 = 1500\,\Omega \cdot 0{,}001\,A = 2{,}5\,V$$

$$I_2 = I_i = 0{,}001\,A \;\rightarrow\; U_2 = R_2 \cdot I_2 = 1000\,\Omega \cdot 0{,}001\,A = 1{,}67\,V$$

$$I_3 = I_i = 0{,}001\,A \;\rightarrow\; U_3 = R_3 \cdot I_3 = 500\,\Omega \cdot 0{,}001\,A = 0{,}835\,V$$

2.1.5) Calculate the sum of the voltage drops across resistors. Compare the calculated sum to the value of voltage output or voltage supply U_i.

$$U_1 + U_2 + U_3 = 2{,}5\,V + 1{,}67\,V + 0{,}835\,V = 5\,V = U_i$$

2.2 Resistors in a parallel electrical circuit

2.2.1) The three resistors are connected in parallel to a direct voltage source. The parallel electrical circuit scheme/diagram is shown in the Figure 2.2.

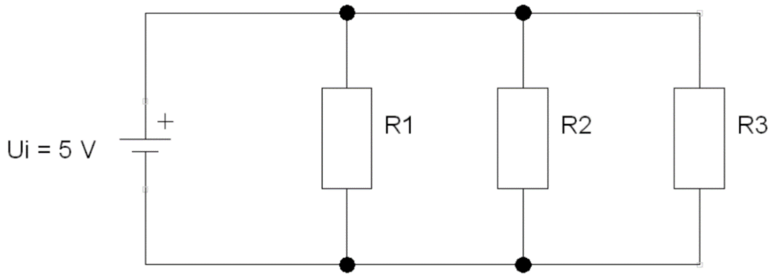


Figure 2.2: Resistors in parallel electrical circuit

2.2.2) Calculate total resistance R_N, minimum total resistance R_{Nmin} and maximum total resistance R_{Nmax} of the parallel electrical circuit based on individual resistance values determined by the resistance color code.

a) Total resistance R_N:

$$G_1 = \frac{1}{R_1} = \frac{1}{1500\ \Omega} = 0{,}00066667\ S$$

$$G_2 = \frac{1}{R_2} = \frac{1}{1000\ \Omega} = 0{,}001\ S$$

$$G_3 = \frac{1}{R_3} = \frac{1}{500\ \Omega} = 0{,}002\ S$$

$$G_N = G_1 + G_2 + G_3 = 0{,}00066667\ S + 0{,}001\ S + 0{,}002\ S = 0{,}00366667\ S$$

$$R_N = \frac{1}{G_N} = \frac{1}{0{,}00366667\ S} = 272{,}73\ \Omega = 273\ \Omega$$

Calculation with the total resistance formula:

$$R_N = \frac{R_1 \cdot R_2 \cdot R_3}{R_1 \cdot R_2 + R_1 \cdot R_3 + R_2 \cdot R_3} =$$

$$R_N = \frac{1500\ \Omega \cdot 1000\ \Omega \cdot 500\ \Omega}{1500\ \Omega \cdot 1000\ \Omega + 1500\ \Omega \cdot 500\ \Omega + 1000\ \Omega \cdot 500\ \Omega} =$$

$$R_N = \frac{750.000.000\ \Omega^3}{1.500.000\ \Omega^2 + 750.000\ \Omega^2 + 500.000\ \Omega^2} = \frac{750.000.000\ \Omega^3}{2.750.000\ \Omega^2} = 272{,}73\ \Omega = 273\ \Omega$$

b) Minimum total resistance R_{Nmin}:

$$G_{1min} = \frac{1}{R_{1min}} = \frac{1}{1470\ \Omega} = 0{,}00068027\ S$$

$$G_{2min} = \frac{1}{R_{2min}} = \frac{1}{980\ \Omega} = 0{,}00102041\ S$$

$$G_{3min} = \frac{1}{R_{3min}} = \frac{1}{490\ \Omega} = 0{,}00204082\ S$$

$$G_{Nmin} = G_{1min} + G_{2min} + G_{3min} =$$

$$G_{Nmin} = 0{,}00068027\ S + 0{,}00102041\ S + 0{,}00204082\ S = 0{,}0037415\ S$$

$$R_{Nmin} = \frac{1}{G_{Nmin}} = \frac{1}{0{,}0037415\ S} = 267{,}27\ \Omega = 267\ \Omega$$

Calculation with the total resistance formula:

$$R_{Nmin} = \frac{R_{1min} \cdot R_{2min} \cdot R_{3min}}{R_{1min} \cdot R_{2min} + R_{1min} \cdot R_{3min} + R_{2min} \cdot R_{3min}} =$$

$$R_{Nmin} = \frac{1470\ \Omega \cdot 980\ \Omega \cdot 490\ \Omega}{1470\ \Omega \cdot 980\ \Omega + 1470\ \Omega \cdot 490\ \Omega + 980\ \Omega \cdot 490\ \Omega} =$$

$$R_{Nmin} = \frac{705.894.000\ \Omega^3}{1.440.600\ \Omega^2 + 720.300\ \Omega^2 + 480.200\ \Omega^2} = \frac{705.894.000\ \Omega^3}{2.641.100\ \Omega^2} = 267{,}27\ \Omega = 267\ \Omega$$

c) Maximum total resistance R_{Nmax}:

$$G_{1max} = \frac{1}{R_{1max}} = \frac{1}{1530\ \Omega} = 0{,}00065359\ S$$

$$G_{2max} = \frac{1}{R_{2max}} = \frac{1}{1020\ \Omega} = 0{,}00098039\ S$$

$$G_{3max} = \frac{1}{R_{3max}} = \frac{1}{510\ \Omega} = 0{,}00196078\ S$$

$$G_{Nmax} = G_{1max} + G_{2max} + G_{3max} =$$

$$G_{Nmax} = 0{,}00065359\ S + 0{,}00098039\ S + 0{,}00196078\ S = 0{,}00359476\ S$$

$$R_{Nmax} = \frac{1}{G_{Nmax}} = \frac{1}{0{,}00359476\ S} = 278{,}18\ \Omega = 278\ \Omega$$

Calculation with the total resistance formula:

$$R_{Nmax} = \frac{R_{1max} \cdot R_{2max} \cdot R_{3max}}{R_{1max} \cdot R_{2max} + R_{1max} \cdot R_{3max} + R_{2max} \cdot R_{3max}} =$$

$$R_{Nmax} = \frac{1530\ \Omega \cdot 1020\ \Omega \cdot 510\ \Omega}{1530\ \Omega \cdot 1020\ \Omega + 1530\ \Omega \cdot 510\ \Omega + 1020\ \Omega \cdot 510\ \Omega} =$$

$$R_{Nmax} = \frac{795.906.000\ \Omega^3}{1.560.600\ \Omega^2 + 780.300\ \Omega^2 + 520.200\ \Omega^2} = \frac{795.906.000\ \Omega^3}{2.861.100\ \Omega^2} = 278{,}18\ \Omega = 278\ \Omega$$

2.2.3) Direct voltage supply or voltage source U_i is 5 V DC. Calculate the value of current supply or current source I_i.

U_i = 5 V DC

$$I_i = \frac{U_i}{R_N} = \frac{5\ V}{273\ \Omega} = 0{,}018\ A = 18\ mA$$

2.2.4) Calculate the values of electrical quantities in the electrical circuit. Fill in the Table 2.2; write down the calculated values of each resistor.

Note: When calculating electrical quantities, write down suitable equations (formulas).

Table 2.2: Calculated values of electrical quantities in parallel electrical circuit
** Copy the determined values from Table 1*

element	electric current *I* [mA]	electric voltage *U* [V]	electric resistance R_m [Ω] *
Resistor 1			
Resistor 2			
Resistor 3			

$$U_1 = U_i = 5\,V \rightarrow \qquad I_1 = \frac{U_1}{R_1} = \frac{5\,V}{1500\,\Omega} = 0{,}0033\,A = 3{,}3\,mA$$

$$U_2 = U_i = 5\,V \rightarrow \qquad I_2 = \frac{U_2}{R_2} = \frac{5\,V}{1000\,\Omega} = 0{,}005\,A = 5\,mA$$

$$U_3 = U_i = 5\,V \rightarrow \qquad I_3 = \frac{U_3}{R_3} = \frac{5\,V}{500\,\Omega} = 0{,}01\,A = 10\,mA$$

2.2.5) Calculate the sum of the current flows through resistors. Compare the calculated sum to the value of current output or current supply I_i.

$$I_1 + I_2 + I_3 = 0{,}0033\,A + 0{,}005\,A + 0{,}01\,A = 0{,}0183\,A \cong 0{,}018\,A = I_i$$

2.3 Resistors in a series-parallel combination electrical circuit

2.3.1) The three resistors are connected in series-parallel combination to a direct voltage source. The series-parallel electrical circuit scheme/diagram is shown in the Figure 2.3.

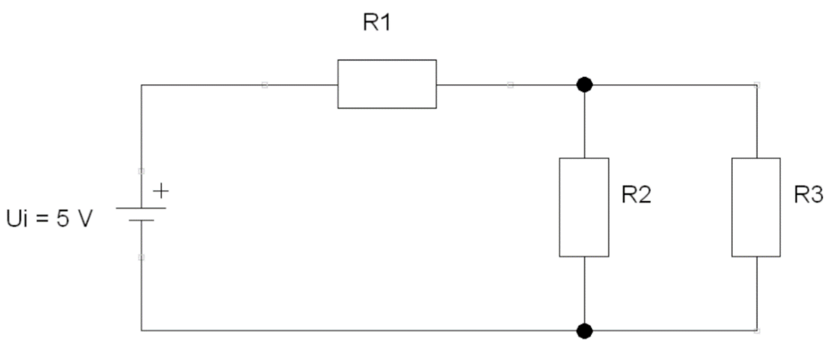


Figure 2.3: Resistors in series-parallel electrical circuit

2.3.2) Calculate total resistance R_N, minimum total resistance R_{Nmin} and maximum total resistance R_{Nmax} of the series-parallel combination circuit based on individual resistance values determined by the resistance color code.

a) Total resistance R_N:

$$G_2 = \frac{1}{R_2} = \frac{1}{1000\ \Omega} = 0{,}001\ S$$

$$G_3 = \frac{1}{R_3} = \frac{1}{500\ \Omega} = 0{,}002\ S$$

$$G_{23} = G_2 + G_3 = 0{,}001\ S + 0{,}002\ S = 0{,}003\ S$$

$$R_{23} = \frac{1}{G_{23}} = \frac{1}{0{,}003\ S} = 333{,}33\ \Omega = 333\ \Omega$$

$$R_N = R_1 + R_{23} = 1500\ \Omega + 333{,}33\ \Omega = 1833{,}33\ \Omega = 1833\ \Omega$$

Calculation with the total resistance formula:

$$R_N = R_1 + \frac{R_2 \cdot R_3}{R_2 + R_3} = 1500\ \Omega + \frac{1000\ \Omega \cdot 500\ \Omega}{1000\ \Omega + 500\ \Omega} = 330\ \Omega + \frac{500.000\ \Omega^2}{1.500\ \Omega} = 1500\ \Omega + 333{,}33\ \Omega$$
$$= 1833{,}33\ \Omega = 1833\ \Omega$$

b) Minimum total resistance R_{Nmin}:

$$G_{2min} = \frac{1}{R_{2min}} = \frac{1}{980\ \Omega} = 0{,}00102041\ S$$

$$G_{3min} = \frac{1}{R_{3min}} = \frac{1}{490\ \Omega} = 0{,}00204082\ S$$

$$G_{23min} = G_{2min} + G_{3min} = 0{,}00102041\ S + 0{,}00204082\ S = 0{,}00306123\ S$$

$$R_{23min} = \frac{1}{G_{23min}} = \frac{1}{0{,}00306123\ S} = 326{,}66\ \Omega = 327\ \Omega$$

$$R_{Nmin} = R_{1min} + R_{23min} = 1470\ \Omega + 326{,}66\ \Omega = 1796{,}66\ \Omega = 1797\ \Omega$$

Calculation with the total resistance formula:

$$R_{Nmin} = R_{1min} + \frac{R_{2min} \cdot R_{3min}}{R_{2min} + R_{3min}} = 1470\ \Omega + \frac{980\ \Omega \cdot 490\ \Omega}{980\ \Omega + 490\ \Omega} = 1470\ \Omega + \frac{480.200\ \Omega^2}{1.470\ \Omega}$$
$$= 1470\ \Omega + 326{,}67\ \Omega = 1796{,}67\ \Omega = 1797\ \Omega$$

c) Maximum total resistance R_{Nmax}:

$$G_{2max} = \frac{1}{R_{2max}} = \frac{1}{1020\ \Omega} = 0{,}00098039\ S$$

$$G_{3max} = \frac{1}{R_{3max}} = \frac{1}{510\ \Omega} = 0{,}00196078\ S$$

$$G_{23max} = G_{2max} + G_{3max} = 0{,}00098039\ S + 0{,}00196078\ S = 0{,}00294117\ S$$

$$R_{23max} = \frac{1}{G_{23max}} = \frac{1}{0{,}00294117\ S} = 340{,}00\ \Omega = 340\ \Omega$$

$$R_{Nmax} = R_{1max} + R_{23max} = 1530\ \Omega + 340\ \Omega = 1870\ \Omega$$

Calculation with the total resistance formula:

$$R_{Nmax} = R_{1max} + \frac{R_{2max} \cdot R_{3max}}{R_{2max} + R_{3max}} = 1530\,\Omega + \frac{1020\,\Omega \cdot 510\,\Omega}{1020\,\Omega + 510\,\Omega} = 1530\,\Omega + \frac{520.200\,\Omega^2}{1.530\,\Omega}$$
$$= 1530\,\Omega + 340\,\Omega = 1870\,\Omega$$

2.3.3) Direct voltage supply or voltage source U_i is 5 V DC. Calculate the value of current supply or current source I_i.

U_i = 5 V DC

$$I_i = \frac{U_i}{R_N} = \frac{5\,V}{1833\,\Omega} = 0{,}0027\,A = 2{,}7\,mA$$

2.3.4) Calculate the values of electrical quantities in the electrical circuit. Fill in the Table 2.3; write down the calculated values of each resistor.
Note: When calculating electrical quantities, write down suitable equations (formulas).

Table 2.3: Calculated values of electrical quantities in series-parallel combination electrical circuit
** Copy the determined values from Table 1*

element	electric current I [mA]	electric voltage U [V]	electric resistance R_m [Ω] *
Resistor 1			
Resistor 2			
Resistor 3			

$I_1 = I_i = 0{,}0027\,A \quad \rightarrow \quad U_1 = R_1 \cdot I_1 = 1500\,\Omega \cdot 0{,}0027\,A = 4{,}05\,V = 4{,}1\,V$

$I_{23} = I_i = 0{,}0027\,A \quad \rightarrow \quad U_{23} = R_{23} \cdot I_{23} = 333\,\Omega \cdot 0{,}0027\,A = 0{,}9\,V$

or $\quad U_i = U_1 + U_{23} \quad \rightarrow \quad U_{23} = U_i - U_1 = 5\,V - 4{,}1\,V = 0{,}9\,V$

$U_2 = U_{23} = 0{,}9\,V \quad \rightarrow \quad I_2 = \frac{U_2}{R_2} = \frac{0{,}9\,V}{1000\,\Omega} = 0{,}0009\,A = 0{,}9\,mA$

$U_3 = U_{23} = 0{,}9\,V \quad \rightarrow \quad I_3 = \frac{U_3}{R_3} = \frac{0{,}9\,V}{500\,\Omega} = 0{,}0018\,A = 1{,}8\,mA$

or $\quad I_{23} = I_2 + I_3 \quad \rightarrow \quad I_3 = I_{23} - I_2 = 0{,}0027\,A - 0{,}0009\,A = 0{,}0018\,A = 1{,}8\,mA$

2.3.5) Calculate the sum of voltage drops in the voltage loop of the series connection. Compare the calculated sum to the value of voltage output or voltage supply U_i.

$U_1 + U_{23} = 4{,}1\,V + 0{,}9\,V = 5\,V = U_i$

2.3.6) Calculate the sum of current flows in the current node/junction of the parallel connection. Compare the calculated sum to the value of current output or current supply I_i.

$I_2 + I_3 = 0{,}0009\,A + 0{,}0018\,A = 0{,}0027\,A = I_1 = I_i$

2.4 Comparation of the results

Upload results to the *Moodle* educational platform.

Compare the results of the (theoretical) calculation, the results of the laboratory measurements and the results of the CORELA simulation.

3. CORELA education platform simulation (student 2)

Conduct a simulation of all three electrical circuits using the CORELA Education Platform.

Installation, structure, and use of the platform is presented in **CORELA Education Platform User Manual**.

3.1 Simulation of a series electrical circuit

Use the CORELA platform to determine and calculate electrical quantities of the elements in the series electrical circuit.

Instructions/steps to conduct the simulation (regarding the CORELA Education Platform User Manual) are listed below.

1. Open the CORELA platform and sign in.

2. Set-up the resistance value.

2.1 *Function list – Electronic components* – Passive – Resistor – click.

2.2 Click on the selected square in the *Function Workspace* (e.g. Resistor 1: row 1, column 1).

2.3 *Resistor.vi* pop-out window appears.

2.4 In *Resistor making* field choose the *5-bend* option.

2.5 In fields named *First band, Second bend*… choose suitable values or colors.

2.6 In *Resistor power* field select or write in value 0,25 W.

2.7 In *Output IV value* choose *Resistor value* option.

2.8 Click on *Record* button. *Resistor.vi* pop-out window disappears.

2.9 Save the value of the Resistor1: *Function list - Controls and indicators – local variables* – Write – Write-VAR-1 – click.

2.10 Click on the square to the right of the resistor square in the *Function Workspace* (e.g. row 1, column 2).

3. Repeat the same procedure to set up the value of the Resistor 2.
Resistor 2: row 2, column 1. Saved value (Write-VAR-2): row 2, column 2.

4. Repeat the same procedure to set up the value of the Resistor 3.

4.1 Resistor 3: row 3, column 1. Saved value (Write-VAR-3): row 3, column 2.

The resistor value set-up in the *Function Workspace* is shown in the Figure 3.1.

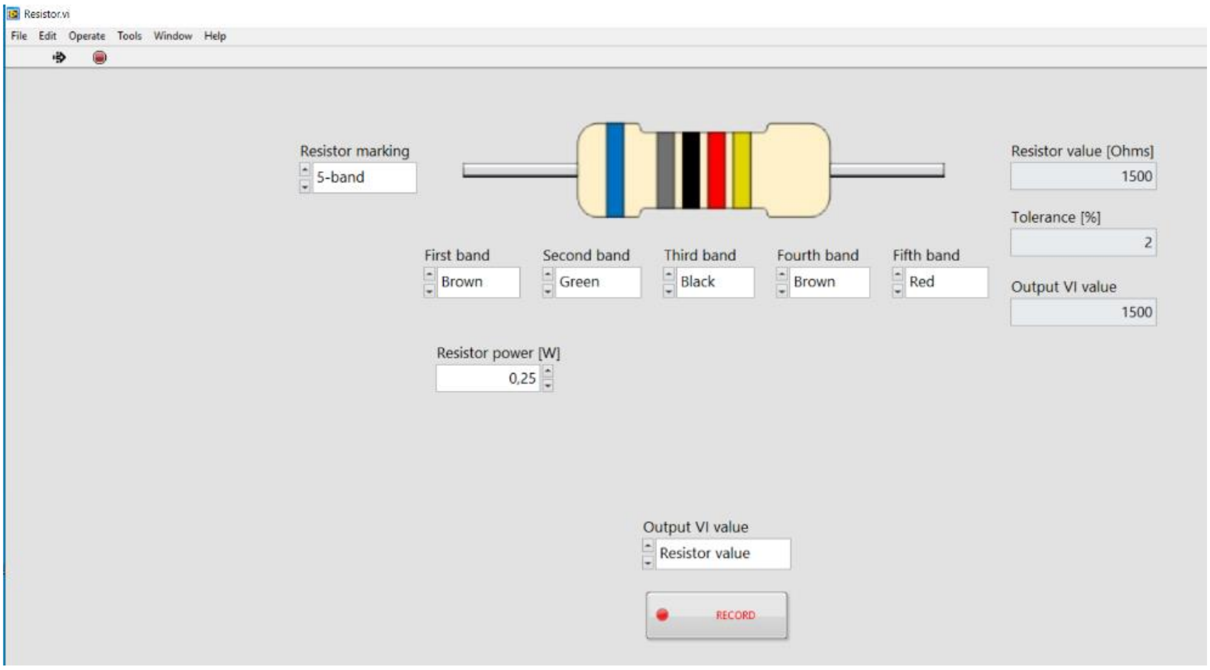


Figure 3.1: Resistor value determination

Values of electric resistors saved as local variables in the *Function Workspace* are shown in the Figure 3.2.

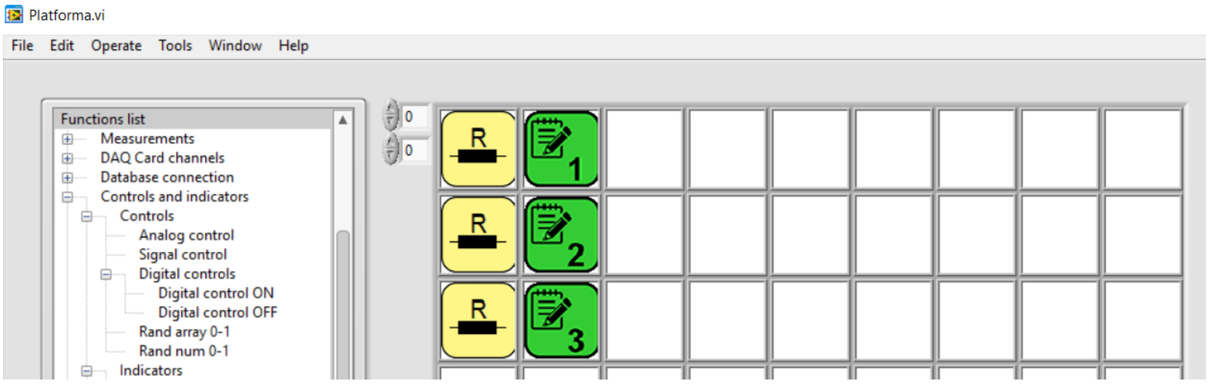


Figure 3.2: Values of electric resistors saved as local variables

5. Calculate the total electric resistance of the electric circuit.

5.1 Read the saved value of the Resistor 1: *Function list - Controls and indicators – local variables –* Read – Read-VAR-1 – click.

5.2 Click on the selected square in the *Function Workspace* (e.g. row 5, column 1).

5.3 Read the saved value of the Resistor 2: *Function list - Controls and indicators – local variables –* Read – Read-VAR-2 – click.

5.4 Click on the square below the “Read-VAR-1” square in the *Function Workspace* (e.g. row 6, column 1).

5.5 Add up the selected (read) values: *Function list - Math - Arithmetic – Add –* click.

5.6 Click on the square to the right of the “Read-VAR-1” square in the *Function Workspace* (e.g. row 5, column 2).

5.7 Read the saved value of the Resistor 3: *Function list - Controls and indicators – local variables –* Read – Read-VAR-3 – click.

5.8 Click on the square below the “Add” square in the *Function Workspace* (e.g. row 6, column 2).

5.9. Add up the previous sum (square “Add”) and the Resistor 3 value (square ”Read-VAR-3”):

Function list - Math - Arithmetic – Add – click.

5.10 Click on the square to the right of the “Add” square in the *Function Workspace* (e.g. row 5, column 3).

5.11 Save the value of the total sum: *Function list - Controls and indicators – local variables –* Write – Write-VAR-4 – click.

5.12 Click on the square to the right of the second “Add” square in the *Function Workspace* (e.g. row 5, column 4).

The total resistance calculation in the *Function Workspace* is shown in the Figure 3.3.

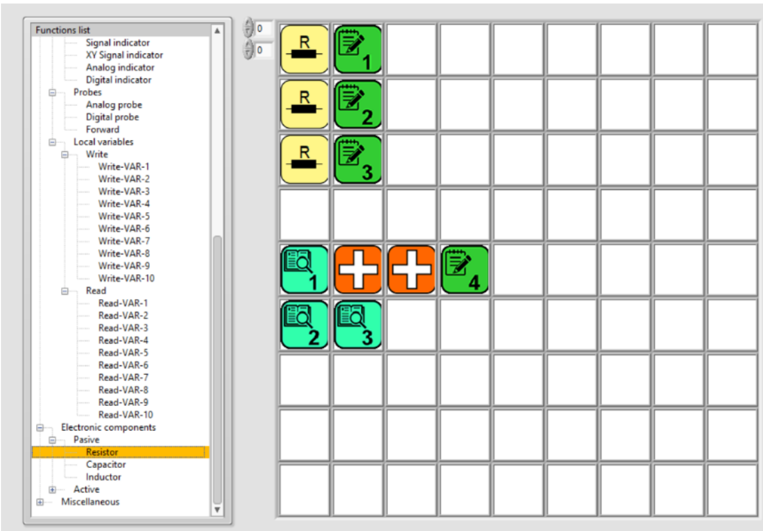


Figure 3.3: Determination of total electric resistance RN of a series electrical circuit

6. Set-up the value of the voltage supply and save it.

6.1 Function list -Controls and indicators – Controls – Analog control – click.

6.2 Click on the selected square in the *Function Workspace* (e.g. row 8, column 1).

6.3 *Analog control.avi* pop-out window appears.

6.4 Write in or select value “5”.

6.5 Click on *Record* button. *Analog control.avi* pop-out window disappears.

6.6 Save the value of the voltage supply: *Function list - Controls and indicators – local variables –* Write – Write-VAR-5 – click.

6.7 Click on the square to the right of the voltage value square in the *Function Workspace* (e.g. row 8, column 2).

The voltage supply set-up in the *Function Workspace* is shown in the Figure 3.4.

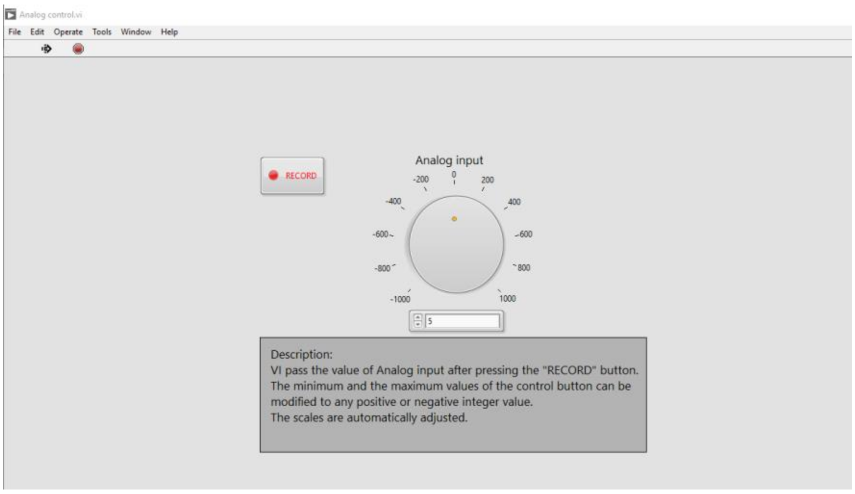


Figure 3.4: Voltage source value determination

Value of voltage source saved as local variables in the *Function Workspace* is shown in the Figure 3.5

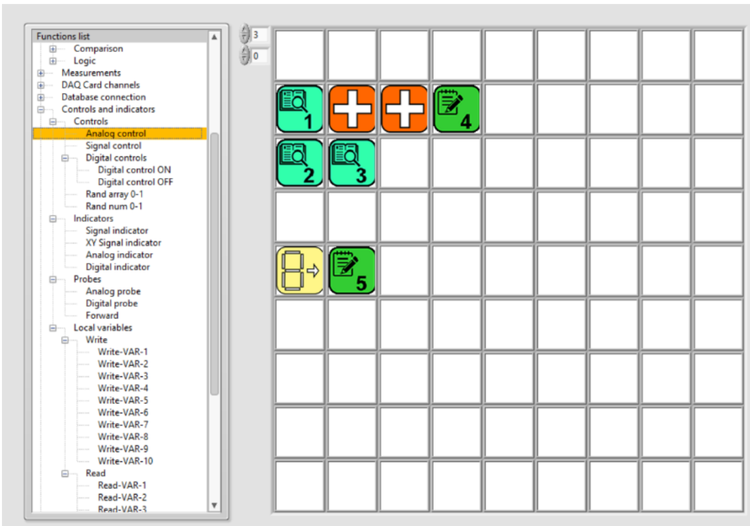


Figure 3.5: Electric voltage value saved as a local variable

7. Calculate the current supply: voltage supply value divided with total resistance value.

7.1 Read the saved value of the voltage supply: *Function list - Controls and indicators – local variables –* Read – Read-VAR-5 – click.

7.2 Click on the selected square in the *Function Workspace* (e.g. row 8, column 4).

7.3 Read the saved value of the total resistance: *Function list - Controls and indicators – local variables* – Read – Read-VAR-4 – click.

7.4 Click on the square below the voltage value or "Read-VAR-5" square in the *Function Workspace* (e.g. row 9, column 4).

7.5 Divide the selected (read) values: *Function list - Math - Arithmetic – Divide* – click.

7.6 Click on the square to the right of voltage value or "Read-VAR-5" square in the *Function Workspace* (e.g. row 8, column 5).

7.7 Save the value of the current supply: *Function list - Controls and indicators – local variables* – Read – Read-VAR-6 – click.

7.8 Click on the square to the right of the "Divide" square in the *Function Workspace* (e.g. row 8, column 6).

The current supply calculation in the *Function Workspace* is shown in the Figure 3.6.

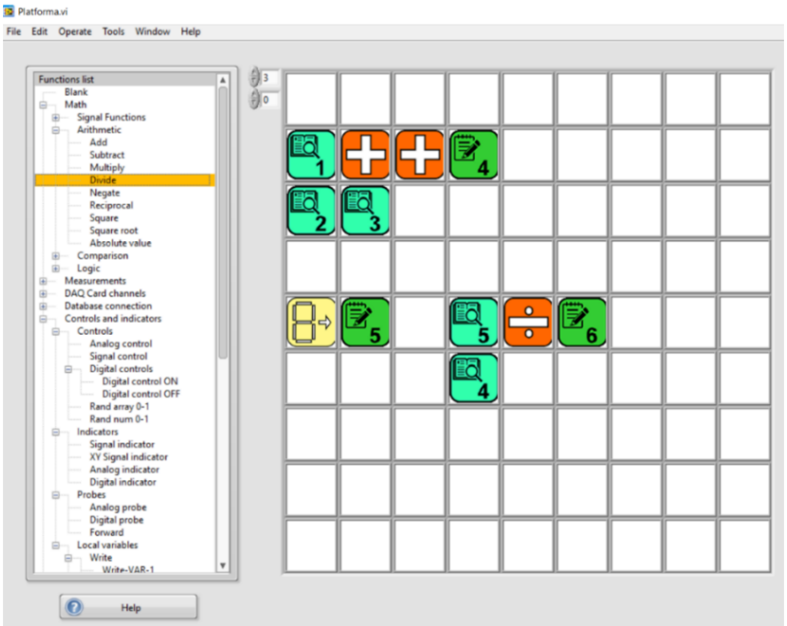


Figure 3.6: Calculation of electric current value

8. Calculate the voltage drop on the Resistor 1.

8.1 Read the saved value of the Resistor 1: *Function list - Controls and indicators – local variables* – Read – Read-VAR-1 – click.

8.2 Click on the selected square in the *Function Workspace* (e.g. row 11, column 1).

8.3 Read the saved value of the current supply: *Function list - Controls and indicators – local variables* – Read – Read-VAR-6 – click.

8.4 Click on the square below the "Read-VAR-1" square in the *Function Workspace* (e.g. row 12, column 1).

Funded by the Erasmus+ Programme of the European Union

8.5 Multiply the selected (read) values: *Function list - Math - Arithmetic – Multiply* – click.

8.6 Click on the square to the right of the "Read-VAR-1" square in the *Function Workspace* (e.g. row 11, column 2).

8.7 Save the value of the voltage drop 1: *Function list - Controls and indicators – local variables* – Write – Write -VAR-7 – click.

8.8 Click on the square to the right of the "Multiply" square in the *Function Workspace* (e.g. row 11, column 3).

9. Calculate the voltage drop on the Resistor 2.

9.1 Read the saved value of the Resistor 2: *Function list - Controls and indicators – local variables* – Read – Read-VAR-2 – click.

9.2 Click on the selected square in the *Function Workspace* (e.g. row 11, column 5).

9.3 Read the saved value of the current supply: *Function list - Controls and indicators – local variables* – Read – Read-VAR-6 – click.

9.4 Click on the square below the "Read-VAR-2" square in the *Function Workspace* (e.g. row 12, column 5).

9.5 Multiply the selected (read) values: *Function list - Math - Arithmetic – Multiply* – click.

9.6 Click on the square to the right of the "Read-VAR-1" square in the *Function Workspace* (e.g. row 11, column 6).

9.7 Save the value of the voltage drop 2: *Function list - Controls and indicators – local variables* – Write – Write -VAR-8 – click.

9.8 Click on the square to the right of the "Multiply" square in the *Function Workspace* (e.g. row 11, column 7).

10. Calculate the voltage drop on the Resistor 3.

10.1 Read the saved value of the Resistor 3: *Function list - Controls and indicators – local variables* – Read – Read-VAR-3 – click.

10.2 Click on the selected square in the *Function Workspace* (e.g. row 11, column 9).

10.3 Read the saved value of the current supply: *Function list - Controls and indicators – local variables* – Read – Read-VAR-6 – click.

10.4 Click on the square below the "Read-VAR-3" square in the *Function Workspace* (e.g. row 12, column 9).

10.5 Multiply the selected (read) values: *Function list - Math - Arithmetic – Multiply* – click.

10.6 Click on the square to the right of the "Read-VAR-3" square in the *Function Workspace* (e.g. row 11, column 10).

10.7 Save the value of the voltage drop 3: *Function list - Controls and indicators – local variables –* Write – Write -VAR-9 – click.

10.8 Click on the square to the right of the “Multiply” square in the *Function Workspace* (e.g. row 11, column 11).

The voltage drops calculation in the *Function Workspace* is shown in the Figure 3.7.

