

**МИНИСТЕРСТВО НАУКИ И ВЫСШЕГО ОБРАЗОВАНИЯ
РОССИЙСКОЙ ФЕДЕРАЦИИ
Федеральное государственное бюджетное
образовательное учреждение высшего образования
«ПЕНЗЕНСКИЙ ГОСУДАРСТВЕННЫЙ УНИВЕРСИТЕТ»**

В. В. Регеда, О. Н. Регеда

Моделирование электрических цепей в среде SimInTech

Методические указания к лабораторным работам

V. V. Regeda, O. N. Regeda

SimInTech simulation of electrical circuits

Methodological guidelines for laboratory work

ПЕНЗА 2025

МИНИСТЕРСТВО НАУКИ И ВЫСШЕГО ОБРАЗОВАНИЯ
РОССИЙСКОЙ ФЕДЕРАЦИИ
Федеральное государственное бюджетное
образовательное учреждение высшего образования
«Пензенский государственный университет» (ПГУ)

В. В. Рееда, О. Н. Рееда

Моделирование электрических цепей в среде SimInTech

Методические указания к лабораторным работам

V. V. Regeda, O. N. Regeda

SimInTech simulation of electrical circuits

Methodological guidelines for laboratory work

Пенза
Издательство ПГУ
2025

Р е ц е н з е н т

кандидат технических наук, доцент,
проректор по учебной работе Пензенского государственного
технологического университета
О. А. Голышевский

Регада, Владимир Викторович.
Р32 Моделирование электрических цепей в среде SimInTech :
метод. указания к лабораторным работам / В. В. Регада,
О. Н. Регада = SimInTech simulation of electrical circuits :
methodological guidelines for laboratory work / V. V. Regeda,
O. N. Regeda. – Пенза : Изд-во ПГУ, 2025. – 52 с.

Рассмотрены основные приемы работы с российской системой модельно-ориентированного проектирования SimInTech. Основное внимание уделено изучению специализированных библиотек, обеспечивающих моделирование электрических схем. Представлены описания лабораторных работ по моделированию цепей постоянного и переменного тока, трехфазных цепей, переходных процессов в линейных электрических цепях, а также по моделированию частотных характеристик. Все описания снабжены методическими указаниями, содержащими необходимый теоретический материал по соответствующей тематике, и списком контрольных вопросов.

Издание подготовлено на кафедрах «Электроэнергетика и электротехника» и «Информационно-измерительная техника и метрология» ПГУ и предназначено для лабораторных и индивидуальных занятий по курсам «Компьютерные технологии» и «Компьютерное моделирование» для обучающихся по направлению подготовки 13.03.02 «Электроэнергетика и электротехника».

The methodological guidelines for laboratory work cover the basic techniques of working with the Russian model-oriented design system SimInTech. The main focus is on the study of specialized libraries that provide modeling of electrical circuits. Descriptions of laboratory work on modeling direct current and alternate current circuits, three-phase circuits, transients in linear electrical circuits, as well as on modeling frequency characteristics are presented. The textbook is provided with methodological guidelines containing the necessary theoretical material on the relevant topic and a list of control questions.

The publication was prepared at the Department of Electric Power Engineering and Electrical Engineering of Penza State University and is intended for laboratory and individual classes in the course “Computer technologies in electrical power engineering” for English-speaking university students majoring in 13.03.02 “Power and Electrical Engineering”.

УДК 001.8 (083.95)

© Пензенский государственный
университет, 2025

INTRODUCTION

The purpose of these guidelines is to teach students how to use the modern Russian model-oriented design system SimInTech for modeling electrical circuits when performing laboratory work, term papers, term papers, final qualifying papers and when conducting scientific research.

Solving the problem of analyzing and modeling an arbitrary electrical circuit in a thermal power plant course includes the following main steps:

1) the transition from the basic electrical circuit to its replacement circuit, consisting only of idealized passive and active elements;

2) selection of a system of independent variables characterizing the processes in the circuit under consideration and compilation of electrical equilibrium equations;

3) solution of electrical equilibrium equations and presentation of the results.

For a long time, computers were used inefficiently only to perform the third of the listed stages, which required a large amount of preparatory work performed manually.