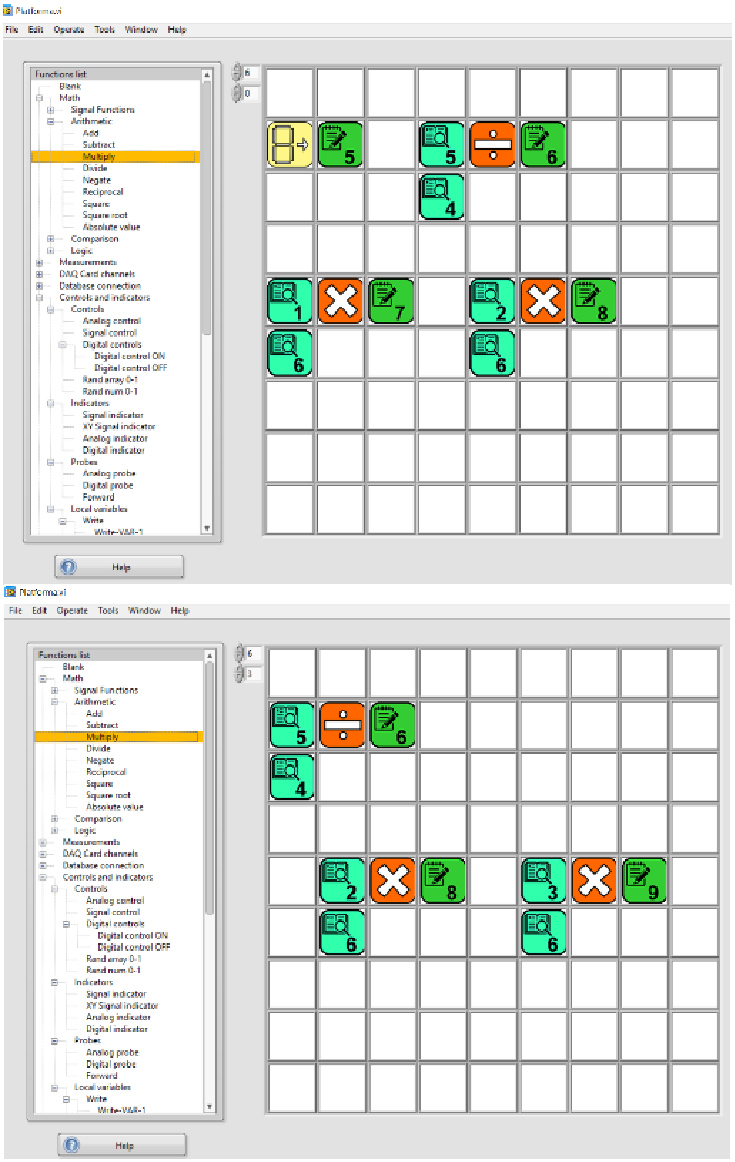


Figure 3.7: Determination of voltage drops values

11. Calculate the sum of the voltage drops of the resistors.

11.1 Read the saved value of the voltage drop 1: *Function list - Controls and indicators – local variables* – Read – Read-VAR-7 – click.

11.2 Click on the selected square in the *Function Workspace* (e.g. row 14, column 1).

11.3 Read the saved value of the voltage drop 2: *Function list - Controls and indicators – local variables* – Read – Read-VAR-8 – click.

11.4 Click on the square below the voltage drop 1 or “Read-VAR-7” square in the *Function Workspace* (e.g. row 15, column 1).

11.5 Add up the selected (read) values: *Function list - Math - Arithmetic – Add* – click.

11.6 Click on the square to the right of the voltage drop 1 or “Read-VAR-7” square in the *Function Workspace* (e.g. row 14, column 2).

11.7 Read the saved value of the voltage drop 3: *Function list - Controls and indicators – local variables* – Read – Read-VAR-8 – click.

11.8 Click on the square below the “Add” square in the *Function Workspace* (e.g. row 15, column 2).

11.9 Add up the previous sum (square “Add”) and the voltage drop 3 value (square ”Read-VAR-9”): *Function list - Math - Arithmetic – Add* – click.

11.10 Click on the square to the right of the second “Add” square in the *Function Workspace* (e.g. row 14, column 3).

11.11 Save the value of the total sum: *Function list - Controls and indicators – local variables* – Write – Write-VAR-10 – click.

11.12 Click on the square to the right of the second “Add” square in the *Function Workspace* (e.g. row 14, column 4).

The sum of the voltage drops of resistors in the *Function Workspace* is shown in the Figure 3.8.

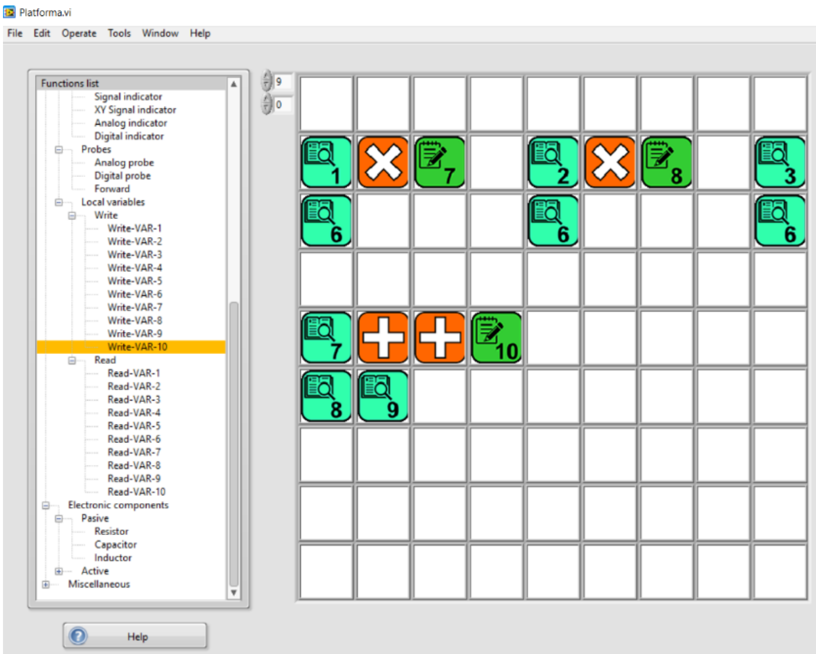


Figure 3.8: Calculation of sum of the voltage drops

12. Compare the value of the voltage supply and the value of the calculated sum of the voltage drops across resistors.

12.1 Explain the difference in compared values.

3.2 Simulation of a parallel electrical circuit

Use the CORELA platform to calculate electrical quantities of the elements in the parallel electrical circuit.

Instructions/steps to conduct the simulation (regarding the CORELA Education Platform User Manual) are similar to instructions in chapter 3.1.

3.3 Simulation of a series-parallel combination electrical circuit

Use the CORELA platform to calculate electrical quantities of the elements in the serious-parallel combination electrical circuit.

Instructions/steps to conduct the simulation (regarding the CORELA Education Platform User Manual) are similar to instructions in chapter 3.1.

3.4 Comparation of the results

The results of the COREALA simulation are saved and uploaded to the *Moodle* educational platform.

Compare the results of the (theoretical) calculation, the results of the laboratory measurements and the results of the CORELA simulation.

4. Measurements of electrical quantities (student 3)

4.1 Measurement of electric resistance

Measure the electric resistance of each resistor with **multimeter - ohmmeter**; use the **direct method**. Fill in the Table 4.1 with the measured values.
Note: Adjust the switch of multimeter to the appropriate quantity and measurement range.

Is the measured electric resistance of each resistor within the tolerance limits (between the minimum and maximum value) based on the *Resistor Color Code* (Table 1)?

Table 4.1: Electric resistance measurements

element	measured resistance R_m [Ω] (direct measurement)	minimum rersistance R_{min} [Ω]	maximum rersistance R_{max} [Ω]	adequacy of the measured resistance value (YES / NO)
Resistor 1		1470 Ω	1530 Ω	
Resistor 2		980 Ω	1020 Ω	
Resistor 3		490 Ω	510 Ω	

Figure 4.1.1: Multimeter

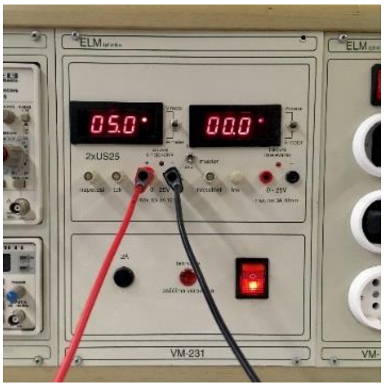


Figure 4.1.2: Voltage source

Controls are intended as inputs for calculation functions and have one output where produces constant or array variables. Variables are analog or digital data. The control group of functions consists also of random generators.

4.2 Resistors in a series electrical circuit

4.2.1) The three resistors are connected in series to a direct voltage source. The series electrical circuit scheme/diagram is shown in Figure 4.2.

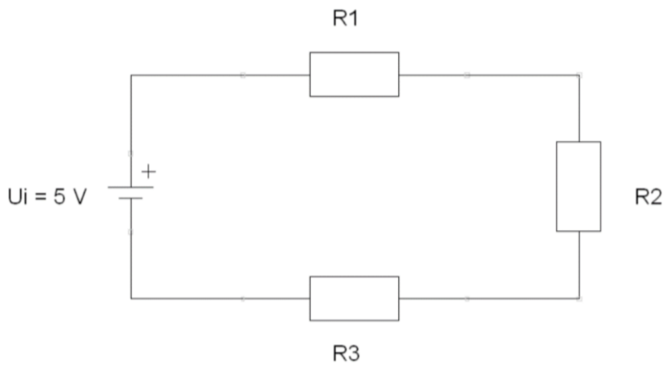


Figure 4.2: Resistors in series electrical circuit

4.2.2) Build the series electrical circuit on the *protoboard*. Measure the total electric resistance of the circuit.

Note: Adjust the switch of multimeter to the appropriate quantity and measurement range.

R_{N-m} = _________________ Ω

Is the measured total electric resistance R_{N-m} within the tolerance limits (between the minimum calculated value R_{Nmin} = 2940 Ω and maximum calculated value R_{Nmax} = 3060 Ω)?

Yes, the measured total electric resistance is within the tolerance limits, that is between 2940 Ω and 3060 Ω.

4.2.3) Prepare the voltage source; set up direct voltage output U_i = 5 V DC. Use the multimeter – voltmeter to measure the value of voltage output.

Note: Adjust the switch to the appropriate quantity type (AC / DC) and measurement range.

$U_{i\text{-}m}$ = ________________ V

4.2.4) Connect the series electrical circuit to the direct voltage supply U_i = 5 V DC. Use the multimeter – amperemeter to measure the value of current supply.

Note: Adjust the switch of multimeter to the appropriate quantity type (AC / DC) and measurement range.

$I_{i\text{-}m}$ = ________________ A = ________________ mA

4.2.5) Conduct the measurements of electrical quantities in the electrical circuit. Fill in the Table 4.2; write down the measured values of each resistor.

Note: Adjust the switch of multimeter to the appropriate quantity type (AC / DC) and measurement range.

4.2.5.1) Measure the voltage drop of each resistor in the electrical circuit.

4.2.5.2) Measure the current flow through each resistor in the electrical circuit.

4.2.5.3) Determine (calculate) measured resistance of each resistor by indirect *U-I* method (based on measured voltage drop and measured current flow).

Table 4.2: Measurements of electrical quantities in series electrical circuit
** Copy the measured values from Table 4.1*

element	electric current I_m [mA]	electric voltage U_m [V]	electric resistance R_m [Ω] indirect measurement	electric resistance R_m [Ω] direct measurement*
Resistor 1				
Resistor 2				
Resistor 3				

$$R_{1m} = \frac{U_{1m}}{I_{1m}} = - = \_\_\_\_\_\_ \; \Omega$$

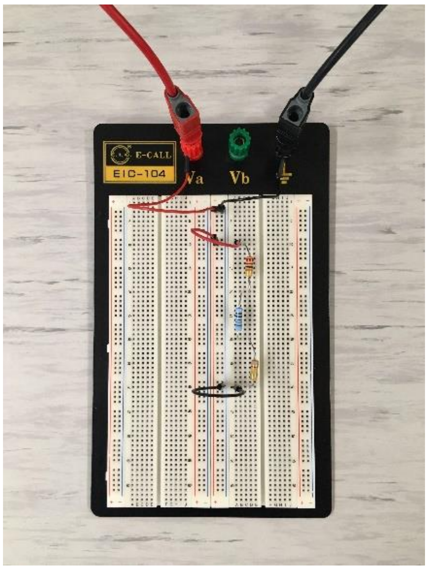
$$R_{2m} = \frac{U_{2m}}{I_{2m}} = - = \_\_\_\_\_\_ \; \Omega$$

$$R_{3m} = \frac{U_{3m}}{I_{3m}} = - = \_\_\_\_\_\_ \; \Omega$$

4.2.6) Calculate the sum of the measured voltage drops across resistors. Compare the calculated sum to the measured voltage output or voltage supply $U_{i\text{-}m}$.

$$U_{1m} + U_{2m} + U_{3m} = \_\_\_\_\_\_ \, V + \_\_\_\_\_\_ \, V + \_\_\_\_\_\_ \, V = \_\_\_\_\_\_ \, V \cong U_{im}$$

The series electrical circuit on protoboard is shown in Figure 4.3.



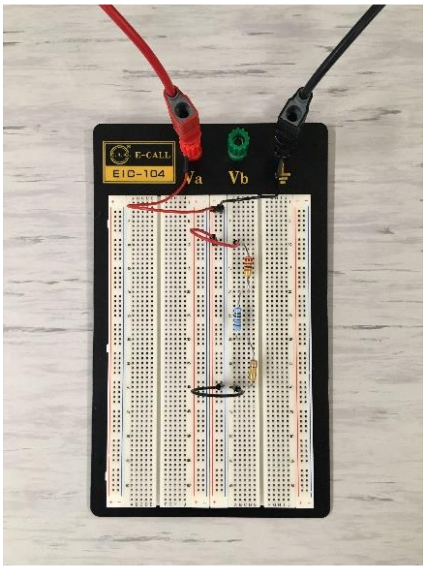


Figure 4.3: The series electrical circuit on protoboard

4.3 Resistors in a parallel electrical circuit

4.3.1) The three resistors are connected in parallel to a direct voltage source. The parallel electrical circuit scheme/diagram is shown in the Figure 4.4.

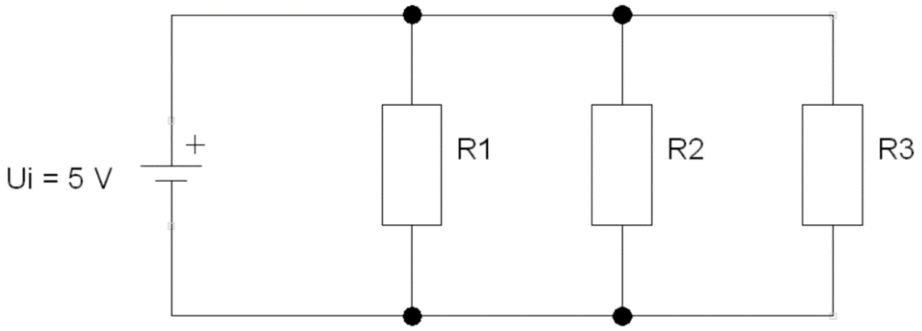


Figure 4.4: Resistors in parallel electrical circuit

4.3.2) Build the parallel electrical circuit on the *protoboard*. Measure the total electric resistance of the circuit.

Note: Adjust the switch of multimeter to the appropriate quantity and measurement range.

$R_{N\text{-}m}$ = ________________ Ω

Is the measured total electric resistance $R_{N\text{-}m}$ within the tolerance limits (between the minimum calculated value R_{Nmin} = 267 Ω and maximum calculated value R_{Nmax} = 278 Ω)?

Yes, the measured total electric resistance is within the tolerance limits, that is between 267 Ω and 278 Ω.

4.3.3) Prepare the voltage source; set up direct voltage output U_i = 5 V DC. Use the multimeter – voltmeter to measure the value of voltage output.

Note: Adjust the switch to the appropriate quantity type (AC / DC) and measurement range.

$U_{i\text{-}m}$ = ________________ V

4.3.4) Connect the parallel electrical circuit to the direct voltage supply U_i = 5 V DC. Use the multimeter – amperemeter to measure the value of current supply.

Note: Adjust the switch of multimeter to the appropriate quantity type (AC / DC) and measurement range.

$I_{i\text{-}m}$ = ________________ A = ________________ mA

4.3.5) Conduct the measurements of electrical quantities in the electrical circuit. Fill in the Table 4.3; write down the measured values of each resistor.

Note: Adjust the switch of multimeter to the appropriate quantity type (AC / DC) and measurement range.

4.3.5.1) Measure the voltage drop of each resistor in the electrical circuit.

4.3.5.2) Measure the current flow through each resistor in the electrical circuit.

4.3.5.3) Determine (calculate) measured resistance of each resistor by indirect *U-I* method (based on measured voltage drop and measured current flow).

Table 4.3: Measurements of electrical quantities in parallel electrical circuit
** Copy the measured values from Table 4.1*

element	electric current I_m [mA]	electric voltage U_m [V]	electric resistance R_m [Ω] indirect measurement	electric resistance R_m [Ω] direct measurement*
Resistor 1				
Resistor 2				
Resistor 3				

$$R_{1m} = \frac{U_{1m}}{I_{1m}} = - = \_\_\_\_\_\_ \ \Omega$$

$$R_{2m} = \frac{U_{2m}}{I_{2m}} = - = \_\_\_\_\_\_ \ \Omega$$

$$R_{3m} = \frac{U_{3m}}{I_{3m}} = - = \_\_\_\_\_\_ \ \Omega$$

4.3.6) Calculate the sum of the measured current flows through resistors. Compare the calculated sum to the measured current output or current supply $I_{i\text{-}m}$.

$$I_{1m} + I_{2m} + I_{3m} = \_\_\_\_\_\_ \ mA + \_\_\_\_\_\_ \ mA + \_\_\_\_\_\_ \ mA = \_\_\_\_\_\_ \ mA \ \triangleq \ I_{im}$$

The parallel electrical circuit on protoboard is shown in the Figure 4.5.

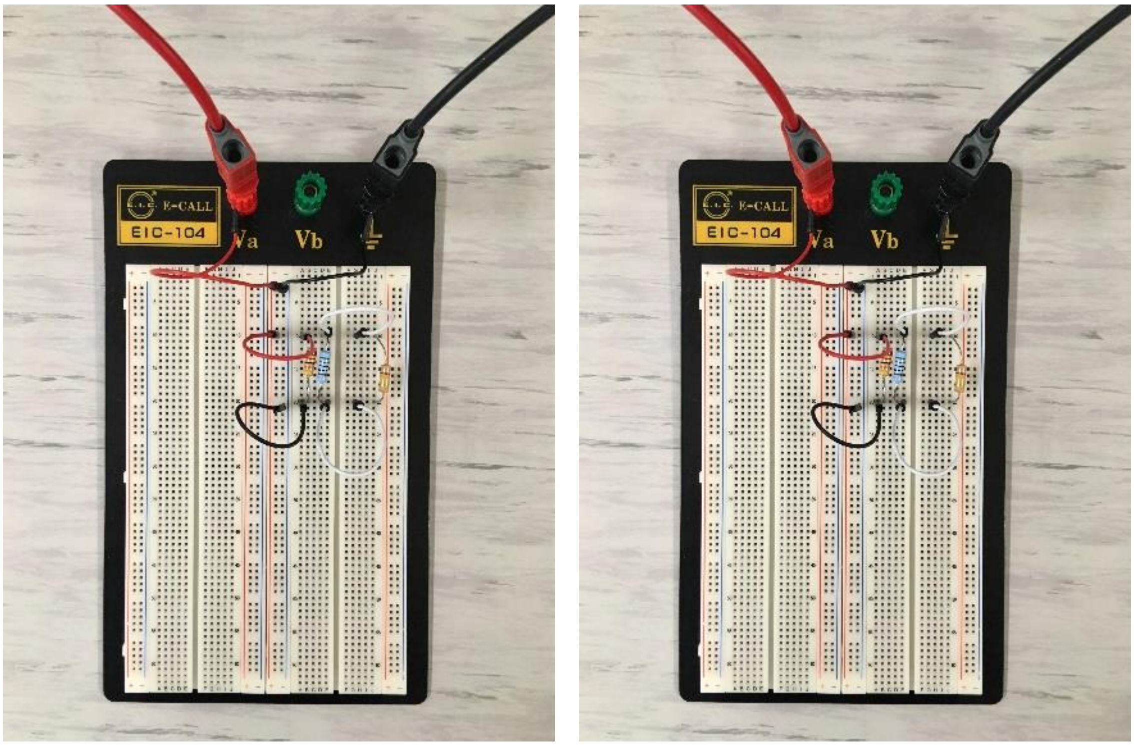


Figure 4.5: The parallel electrical circuit on protoboard

4.4 Resistors in a series-parallel combination electrical circuit

4.4.1) The three resistors are connected in series-parallel combination to a direct voltage source. The series-parallel electrical circuit scheme/diagram is shown in the Figure 4.6.

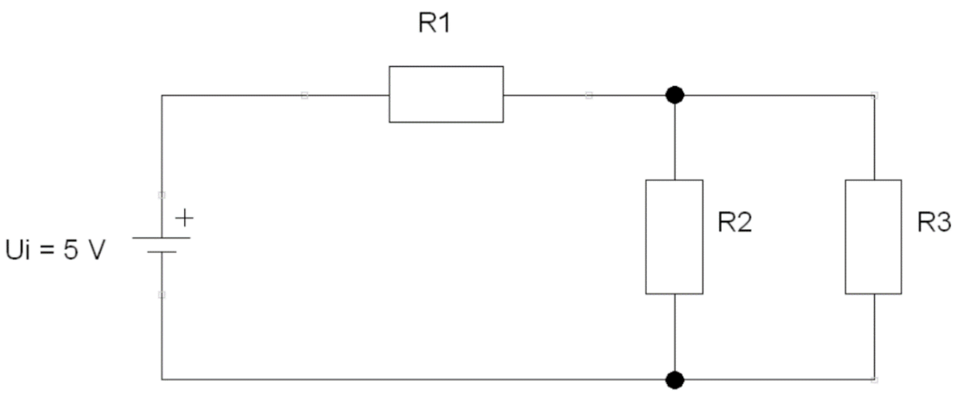


Figure 4.6: Resistors in series-parallel electrical circuit

4.4.2) Build the series-parallel combination electrical circuit on the *protoboard*. Measure the total electric resistance of the circuit.
Note: Adjust the switch of multimeter to the appropriate quantity and measurement range.

$R_{N\text{-}m}$ = ________________ Ω

Is the measured total electric resistance $R_{N\text{-}m}$ within the tolerance limits (between the minimum calculated value R_{Nmin} = 1797 Ω and maximum calculated value R_{Nmax} = 1870 Ω)?

Yes, the measured total electric resistance is within the tolerance limits, that is between 1797 Ω and 1870 Ω.

4.4.3) Prepare the voltage source; set up direct voltage output U_i = 5 V DC. Use the multimeter – voltmeter to measure the value of voltage output.

Note: Adjust the switch to the appropriate quantity type (AC / DC) and measurement range.

$U_{i\text{-}m}$ = ________________ V

4.4.4) Connect the series-parallel combination electrical circuit to the direct voltage supply U_i = 5 V DC. Use the multimeter – amperemeter to measure the value of current supply.
Note: Adjust the switch of multimeter to the appropriate quantity type (AC / DC) and measurement range.

$I_{i\text{-}m}$ = ________________ A = ________________ mA

4.4.5) Conduct the measurements of electrical quantities in the electrical circuit. Fill in the Table 4.4; write down the measured values of each resistor.

Note: Adjust the switch of multimeter to the appropriate quantity type (AC / DC) and measurement range.

4.4.5.1) Measure the voltage drop of each resistor in the electrical circuit.

4.4.5.2) Measure the current flow through each resistor in the electrical circuit.

4.4.5.3) Determine (calculate) measured resistance of each resistor by indirect *U-I* method (based on measured voltage drop and measured current flow).

Table 4.4: Measurements of electrical quantities in series-parallel combination electrical circuit
** Copy the measured values from Table 4.1*

element	electric current I_m [mA]	electric voltage U_m [V]	electric resistance R_m [Ω] indirect measurement	electric resistance R_m [Ω] direct measurement*
Resistor 1				
Resistor 2				
Resistor 3				

$$R_{1m} = \frac{U_{1m}}{I_{1m}} = - = \_\_\_\_\_\_ \ \Omega$$

$$R_{2m} = \frac{U_{2m}}{I_{2m}} = - = \_\_\_\_\_\_ \ \Omega$$

$$R_{3m} = \frac{U_{3m}}{I_{3m}} = - = \_\_\_\_\_\_ \ \Omega$$

4.4.6) Calculate the sum of the measured voltage drops in the voltage loop of the series connection. Compare the calculated sum to the measured voltage output or voltage supply $U_{i\text{-}m}$.

Funded by the
Erasmus+ Programme
of the European Union

$U_{1m} + U_{23m} = \_\_\_\_\_\_ V + \_\_\_\_\_\_ V = \_\_\_\_\_\_ V \cong U_{im}$

4.4.7) Calculate the sum of the measured current flows in the current node/junction of the parallel connection. Compare the calculated sum to the measured current output or current supply $I_{i\text{-}m}$.

$I_{2m} + I_{3m} = \_\_\_\_\_\_ mA + \_\_\_\_\_\_ mA = \_\_\_\_\_\_ mA \cong I_{1m} \cong I_{im}$

The series-parallel electrical circuit on protoboard is shown in the Figure 4.7.

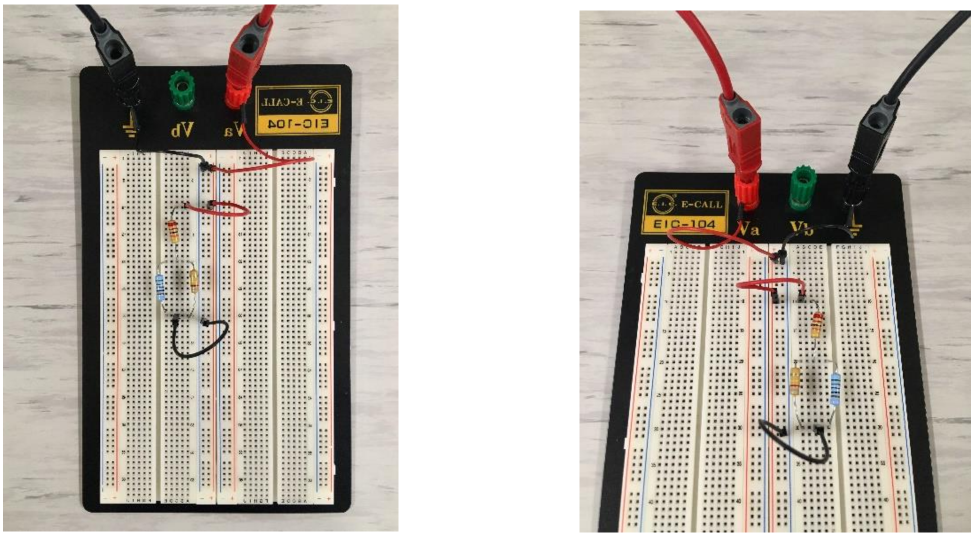


Figure 4.7: The series-parallel electrical circuit on protoboard

4.5 Comparation of the results

Upload results to the *Moodle* educational platform.

Compare the results of the (theoretical) calculation, the results of the laboratory measurements and the results of the CORELA simulation.

Video material:

https://www.youtube.com/watch?v=8P3Clw4ZEHk&feature=youtu.be

Funded by the Erasmus+ Programme of the European Union

III Impedance measurement

1. Introduction

The alternating excitation voltage source in electric circuit generates an alternating current flow. That current is time-varying and alternate both in intensity and direction (Figure 1.1).

In electrical engineering, the usual waveform of alternating current is a sine wave.

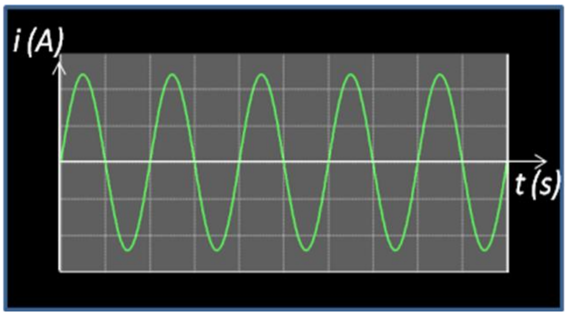
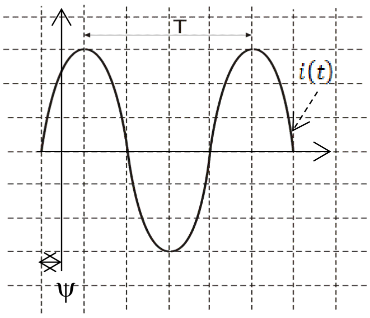


Figure 1.1: Sine wave alternating current

Alternating quantities can be presented in three ways (Figures 1.2, 1.3 and 1.4)



$$i(t) = I_m \cdot \sin(\omega t + \psi)\,[A]$$

Figure.1.2
Temporal (trigonometric) representation

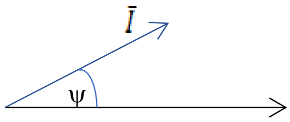
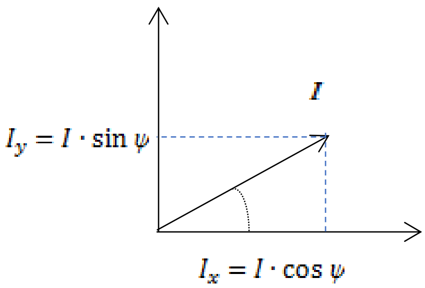


Figure.1.3
Phasoric (geometric) representation



$$I = I \cdot \cos\psi + jI \cdot \sin\psi$$

Figure1.4
Complex (arithmetic) representation

The values of a sine wave at a given point of time are determined by the following relations:

$i(t) = I_m \cdot \sin(\omega t + \psi)\,[A]$

$u(t) = U_m \cdot \sin(\omega t + \theta)\,[V]$

I_m, U_m- amplitude or maximum value of alternating current, i.e. voltage;

ω – angular frequency of alternating variables ($\omega = 2\pi f\ \left[\frac{rad}{s}\right]$);

f - frequency of alternating variables $[Hz]$;

ψ, θ - phase of alternating variables at the moment t=0 $[rad]$;

φ - phase difference between voltage and current ($\varphi = \theta - \psi$)$[rad]$.

Funded by the Erasmus+ Programme of the European Union

Ammeters and voltmeters for measuring alternating current, i.e. voltage, measure their effective value.

$$I = \frac{I_m}{\sqrt{2}} = 0.707 \cdot I_m [A]$$

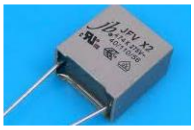
$$U = \frac{U_m}{\sqrt{2}} = 0.707 \cdot U_m [V]$$

I , U - effective value of alternating current, ie voltage.

Elements in alternating current circuits

Thermogenic (ohmic) resistor

Inductor (coil)



Capacitor

The impedance calculation in an AC circuit depends on type and number of elements, and on the way, they are interconnected.

The impedance can be also, determined in a practical manner by measuring the effective voltage and current values using the UI method:

$$Z = \frac{U}{I} [\Omega]$$

Z - impedance module, i.e. the complete resistance of the circuit;

U, I - effective values of voltage and current in the circuit.

2. Theoretical tasks (all students)

Determine the impedance for different frequencies of the AC power source, in Figure 2, when connected to a circuit consisting of:

a) Ohmic resistor with resistance R = 470Ω (Figure 2a)

b) Inductor with inductance L = 1 mH (Figure 2b)

c) Capacitor with capacitance C = 0,47 μF (Figure 2c)

d) A serial connection of all three elements (Figure 2d)

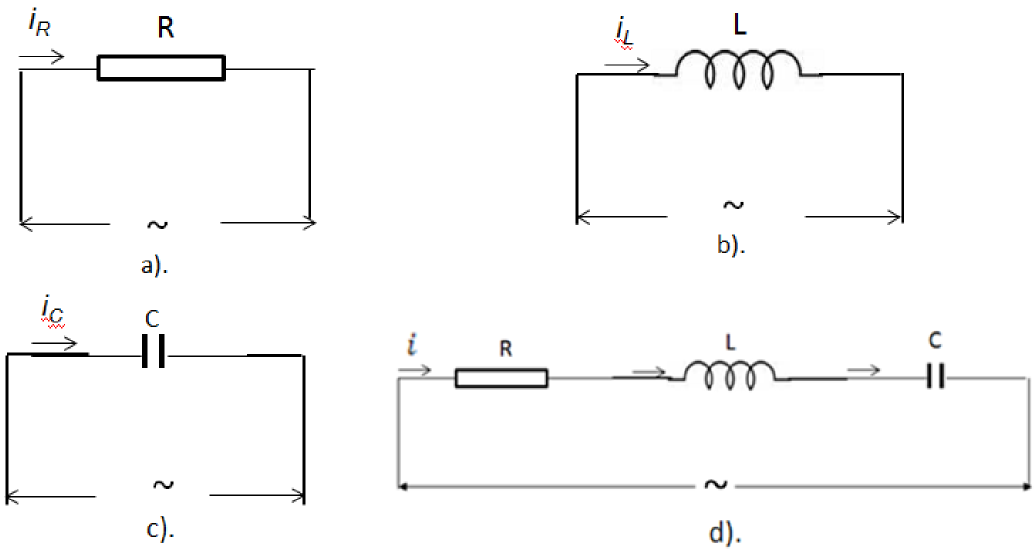


Figure 2: Electrical circuits

3. Calculating impedance (student 1)

3.1 Calculations

The impedance is calculated for each electrical circuit in Figure 2 and different frequencies for the AC voltage.

a. An alternating current circuit with an ohmic resistor is shown in Figure 2.a. The impedance in the circuit is calculated by the formula:

$Z = R[\Omega]$

The calculated values are written in Table 3a.

b. An alternating current circuit with a coil is shown in Figure 2.b. The impedance in the circuit is calculated by the formula:

$Z = X_L[\Omega] \qquad X_L = \omega \cdot L\ [\Omega], \quad \omega = 2\pi f\ \left[\frac{rad}{s}\right]$

where X_L is the inductive resistance of the coil $[\Omega]$

The calculated values are written in Table 3b.

c. An alternating current circuit with a capacitor is shown in Figure 2.c. The impedance in the circuit is calculated by the formula:

$Z = X_c[\Omega]$

where X_c is the capacitive resistance of the capacitor $[\Omega]$

$X_C = \frac{1}{\omega C}\ [\Omega]$

The calculated values are written in Table 3c.

d. An alternating current circuit with a resistor, coil and capacitor connection (serial RLC circuit) is shown in Figure 2.d. The impedance in the circuit is calculated by the formula:

$$Z = \sqrt{R^2 + X^2} = \sqrt{R^2 + (X_L - X_C)^2}\ [\Omega]$$

R – active (ohmic) resistance in the circuit $[\Omega]$

X – reactive resistance in the circuit

$$X = X_L - X_C\ [\Omega]$$

The calculated values are written in Table 3d.

Determine the character of the serial RLC electrical circuit, if it is known that:

for $X_L > X_C$ the circuit has an inductive character;

for $X_L < X_C$ the circuit has a capacitive character;

for $X_L = X_C$ the circuit has an ohmic character.

Write down the calculated values in a table

Table 3a: Electrical circuit with ohmic resistance

f(Hz)	100	1000	10000	100000
Z(Ω)				

Table 3b: Electrical circuit with inductor

f(Hz)	5000	10000	50000	100000
Z(Ω)				

Table 3c: Electrical circuit with capacitor

f(Hz)	100	200	500	1000
Z(Ω)				

Table 3d: Serial RLC circuit

f(Hz)	100	1000	10000	100000
Z(Ω)				

Question 1: *How does the capacitive resistance change by increasing the frequency?*

Question 2: *How does the inductive resistance change by increasing the frequency?*

Question 3: *What character has a serial RLC circuit from the task?*

3.2 Comparation of the results

Sign in to the CORELA platform to enter the values.

The values of frequencies and impedances are entered and sent to the educational platform according to the procedure explained in point 6 of this manual.

Compare the obtained impedance data with the data obtained by student 1 and student 3.

4. Determination of impedance by simulation (student 2)

4.1 Activation of the exercise

After logging in to the CORELA platform, select the RLC impedance exercise from the list of functions in Miscellaneous, and left-click on it to place it in a desktop field, as shown in Figure 4.1.

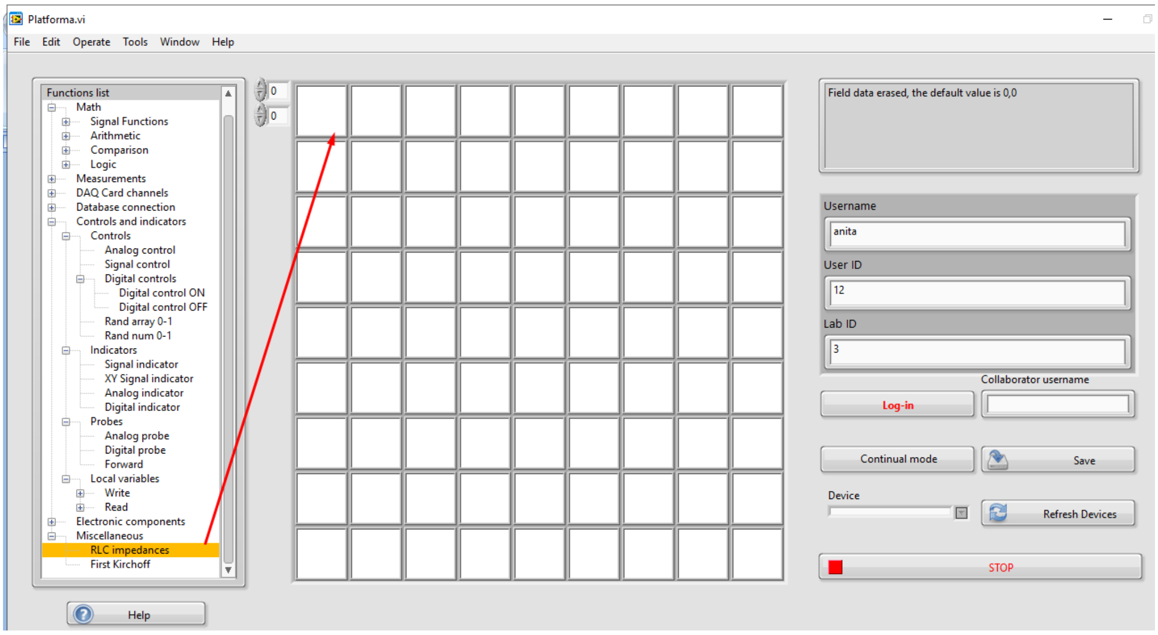


Figure 4.1: Setting exercise on desktop

A pop-up window with the RLC impedance exercise is displayed on the screen, as in Figure 4.2.

With the support of the Erasmus+ Programme of the European Union

4.2 Selecting the type of electrical circuit and reading the results

The type of electrical circuit is selected from the “Connection” field.

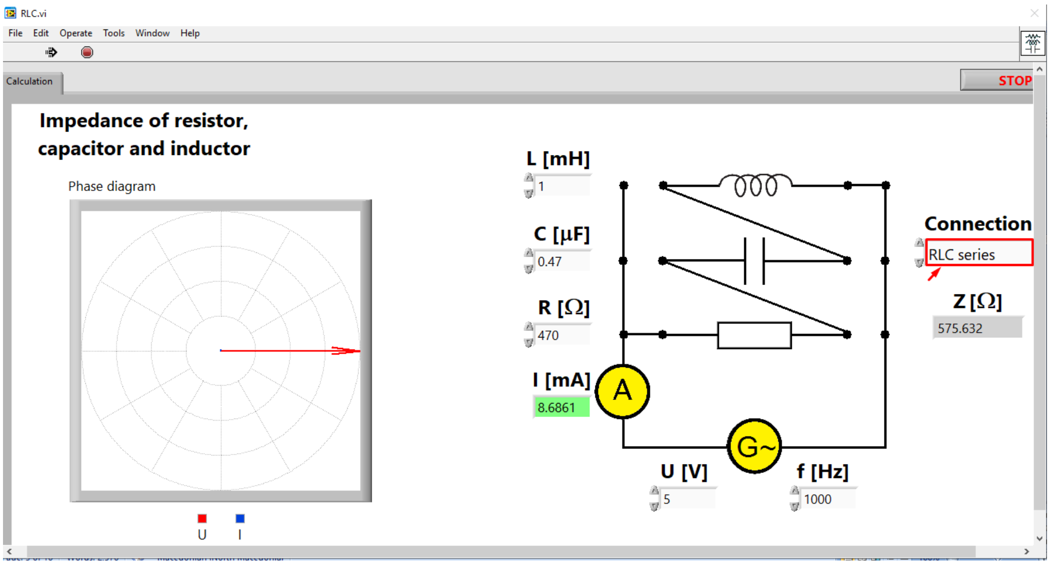


Figure 4.2: Selection of type of electrical circuit

The simulation of the measurements is performed separately for each electrical circuit, according the order in Figure 2.

After selecting the required electrical circuit, the effective value of an alternating voltage *U* = 5 V, frequency f, resistance of resistor R, inductance of coil L and capacitance of capacitor C are entered in the appropriate fields of the work surface, according to the characteristics of elements specified in the task.

For each electrical circuit, several measurements are made for different frequency values, given in the corresponding tables. The current and impedance values are changing by the change of the frequency and are read and entered in the corresponding tables (2a, 2b, 2c and 2d).

The phase difference φ, between voltage and current, can be seen from the phase diagram drawn on the left side of the screen (Figure 4.3).

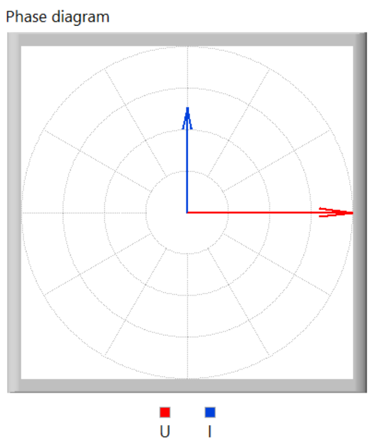


Figure 4.3: Phase diagram of an electrical circuit with a capacitor

In the space provided, on the right side of the tables, write the phase difference (φ) between the voltage and current for a circuit with an ideal coil (Figure 2b) and for a circuit with an ideal capacitor (Figure 2c).

Table 4a: Electrical circuit with ohmic resistance

f(Hz)	100	1000	10000	100000
I(A)				
Z(Ω)				

Table 4b: Electrical circuit with inductor

f(Hz)	5000	10000	50000	100000
I(A)				
Z(Ω)				

φ = -----------

Table 4c: Electrical circuit with capacitor

f(Hz)	100	200	500	1000
I(A)				
Z(Ω)				

φ = -----------

Table 4d: Serial RLC circuit

f(Hz)	100	1000	10000	100000
I(A)				
Z(Ω)				

After reading and writing the measured values for I and Z, for all four electrical circuits, the pop-up window is closed by pressing the "STOP" button and the exercise icon is displayed on the desktop (Figure 4.4).

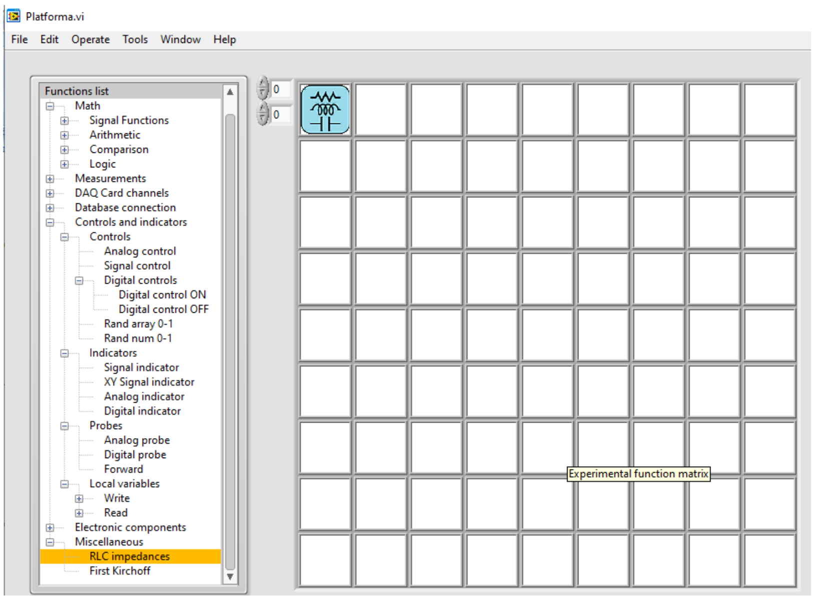


Figure 4.4: Desktop after simulation measurement

Question 1: *What is the phase difference between voltage and current in an electrical circuit with an ideal coil (Figure 2b)?*

Question 2: *What is the phase difference between the voltage and current in an electrical circuit with an ideal capacitor (Figure 2c)?*

Question 3: *How does the value of the impedance change with the change of the frequency in the electrical circuit in figure 2b?*

Question 4: *How does the value of the impedance change with the change of the frequency in the electrical circuit in figure 2c?*

4.3 Comparation of the results

The values for the frequencies and the values of the impedance, for all four electrical circuits of the exercise, are entered in the CORELA platform and sent to the educational platform according to the procedure explained in point 6 of this manual.

Compare the obtained impedance data with the data obtained by student 1 and student 3.

5. Impedance measurement (student 3)

Determine the impedance in practice by measuring the current, for different frequencies of the connected alternating voltage source.

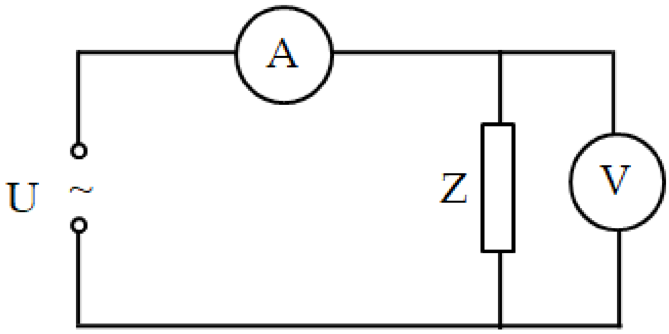


Figure 5.1: Electrical circuit for impedance measurement

The following hardware is used in the experiment:

- Function generator (figure 5.2 a)
- Oscilloscope (figure 5.2 b)
- Experimental board (Figure 5.2 s) with following elements:
    - Resistor R = 470 Ω
    - Inductor L = 1 mH
    - Capacitor C = 0,47 µF
- Digital multi-meter- AC Voltage (figure 5.2 d)

- Data Acquisition Card NI-myDAQ (figure 5.2 e)
- Conductors

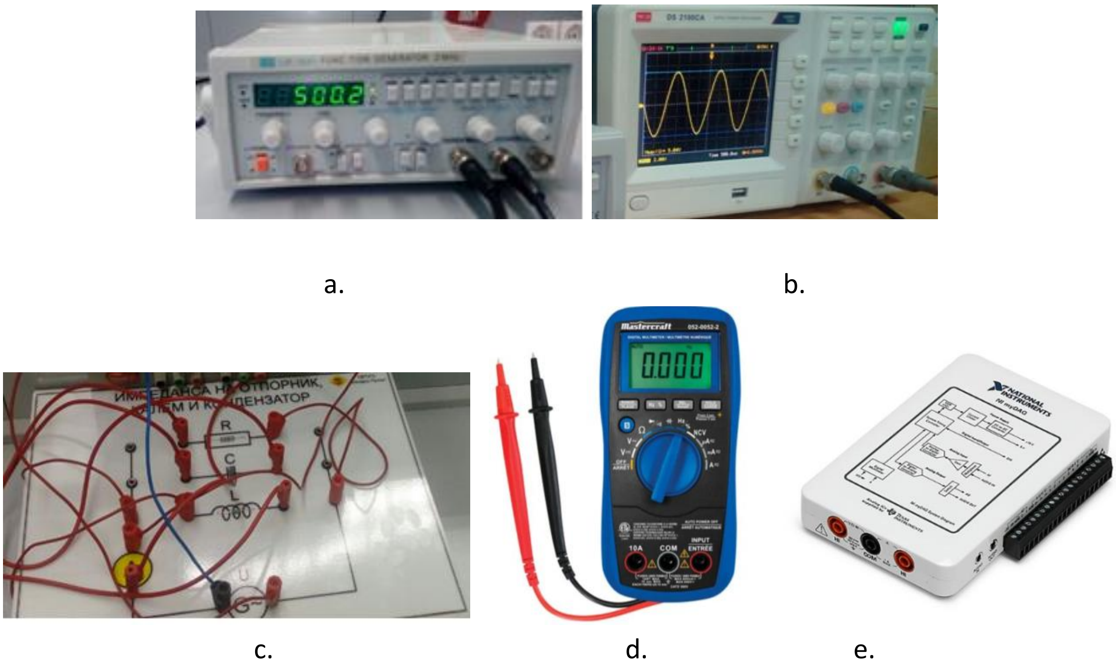


Figure 5.2 Hardware for impedance measurement

a). Function generator; b). Oscilloscope; c). Experimental board; d). Digital multi-meter

e). Acquisition Card

5.1 Procedure

The practical realization of the exercise is done by following the steps:

Step 1. Connect the function generator and the oscilloscope. The sinusoidal shape of the signal is selected and the effective value of the voltage U = 5 V is set.

Step 2. The elements in Figure 5.6c are connected by conductors, which form a series of electrical circuits with different types of impedances, shown in Figure 2.

Step 3. The NI-myDAQ acquisition card is connected as an amperemeter in the circuit, to measure the effective value of alternating current, with a measuring range of 200 mA.

Note: Be careful when connecting the NI myDAQ acquisition card. In order to avoid permanent damage to the equipment, care should be taken when properly connecting its terminals and selecting the measuring range. The acquisition card is used as an ammeter, with measuring range 200 mA.

Step 4. The functional generator is connected to the circuit, with a pre-set effective voltage value. During the measurement, we adjust the frequency values according to the instructions in the task.

The following steps are related to the measurement of the electric current in the circuit.

Note: During practical performance of the exercise, make sure that the current through the individual elements in the electrical circuit does not exceed the permissible value, in accordance with the characteristics of the electrical elements. For these reasons, for the elements which characteristics are listed in the exercise, the measurements of Z and I, for the electrical circuits of Figure 2b and Figure 2c, should be performed only for the given frequency values in the tables.

Step 5. After logging in the CORELA platform, select NI myDAQ in the External Device Selection Window (Figure 5.3). If no devices are displayed in the window, press the "Refresh Devices" control button.

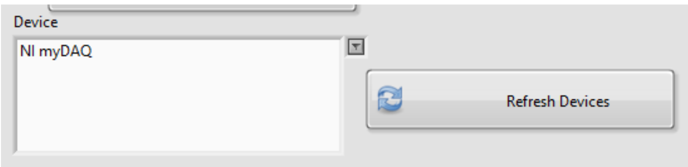


Figure 5.3: Selection of Acquisition Card

In order to be able to measure the effective value of alternating current through the acquisition card, it is necessary to activate the DAQ Card channel's function. From the "DMM input" function list, select an ammeter for alternating current and a measuring range of 200 mA (Figure 5.4).

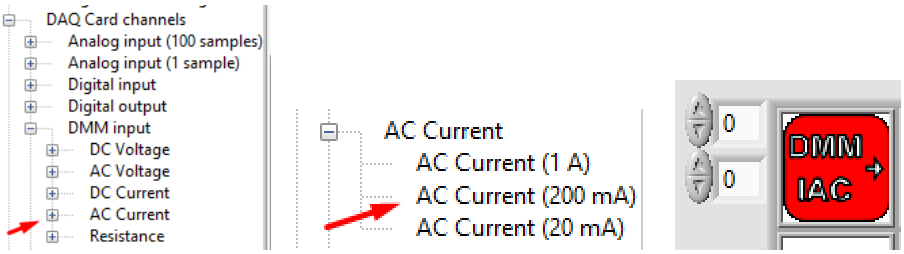


Figure 5.4: Activation of digital amperemeter with a measuring range of 200mA

Step 6. The measured effective value of the current is read by placing the "Forward" function on the desktop, next to the digital ammeter (Figure 5.5).

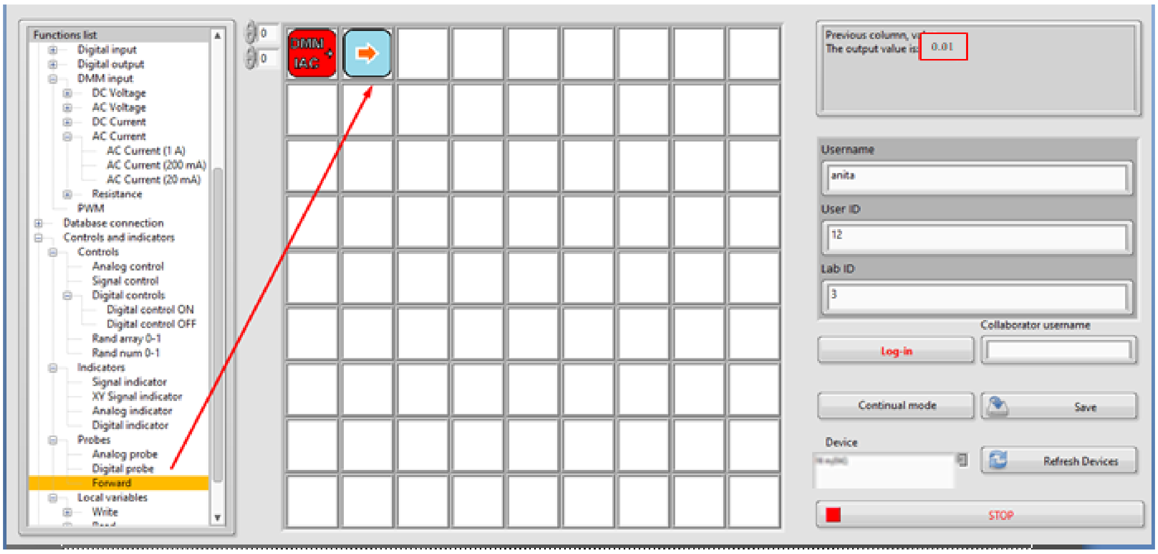


Figure 5.5: Reading the measured value of the current

The procedure from step 6 is repeated for each value of the frequency, given in the tables, according the selected electrical circuit.

The measured currents are entered in Table 5 (a; b; c; d).

Table 5a: Electrical circuit with ohmic resistance

f(Hz)	100	1000	10000	100000
I(A)				
Z(Ω)				

Table 5b: Electrical circuit with inductor

f(Hz)	5000	10000	50000	100000
I(A)				
Z(Ω)				

Table 5c: Electrical circuit with capacitor

f(Hz)	100	200	500	1000
I(A)				
Z(Ω)				

Table 5d: Serial RLC circuit

f(Hz)	100	1000	10000	100000
I(A)				
Z(Ω)				

The impedance *Z*, for each of the given frequencies in the tables, is calculated according to the formula:

$$Z = \frac{U}{I}[\Omega]$$

Question 1: *How does the current change by increasing the frequency in electrical circuit in figure 2b?*

Question 2: *How does the current change by increasing the frequency in electrical circuit in figure 2c?*

5.2 Comparation of the results

Frequency and impedance data values shall be recorded on the CORELA platform and sent to the educational platform in accordance with the procedure described in item 6 of this manual.

Compare the obtained impedance data with the data obtained by student 1 and student 2.

6. Procedure of writing and sending data to the educational platform

Step 1: Write and saving the data

From the function list, from the "Database connection" we select the function "Signal control" and with the left mouse-click on an empty field on the desktop, we set the function.

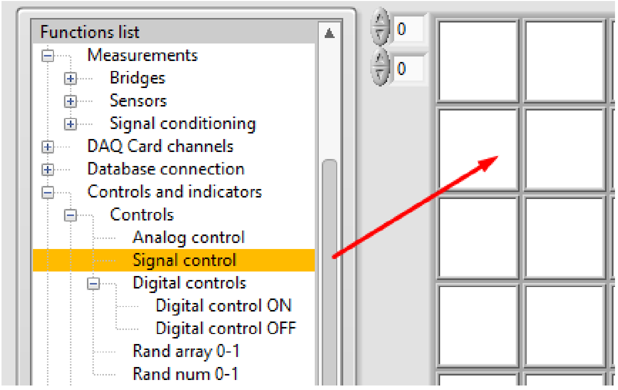


Figure 6.1: Setting a "Signal Control" function

A pop-up window opens where we enter the frequency values in form of an array (Figure 6.2).

Figure 6.2: Data entry

We record the data by pressing the "Record" button (Figure 6.2).

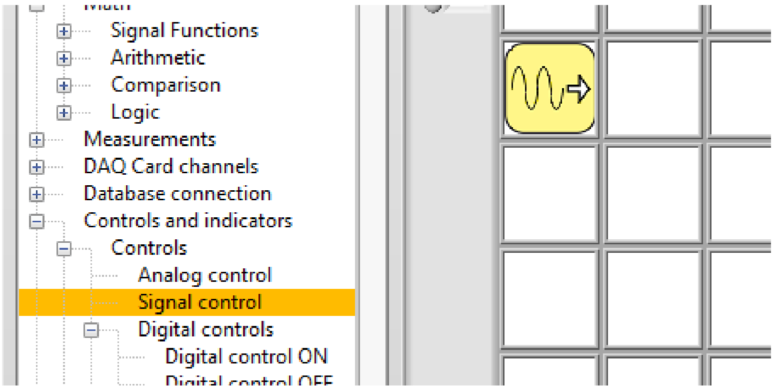


Figure 6.3: Screen layout after data capture

Step 2: Reviewing the recorded data

By setting the "Signal indicator" function, in the field to the right, next to the "Signal Control" function (Figure 6.4), the data entered on the CORELA platform can be accessed.

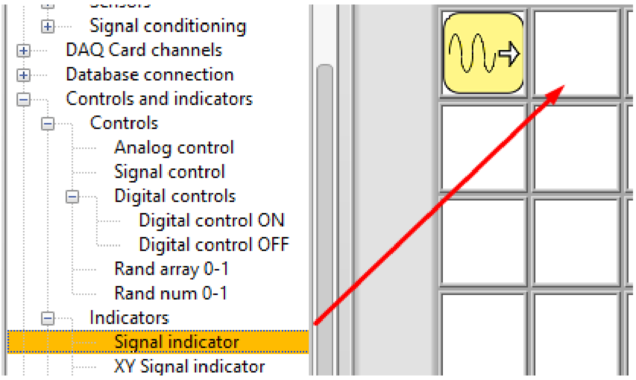


Figure 6.4: Setting a "Signal indicator" function

A pop-up window opens with a graphical representation of the entered frequency values (Figure 6.5).

Figure 6.5: Graphic display of entered frequency values

Step 3: Saving the recorded data

Selection of the output function "Signal Indicator" and pressing the "SAVE TO FILE" button writes the results in a separate document which can be further used for comparison and processing Figure 6.6.

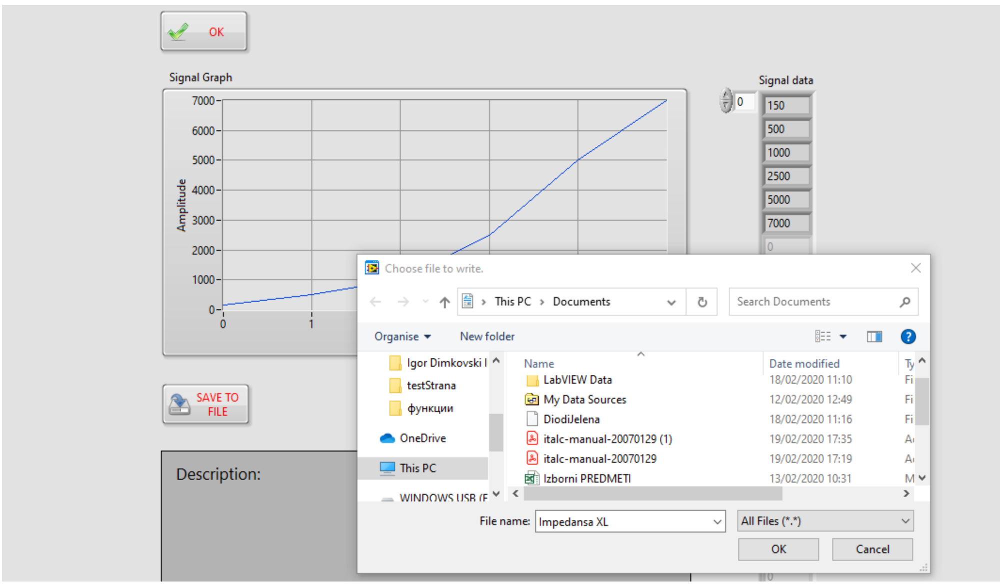


Figure 6.6: Recording results

Then press the "OK" button and return to desktop (Figure 6.7).

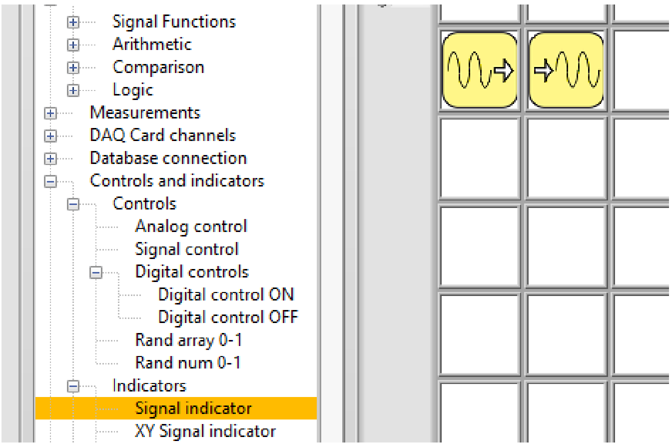


Figure 6.7: Desktop after setting the "Signal indicator" function

Step 4: Entering the impedance data

The procedure is repeated to enter calculated values for the impedances. By setting new "Signal control" and "Signal indicator" functions on desktop, the CORELA platform records and reviews the impedance data for all four electrical circuits from the task (Figure 6.8).

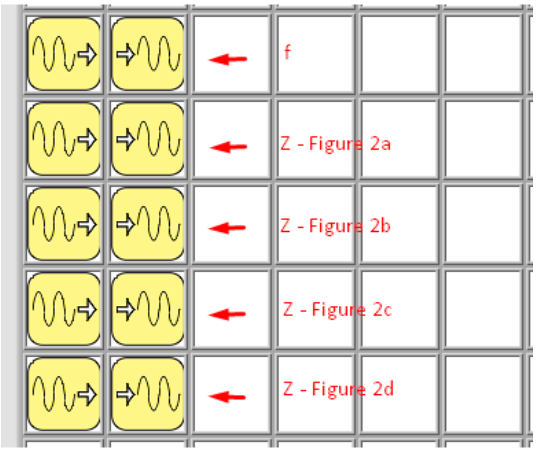


Figure 6.8: Display of CORELA platform task data entry

Step 5: Graphical representation of dependence of impedance on the change of frequency (optional).

To get a graphical representation of the dependence of one of the impedances on frequency, it is necessary to enter data on both the frequency and the impedance, and then set the XY Signal Indicator function (Figure 6.9).

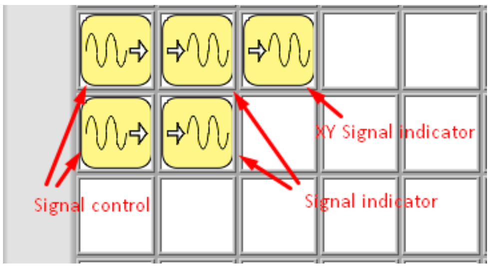


Figure 6.9: Sequence of functions for graphical display of dependence between two quantities

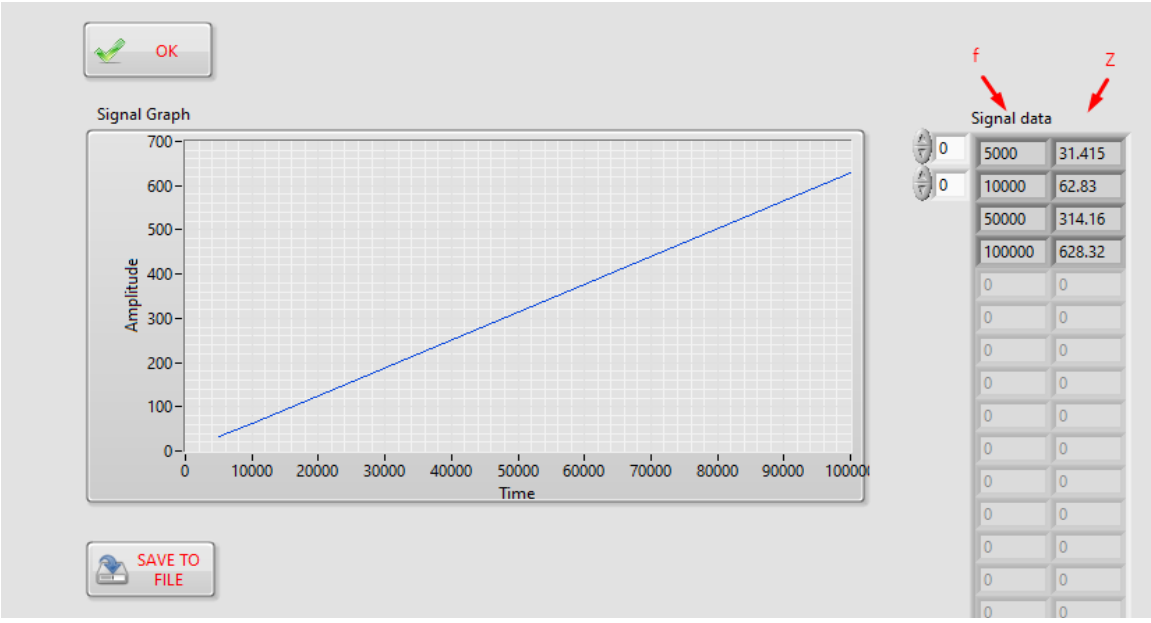


Figure 6.10: Graphic representation of impedance dependence on frequency for electrical circuit with an ideal coil

Step 6: Sending data to the educational platform

Sending data to the educational platform (Moodle) is done with the function "Database connection" (figure 6.11), by selection of output channels from the "Output channel". To send the frequency data, select CH out 1.

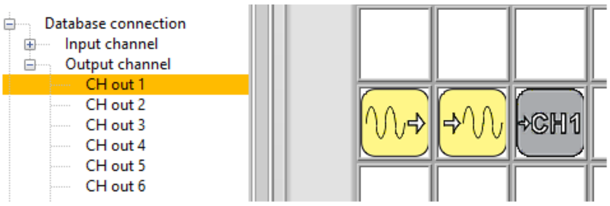


Figure 6.11: Sending data to the educational platform

The impedance data, from all four types of electrical circuits in the task, is sent to the educational platform through the remaining output channels (CH out2, CH out3, …….).

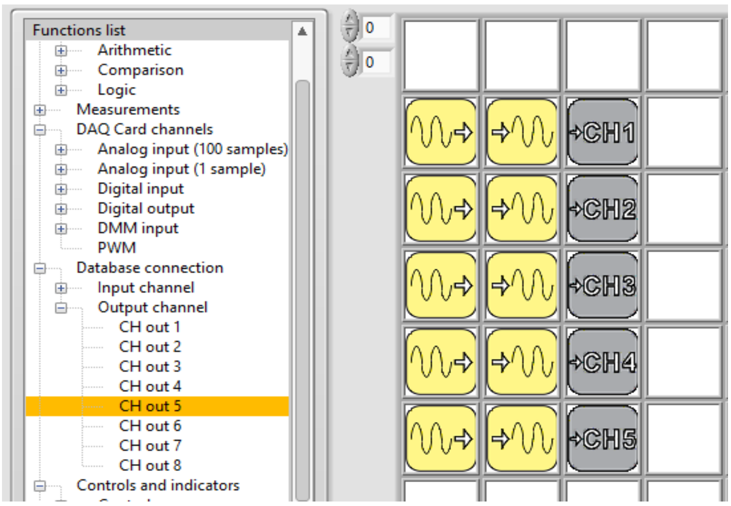


Figure 6.12: Screen after sending the results on the educational platform

After completing the task, enter the „chat room“.

Compare and comment on the results obtained in three ways:

1. *by calculation (student 1)*
2. *by simulation (student 2)*
3. *by measurement (student 3)*

Video material:

https://youtu.be/sW_nFX7Ka-k

IV Static current-voltage characteristic of a rectifier diode

Measuring of static current-voltage characteristics of a rectifier diode

1. Introduction

Diodes are electronic semiconductor elements with one PN junction and two terminals - anode and cathode.

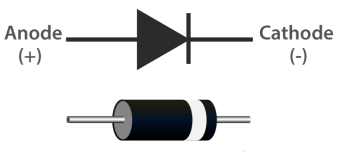


Figure 1.1: Diode symbol

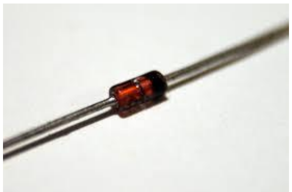


Figure 1.2: Appearance of diode

Depending on whether the anode is on a higher or lower potential than the cathode, diode can be direct or inverse polarized.