Modern methods of machine (automated) circuit analysis involve the widespread use of computers not only to solve electrical equilibrium equations, but also to form these equations according to a given circuit replacement scheme, or to draw up a circuit replacement scheme and the corresponding electrical equilibrium equations according to a given schematic electrical circuit. This allows not only to significantly reduce the analysis time, but also to significantly improve its accuracy.

Modern circuit machine analysis programs are part of modern CAD systems and are conditionally divided into two large groups:

- general-purpose programs designed to solve a wide range of circuit analysis tasks of various types;

- specialized programs focused on solving certain specific analysis tasks, such as studying the time or frequency characteristics of linear circuits, finding working conditions, checking of nonlinear elements, determining the sensitivity of the circuit to changes in the parameters of the elements, etc.

Regardless of their purpose, they consist of several main blocks corresponding to the main stages of circuit analysis:

- a block for preparing initial data;
- a block for forming electrical equilibrium equations;
- a block for solving electrical equilibrium equations;
- a block for presenting analysis results.

Modern software products for modeling electrical circuits include such as Micro-Cap, OrCAD, NI Multisim, LabVIEW, Simulink, as well as domestic Delta Design, Symica and SimInTech systems.

On December 15, 2016, the SimInTech system was included in the Unified Register of Russian Programs for Electronic Computers and Databases. The copyright holder is “3B SERVICE LLC”.

The SimInTech system can be used for functional and dynamic modeling of various systems: military-industrial complex / defense industry facilities, energy facilities, nuclear facilities, oil and gas facilities, aerospace facilities, transport facilities, and others. In addition, the SimInTech system allows you to develop algorithms and control interfaces, as well as automatically generate code for programmable controllers and graphic displays [1].

Models in the SimInTech system are created from standard blocks placed in various libraries. The following libraries are currently included in SimInTech for modeling electrical circuits:

- electrical equipment for the analysis of electrical circuits in current values;
- electrical equipment for the analysis of electrical circuits in instantaneous values;
- electric drives.

SimInTech has an open interface for developing its own libraries and for connecting third-party software modules in various programming languages (C, Pascal, Fortran, and others).

In SimInTech, it is possible to develop both simple models that describe in detail the behavior of a system (for example, an RLC circuit with a power source), and complex models that partially describe not only the system itself, but also control algorithms, protection, control interface, etc. To organize synchronous data exchange between calculation schemes, a database of signals is used - a structured repository of variables used in one or more projects that are part of a complex model [1].

The SimInTech program can be downloaded for free at <https://simintech.ru>.

These guidelines describe the application of the SimInTech system for modeling direct current (DC) and alternate current (AC) electrical circuits in steady-state and transient modes.

Laboratory work № 1

Fundamentals of SimInTech Modeling

The purpose of the work: familiarization with the basic rules of the SimInTech system for creating electrical circuits.

Work assignment

1. Launch the SimInTech program.

As a result, the main window with the SimInTech interface appears on the screen, shown in Figure 1.1 [2].

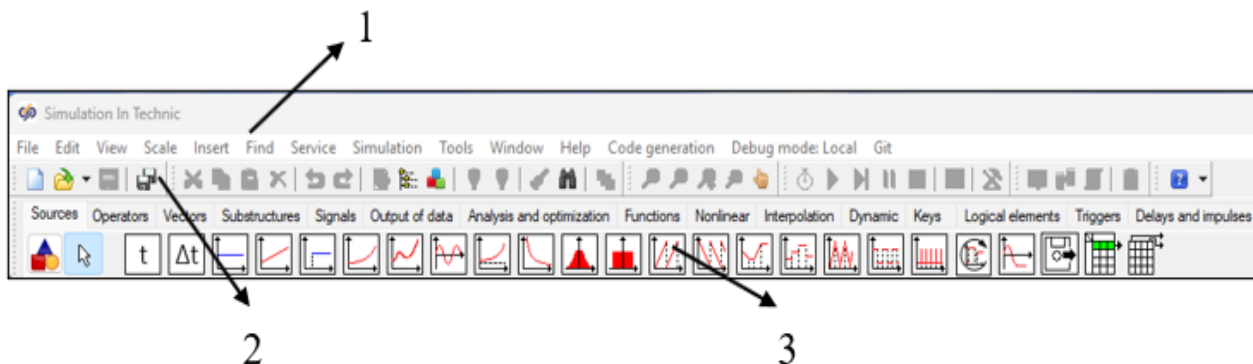


Figure 1.1

The first line of the main window contains the **Menu area of the main window** (1), from which, in particular, you can control the creation, editing and modeling modes of created models, which are called calculation schemes in SimInTech.