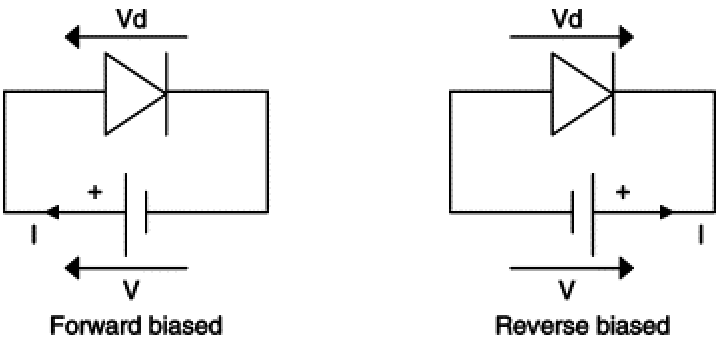


Figure 1.3: Direct and inverse polarized diode

The value of the diode current can be calculated by the equations:

$$i_D = I_S\left[e^{\frac{V_D}{nV_T}} - 1\right]$$

$$V_T = \frac{kT}{q} \approx 26mV$$

$$i_D \approx I_S e^{\frac{V_D}{nV_T}}$$

i_D and V_D are current and voltage of a diode. *Is* is an inverse saturation current of the diode in range 10^{-12} A. n is 1 for a germanium diode and n is 2 for a silicon diode. V_T is a voltage equivalent for the temperature.

The diode current–voltage characteristic graphics is a representation of the dependence of current on voltage, both, for forward and reverse polarized diode.

Figure 1.4 shows a static current - voltage characteristic of 1N4001 diode.

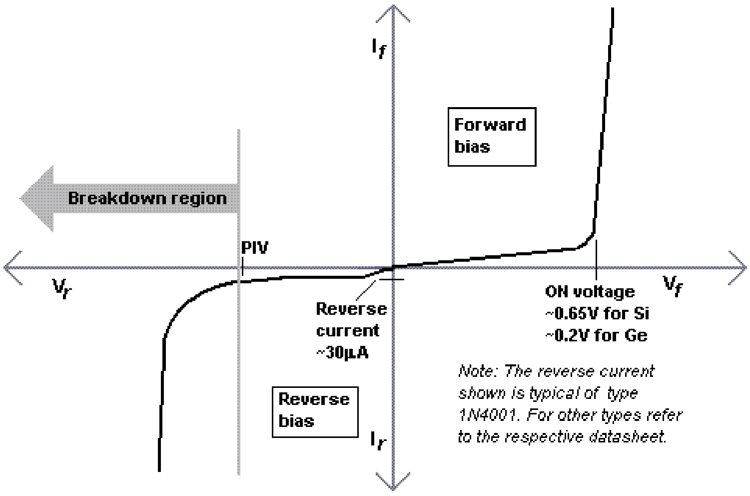


Figure 1.4: Static current - voltage characteristic of the 1N4001 diode

2. Theoretical tasks (all students)

On the diode static characteristics graph, highlight the characteristic parameters and explain the terms:

- threshold voltage U_T (switching voltage),

- reverse saturation current and

- breakdown reverse voltage U_{BR}.

3. Calculation of 1N4001 diode current value depending on the biasing voltage (student 1)

Using the circuit shown in Figure 3.1, record the static current voltage characteristic of the diode in case of:

A) forward polarized diode (short circuit at position ks1),

B) reverse polarized diode (short circuit at position ks2),

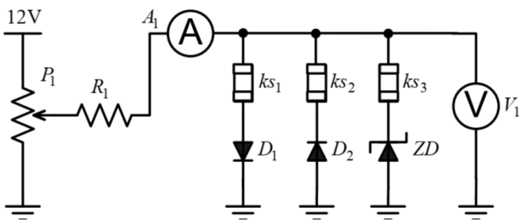


Figure 3.1: Electrical circuit for measuring static diode characteristics

Procedure

In Table 3.1 enter the calculated current values, for the values of power supply voltage, in case of a forward biased diode.

Table 3.1: Current for forward biased diode

Vd [V]	0.30	0.40	0.45	0.50	0.55	0.60	0.65	0.7
Id [mA]								

Represent the computed values graphically in the given coordinate system of Figure 3.2 and sketch the current voltage characteristic for a forward and reverse biased diode.

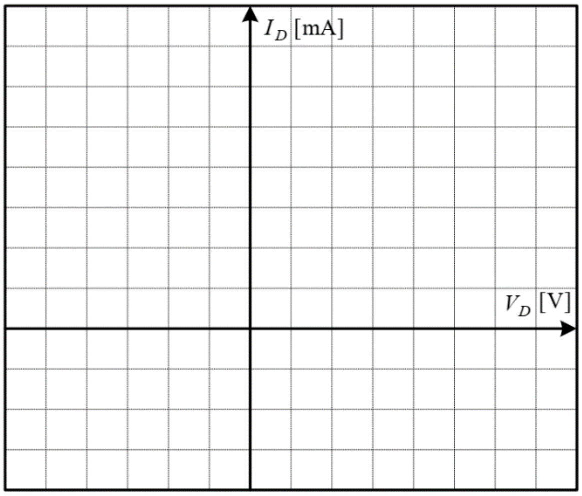


Figure 3.2: Current-voltage characteristic of 1N4001 diode

Determine the value of the threshold voltage U_T and breakdown reverse voltage U_{BR} on the graph.

U_T = _____ U_{invp} = _____

Write the default values for the biasing voltage V_d and the calculated values for the current Id on the Moodle platform. For this purpose, you need to login to the CORELA platform.

By selecting the Signal control function from the menu on the left side of the educational platform, as in Figure 3.3 A and 3.3 B, the results are entered as an array of values shown in Figure 3.4, one array for V_d and the other for I_d.

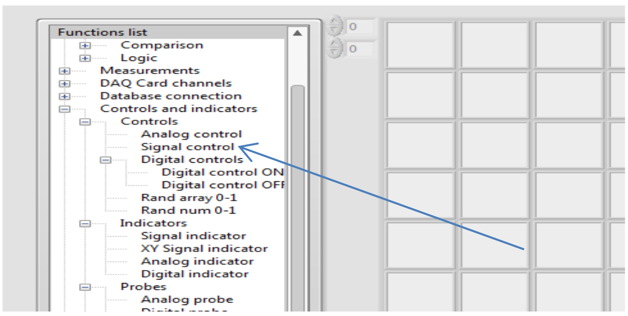


Figure 3.3 A

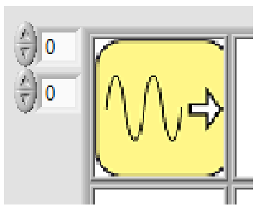


Figure 3.3B

Figure 3.3: Selection of Signal control function

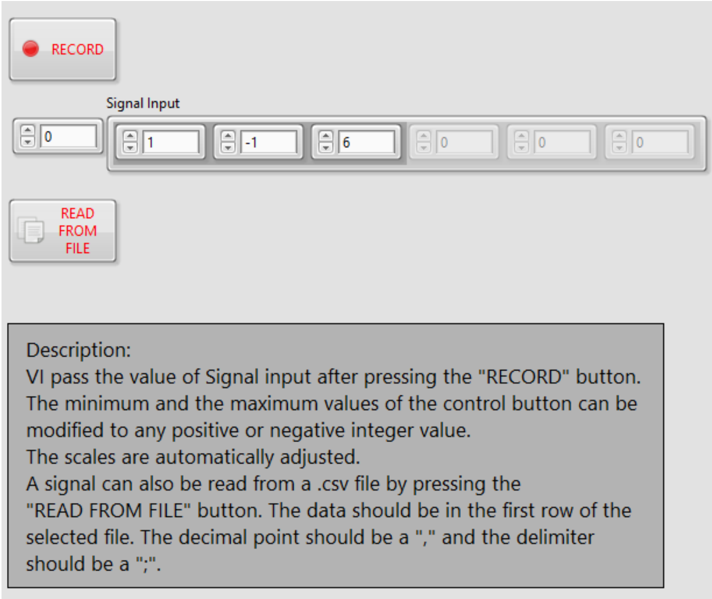


Figure 3.4: Entering data as string of values

By setting the "Signal indicator" function, in the field to the right, next to "Signal Control" function, Figure 3.5, the entered data on the CORELA platform can be accessed. A pop-up window opens with a graphical display of the entered values for voltage, i.e. current through the diode.

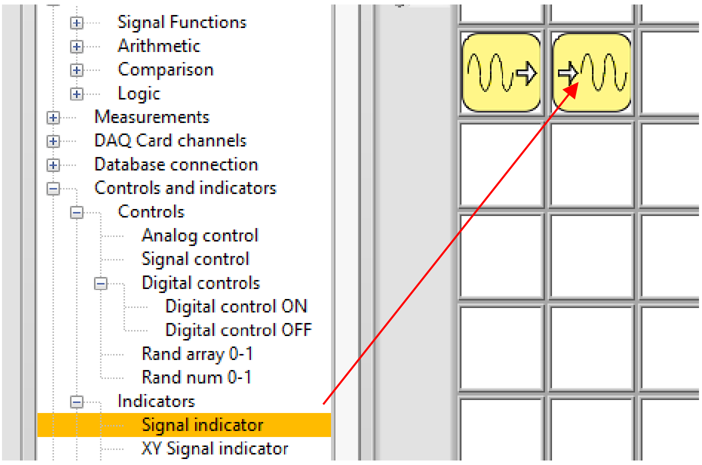


Figure 3.5: Access to entered data with Signal indicator function

The Signal indicator function enables graphical display of the entered data as a time function, Figure 3.6.

Note: The graphical display of the data with Signal indicator function does not refer to this exercise!

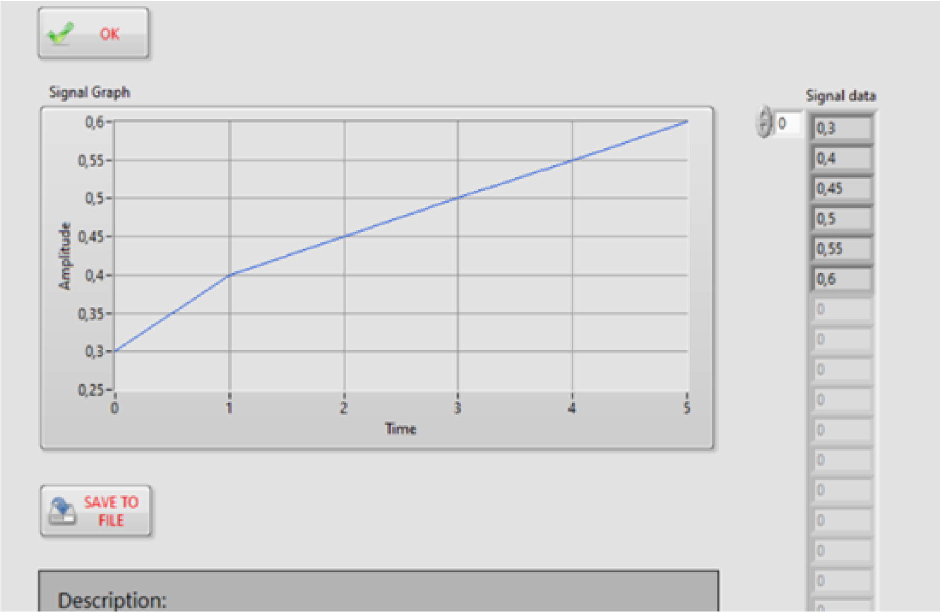


Figure 3.6: Display of entered data with Signal indicator function

➢ Draw a current-voltage characteristic of a diode using the XY Signal indicator function from the educational platform, as it is described in section 3.2 of the exercise!

➢ In order, the results to be visible to other users, they need to be placed on the educational platform. This is achieved with help of Database connection → Output channel → CH out x, where x- is one of the output channels.

➢ Explain the dependence of the current on the voltage of the diode during direct and inverse polarization!

4. Simulation of measuring of diode static characteristic using the functions implemented in CORELA platform (student 2)

Sign in with the assigned username and ID number and select an exercise number to perform.

From the Function List select→Electronic components →Active, activate the Diode function Figure 4.1A and with the left mouse-click place it in a field on the desktop of the platform, as shown in Figure 4.1B.

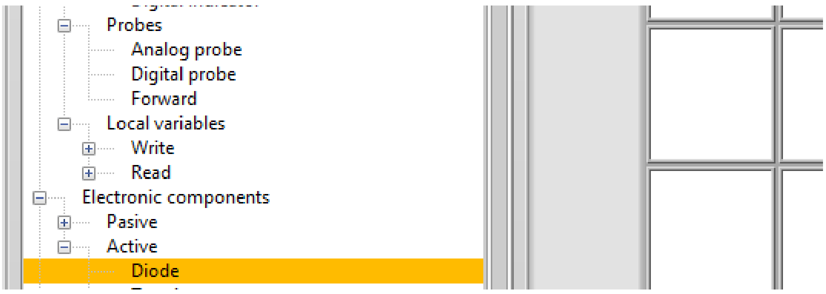


Figure 4.1A

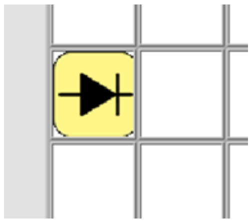


Figure 4.1B

Figure 4.1: Diode tag mounted on the desktop

The components of Diode function are shown in Figure 4.2.

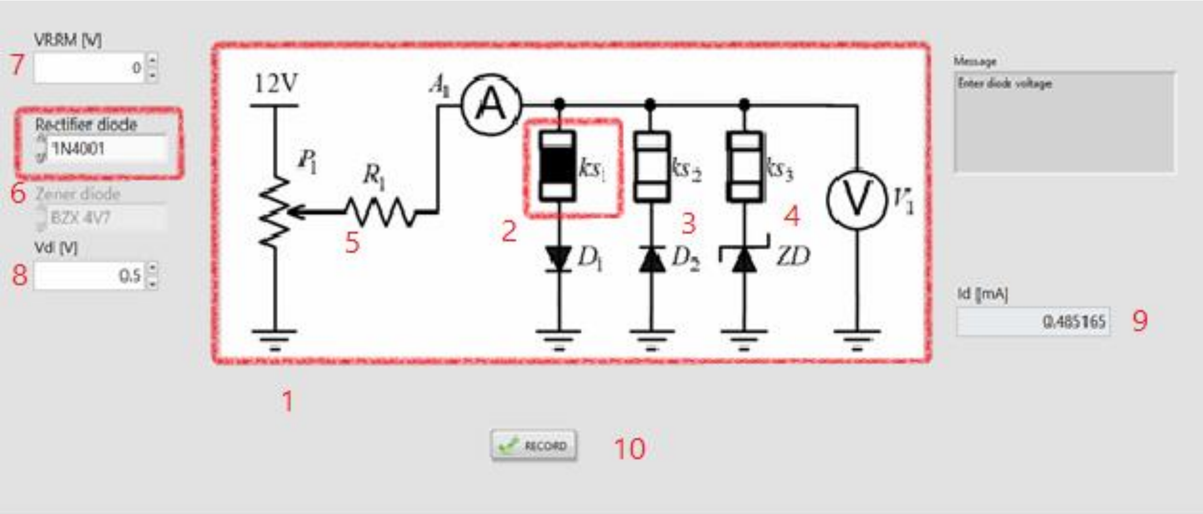


Figure 4.2 Diode function components

Description of Diode function components:

1. Electrical circuit for testing diode current voltage characteristics
2. Switch ks1, for forward biased diode configuration
3. Switch ks2, for the reverse biased diode configuration
4. Switch ks3, for Zener diode configuration
5. Resistor with resistivity $R1$ = 470Ω
6. Rectifier diode – diode type selection field
7. Voltage defining field (only if the diode selection is Custom type). In other cases, choose a value from the catalog in Annex 1, according to the diode type.
8. A field for entering the voltage values *Vd*
9. A field for reading the current values *Id*
10. Record button, when you click on it, the simulation is over.

Procedure

- By setting Diode function, a window as in Figure 4.2 opens on the desktop.
- By selecting one of the ks switches, the circuit configuration is selected, i.e. diode polarization. By selecting the ks1 switch, the current will be measured, depending on the voltage of a forward biased diode.
- Select the diode type; for this exercise it is 1N4001 diode (Fig. 4.3).

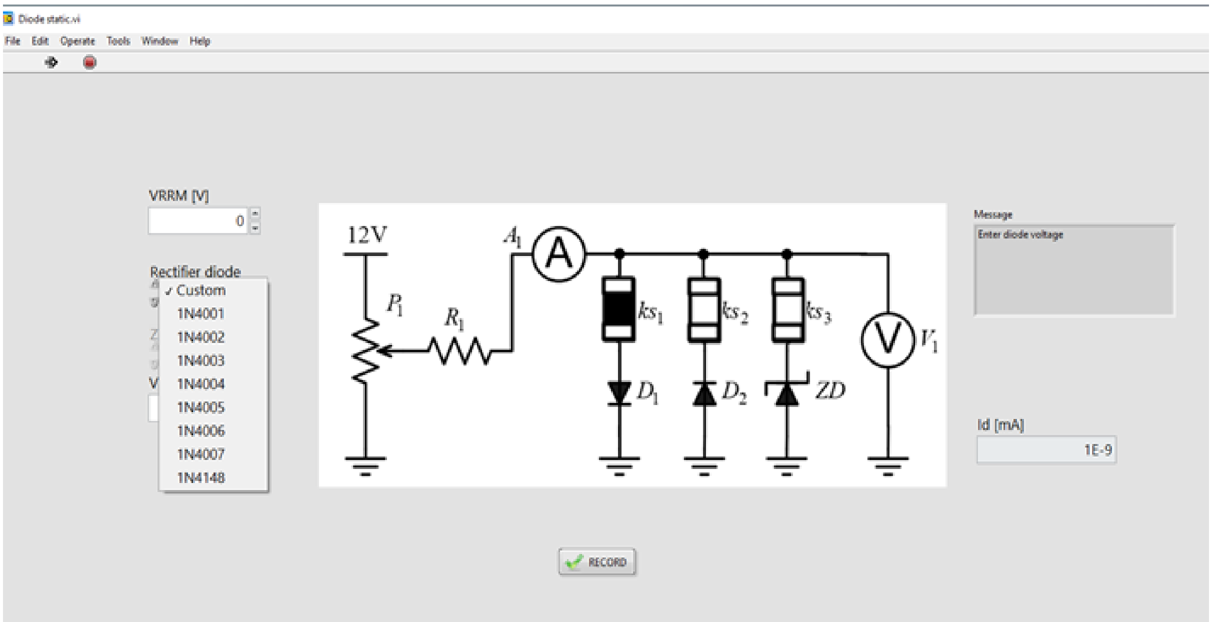


Figure 4.3 Diode type selection for recording static characteristic of forward biased diode

- According to the catalog in Appendix 1, the *Vrrm* of the diode is determined.
- The values of Vd are determined in Table 4.1.

Table 4.1: Id values obtained depending on Vd, for forward biased diode

Vd [V]	0.30	0.40	0.45	0.50	0.55	0.60
Id [mA]						

Note: For voltage value 0.7 V, diode breaks down.

- For each value of voltage *Vd*, by pressing RECORD, the value of current *Id* is obtained. The obtained results for the current *Id* will be read on the screen fig. 4.4 and will be entered in table 4.2.

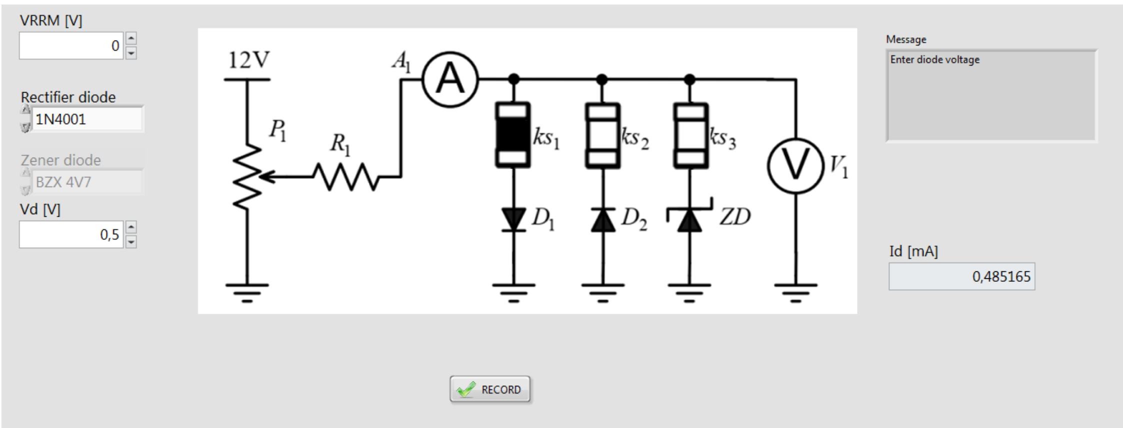


Figure 4.4: Simulation of current measurement of current-voltage characteristic for forward biased diode

The same procedure is used for a reverse biased diode by selecting ks 2. Enter the diode current values in Table 4.2.

Table 4.2: Id values obtained depending on Vd, for reverse biased diode

Vd [V]	-2	-4	-6	-8	-10
Id [µA]					

Task: Represent the current dependence on voltage for a diode, i.e. the current-voltage characteristic of the diode represented by a **2D graph!**

This is achieved in following order:

➢ Selecting the **Signal Control** function, figure 4.5, opens a window where the values of the voltage *Vd* are written, figure 4.6.

➢ Using the **Signal Indicator** function, which is placed on the desktop next to Signal Control, Figure 4.5, the entered data on the CORELA platform is accessed.

➢ The procedure is repeated for the values of the diode current, Figure 4.5 and 4.7

➢ By selecting the function **XY- Signal indicator**, and its placement on the work-surface, figure 4.5, the display clearly shows the current - voltage dependence, figure 4.8.

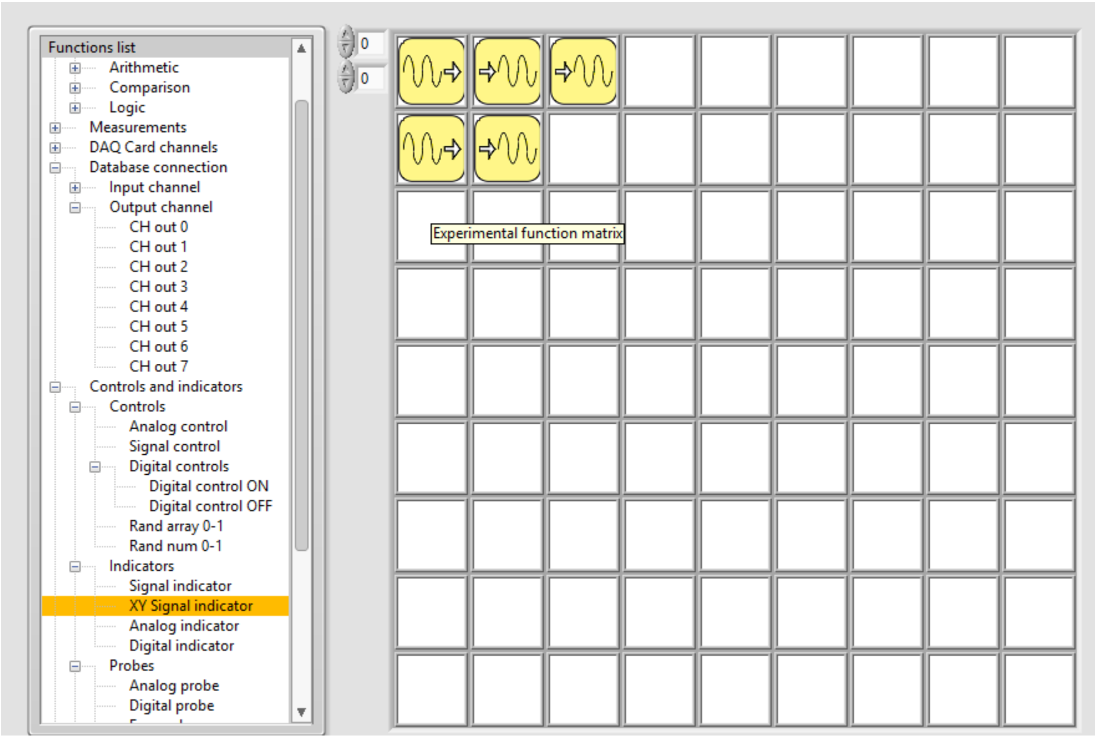


Figure 4.5: Selection of XY - Signal indicator

Figure 4.6: Entering values of Vd

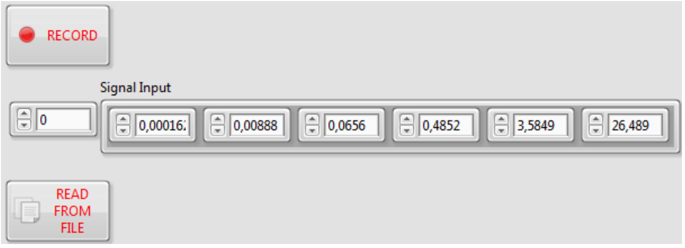


Figure 4.7: Entering values of Id

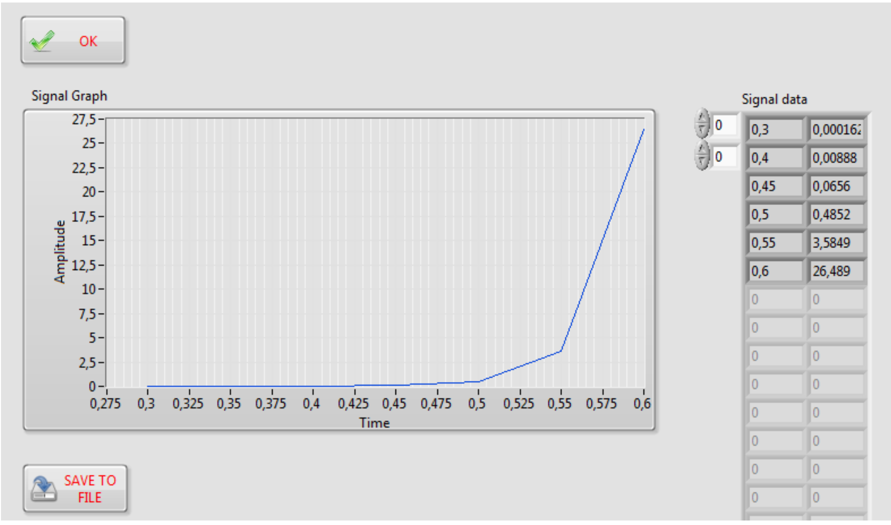


Figure 4.8: Current - voltage characteristic displayed on 2D graph by XY Signal indicator function

Task: Determine diode conduction threshold, breakdown voltage and inverse breakdown voltage for the diode in the obtained graph.

The data can be transferred to the educational platform using the function **Database connection**, figure 4.9, by selection of the output channels from the **Output channel** .

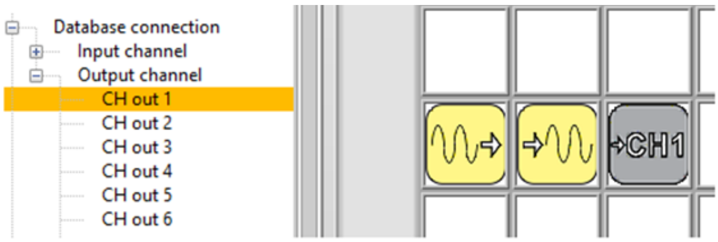


Figure 4.9: Transferring measured values to the educational platform

Note: Each group of students should use different channels for their data. The platform offers 8 input and 8 output channels.

Figure 4.10. shows by selection of output function **Signal Indicator** and button **SAVE TO FILE**, results can be written in a document and further used for comparison and processing.

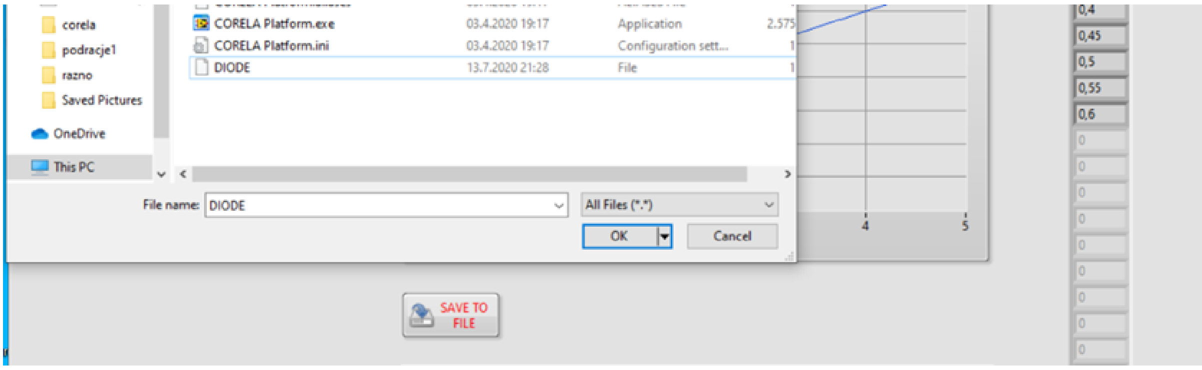


Figure 4.10 Recording results in a document

Pressing the "OK" button returns you to the desktop.

5. Practical realization and measurement (student 3)

Required material for practical realization of the exercise:

- Resistor $R1$ = 470 Ω

- Diode 1N4001

- Variable DC power supply, Figure 5.1

- Data Acquisition card NI myDAQ, Figure 5.2

- Digital multi-meter

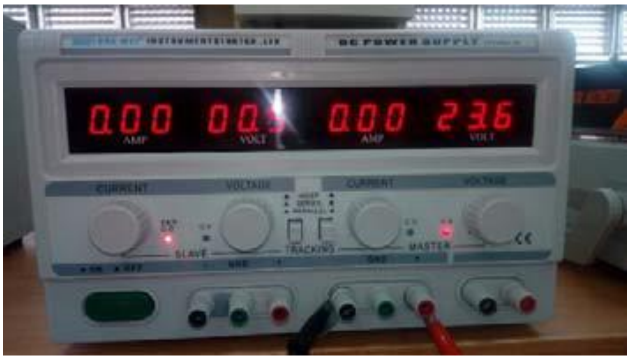


Figure 5.1 Variable DC power supply

Figure 5.2 NI myDAQ device

Connect the electrical circuit according to the scheme given in Figure 5.3.

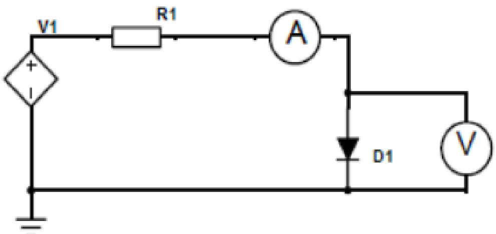


Figure 5.3: Electrical connection diagram

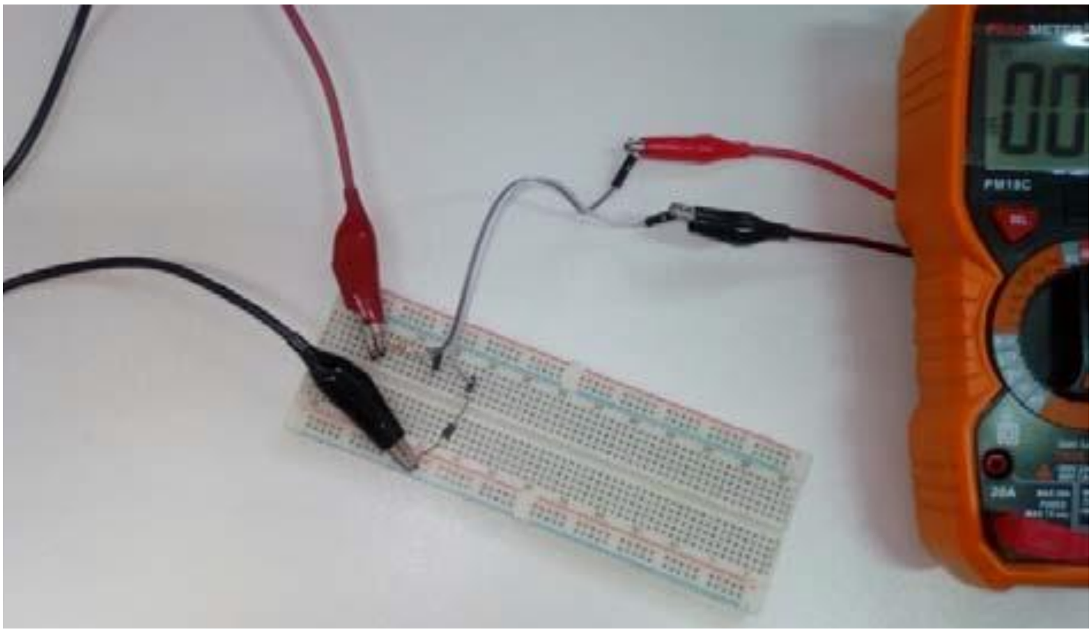


Figure 5.4: Electrical circuit for practical realization of the exercise

1. Connect the DC power supply according the electrical circuit in Figure 5.3

Note: Be careful with the polarity of the terminals of DC power supply. The positive voltage terminal is in red color and connected to resistor R1, and the negative one is in black color and connected to GND.

2. The digital multi-meter, used as an ammeter, is connected in series with the elements of the circuit. It is used to measure the current through the diode Figure 5.3 and Figure 5.4.
3. Using the CORELA platform, the voltage on diode terminals is measured with the NI myDAQ acquisition card, as a voltmeter. Its terminals are set in parallel to the diode, at the location marked for voltmeter V. DMM functions are used for voltage measurement - DC Voltage from the DMM input, Figure 3.3.6.
4. Once you have made all necessary connections of elements and devices and after you turn on devices, log on to the CORELA platform. In the selection window of external device "Device window" select NI myDAQ card (Figure 5.5). If no devices are displayed in the window, press the control button "Refresh Devices".

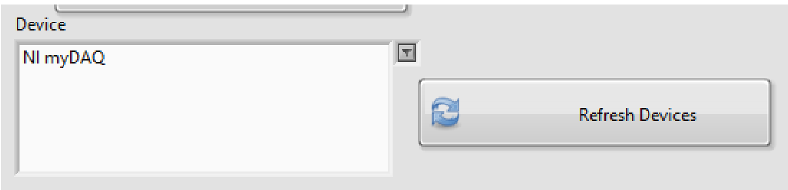


Figure 5.5: Selection of acquisition card

In order to be able to measure the diode voltage through the acquisition card, it is necessary to activate the DAQ Card channel's function. Select from the "DMM input" function list DC Voltage with measuring range of 2V, figure 5.6.

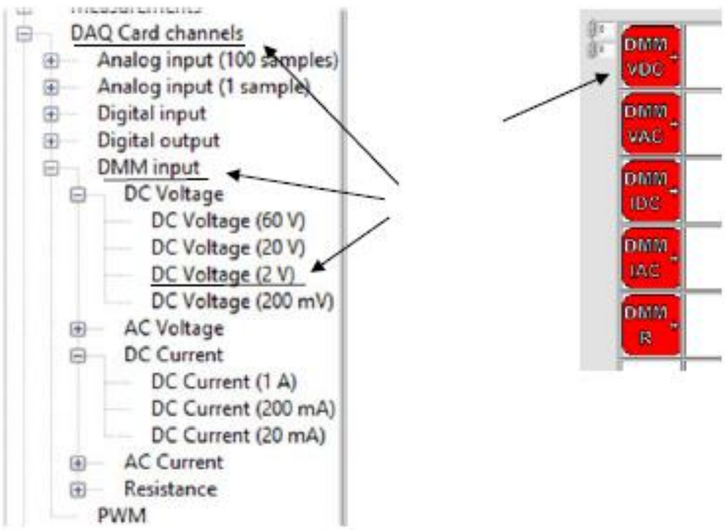


Figure 5.6: DMM input/ Appearance of an icon in a selected field

Note: Use the proper connection terminals of NI myDAQ acquisition card. Be careful with their polarity, connection and measuring range.

Once you have completed the electrical circuit, proceed to measure the current that flows through the diode. Follow the steps for current measurement:

5. Activate the "Diodes" tab of the virtual instrument. The following front panel appears:

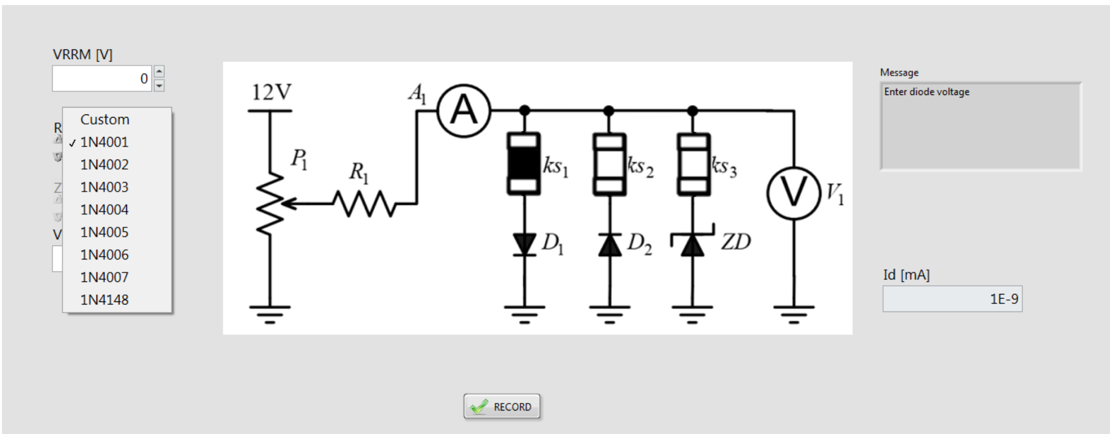


Figure.5.7 Front panel when "Diode" tab is selected from the virtual instrument

6. By changing the voltage of the DC power supply, the input voltage Vd is adjusted for values given in Table 5.1, and measured with the virtual instrument.

The measured voltage value is read from the message box by setting the "Forward" function to desktop, next to the digital voltmeter, Figure 5.8.

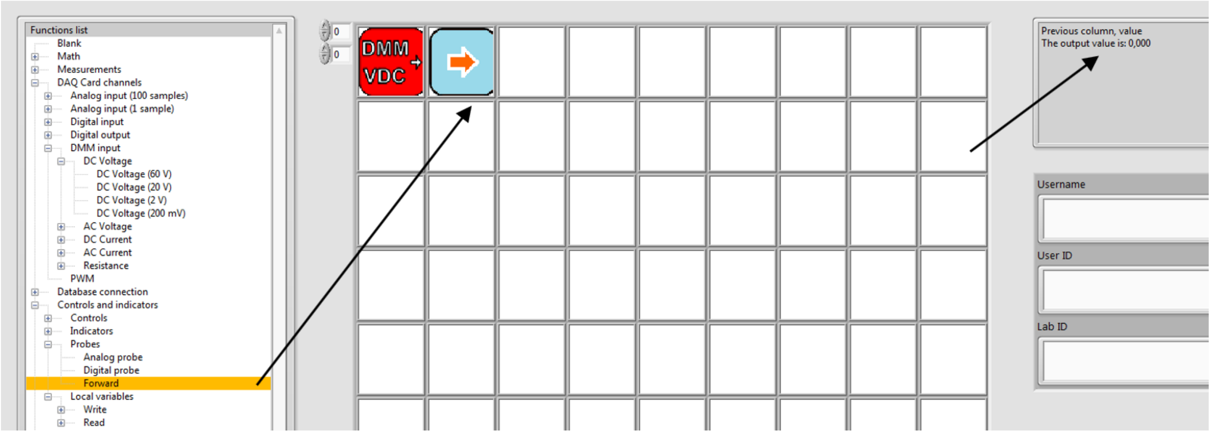


Figure 5.8: Reading the measured voltage value from message box

7. Measure the value of the diode current Id with digital multi-meter, used as ammeter, Figure 5.9, and write it in Table 5.1.

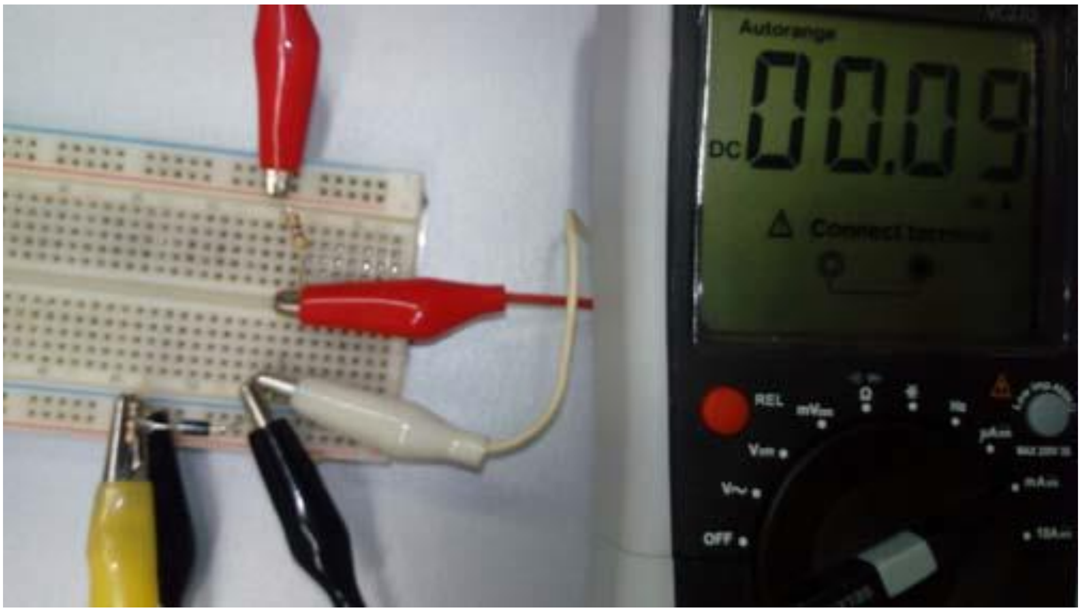


Figure 5.9: Measurement of diode current with digital multi-meter/ammeter

Table 5.1: Current values for forward biased diode

Vd [V]	0.30	0.40	0.45	0.50	0.55	0.60
Id [mA]						

8. Steps 5, 6 and 7 are repeated for each of the given voltage values *Vd* in Table3.1.
9. Compare the results obtained from the performed measurements of diode current with the theoretical calculations from part 1 and the simulations from part 2 of the exercise!

The procedure described above is repeated for the reverse biased diode, when the switch ks2 from the electrical circuit in Figure 2.1 is closed. The values of the current Id for given voltage values *Vd* are written in table 5.2.

Table 5.2: Current values for reverse biased diode

Vd [V]	-2	-4	-6	-8	-10
Id [µA]					

10. Send the measured data to the educational platform!

11. Draw the current-voltage characteristic of a diode using the XY function Signal indicator from the educational platform

Question 1: Is the obtained current-voltage characteristic of the diode identical when obtained by simulation?

Question 2: If the obtained current-voltage characteristic of a diode is not identical when obtained by simulation, what is the reason for that?

Question 3: How can the diode current-voltage characteristics, obtained by simulation and practical measurement, be as similar as possible?

Question 4: What difficulties and problems did you encounter during the realization of this exercise?

Video material:

https://www.youtube.com/watch?v=nSKkV7USLFA&feature=youtu.be

6. Appendix 1: Catalogue values for diodes implemented in the platform

Table 1: Diodes

Symbol	U_{RRM} [V]	I_{FAV} [A]	I_{FSM} [A]	U_F [V] ат I_{FAV}, T=25^{0}C	I_R[μA] ат U_{RRM}, T=25^{0}C
1N4001	50	1	30	1	5
1N4002	100	1	30	1	5
1N4003	200	1	30	1	5
1N4004	400	1	30	1	5
1N4005	600	1	30	1	5
1N4006	800	1	30	1	5
1N4007	1000	1	30	1	5
1N4148	100	0,15	2	75	0,025/U_R=20V
custom					

Table 2: Zener diodes

Symbol	U_Z [V] min	U_Z [V] max	I_Z [mA]	I_{Zmax} [mA]	r_Z(max) [Ω]
BZX 4V7	4,4	5,0	45	215	13
BZX 5V6	5,2	6,0	45	190	7,0
BZX 6V8	6,4	7,2	35	155	3,5
BZX 7V5	7,0	7,9	35	140	3,0
BZX 8V2	7,7	8,7	25	130	5,0
BZX 9V1	8,5	9,6	25	120	5,0
BZX 10V	9,4	10,6	25	105	7,5
BZX 11V	10,4	11,6	20	97	8,0
BZX 12V	11,4	12,7	20	88	9,0
BZX 13V	12,4	14,1	20	79	10,0
BZX 15V	13,8	15,6	15	71	15,0
BZX 18V	16,8	19,1	15	62	20,0
BZX 20V	18,8	21,2	10	56	24,0
BZX 22V	20,8	23,3	10	52	25,0
BZX 24V	22,8	25,6	10	47	25,0
BZX 27V	25,1	28,9	8	41	30,0
BZX 30V	28,0	32,0	8	36	30,0

V Logic functions

1. Theoretical tasks (*student 1*)

1.1 Realisation of the logic function

1.1.1) The device should start operating when sensor A and sensor B are both activated at the same time or sensor C is deactivated.

a) Write down the appropriate logic function.

Y =

SOLUTION:

$$Y = (A\,B) + \bar{C}$$

b) Fill in the logic function truth table.

Truth table:

SOLUTION:

A	B	C	Y
0	0	0	1
0	0	1	0
0	1	0	1
0	1	1	0
1	0	0	1
1	0	1	0
1	1	0	1
1	1	1	1

c) Draw a schematic diagram of the logic function and the electronic circuit using TinyCAD software environment.

Conduct a simulation of the logic function and the electronic circuit using TinkerCad software environment.

How many chip-units have you used? What was the type of used chip-units?

SOLUTION: 3 chip units, 7432,7408,7404

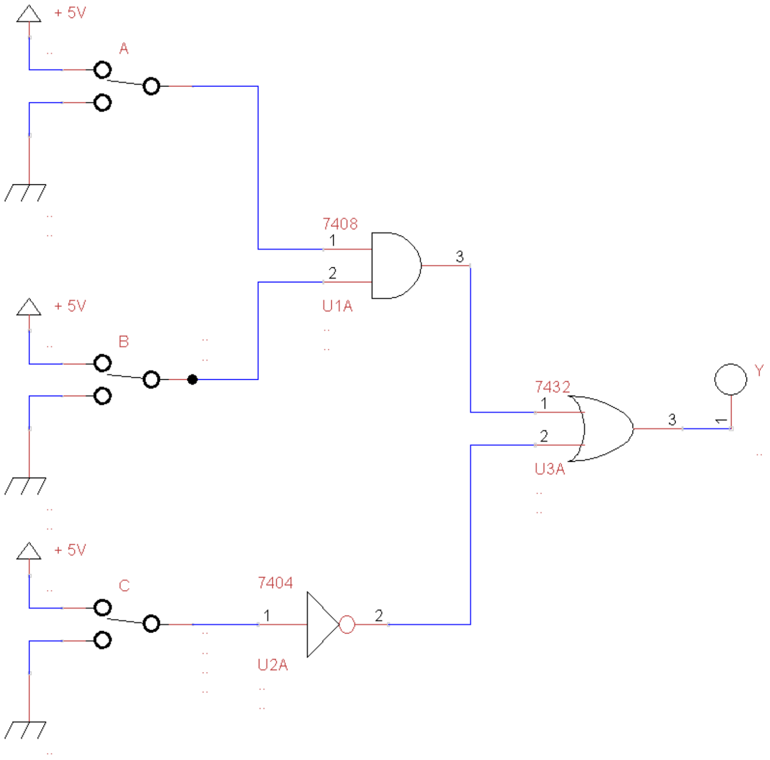


Figure 1.1.1: Schematic diagram of the logic function and the electronic circuit

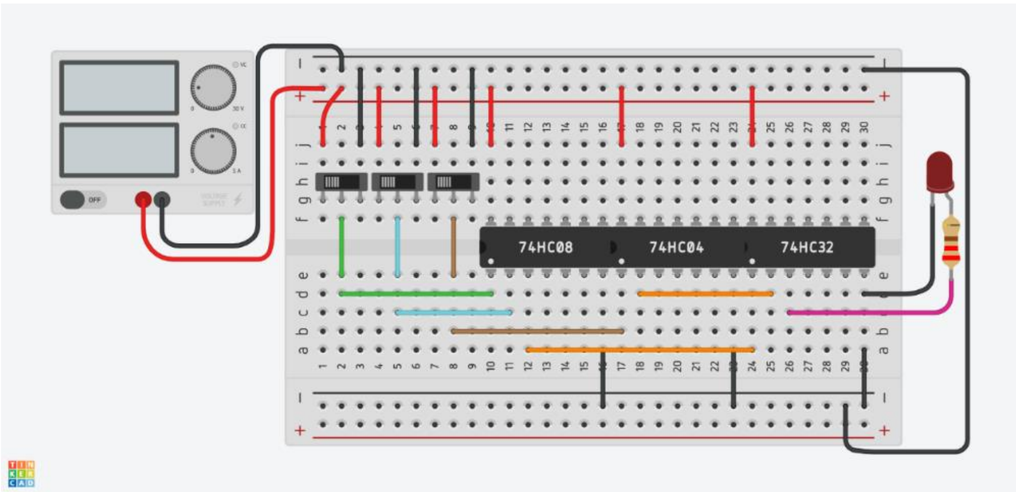


Figure 1.1.2: Scheme of the electric circuit in Tinkercad circuit simulator

1.1.2) Convert the logic function to implement a purely NAND gate form.

Advice: Use double negation and De Morgan's theorem.

$Y = \overline{A + B} = \bar{A} * \bar{B}$ *De Morgan's theorem*

SOLUTION:

$Y = \overline{\overline{AB + \bar{C}}} = \overline{\overline{AB} * \overline{\bar{C}}} = \overline{\overline{AB} * C}$

b) Draw a schematic diagram of the logic function in purely NAND form and the electronic circuit using TinyCAD software environment.

With the support of the Erasmus+ Programme of the European Union

Conduct a simulation of the logic function in purely NAND form and the electronic circuit using TinkerCad software environment.

How many chip-units have you used? What was the type of used chip-units?

SOLUTION: 1 chip, 7400

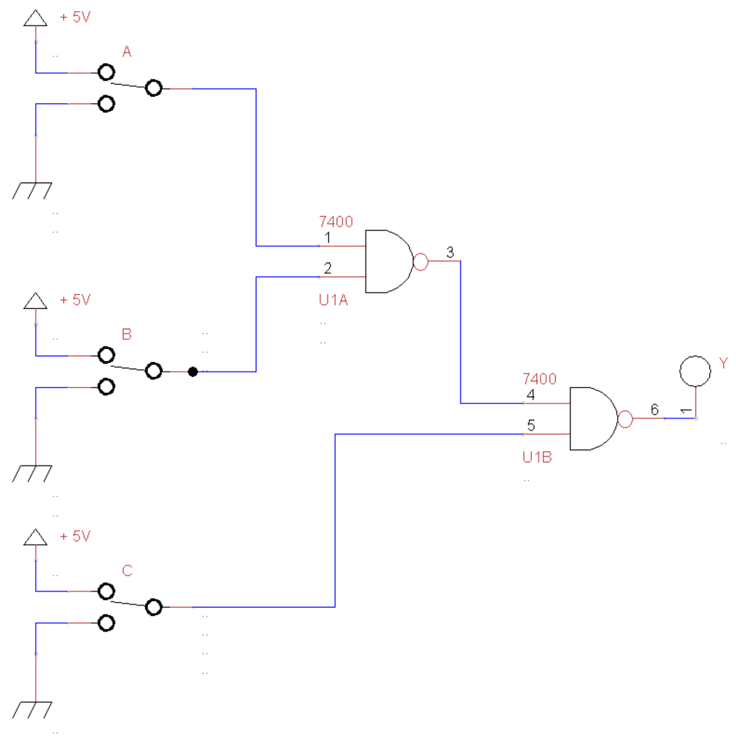


Figure 1.1.3: Schematic diagram of the logic function in purely NAND form and the electronic circuit

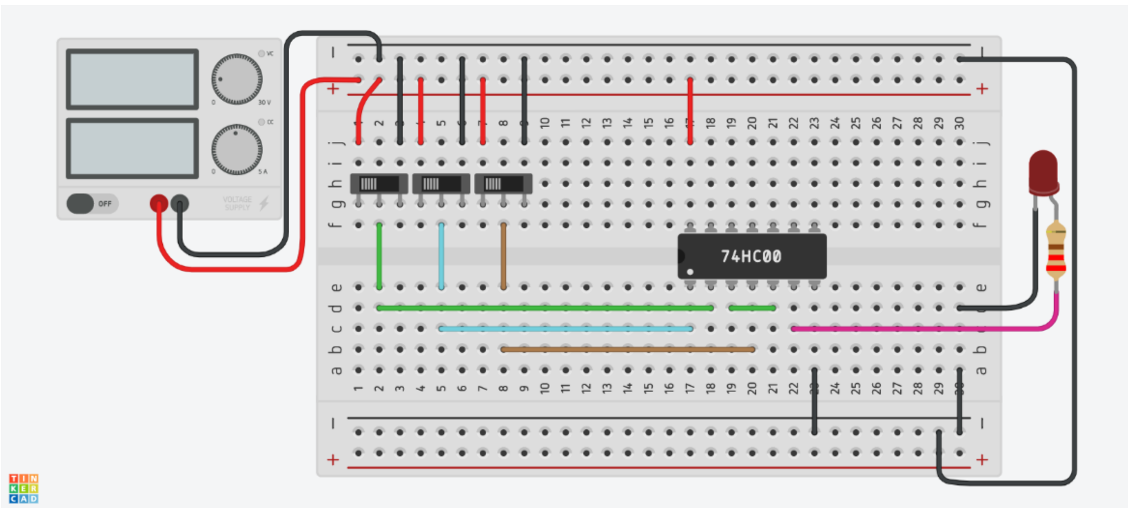


Figure 1.1.4: Scheme of the electric circuit in Tinkercad circuit simulator

With the support of the Erasmus+ Programme of the European Union

1.2 Realisation of the logic function using NAND gates

1.2.1) The logic function is described by the following equation:

$\mathrm{Y} = \bar{A}\,\mathrm{B} + \mathrm{C}$

Fill in the logic function truth table.

Truth table:

SOLUTION:

A	B	C	Y
0	0	0	0
0	0	1	1
0	1	0	1
0	1	1	1
1	0	0	0
1	0	1	1
1	1	0	0
1	1	1	1

1.2.2) a) Convert the logic function to implement a purely NAND gate form.

Advice: Use double negation and De Morgan's theorem.

SOLUTION:

$\mathrm{Y} = \overline{\overline{\bar{\mathrm{A}}\,\mathrm{B} + \mathrm{C}}} = \overline{\overline{\bar{A}B} * \bar{C}}$

Draw a schematic diagram of the logic function in purely NAND form and the electronic circuit using TinyCAD software environment.

Conduct a simulation of the logic function in purely NAND form and the electronic circuit using TinkerCad software environment.

SOLUTION:

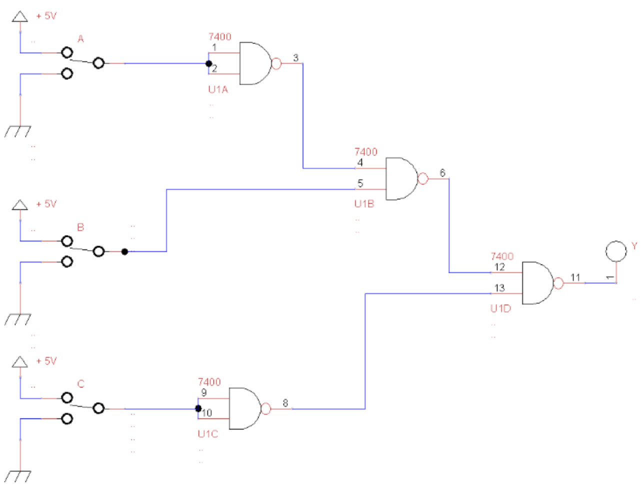


Figure 1.2.1: Schematic diagram of the logic function in purely NAND form and the electronic circuit

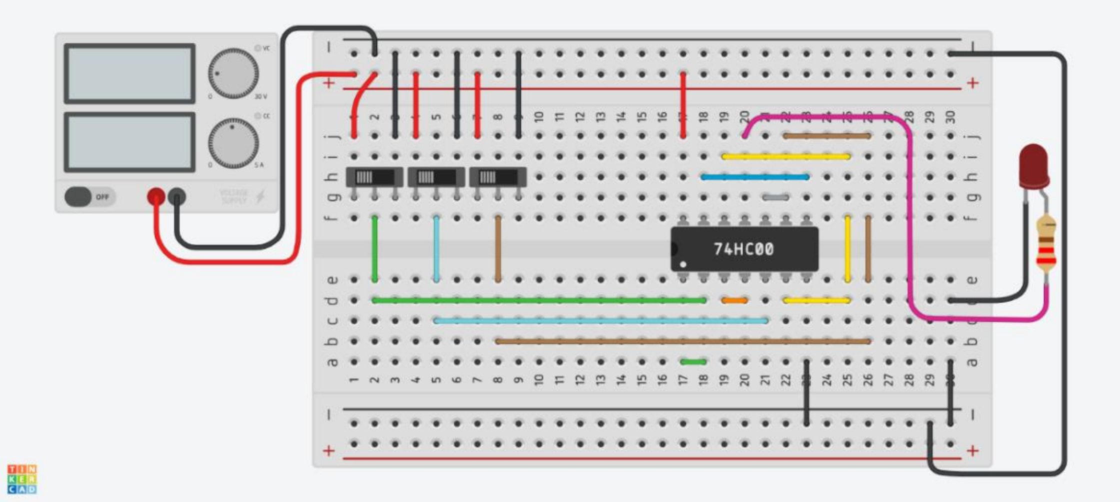


Figure 1.2.2: Scheme of the electric circuit in Tinkercad circuit simulator

1.3 Simplification of the logic function using Boolean algebra laws

1.3.1) a) Write down the appropriate logic equation for the following logic circuit.

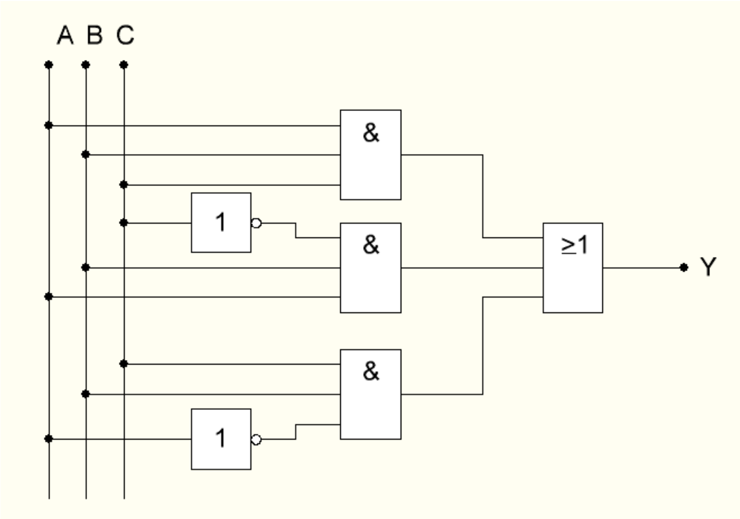


Figure 1.3.1: Schematic diagram of the logic function

SOLUTION:

$Y = A\,B\,C + A\,B\,\bar{C} + \bar{A}\,B\,C$

b) Fill in the logic function truth table.

Truth table:

SOLUTION:

A	B	C	Y
0	0	0	0
0	0	1	0
0	1	0	0
0	1	1	1
1	0	0	0
1	0	1	0
1	1	0	1
1	1	1	1

1.3.2) Simplify the logic function using Boolean algebra laws.

$\mathrm{Y} = \mathrm{A\,B\,C} + \mathrm{A\,B}\,\bar{C} + \bar{A}\,\mathrm{B\,C}$

SOLUTION:

Y= BA+BC

1.3.3) a) Convert the logic function to implement a purely NAND gate form.
Advice: Use double negation and De Morgan's theorem.
SOLUTION:
$\mathrm{Y} = \overline{\overline{\mathrm{B\,A} + \mathrm{B\,C}}} = \overline{(\overline{B\,A}) * (\overline{B\,C})}$

b) Draw a schematic diagram of the simplified logic function implemented in purely NAND gate form and the electronic circuit using TinyCAD software environment.

Conduct a simulation of the simplified logic function implemented in purely NAND gate form and the electronic circuit using TinkerCad software environment. What type and how many chip-units would you use to realise the logic function in the NAND gate form?

What type and how many chip-units have you used for implementing the logic function in its original form? There is no “Draw a schematic diagram” and simulation task correspondent to this question.

SOLUTION: 3 logic gates, 1 chip, 7400

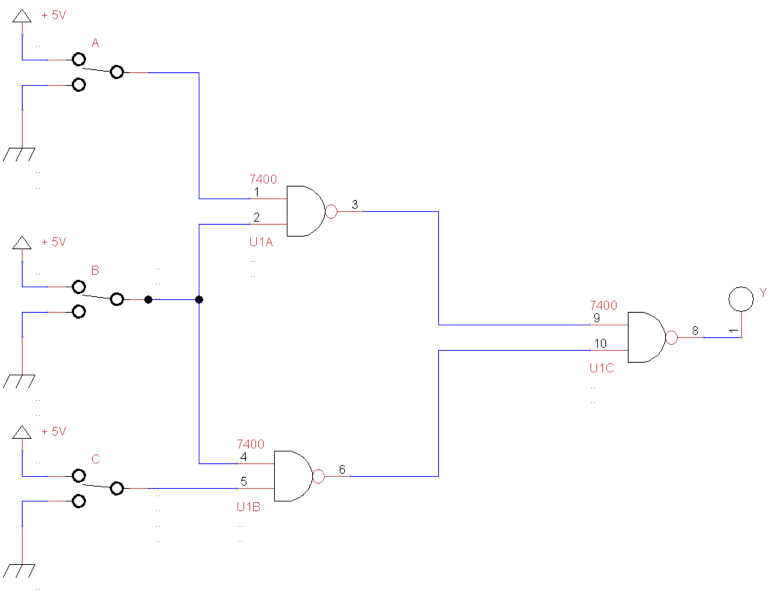


Figure 1.3.2: Schematic diagram of the simplified logic function in purely NAND form and the electronic circuit

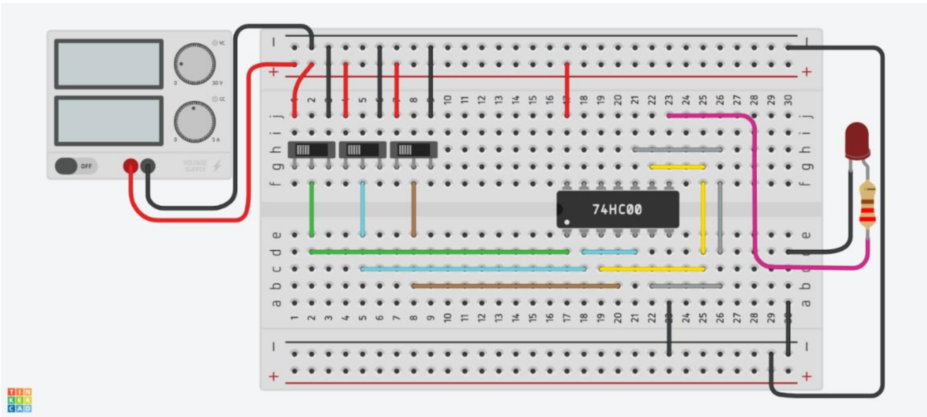


Figure 1.3.3: Scheme of the electric circuit in Tinkercad circuit simulator

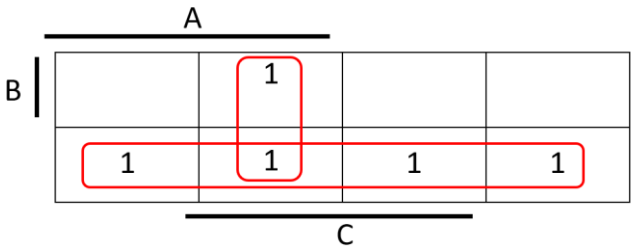
1.4 Simplification of the logic function using Veitch diagram

1.4.1) The Logic Truth table is given:

A	B	C	Y
0	0	0	1
0	0	1	1
0	1	0	0
0	1	1	0
1	0	0	1
1	0	1	1
1	1	0	0
1	1	1	1

1.4.2) Minimise the logic function by using Veitch diagram.

SOLUTION:



1.4.3) Write down the full disjunctive normal form. Simplify the logic function using Boolean algebra laws.

SOLUTION:

$\mathrm{Y} = \mathrm{A}\,\mathrm{C} + \overline{\mathrm{B}}$

1.4.4) a) Convert the logic function to implement a purely NAND gate form.

Advice: Use double negation and De Morgan's theorem.

SOLUTION:

$\mathrm{Y} = \overline{\overline{(A\,C + \overline{B})}} = \overline{(\overline{A\,C}) * (\overline{\overline{B}})}$

b) Draw a schematic diagram of the logic function in disjunctive normal form implemented in purely NAND gate form and the electronic circuit using TinyCAD software environment.
Conduct a simulation of the logic function in the full disjunctive normal form implemented in purely NAND gate form and the electronic circuit using TinkerCad software environment.

SOLUTION:

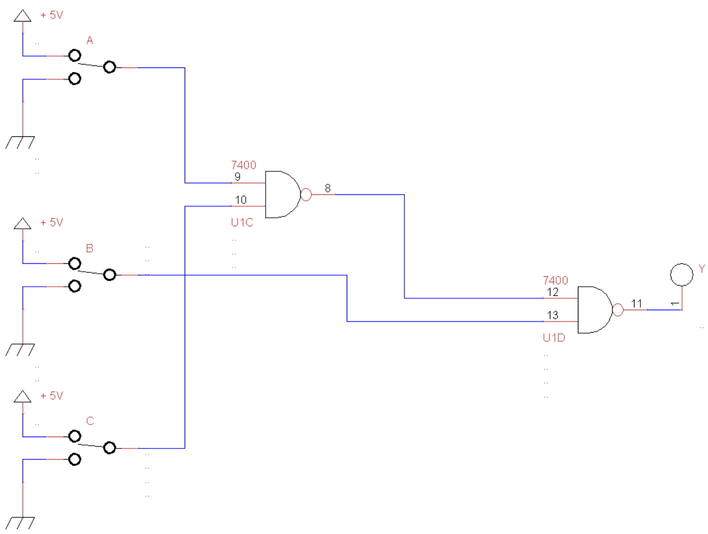


Figure 1.4.1: Schematic diagram of the logic function in the full disjunctive normal form implemented in purely NAND gate form and the electronic circuit

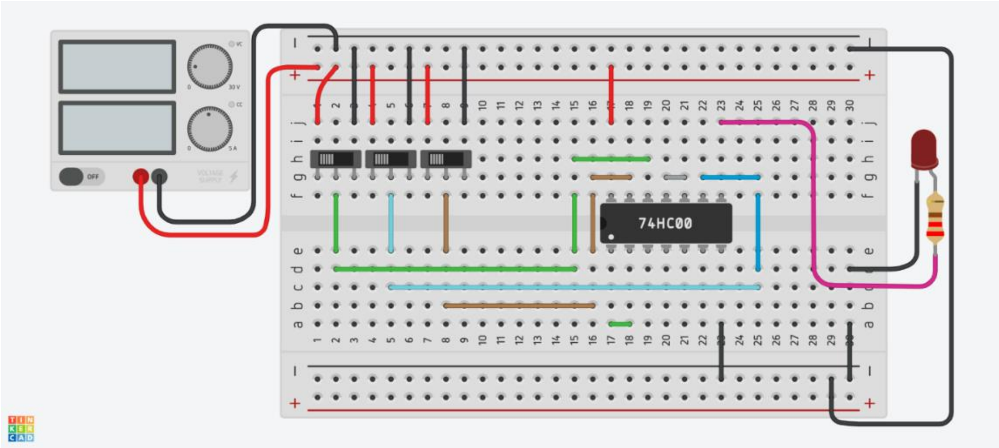


Figure 1.4.2: Scheme of the electric circuit in Tinkercad circuit simulator.

2. CORELA education platform simulation (student 2)

Conduct a simulation of all five electrical circuits using the CORELA Education Platform.

Installation, structure, and use of the platform are presented in **CORELA Education Platform User Manual**.

2.1 Realisation of the logic function

2.1.1) The device should start operating when sensor A and sensor B are both activated at the same time or sensor C is deactivated.

The logic function is determined by the equation and the truth-table below.

Equation:

$$Y = (A\,B) + \bar{C}$$

Truth table:

A	B	C	Y
0	0	0	1
0	0	1	0
0	1	0	1
0	1	1	0
1	0	0	1
1	0	1	0
1	1	0	1
1	1	1	1

The schematic diagram of the logic function and the electronic circuit is shown in Figure 2.1.1.

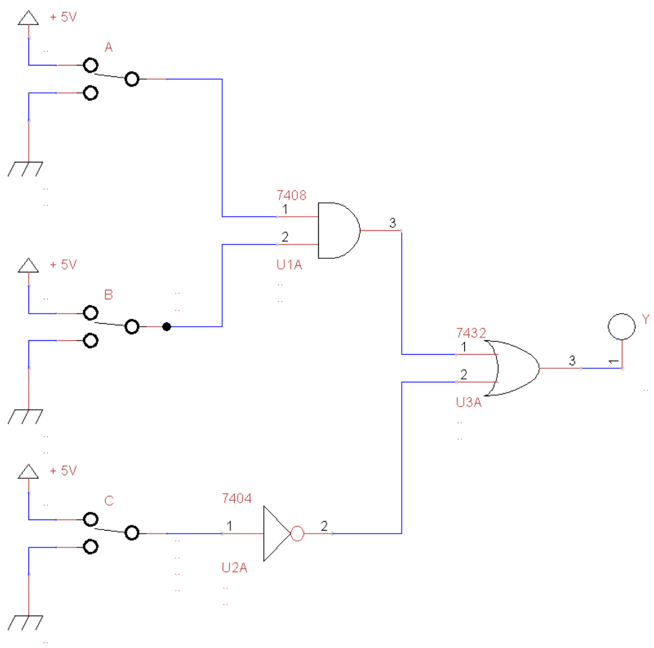


Figure 2.1.1: Schematic diagram of the logic function and the electronic circuit

2.1.2) Conduct a simulation of the described logic function in the CORELA platform.