The second line of the window contains the **Button Panel area** (2), with grouped sets of buttons that invoke the most frequently used commands.

The third line of the window contains the **Block Palette** area (3), where the blocks corresponding to the selected calculation scheme appear on the tabs.

2. To solve a specific engineering problem, an appropriate calculation scheme with a solver and its own settings should be selected.

To create a calculation scheme, use the command **File → New Project**, after selecting which it is necessary to indicate the appropriate calculation scheme in the drop-down list.

Select the calculation scheme **Control system diagram**. As a result, a schematic window of the project will open (Figure 1.2).

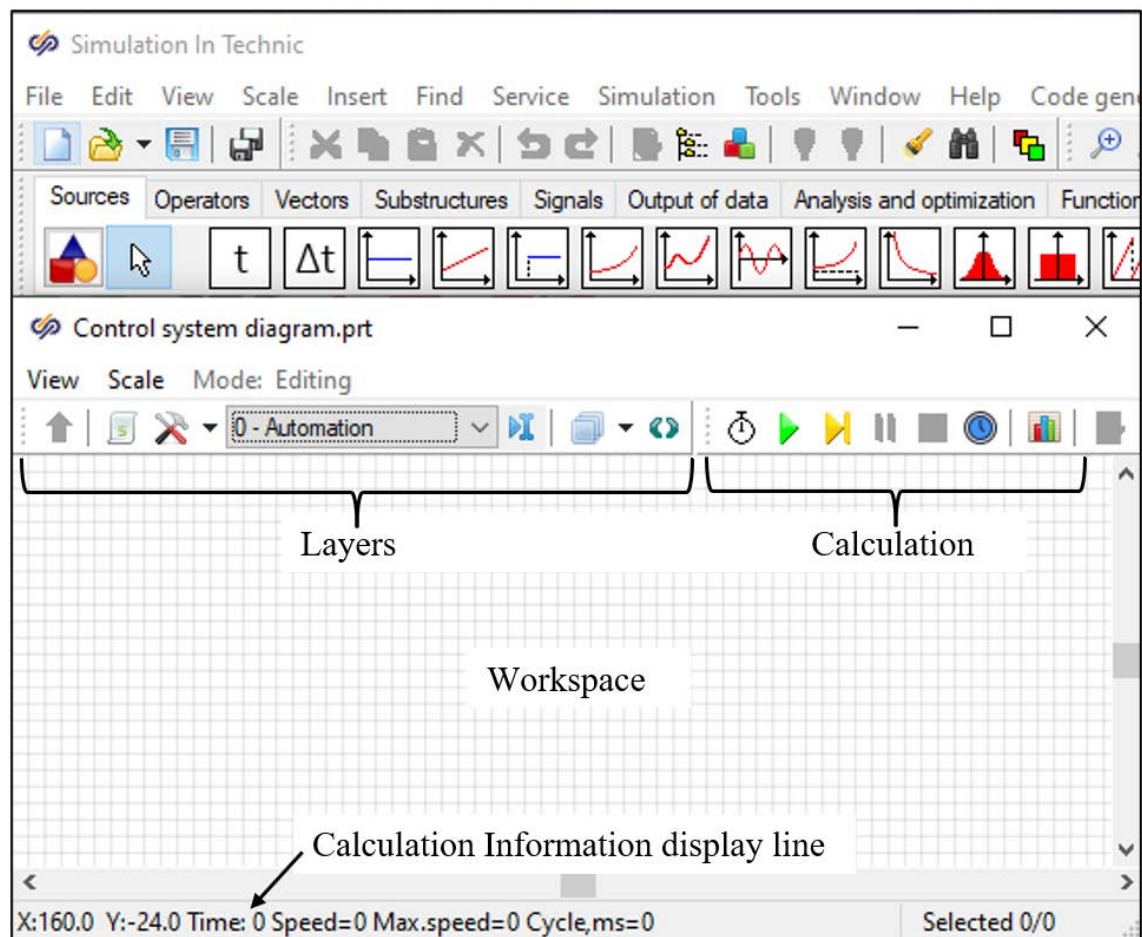


Figure 1.2

Save the project as a file by selecting the **File** command in the main window menu **Save project as...** and specifying the access path and file name as directed by the teacher.