In the CORELA platform editor, assemble the circuit according to Figure 2.1.1: (Schematic diagram of the logic function and the electronic circuit).

Note that it is necessary to test all combinations from the logical table (Truth table) from point 2.1.1, step by step and check the results. This means making simulation circuits for combinations of input variables from 000, 001, 010 to 111.

An example of a combination 110 is shown in Figure 2.1.2.

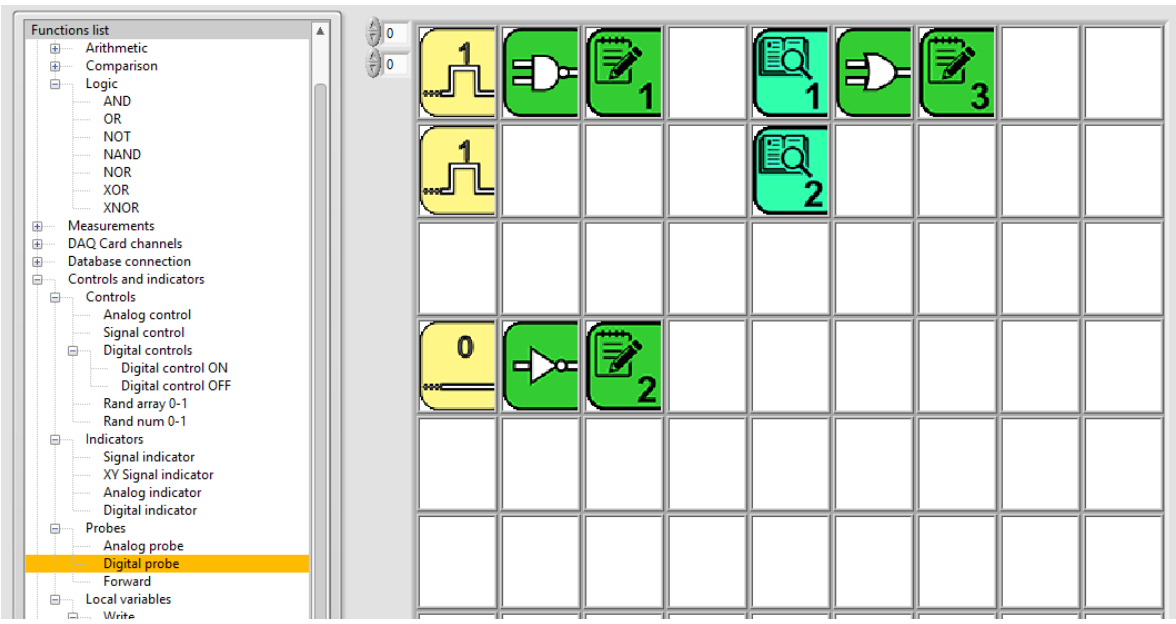


Figure 2.1.2: Notation of the logic function in the CORELA simulation editor

2.1.3) The logic function in purely NAND form is determined by the equation and truth table below.

Equation:

$$Y = \overline{\overline{\mathrm{AB} + \overline{\mathrm{C}}}} = \overline{\overline{AB} * \overline{\overline{C}}} = \overline{\overline{AB} * C}$$

Truth table:

A	B	C	Y
0	0	0	1
0	0	1	0
0	1	0	1
0	1	1	0
1	0	0	1
1	0	1	0
1	1	0	1
1	1	1	1

The schematic diagram of the logic function in purely NAND form and the electronic circuit is shown in Figure 2.1.3.

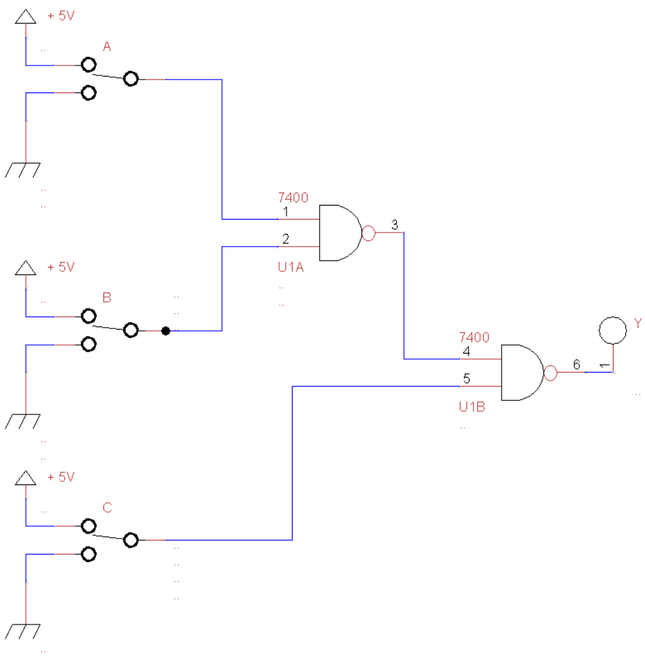


Figure 2.1.3: Schematic diagram of the logic function in purely NAND form and the electronic circuit

2.1.4) Conduct a simulation of the logic function in purely NAND form in the CORELA platform.

In the CORELA platform editor, assemble the circuit according to Figure 2.1.3: (Schematic diagram of the logic function in purely NAND form and the electronic circuit.).

Note that it is necessary to test all combinations from the logic table (Truth table) from point 2.1.3, step by step and check the results. This means making simulation circuits for combinations of input variables from 000, 001, 010 to 111.

An example of a combination 110 is shown in Figure 2.1.4.

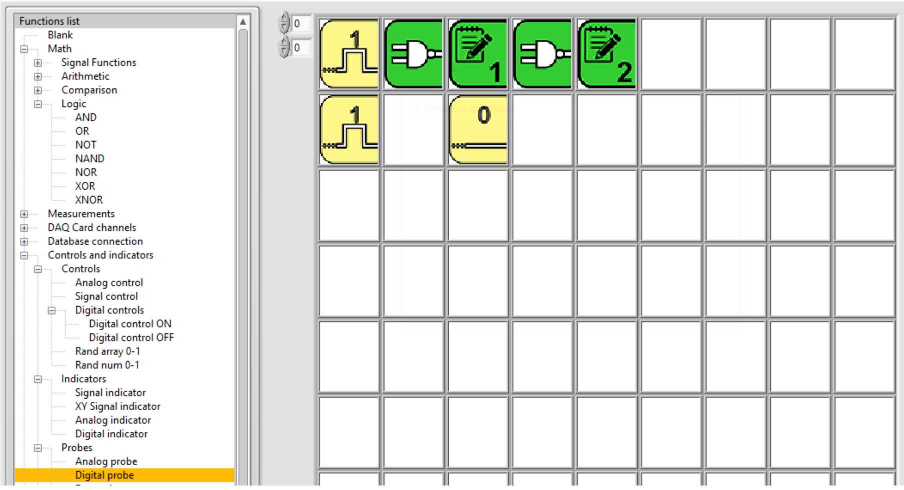


Figure 2.1.4: Notation of the logic function in the CORELA simulation editor

2.2 Realisation of the logic function using NAND gates

2.2.1) The logic function is described by the following equation and truth table:

Equation:

$$Y = \bar{A}\,B + C$$

Truth table:

A	B	C	Y
0	0	0	0
0	0	1	1
0	1	0	1
0	1	1	1
1	0	0	0
1	0	1	1
1	1	0	0
1	1	1	1

2.2.2) The logic function in purely NAND form is determined by the equation below.

Equation:

$$Y = \overline{\overline{\bar{A}\,B + C}} = \overline{\overline{\bar{A}B} * \bar{C}}$$

The schematic diagram of the logic function in purely NAND form and the electronic circuit is shown in Figure 2.2.1.

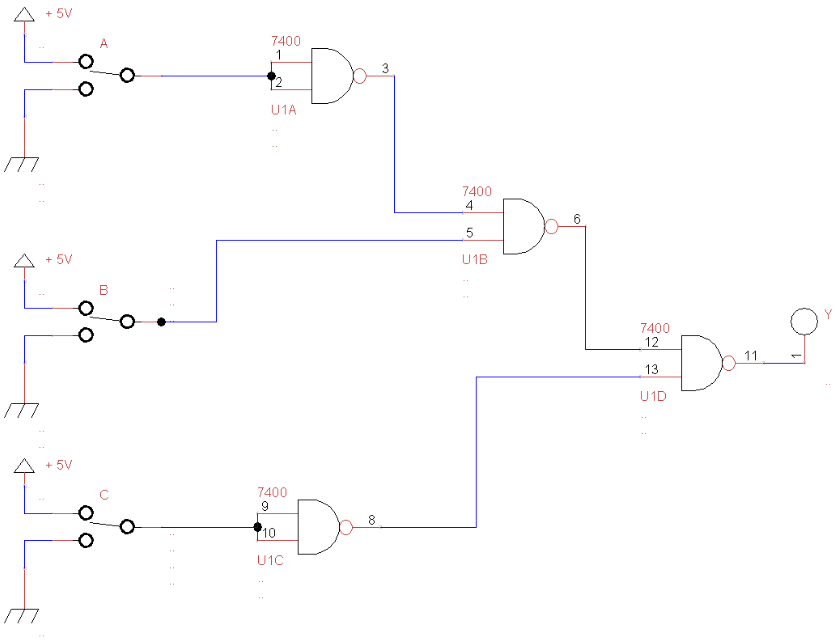


Figure 2.2.1: Schematic diagram of the logic function in purely NAND form and the electronic circuit

2.2.3) Conduct a simulation of the logic function in purely NAND form in the CORELA platform.

In the CORELA platform editor, assemble the circuit according to Figure 2.2.1: (Schematic diagram of the logic function in purely NAND form and the electronic circuit.).

Note that it is necessary to test all combinations from the logical table (Truth table) from point

2.2.1, step by step and check the results. This means making simulation circuits for combinations of input variables from 000, 001, 010 to 111.

An example of a combination 011 is shown in Figure 2.2.2.

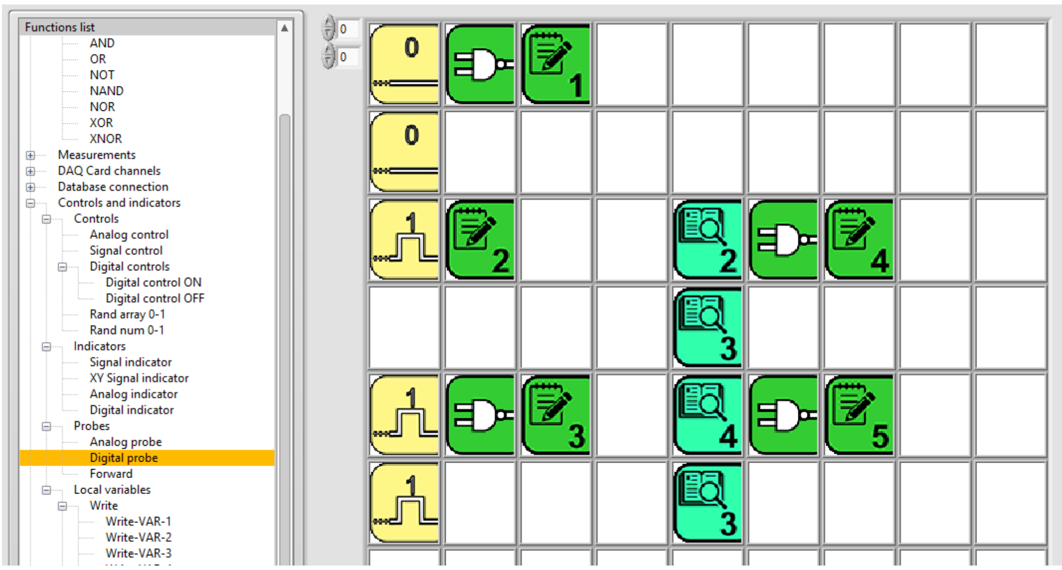


Figure 2.2.2: Notation of the logic function in the CORELA simulation editor

2.3 Simplification of the logic function using Boolean algebra laws

2.3.1) The schematic diagram of the logic function is shown in Figure 2.3.1.

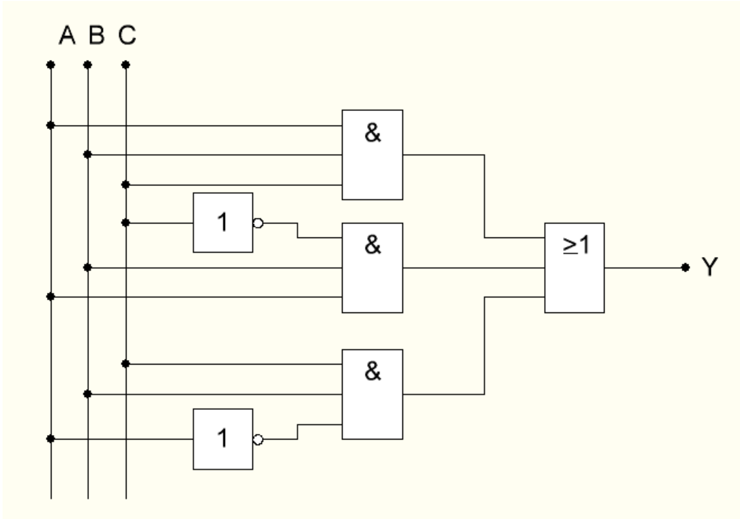


Figure 2.3.1: Schematic diagram of the logic function

The logic function presented in a schematic diagram shown in Figure 2.3.1 is described by the following equation and truth table:

Equation:

$$Y = A\,B\,C + A\,B\,\bar{C} + \bar{A}\,B\,C$$

Truth table:

A	B	C	Y
0	0	0	0
0	0	1	0
0	1	0	0
0	1	1	1
1	0	0	0
1	0	1	0
1	1	0	1
1	1	1	1

2.3.2) The simplified logic function is determined by the following equation.

Equation:

Y= BA+BC

2.3.3) The simplified logic function in purely NAND form is determined by the equation below.

Equation:

$$Y = \overline{\overline{B\,A + B\,C}} = \overline{(\overline{B\,A}) * (\overline{B\,C})}$$

The schematic diagram of the simplified logic function in purely NAND form and the electronic circuit is shown in Figure 2.3.2.

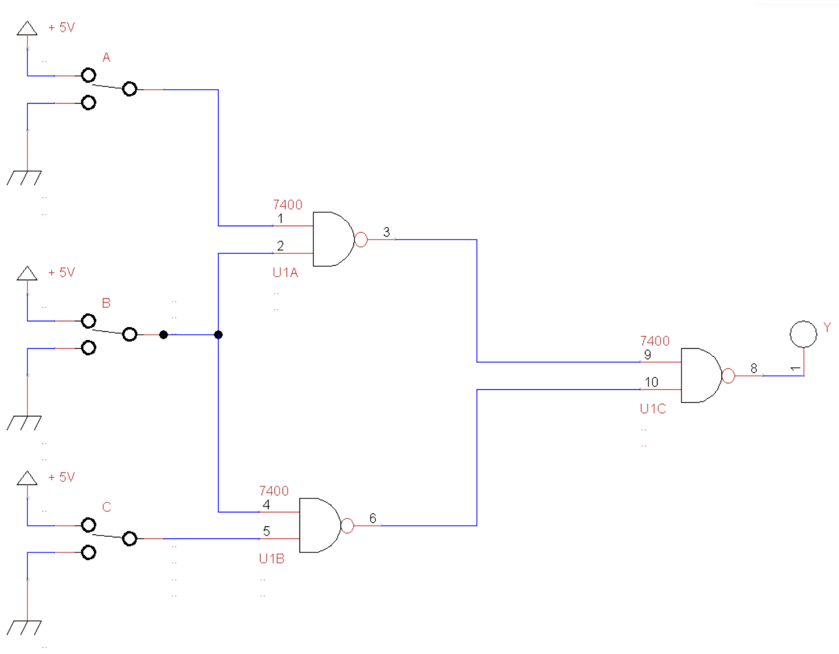


Figure 2.3.2: Schematic diagram of the simplified logic function in purely NAND form and the electronic circuit

2.3.3) Conduct a simulation of the logic function in purely NAND form in the CORELA platform.

In the CORELA platform editor, assemble the circuit according to Figure 2.3.2: (Schematic diagram of the logic function in purely NAND form and the electronic circuit.).

Note that it is necessary to test all combinations from the logical table (Truth table) from point 2.3.1, step by step and check the results. This means making simulation circuits for combinations of input variables from 000, 001, 010 to 111.

An example of a combination 101 is shown in Figure 2.3.3.

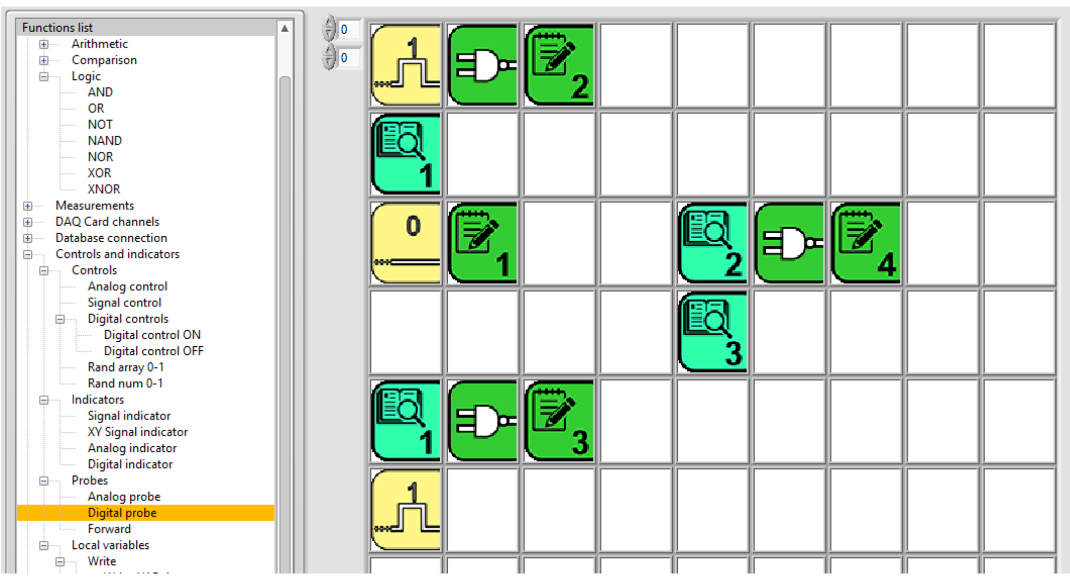


Figure 2.3.3: Notation of the logic function in the CORELA simulation editor

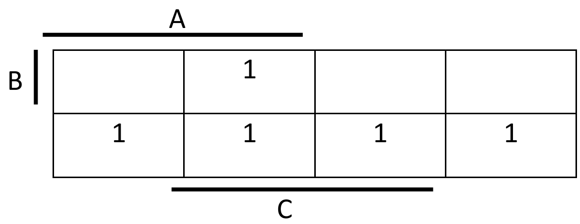
2.4 Simplification of the logic function using Veitch diagram

2.4.1) The logic function is determined by the following truth table:

Truth table:

A	B	C	Y
0	0	0	1
0	0	1	1
0	1	0	0
0	1	1	0
1	0	0	1
1	0	1	1
1	1	0	0
1	1	1	1

2.4.2) The Veitch diagram of the logic function:



2.4.3) Disjunctive normal form of the logic function:

Equation:

$Y = A\,C + \overline{B}$

2.4.4) Disjunctive normal form of the logic function implemented in purely NAND gate form is determined by the equation below.

$$Y = \left(\overline{\overline{A\,C + \overline{\overline{B}}}}\right) = \overline{(\overline{A\,C}) * (\overline{\overline{\overline{B}}})} = \overline{\overline{A\,C} * B}$$

The schematic diagram of the logic function in the full disjunctive normal form implemented in purely NAND gate form and the electronic circuit is shown in Figure 2.4.1.

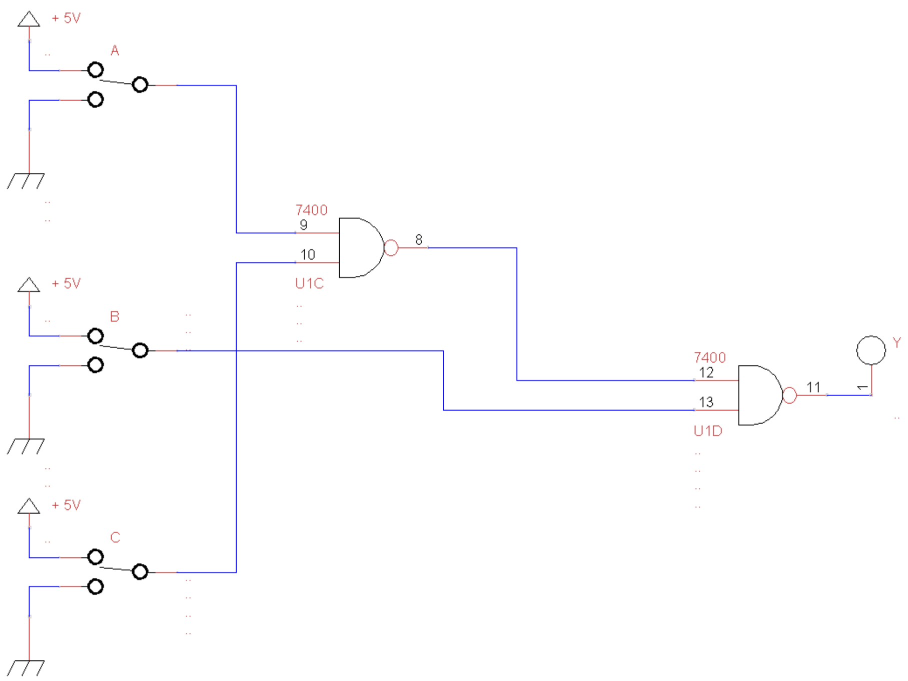


Figure 2.4.1: Schematic diagram of the logic function in disjunctive normal form implemented in purely NAND gate form and the electronic circuit

2.4.5) Conduct a simulation of the logic function in purely NAND form in the CORELA platform.

In the CORELA platform editor, assemble the circuit according to Figure 2.4.1: (Schematic diagram of the logic function in purely NAND form and the electronic circuit.).

Note that it is necessary to test all combinations from the logical table (Truth table) from point 2.4.1, step by step and check the results. This means making simulation circuits for combinations of input variables from 000, 001, 010 to 111.

An example of a combination 101 is shown in Figure 2.4.2.

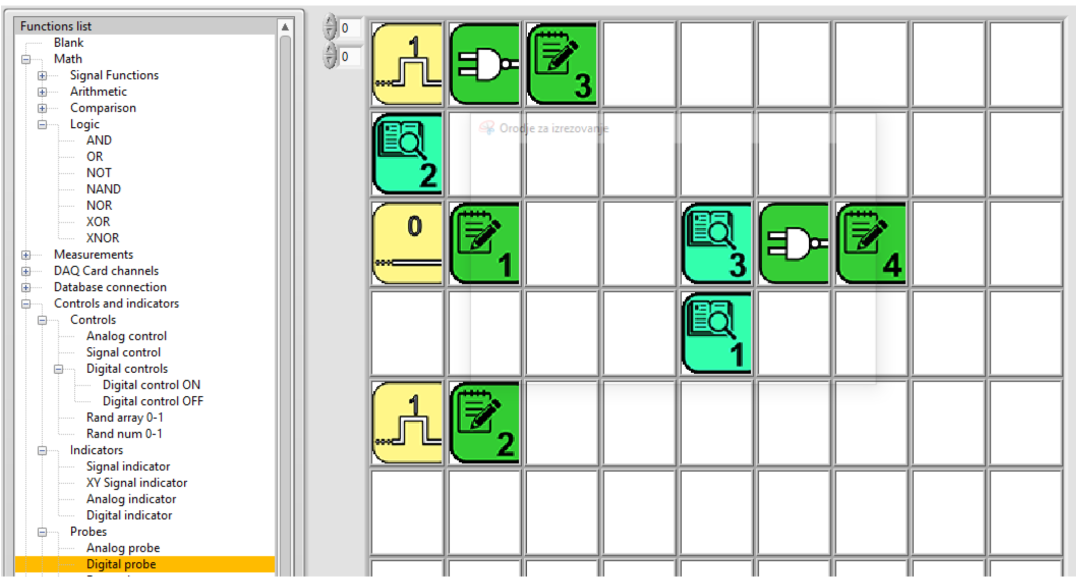


Figure 2.4.2: Notation of the logic function in the CORELA simulation editor

With the support of the Erasmus+ Programme of the European Union

3 Laboratory practise (student 3)

3.1 Realisation of the logic function

3.1.1) The device should start operating when sensor A and sensor B are both activated at the same time or sensor C is deactivated.

The logic function is determined by the equation and the truth table below.

Equation:

$$Y = (A\,B) + \overline{C}$$

Truth table:

A	B	C	Y
0	0	0	1
0	0	1	0
0	1	0	1
0	1	1	0
1	0	0	1
1	0	1	0
1	1	0	1
1	1	1	1

The schematic diagram of the logic function and the electronic circuit is shown in Figure 3.1.1.

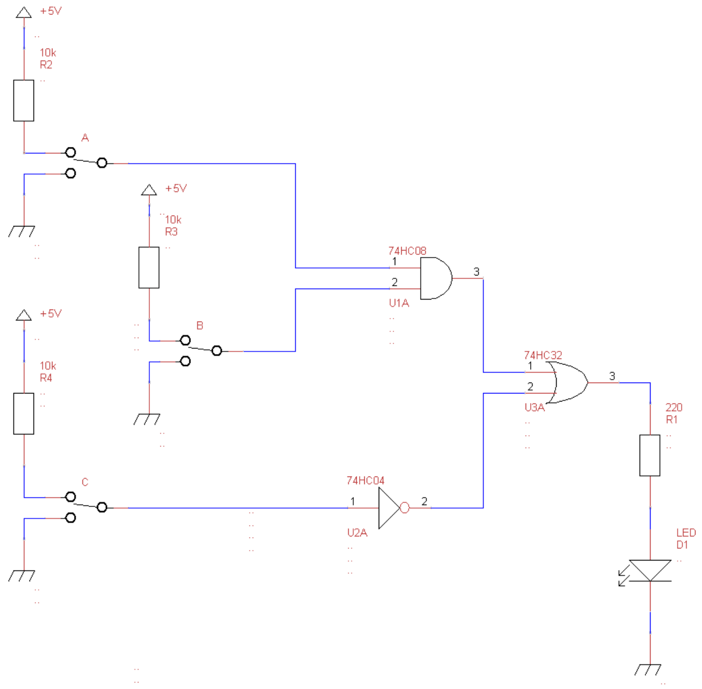


Figure 3.1.1: Schematic diagram of the logic function and the electronic circuit

Build/realise the described logic function and the electronic circuit on protoboard; use the appropriate chip-units and other electric elements.

Elements:
- protoboard,
- chip-units: IC 74HC08, 74HC04, 74HC32,
- 3 x 3 pole slide switch,
- 3 x resistor 10 kΩ,
- resistor 220 Ω,
- LED (Light Emitting Diode),
- wires,
- DC voltage power supply (5 V).

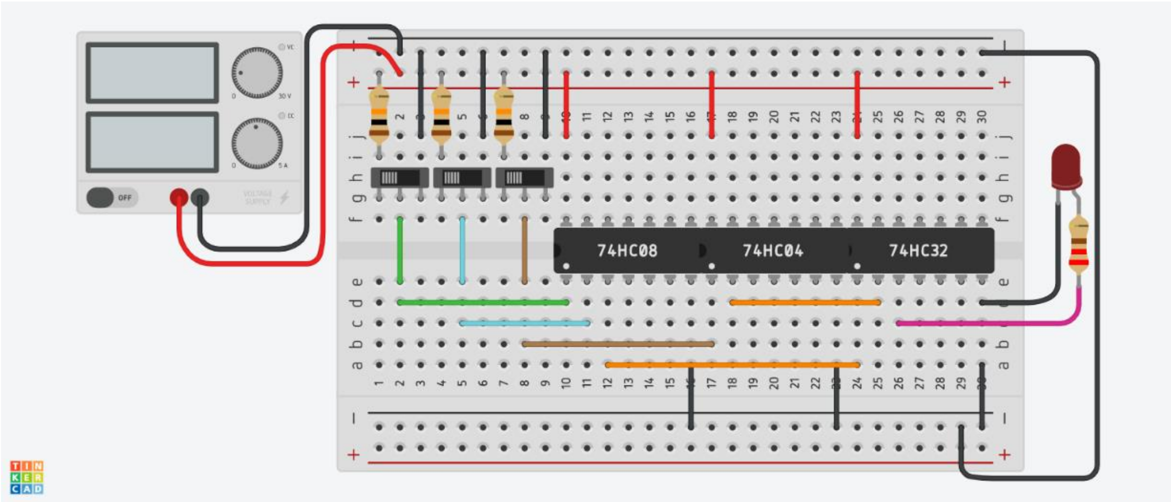


Figure 3.1.2: Scheme of the electric circuit on protoboard

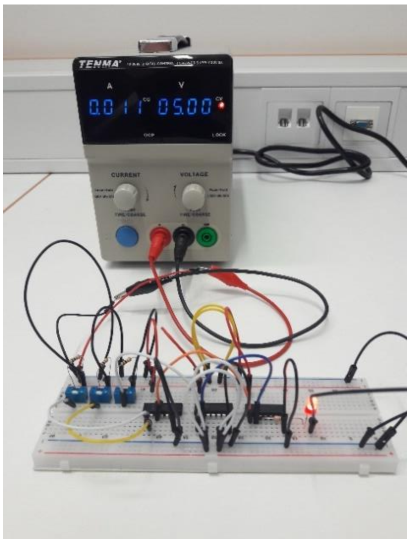


Figure 3.1.3: DC voltage power supply and the electric circuit on protoboard

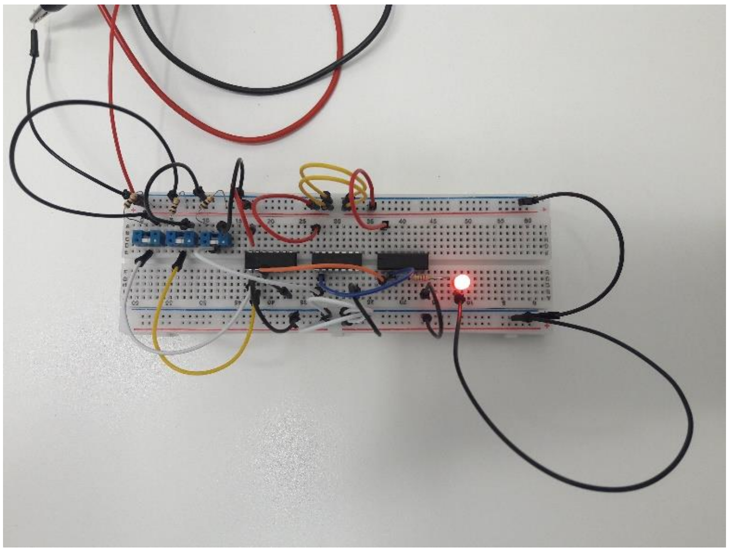


Figure 3.1.4: The electric circuit on protoboard

3.1.2) The logic function is described by the following equation:

$Y = \bar{A}\,B + C$

The logic function in purely NAND form is determined by the equation and truth table below.

Equation:

$$Y = \overline{\overline{AB + \overline{C}}} = \overline{\overline{AB} * \overline{\overline{C}}} = \overline{\overline{AB} * C}$$

Truth table:

A	B	C	Y
0	0	0	1
0	0	1	0
0	1	0	1
0	1	1	0
1	0	0	1
1	0	1	0
1	1	0	1
1	1	1	1

The schematic diagram of the logic function in purely NAND form and the electronic circuit is shown in Figure 3.1.5.

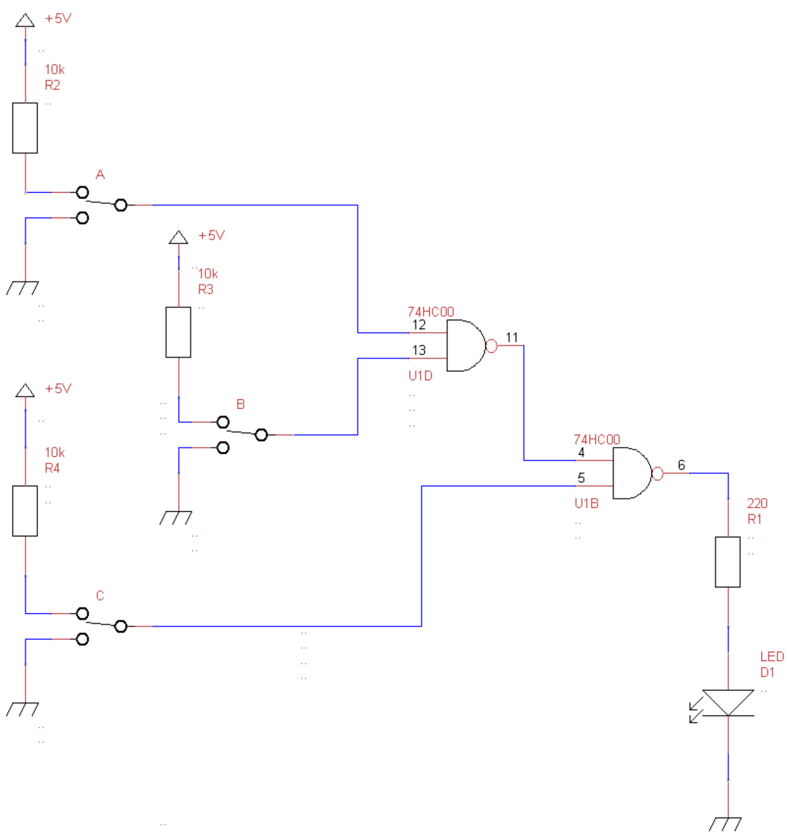


Figure 3.1.5: Schematic diagram of the logic function in purely NAND form and the electronic circuit

Build/realise the logic function in purely NAND form and the electronic circuit on protoboard; use the appropriate chip-units and other electric elements.

Elements:
- protoboard,
- chip-units: IC 74HC00,
- 3 x 3 pole slide switch,
- 3 x resistor 10 kΩ,
- resistor 220 Ω,
- LED (Light Emitting Diode),
- wires,
- DC voltage power supply (5 V).

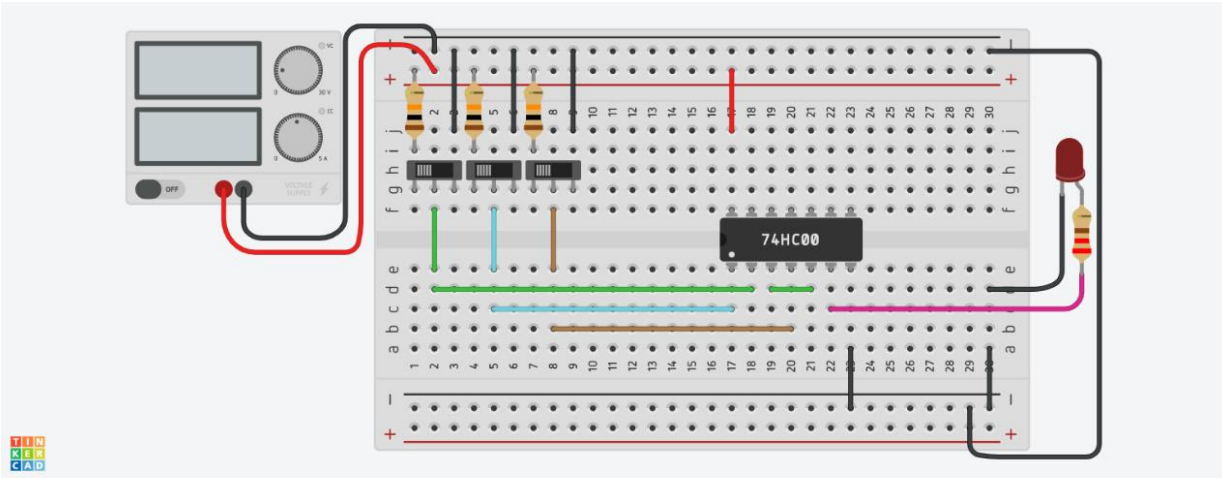


Figure 3.1.6: Scheme of the electric circuit on protoboard

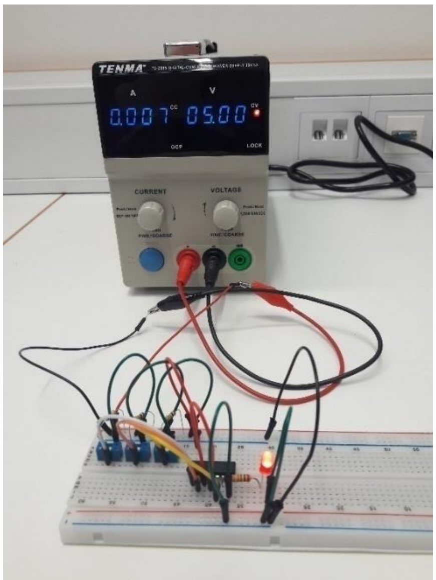


Figure 3.1.7: DC voltage power supply and the electric circuit on protoboard

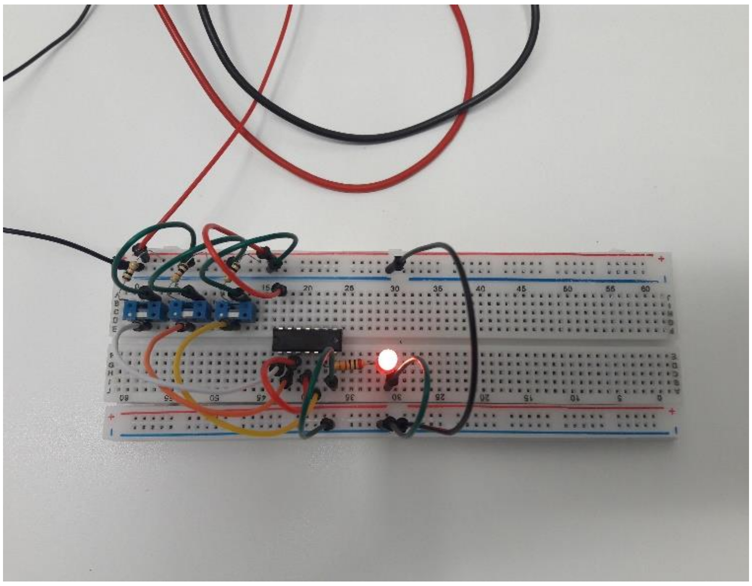


Figure 3.1.8: The electric circuit on protoboard

With the support of the Erasmus+ Programme of the European Union

3.2 Realisation of the logic function using NAND gates

3.2.1) The logic function is described by the following equation and truth table:

Equation:

$Y = \bar{A}\,B + C$

Truth table:

A	B	C	Y
0	0	0	0
0	0	1	1
0	1	0	1
0	1	1	1
1	0	0	0
1	0	1	1
1	1	0	0
1	1	1	1

3.2.2) The logic function in purely NAND form is determined by the equation below.

Equation:

$Y = \overline{\overline{\bar{A}\,B + C}} = \overline{\overline{\bar{A}B} * \bar{C}}$

The schematic diagram of the logic function in purely NAND form and the electronic circuit is shown in Figure 3.2.1.

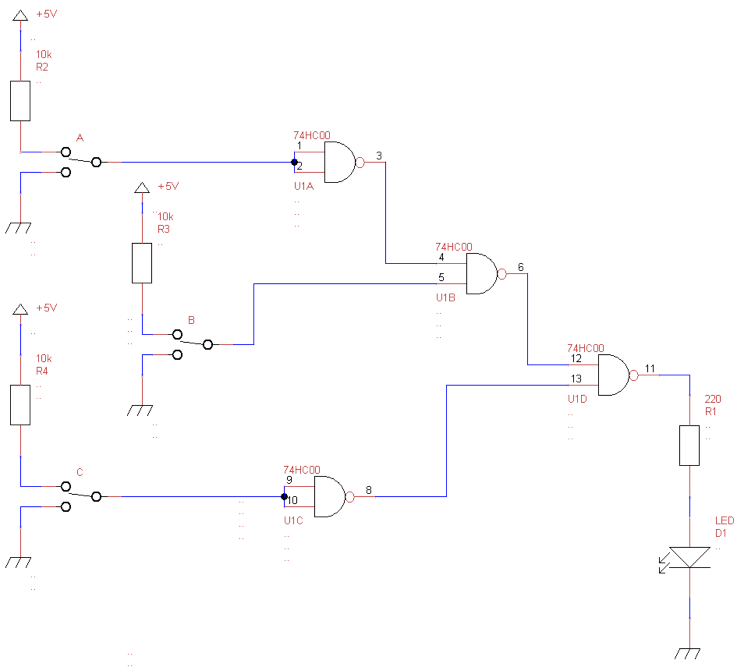


Figure 3.2.1: Schematic diagram of the logic function in purely NAND form and the electronic circuit

With the support of the Erasmus+ Programme of the European Union

Build/realise the logic function in purely NAND form and the electronic circuit on protoboard; use the appropriate chip-units and other electric elements.

Elements:
- protoboard,
- chip-units: IC 74HC00,
- 3 x 3 pole slide switch,
- 3 x resistor 10 kΩ,
- resistor 220 Ω,
- LED (Light Emitting Diode),
- wires,
- DC voltage power supply (5 V).

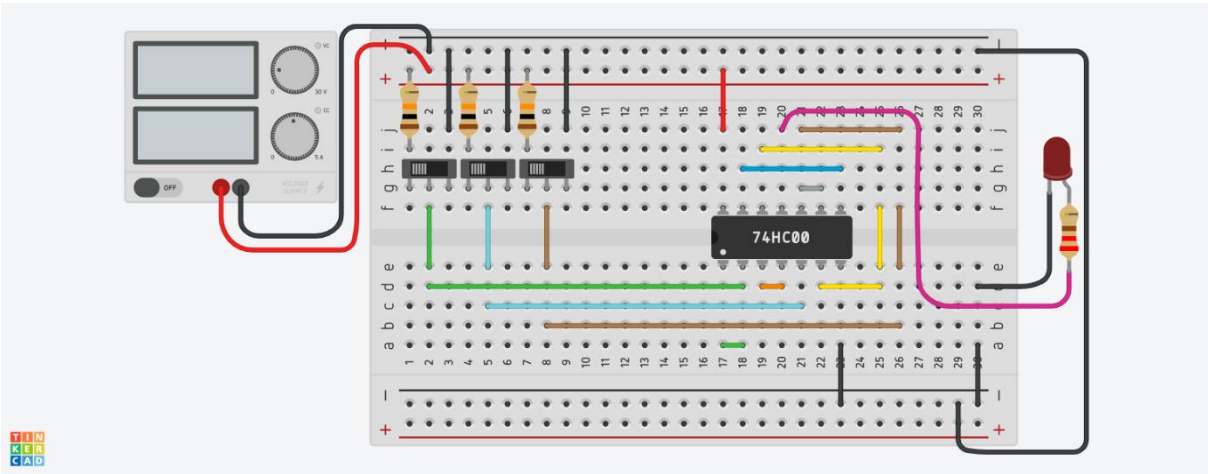


Figure 3.2.2: Scheme of the electric circuit on protoboard.

3.3 Simplification of the logic function using Boolean algebra laws

3.3.1) The schematic diagram of the logic function is shown in Figure 3.3.1.

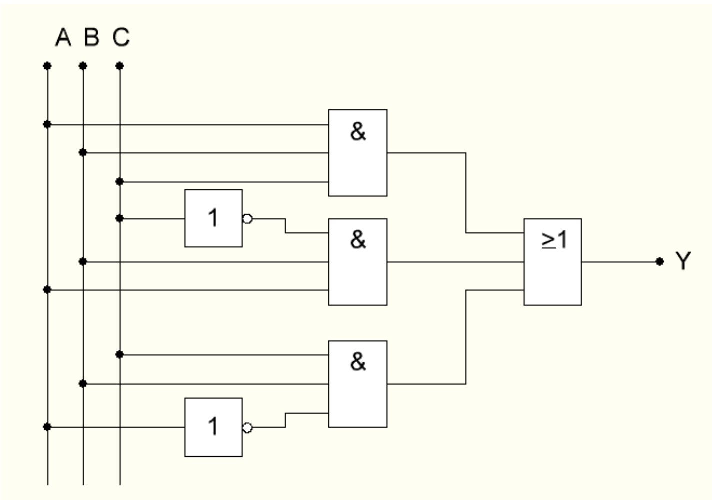


Figure 3.3.1: Schematic diagram of the logic function

The logic function presented in a schematic diagram shown in Figure 3.3.1 is described by the following equation and truth table:

Equation:

$$Y = A\,B\,C + A\,B\,\bar{C} + \bar{A}\,B\,C$$

Truth table:

A	B	C	Y
0	0	0	0
0	0	1	0
0	1	0	0
0	1	1	1
1	0	0	0
1	0	1	0
1	1	0	1
1	1	1	1

3.3.2) The simplified logic function is determined by the following equation.

Equation:

$$Y = B\,A \; + \; B\,C$$

3.3.3) The simplified logic function in purely NAND form is determined by the equation below.
Equation:

$$Y = \overline{\overline{B\,A + B\,C}} = \overline{\overline{(B\,A)} * \overline{(B\,C)}}$$

The schematic diagram of the simplified logic function in purely NAND form and the electronic circuit is shown in Figure 3.3.2.

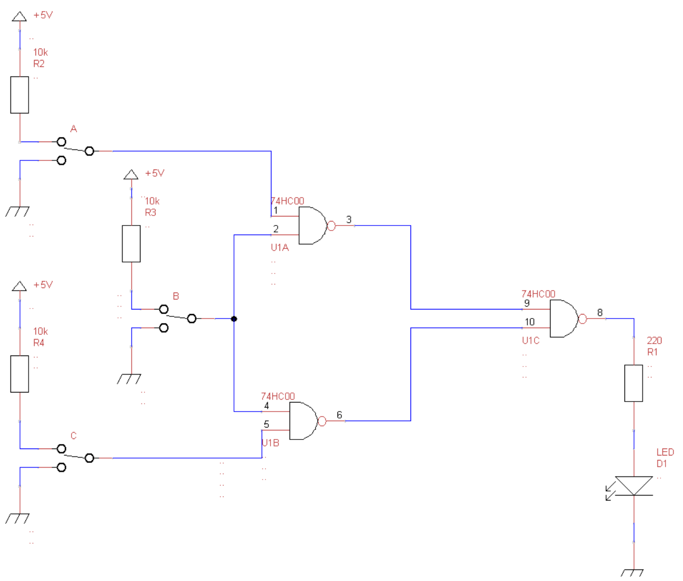


Figure 3.3.2: Schematic diagram of the simplified logic function in purely NAND form and the electronic circuit

Build/realise the simplified logic function implemented in purely NAND gate form and the electronic circuit on protoboard; use the appropriate chip-units and other electric elements.

Elements:
- protoboard,
- chip-units: IC 74HC00,
- 3 x 3 pole slide switch,
- 3 x resistor 10 kΩ,
- resistor 220 Ω,
- LED (Light Emitting Diode),
- wires,
- DC voltage power supply (5 V).

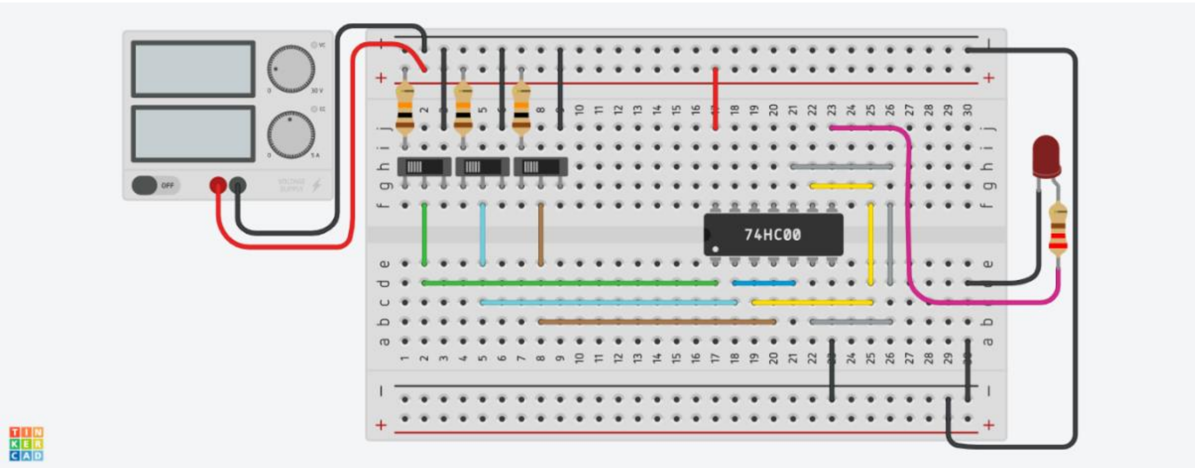


Figure 3.3.3: Scheme of the electric circuit on protoboard

3.4 Simplification of the logic function using Veitch diagram

3.4.1) The logic function is determined by the following truth table:

Truth table:

A	B	C	Y
0	0	0	1
0	0	1	1
0	1	0	0
0	1	1	0
1	0	0	1
1	0	1	1
1	1	0	0
1	1	1	1

3.4.2) The Veitch diagram of the logic function:

A

B				
B		1		
	1	1	1	1

C

3.4.3) The full disjunctive normal form of the logic function:

Equation:

$$Y = A\,C + \overline{B}$$

3.4.4) The full disjunctive normal form of the logic function implemented in purely NAND gate forms determined by the equation below.

$$Y = \left(\overline{\overline{A\,C + \overline{B}}}\right) = \overline{(\overline{A\,C}) * (\overline{\overline{B}})} = \overline{\overline{A\,C} * B}$$

The schematic diagram of the logic function is shown in Figure 3.4.

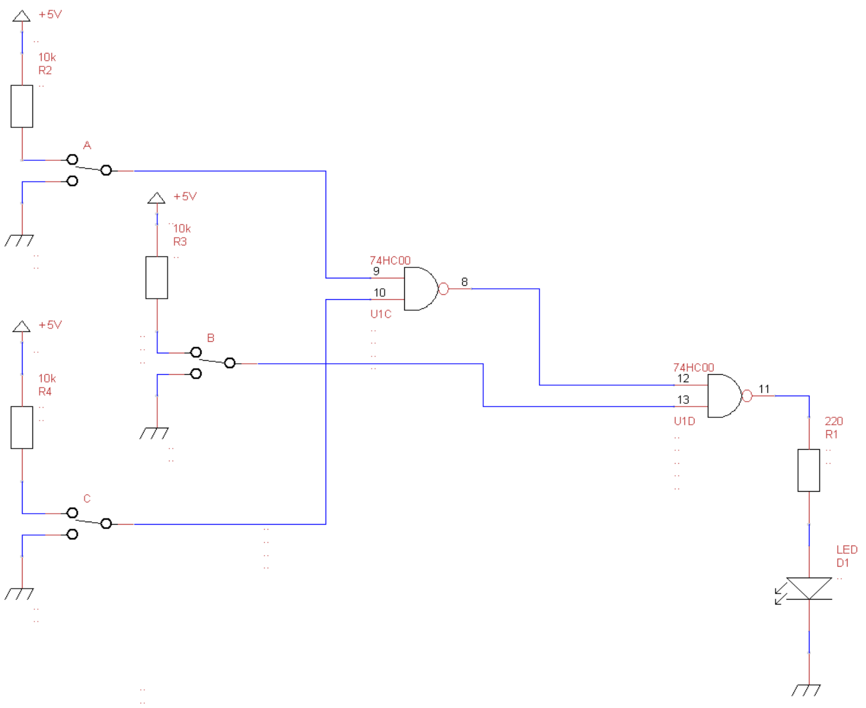


Figure 3.4.1: Schematic diagram of the logic function in the full disjunctive normal form implemented in purely NAND gate form and the electronic circuit

Build/realise the logic function in the full disjunctive normal form implemented in purely NAND gate form and the electronic circuit on protoboard; use the appropriate chip-units and other electric elements.

Elements:
- protoboard,
- chip-units: IC 74HC00,
- 3 x 3 pole slide switch,

With the support of the Erasmus+ Programme of the European Union

- 3 x resistor 10 kΩ,
- resistor 220 Ω,
- LED (Light Emitting Diode),
- DC voltage power supply (5 V),
-wires.

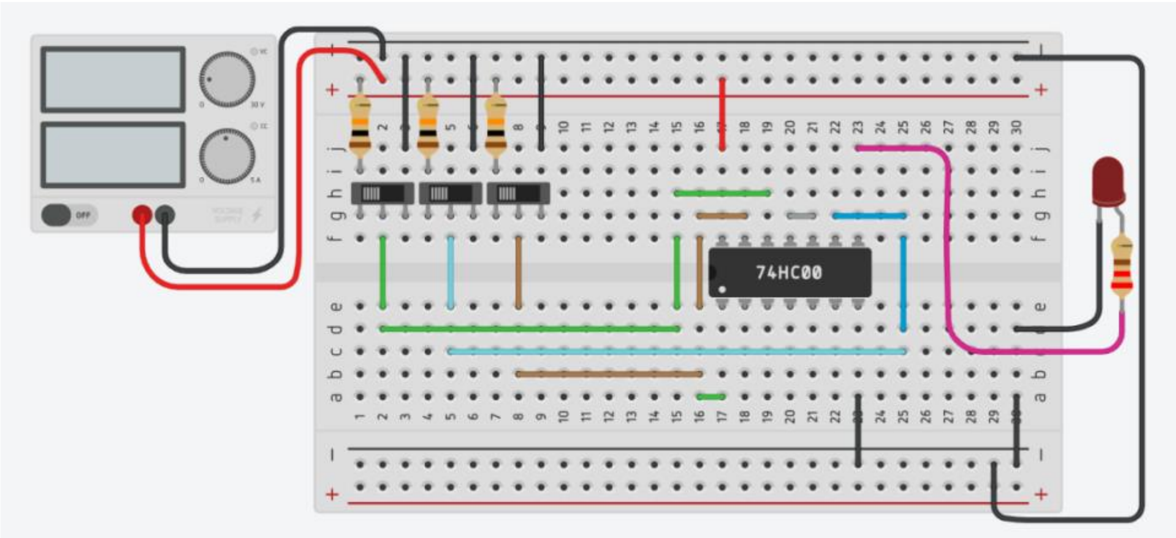


Figure 3.4.2: Scheme of the electric circuit on protoboard

Video material:

https://www.youtube.com/watch?v=EyWW7VOmR0c

VI PWM DC motor control

1. Introduction

PWM (Pulse-width modulation) control is a very commonly used method for controlling the power across loads. This method is very easy to implement and has high efficiency. PWM signal is essentially a high frequency square wave (typically greater than 1KHz). The duty cycle of this square wave is varied in order to vary the power supplied to the load. Duty cycle (hereinafter referred to as D) is usually stated in percentage and it can be expressed using the equation: D[%] = (TON/(TON + TOFF)) *100. where TON is the time for which the square wave is high and TOFF is the time for which the square wave is low. When duty cycle is increased the power dropped across the load increases and when duty cycle is reduced, power across the load decreases. The block diagram of a typical PWM power controller scheme is shown in Figure 1.1.

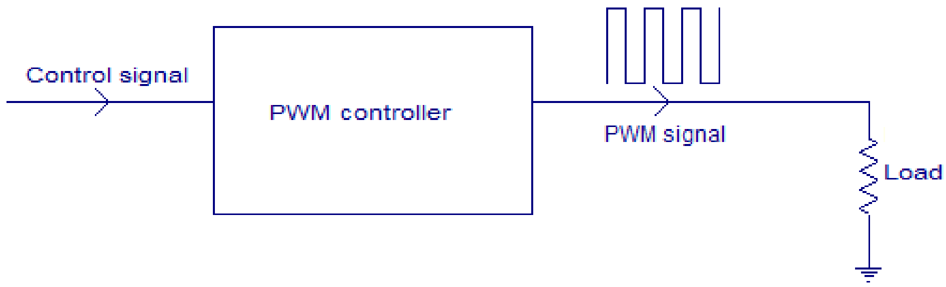


Figure 1.1

Control signal is what we give to the PWM controller as the input. It might be an analog or digital signal according to the design of the PWM controller. The control signal contains information on how much power has to be applied to the load. The PWM controller accepts the control signal and adjusts the duty cycle of the PWM signal according to the requirements. PWM waves with various duty cycle are shown in the Figure 1.2.

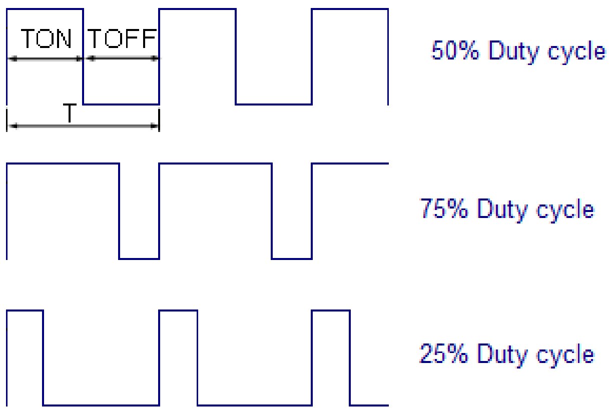


Figure 1.2

In the above wave forms you can see that the frequency is same but ON time and OFF time are different.

2. PWM control for DC motor (student 1)

Task: Determine the values of TON and TOFF as well as the total time TON + TOFF = T where T is period of the rectangular waveform with frequency of 1000 Hz. Specified times should be set for D_{25}[%]=25, and D_{75}[%]=75 of the total clock time.

Task: Calculate the values of the analog voltage control signal (U_D) on the basis of which the PWM control device constructs a PWM signal, if the range of values of the analog signal is 0-5V, where a voltage of 5V indicates a duty cycle of 100%. Also the 0-5V voltage interval corresponds after the AD conversion (10 bits) to the numeric interval 0-1023.

Note: Due to different types of DC motors, we cannot set a general formula for the dependence of their speed on the amount of duty cycle, but we can say that a higher duty cycle also means a higher motor speed.

Note: CORELA platform controls the ARDUINO module and the <PWM> function controls the PWM outputs of the module. We forward the amount of the duty cycle to this function and the function itself does the rest of the work in order to obtain the correct PWM waveform of the control signal.

Task: Draw the waveforms of the PWM signal for both required cases with the indicated times TON and TOFF (μs)

For a given PWM signal frequency (f) of 1kHz, the formula for determining the period (T) is:

$$T = \frac{1}{f} \cdot 10^6 [\mu s]$$

TON and TOFF times are calculated according to the formulas:

$$TON = \frac{D[\%]}{100} \cdot T\ [\mu s]$$

$$TOFF = T - TON\ [\mu s]$$

The input voltage of the control signal is calculated according to the formula:

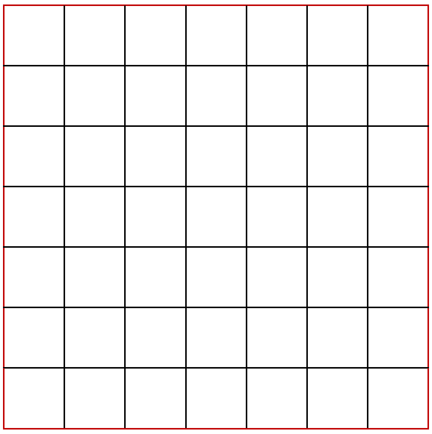
$$\mathrm{U}_D = 5\mathrm{V} \cdot \frac{D[\%]}{100} = [V]$$

Fill in Table 1 with the calculated data.

Table 1: PWM signal parameters

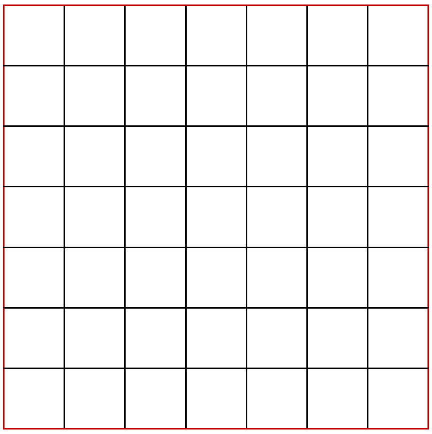
D [%]	T [μs]	TON [μs]	TOFF [μs]	U_D [V]
25				
75				

For a 25% duty cycle, draw the waveform of the PWM signal with the indicated times in the Grid 1.



Grid 1

For a 75% duty cycle, draw the waveform of the PWM signal with the indicated times in the Grid 2.



Grid 2

Question 1: How does the delivered electricity to the consumer depend on the PWM signal Duty Cycle?

Question 2: What value of the PWM signal is measured by the voltmeter in DC mode?

2.1 Comparation of the results

Sign in to the CORELA platform to enter values. The values of the period, ON and OFF times, and the mean values of the control voltage are entered and send to the educational platform according to the procedure explained in section 5 of this manual.

Compare the obtained values with those obtained by Student 2 and Student 3.

3. Simulation of PWM control for DC motor (student 2)

Syntax used in the rest of the manual:

<XXX> name of the available executive function in CORELA platform, or a button

<XXX> name of the executive function in CORELA platform when it is set on desktop

[XXX.YY] hierarchical path to the available executive function in CORELA platform

In this part of the exercise, we perform a simulation of the ideal electrical circuit.

The aim of the simulation is to check the theoretical calculations from chapter 2. The simulation of the electrical circuit is realized with the CORELA virtual platform.

At first by USB port on computer you must connect, well done prepared (ASRL4::INSTR), ARDUINO UNO circuit bord.

Run the application and press the button <Continual mode>.

After that insert <Analog control> control located in the [Functions list->Controls and indicators->Controls]. Record analog value of 6 (one of PWM channels on ARDUINO UNO circuit bord). Insert again <Analog control> control. Record duty cycle analog value of 0,25.

Insert <PWM> function located in the [Functions list->DAQ card channels] into place according to the picture. This control needs two input values, previously recorded and, realized by two <Analog control> controls.

Right from <PWM> function insert <Signal indicator> indicator, located in the [Functions list->Controls and indicators->Indicators].

Repeat the entire procedure according to the figure and change only the <Analog control> control value for the PWM channel to number 5.

The layout in the application is shown in Figure 3.1.

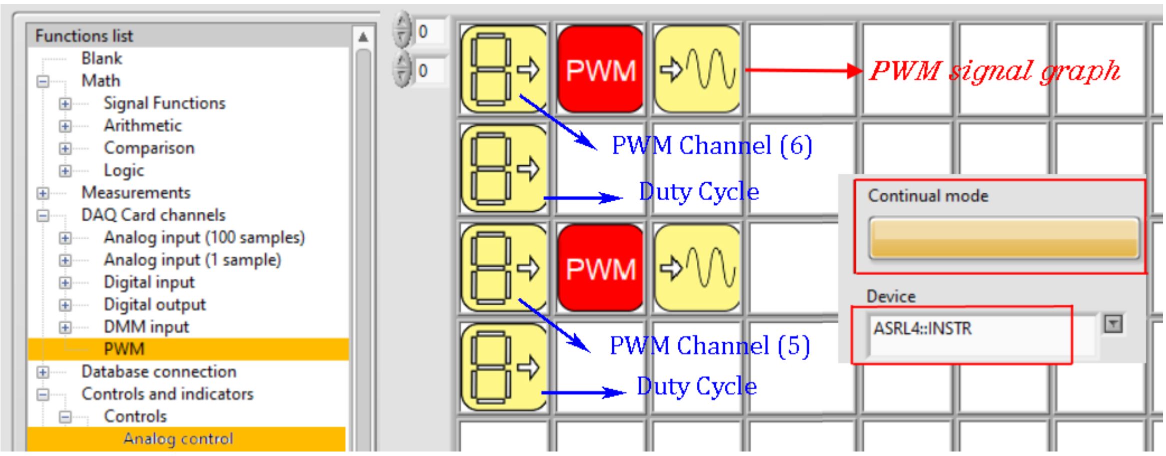


Figure 3.1

Connect the oscilloscope to the selected PWM channels (5 and 6) on the UNO board. Connect the voltmeter according to the connection diagram.

Figure 3.2 shows a schematic diagram for performing the simulation.

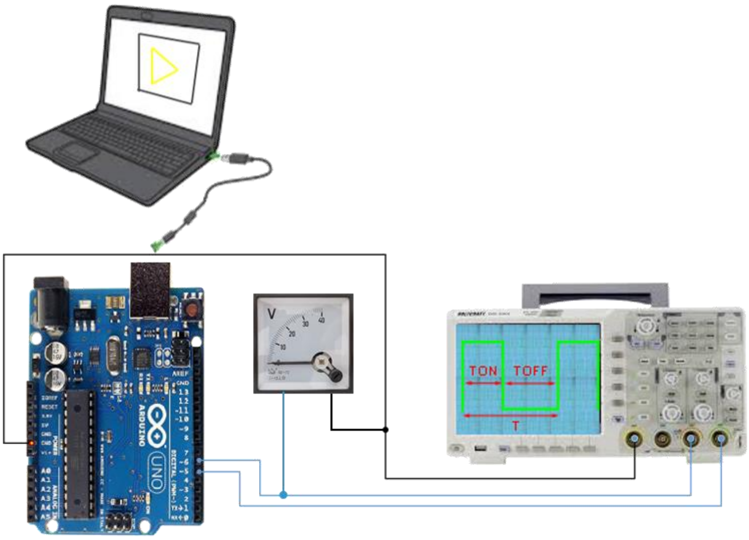
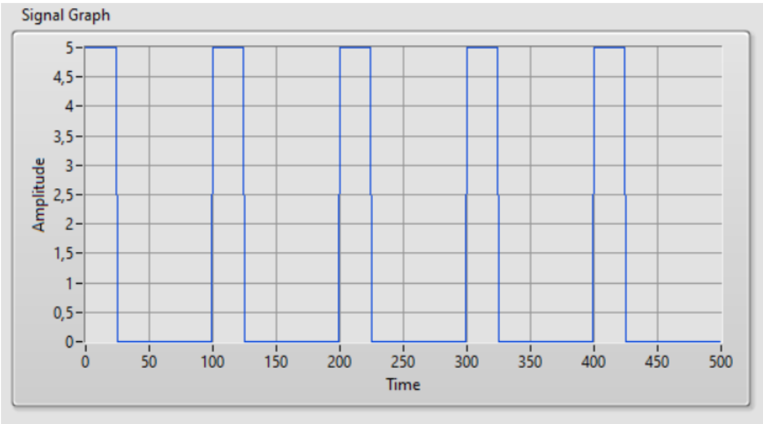


Figure 3.2

Step 1)

Compare the waveform of Graph 1 with the shape drawn in the Grid 1 of Chapter 2. According to the waveform on the oscilloscope screen, determine the times T, TON and TOFF, and enter them in Table 2.

Voltage reading (U_D) on the voltmeter also enter in Table 2.



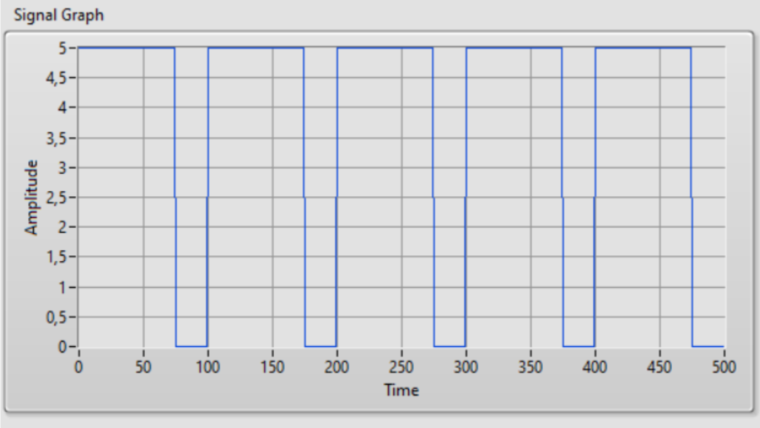
Graph 1

Step 2)

Change analog control values for duty cycle to 0,75.

Compare the waveform of graph 2 with the shape drawn in the grid 2 of Chapter 2. According to the waveform on the oscilloscope screen, determine the times T, TON and TOFF, and enter them in Table 2.

Voltage reading (U_D) on the voltmeter also enter in Table 2.



Graph 2

Table 2: PWM signal parameters

D [%]	T [µs]	TON [µs]	TOFF [µs]	U_D [V]
25				
75				

Question 1: Why do we have to determine the T, TON and TOFF times from the oscilloscope display and not from the PWM function output graph?

Question 2: Why does the operating motor not mind the rectangular shape of the PWM voltage signal instead of the DC voltage of the constant level?

3.1 Comparation of the results

Sign in to the CORELA platform to enter values. The values of the period, ON and OFF times, and the mean values of the control voltage are entered and send to the educational platform according to the procedure explained in section 5 of this manual.

Compare the obtained values with those obtained by Student 1 and Student 3.