From Figure 1.2, it can be seen that the title of the project window contains the file name **Control system diagram** and the default file extension **.prt**.

The button panel at the top of the project window contains the tools for configuring the project **Layers** and managing the calculation **Calculation** (Figure 1.2).

At the bottom of the projects window there is a line for displaying calculated information **Calculation Information display line**.

In the center of the project window is a **Workspace** in which, using blocks selected from a palette of blocks, you can create and edit previously created calculation models, create your own blocks, etc.

In the area of the **Block Palette** (see 3 on Figure 1.1), you can find many block tab for modeling various systems.

SimInTech includes a set of specialized libraries that provide modeling of electrical circuits.

The **EC – Static v.2.0** library contains models that provide calculation of the effective values of current, voltage and power (Figure 1.3).

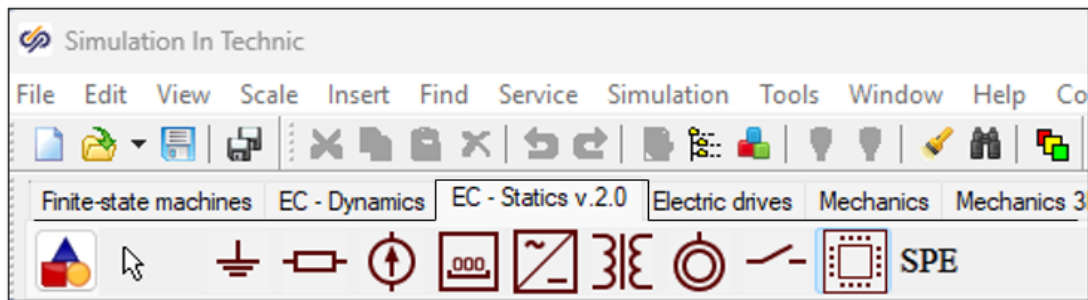


Figure 1.3

The **EC – Dynamics** library contains models that provide the calculation of instantaneous values of AC circuits in steady-state and transient modes (Figure 1.4).

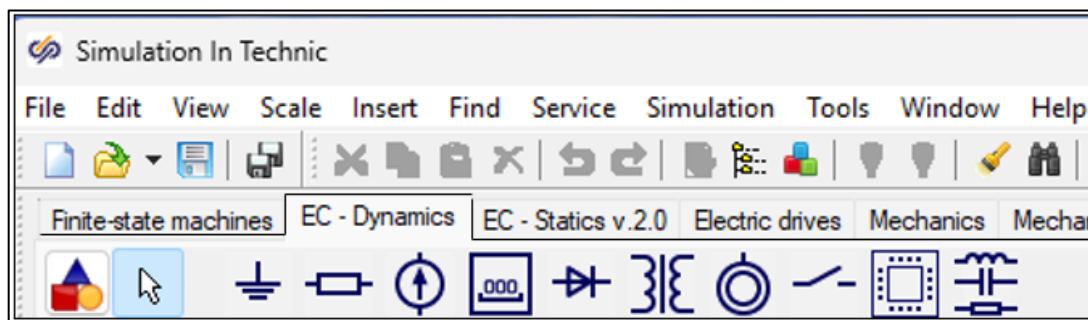


Figure 1.4

Save your project as a file named **Sample** with the extension **.prt**.

In this laboratory work, the circuits created from the blocks of the **EC – Dynamics** library from the bookmarks shown in Figures 1.5–1.8 will be modeled.