Funded by the
Erasmus+ Programme
of the European Union

4. Practical realization and measurement (student 3)

This part of the exercise is related to the practical realization of the electrical circuit and performing realistic measurements. The aim is to test the PWM control once again, but this time in realistic conditions. To perform the exercise, we use the ARDUINO platform and equipment shown in the Figure 4.1.

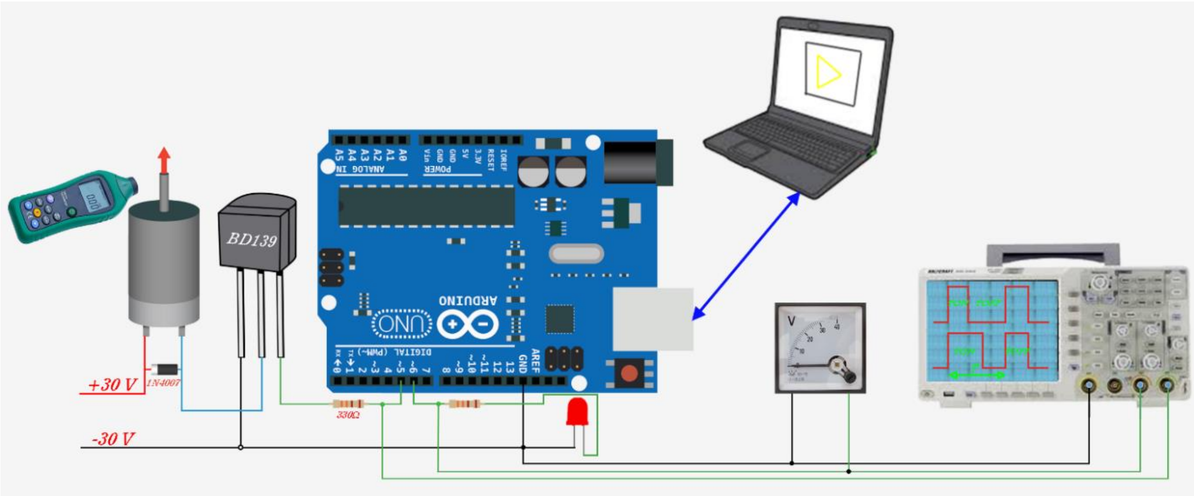


Figure 4.1

The following hardware and software are used to perform the experiment:

1. Experimental board
2. DC power supply (0-30 V)
3. Arduino UNO / NANO electronic board with CORELA platform
4. Electrical resistor 330 Ω x2
5. Transistor BD139 / 2N2222 (I_{cmax}=1,5A / 800 mA)
6. Diode 1N4007
7. DC motor 30V / max. 200 mA /rated speed 1000 rpm
8. LED red
9. DC voltage meter
10. Oscilloscope
11. Electronic revolute counter
12. Computer with CORELA virtual platform

Note: The frequency of the PWM signal is 1000 Hz. An LED was added to visualize the control with a PWM signal.

Funded by the
Erasmus+ Programme
of the European Union

Step 1)

At first by USB port on computer you must connect, well done prepared (ASRL4::INSTR), ARDUINO UNO circuit bord.

Run the application and press the button <Continual mode>.

After that insert <Analog control> control located in the [Functions list->Controls and indicators->Controls]. Record analog value of 6 (one of PWM channels on ARDUINO UNO circuit bord).

Insert again <Analog control> control. Record duty cycle analog value of 0,25.

Insert <PWM> function located in the [Functions list->DAQ card channels] into place according to the picture. This control needs two input values, previously recorded and, realized by two <Analog control> controls.

Right from <PWM> function insert <Signal indicator> indicator, located in the [Functions list->Controls and indicators->Indicators].

Repeat the entire procedure according to the figure and change only the <Analog control> control value for the PWM channel to number 5.

The layout in the application is shown in Figure 4.2.

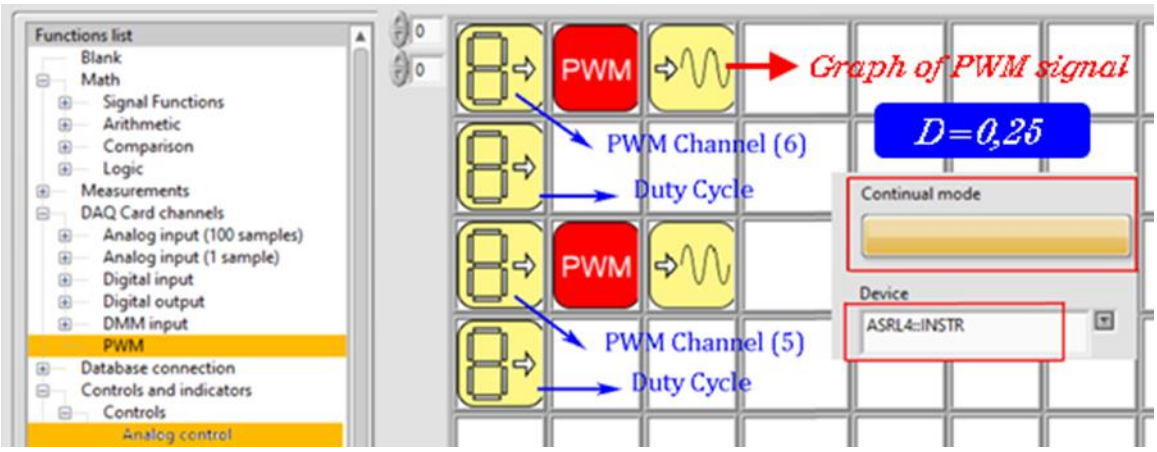


Figure 4.2

Read the speed (v_D) on the electronic counter and enter it in Table 3.

Voltage reading (U_D) on the voltmeter also enter in Table 3.

Determine the times T, TON and TOFF according to the waveform on the oscilloscope screen and enter them in Table 3.

Step 2)

In CORELA, change only the value of both analog controls for the duty cycle to 0,75.

Read the speed (v_D) on the electronic counter and enter it in Table 3.

Voltage reading (U_D) on the voltmeter also enter in Table 3.

Determine the times T, TON and TOFF according to the waveform on the oscilloscope screen and enter them in Table 3.

Table 3: PWM motor control

D [%]	T [µs]	TON [µs]	TOFF [µs]	U_D [V]	v_D [o/min]
25					
75					

Question 1: Is the change in rotational speed proportional to the changed values of the Duty Cycle?

Question 2: Write formulas for determining the speed of one DC motor with serial and one DC motor with parallel excitation and connect them with the answer from question 1.

4.1 Comparation of the results

Sign in to the CORELA platform to enter values. The values of the period, ON and OFF times, and the mean values of the control voltage are entered and send to the educational platform according to the procedure explained in section 5 of this manual.

Compare the obtained values with those obtained by Student 1 and Student 2.

5. Procedure of writing and sending dana to the educational platform

Step 1: Write and saving the data

From the menu [Functions list->Controls and indicators->Controls] select the control <Analog control> and with the left mouse-click on an empty field on the desktop, we set the control.

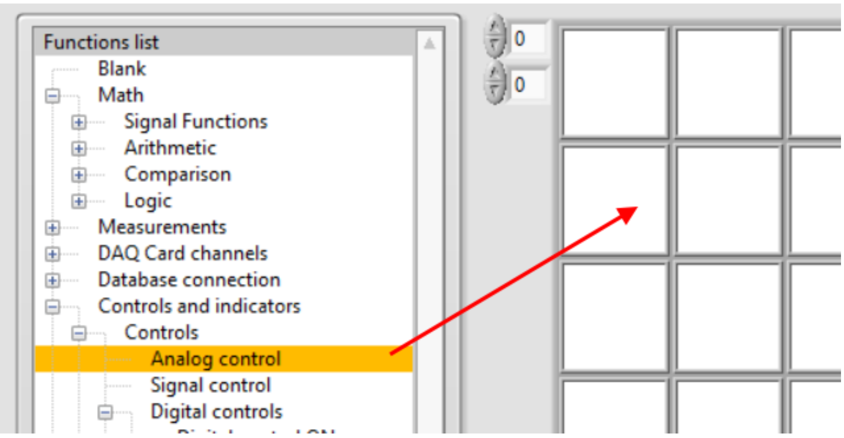


Figure 5.1 Setting a <Analog control> control

A pop-up window opens where we enter the frequency values of the PWM signal (Figure 5.2).

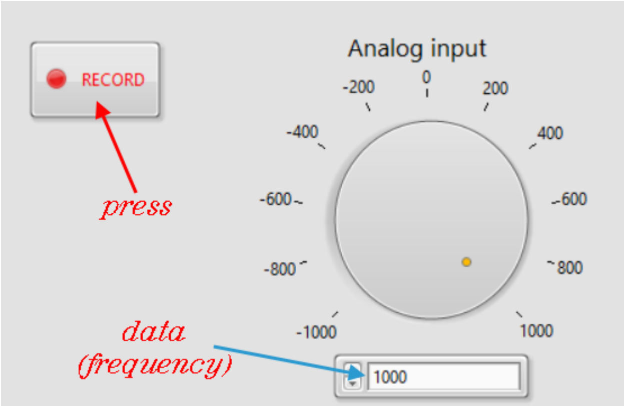


Figure 5.2 Data entry

We record the data by pressing the <Record> button (Figure 5.2).

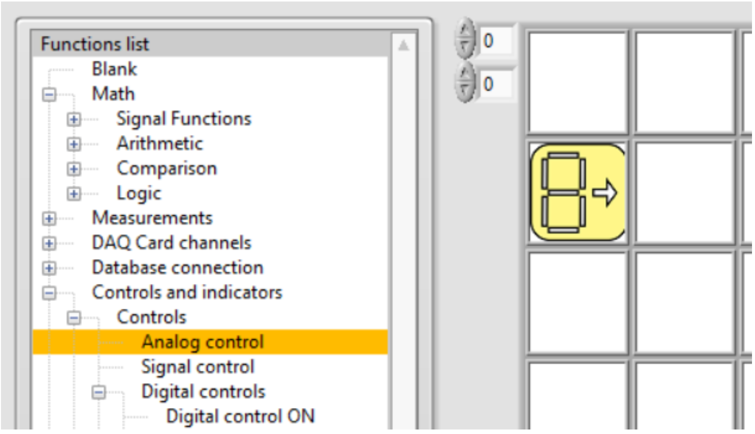


Figure 5.3 Screen layout after data capture

Step 2: Reviewing the recorded data

By setting the <Analog indicator> indicator, in the field to the right, next to <Analog control> control (Figure 5.4), the data entered on the CORELA platform can be accessed.

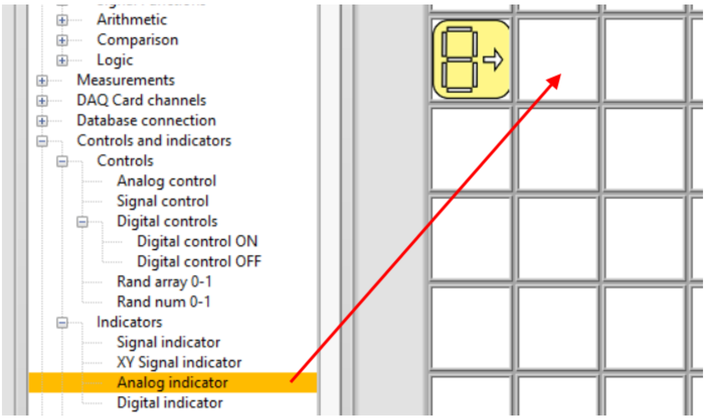


Figure 5.4 Setting a <Analog indicator> indicator

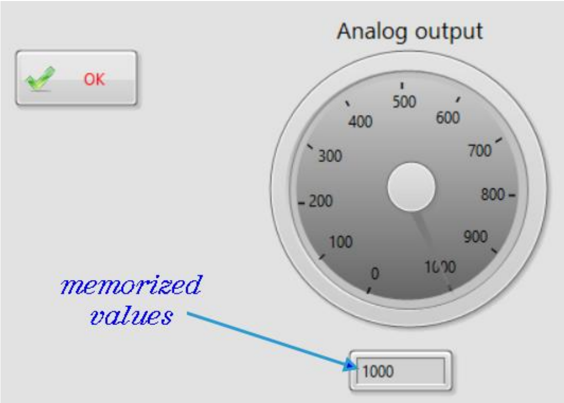


Figure 5.5 Display of entered frequency values

Then press the <OK> button and return to desktop (Figure 5.6).

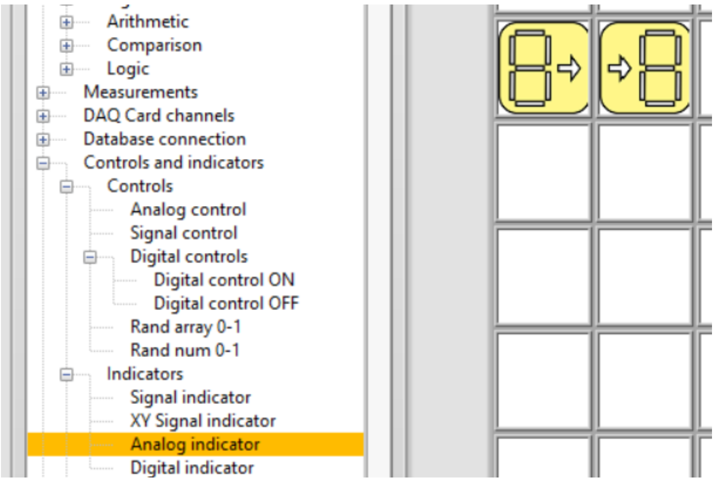


Figure 5.6 Desktop after setting < Analog indicator > indicator

Step 3: Enter time T, TON, TOFF and mean voltage values U_D

The procedure is repeated to enter T, TON, TOFF and U_D. By setting a new <Analog control> control and a new <Analog indicator> indicator on the desktop, the CORELA platform records and reviews all entered data (Figure 5.7).

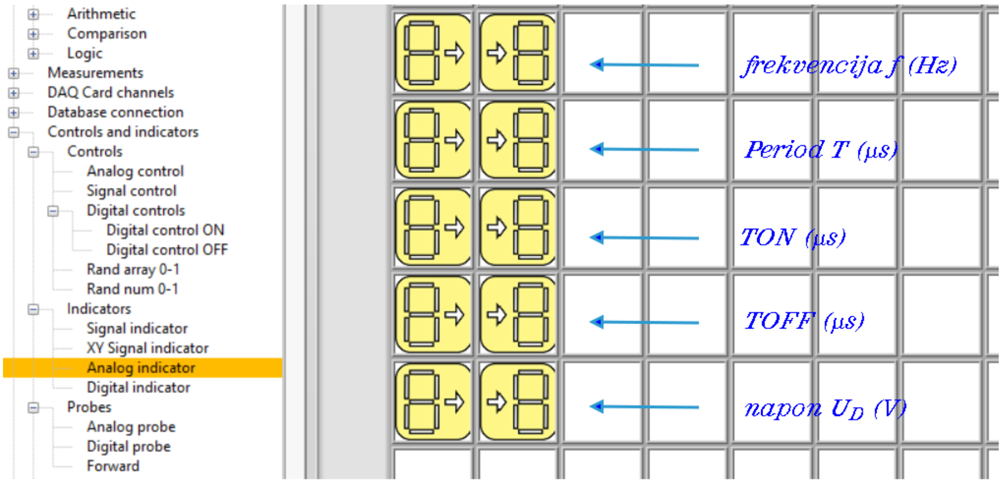


Figure 5.7 Display of CORELA platform task dana entry

Step 4: Sending dana to the educational platform

Sending data to the educational platform (Moodle) is done by selection one of output channels from the menu [Functions list->Database connection->Output channel]. To send the frequency data, select <CH out 1> (Figure 5.8).

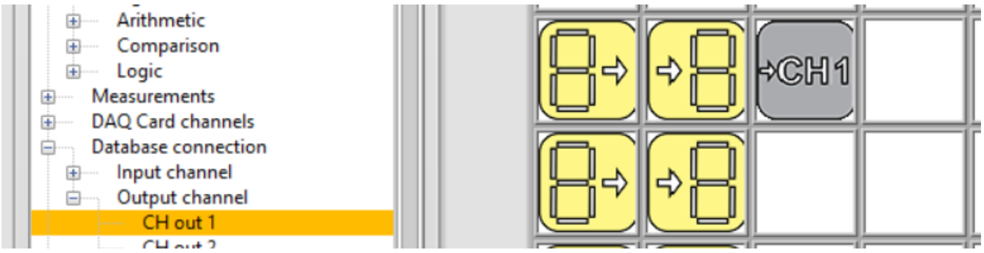


Figure 5.8 Sending data to the educational platform

All other dana, is sent to the educational platform through the remaining output channels (<CH out2>, <CH out3>, …).

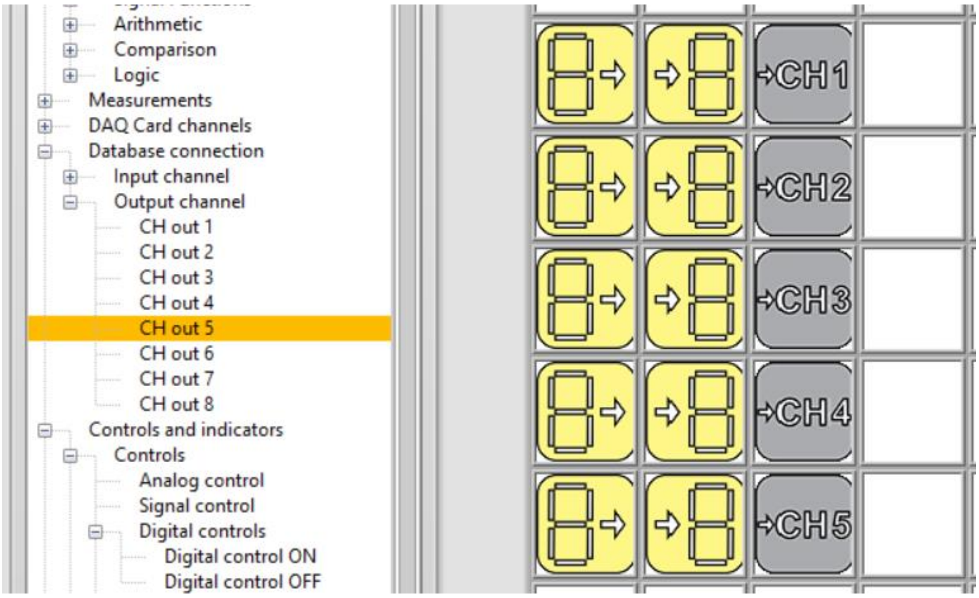


Figure 5.9 Screen after sending the results data on the educational platform

After completing the task, enter the „chat room".

Compare and comment on the results obtained in three ways:

- *by calculation (Student 1)*
- *by simulation (Student 2)*
- *by measurement (Student 3)*

Video material:

https://www.youtube.com/watch?v=DWw8bTcjo8s

VII Maximum Power Transfer

1. Introduction

When we connect a load resistance, R across the output terminals of the power source, the impedance of the load will vary from an open-circuit state to a short-circuit state resulting in the power being absorbed by the load becoming dependent on the impedance of the actual power source. Then for the load resistance to absorb the maximum power possible it has to be “Matched” to the impedance of the power source and this forms the basis of maximum power transfer.

The Maximum Power Transfer Theorem is useful circuit analysis method to ensure that the maximum amount of power will be dissipated in the load resistance R when the value of the load resistance is exactly equal to the resistance of the power source RI. The relationship between the load impedance and the internal impedance of the energy source will give the power in the load. Consider the circuit below.

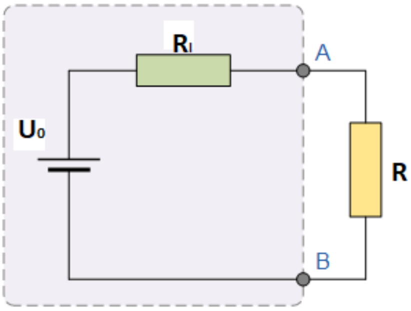


Figure 1.1: Replacement scheme of real source with load

Maximum power transfer example

U_0 = 100 V

R_i = 25 Ω

R is variable between 0 – 100Ω

By using the following Ohm’s Law equations:

$I = \frac{U_0}{R+R_i}$ and $P = I^2R$; $P = \frac{U_0^2}{(R+R_i)^2}R$

we can complete the following table to determine the current and power in the circuit for different values of load resistance.

R (Ω)	I (A)	P (W)
0	4.0	0
5	3.3	55
10	2.8	78
15	2.5	93
20	2.2	97
25	2.0	100
30	1.8	97
40	1.5	94
60	1.2	83
100	0.8	64

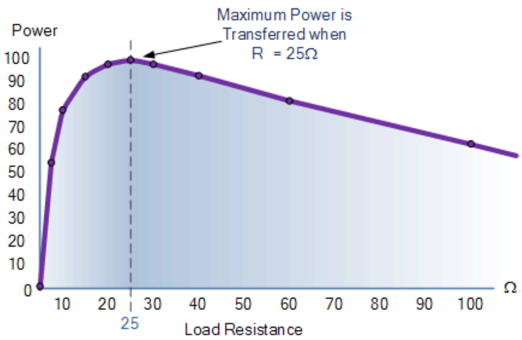


Figure 1.2: Graph of power against load resistance

Funded by the Erasmus+ Programme of the European Union

2. Calculation of maximum power transfer (student 1)

The student will analyze an electrical circuit composed of a real source and variable resistance (figure 2.1).

Task: Calculate the power *P* dissipated in resistance according to table. Parameters of source are e.g.

U_0 = 12V and R_i = 10Ω (this data is given by student 3 who performs measurements with a real source).

Figure 2.1: Circuit for maximum power transfer analysis

R (Ω)	P (W)
1	
2	
4	
6	
8	
10	
12	
14	
16	
20	
24	
30	
40	
60	
100	

The second task is to draw a graph of the dependence of power and resistance.

2.1 Comparation of the results

Log in to the CORELA data submission platform. Enter the values of maximum power and associated resistance and send them to the educational platform (see instructions in Chapter 5).

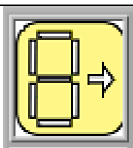
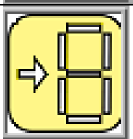
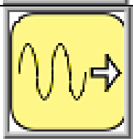
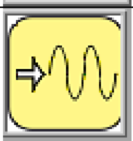
Compare the obtained values with those obtained from students 2 and 3.

3. Maximum power transfer simulation (student 2)

In this part of the exercise, we perform a simulation of the ideal electrical circuit.
The aim of the simulation is to check the theoretical calculations from chapter 2. The simulation of the electrical circuit is realized with the CORELA virtual platform.
Therefore, we will show the formula below in a blockchain.

$$P = \frac{U_0^2}{(R + R_i)^2} R$$

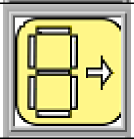
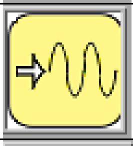
Run the application and insert Analog control located in the functions list under the menu Controls and indicators->Controls. Record analog value of 12 (U_0 = 12V). On second place insert Square (Math->Arithmetic). Keep to follow instructions below.

1.		Controls and indications->Controls->Analog control	Record analog value of U_0=12V.
2.		Math->Arithmetic-> Square	
3.		Controls and indications->Probes->Analog probe	The value of U_0^2.
4.		Controls and indications->Controls->Signal control	Resistance R values.
5.		Math->Arithmetic-> Multiply	
6.		Controls and indications->Indicators->Signal indicator	Result of $U_0^2 \times R$.
7.		Controls and indications->Local variables->Write-VAR	Local variables 1 – numerator values. Local variables2 – R values.

So, we get the value of the numerator.

Figure 3.1: Function layout

Let's move on to the denominator now.

1.		Controls and indications->Local variables->Read-VAR	Read local variables 2.
2.		Controls and indications->Controls->Analog control	Record the value R_i=10Ω
3.		Math->Arithmetic-> Add	
4.		Controls and indications->Indicators->Signal indicator	1.Result of $R+R_i$ 2. Result of $(R+R_i)^2$
5.		Math->Arithmetic-> Square	
6.		Controls and indications->Local variables->Write-VAR	Local variables 3 – denominator values.

So, we get the value of the denominator.

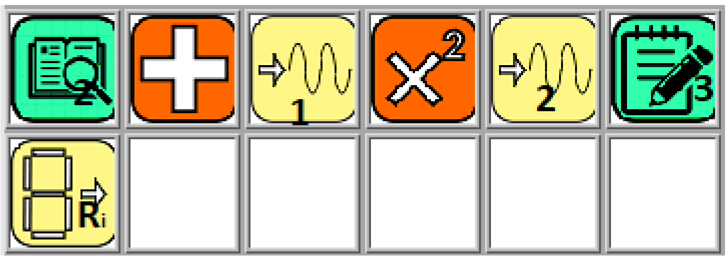
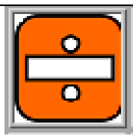
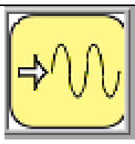
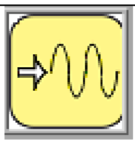


Figure 3.2: Function layout

Finally, by dividing the two values we get the result and power characteristic depending on the resistance attached.

1.		Controls and indications->Local variables->Read-VAR	Read local variables 3. Read local variables 1. Read local variables 2.
2.		Math->Arithmetic-> Divide	
3.		Controls and indications->Indicators-> Signal indicator	1.Values of P. 2. Values of R.
4.		Controls and indications->Local variables->Write-VAR	Local variables 4 – P values. Local variables 5 – R values
5.		Controls and indications->Indicators->XY Signal indicator	P=f(R) graph

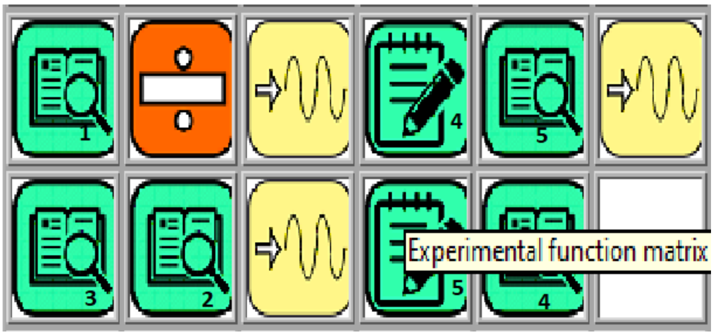


Figure 3.3: Function layout

3.1 Comparation of the results

Log in to the CORELA data submission platform. Read the value of maximum power and resistance at which this data is achieved from the graph, enter it and send it to the educational platform (see instructions in Chapter 5).

Compare the obtained values with those obtained from students 1 and 3.

4. Practical realization and measurement (student 3)

This part of the exercise is related to the practical realization of the electrical circuit and performing realistic measurements. The aim is to test the **maximum power transfer** in realistic conditions. To perform the experiment, we use schematics viewed on the following figure:

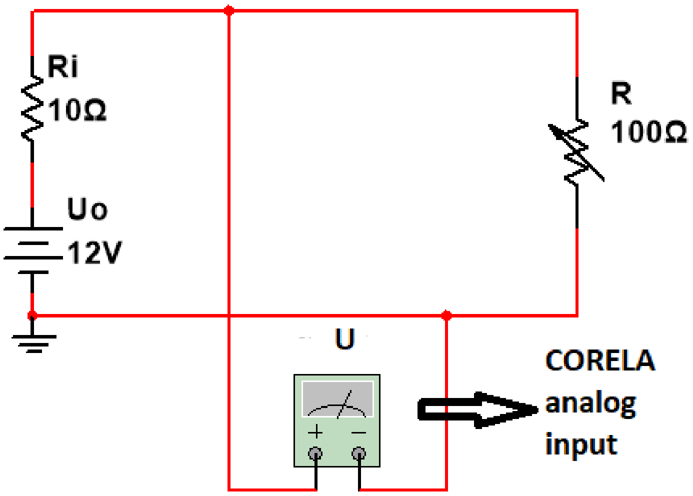


Figure 4.1: Wiring diagram for performing practical measurements

We have a voltage source of known parameters (*U_0, R_i*) and a variable resistor *R*. Changing the resistance *R,* we will determine its amount for maximum power transfer. *R* and *R_i* form the voltage divider, where the expression is valid:

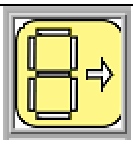
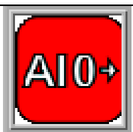
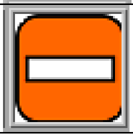
$$\frac{R}{R_i} = \frac{U}{U_{R_i}} = \frac{U}{U_0 - U}$$

So, by measuring the voltage *U* we can calculate:

$R = \frac{R_i \times U}{U_0 - U}$ and finally, $P = \frac{U^2}{R}$

Voltage *U* will be entered in the CORELA using function Analog input (if we have a data acquisition card / DAQ card) otherwise we use a voltmeter and the Signal control function, and we will perform the calculation as follows:

Calculation of resistance *R*.

1.		Controls and indications->Controls->Analog control	1.Record the value U_0=12V. 2. Record the value R_i=10Ω. 3. Record the value of R.
2.		DAQ card channels->Analog input	Measured voltage U.
3.		Math->Arithmetic-> Subtract	

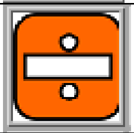
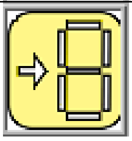
4.		Math->Arithmetic-> Multiply	
5.		Math->Arithmetic-> Divide	
6.		Controls and indications->Probes->Analog probe	1.Result of U_0-U. 2. Result of R_ixU. 3. Result of U^2.
7.		Math->Arithmetic-> Square	
8.		Controls and indications->Local variables->Write-VAR	Local variables 1 – denominator value. Local variables 2 – numerator value. Local variables 3 – value of R. Local variables 4 – value of P.

Figure 4.2: Calculation of resistance R

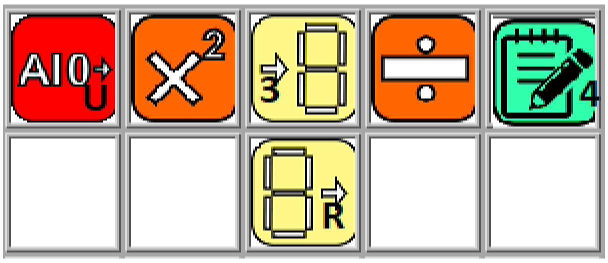


Figure 4.3: Calculation of power P

4.1 Comparation of the results

Log in to the CORELA data submission platform. Enter the values of maximum power and associated resistance and send them to the educational platform (see instructions in Chapter 5).

Compare the obtained values with those obtained from students 1 and 2.

5. Procedure of writing and sending dana to the educational platform

Step 1: Write and saving the data

From the menu list of functions <Functions list-> Controls and indicators-> Controls> select the analog control <Analog control> and left-click to place it on an empty field of the desktop.

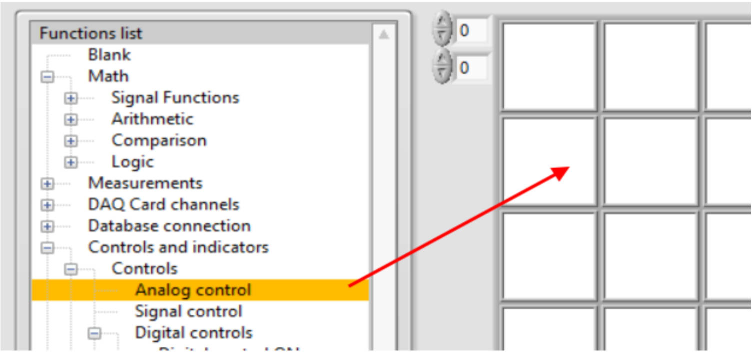


Figure 5.1: Setting the <Analog control> function - control

A pop-up window opens where we enter the value, we want to set on the CORELA platform. Since we have two data, maximum power and resistance value, we use the same function twice.

Step 2: Reviewing the recorded data

By placing the <Analog indicator> in the field to the right of the already set analog control <Analog control> (Figure 5.2) the data entered in the CORELA platform is available.

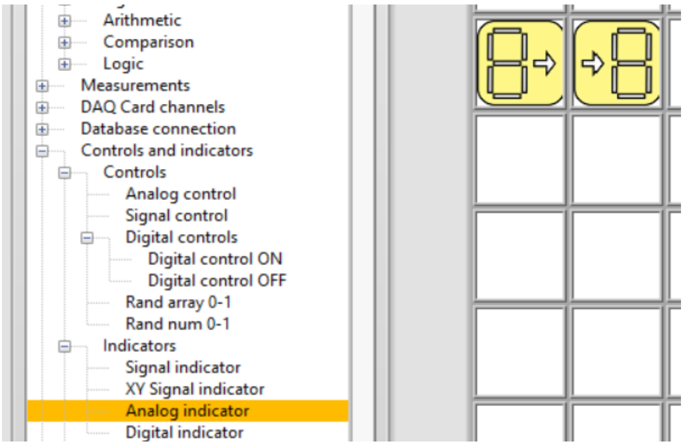


Figure 5.2: Screen layout after setting the <Analog indicator> function - control

Funded by the
Erasmus+ Programme
of the European Union

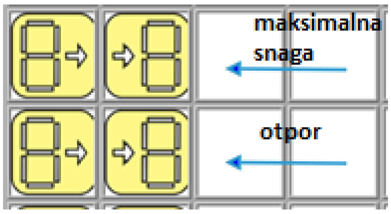


Figure 5.3: Display the CORELA platform data entry task

Step 3: Sending data to the educational platform

From the function list menu <Functions list-> Database connection-> Output channel> select output channel 1 <CH out 1> for the first data and output channel 2 <CH out 2> for the second data.

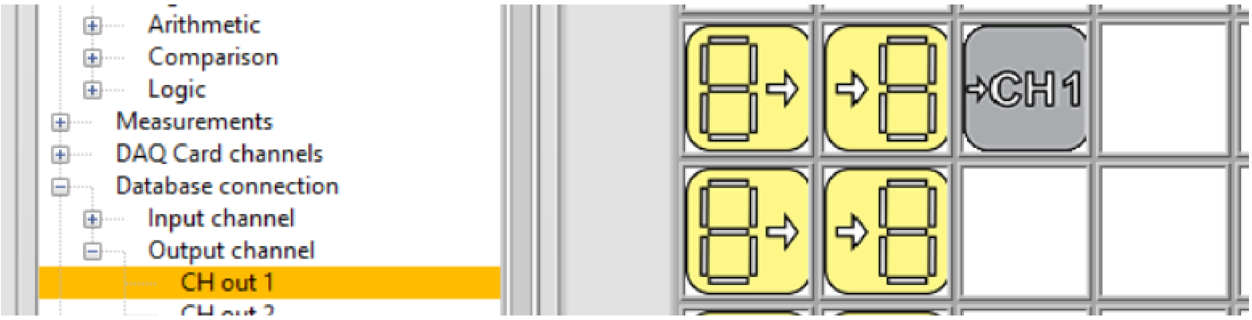


Figure 5.4: Sending data to the educational platform

Video material:

https://youtu.be/HE4i_2nEbDc