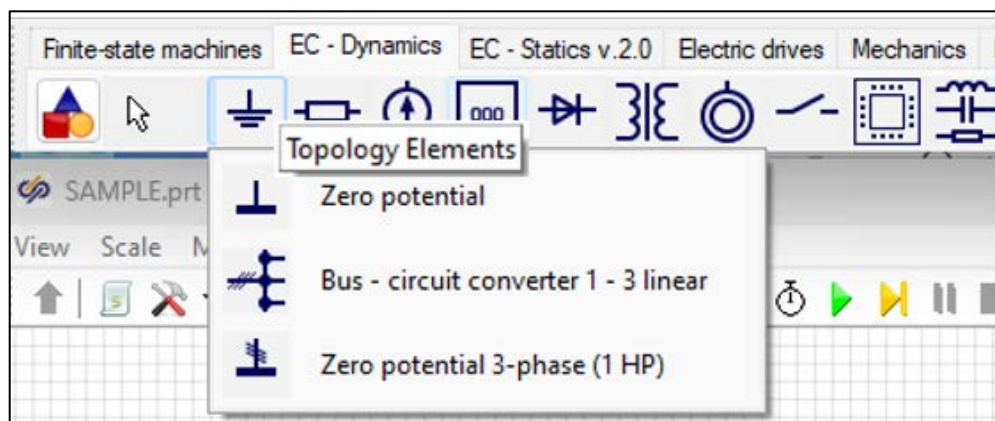


Figure 1.5

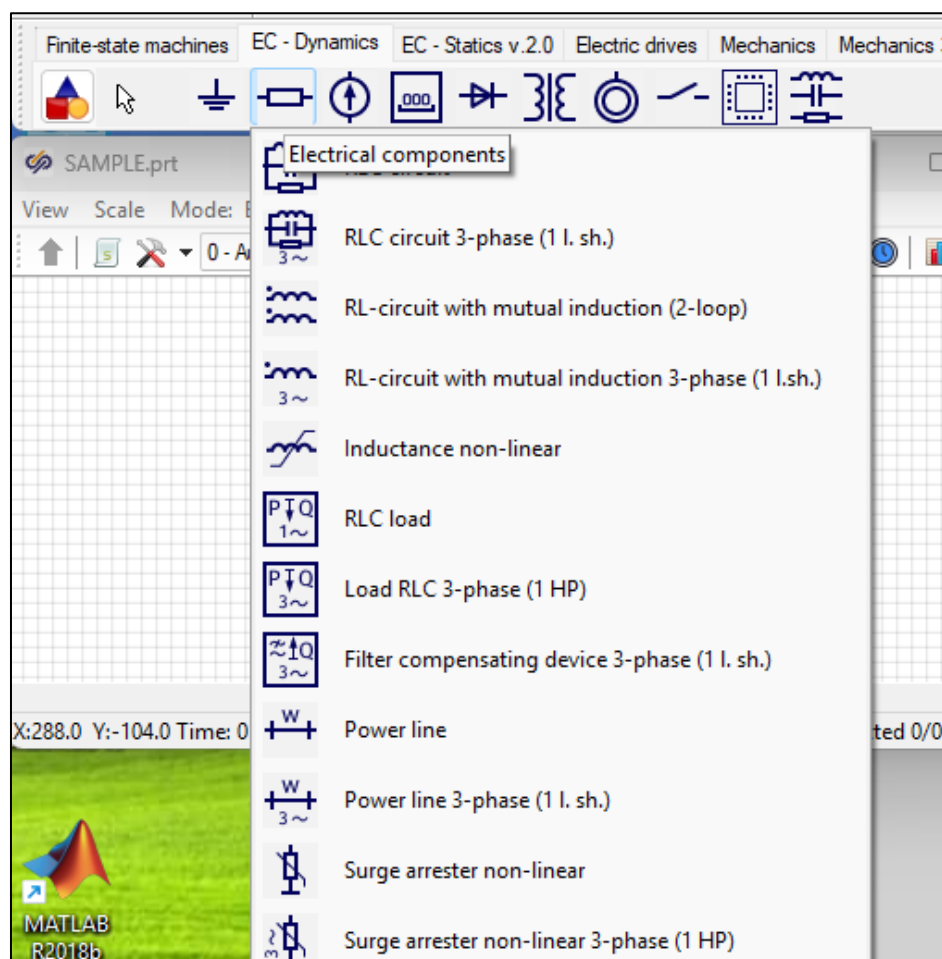


Figure 1.6

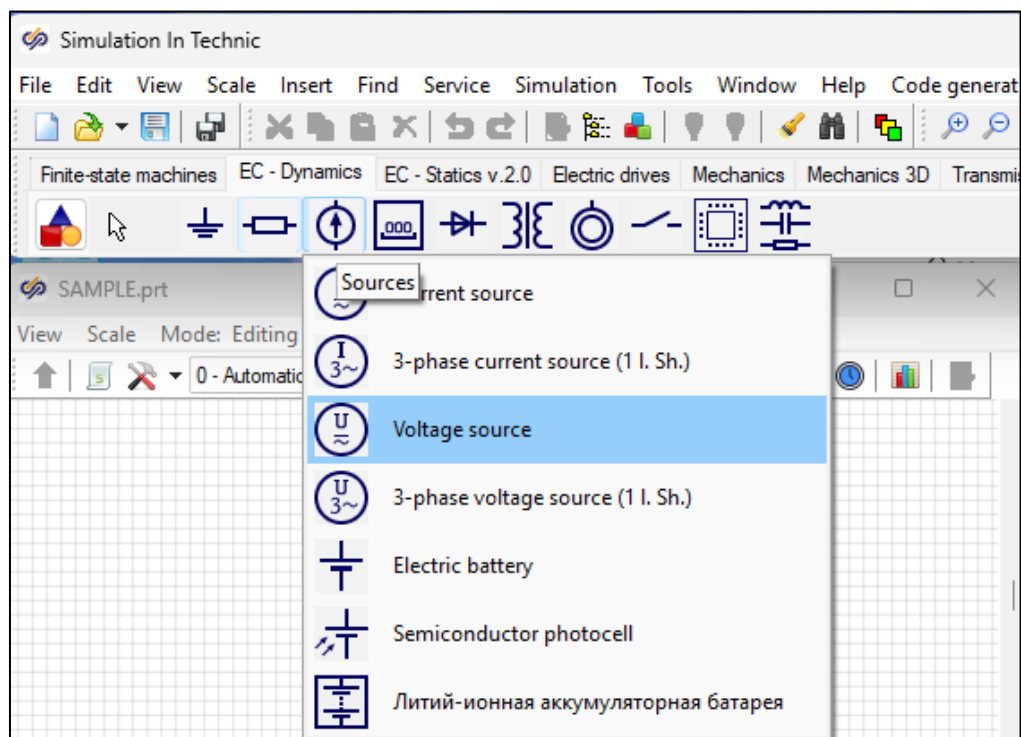


Figure 1.7

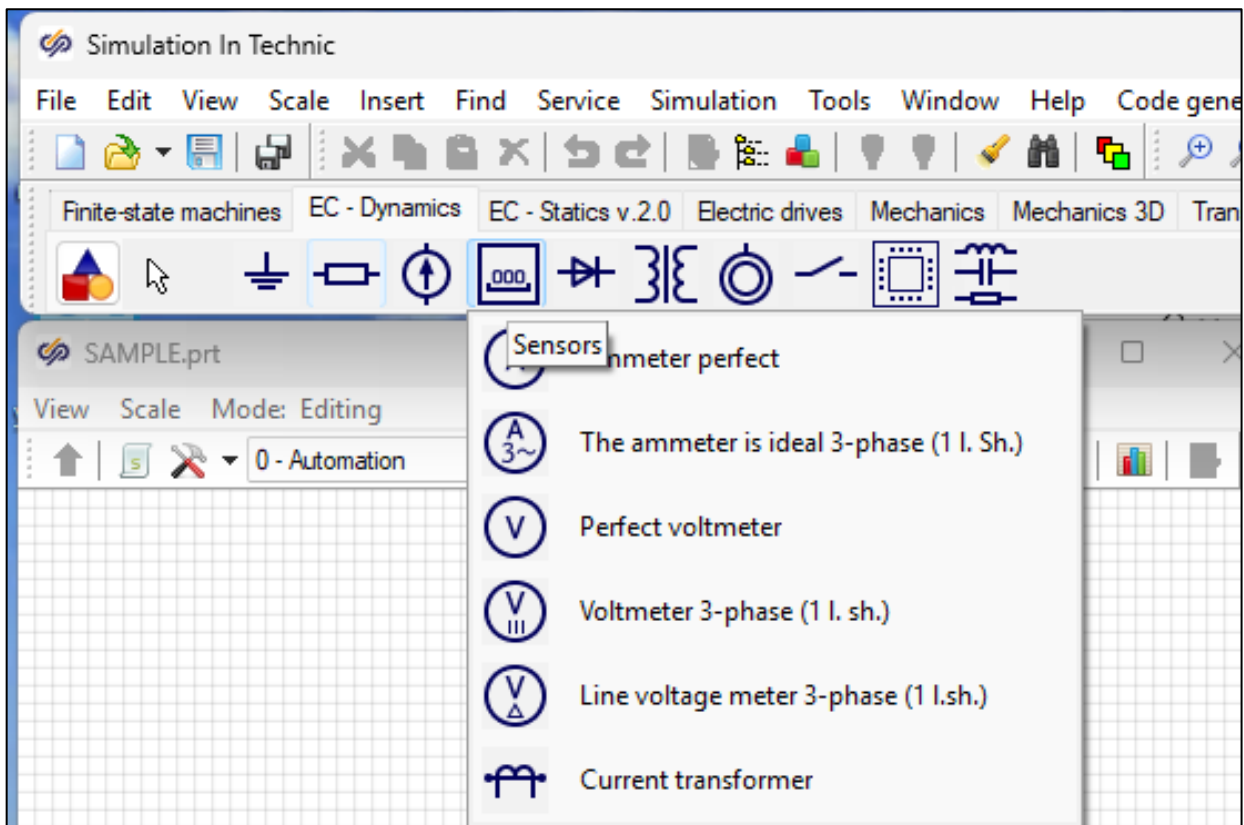


Figure 1.8

3. Open the **EC – Dynamics** library and left-click (LMB) to select the following blocks:

- **Voltage source** block from the **Sources** tab;
- the **Zero potential** block from the **Topology Elements** tab;
- the **RLC-circuit** block from the **Electrical component** tab;
- the **Perfect voltmeter** block from the **Sensors** tab.

After selecting the required block, move the cursor to the project window and click the LMB to indicate the place where the block is inserted (Figure 1.9). If necessary, you can move the blocks by clicking on its image and dragging it to the desired location without taking your finger off the button.

4. Select the **RLC-circuit** block, right-click and select the command **Rotate ⇒ 90 degrees clockwise** from the context menu that appears (Figure 1.10).

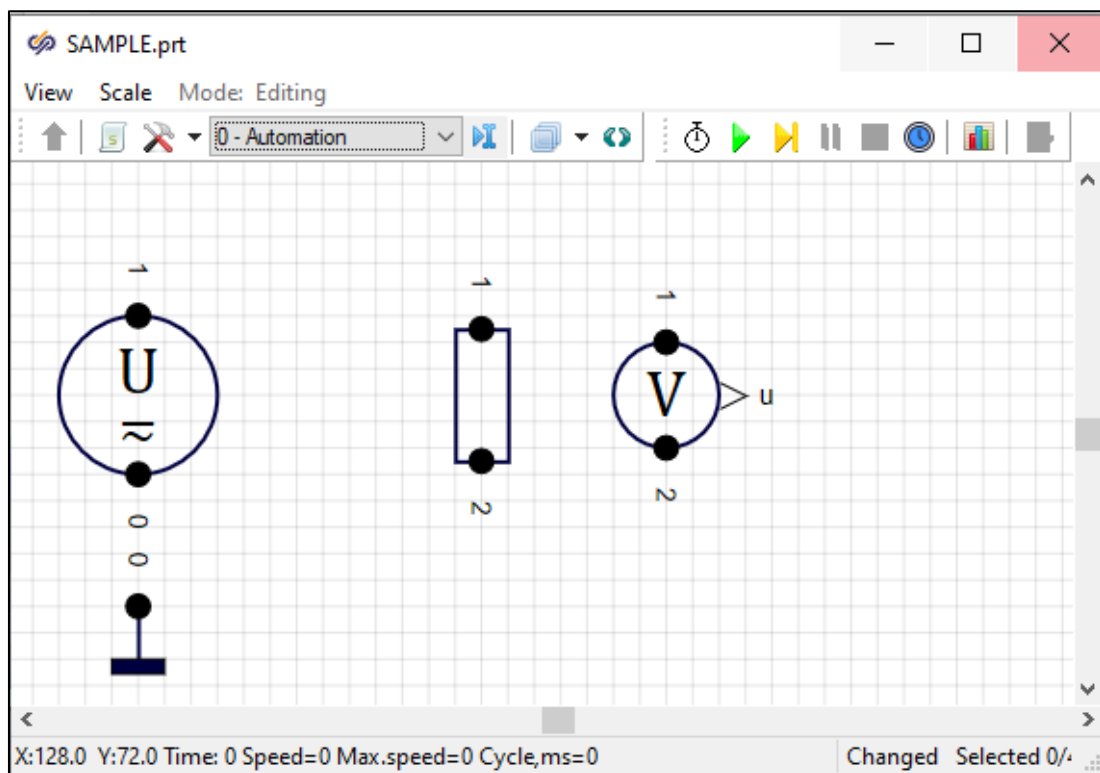


Figure 1.9

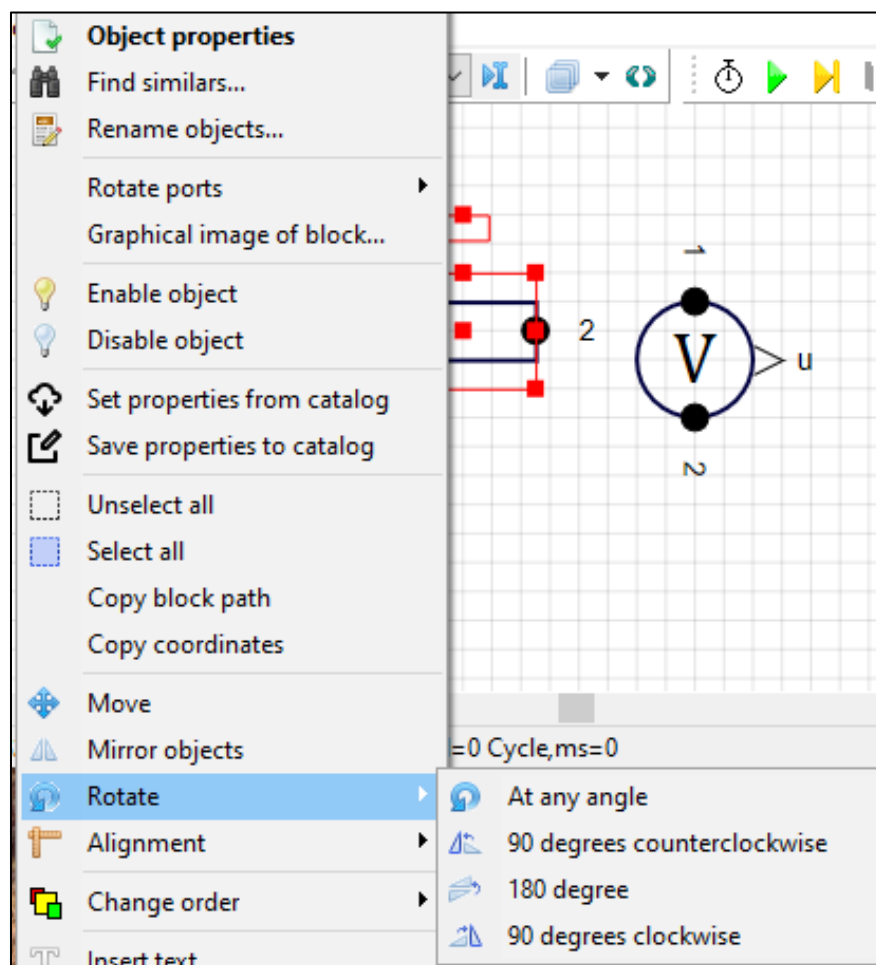


Figure 1.10

5. Block properties are set via the **Properties** window, which is either selected in the context menu for the selected block, or opens by double-clicking on the selected block. Select the **RLC-circuit** block, right-click and select **Object Properties** from the context menu that appears (Figure 1.11).

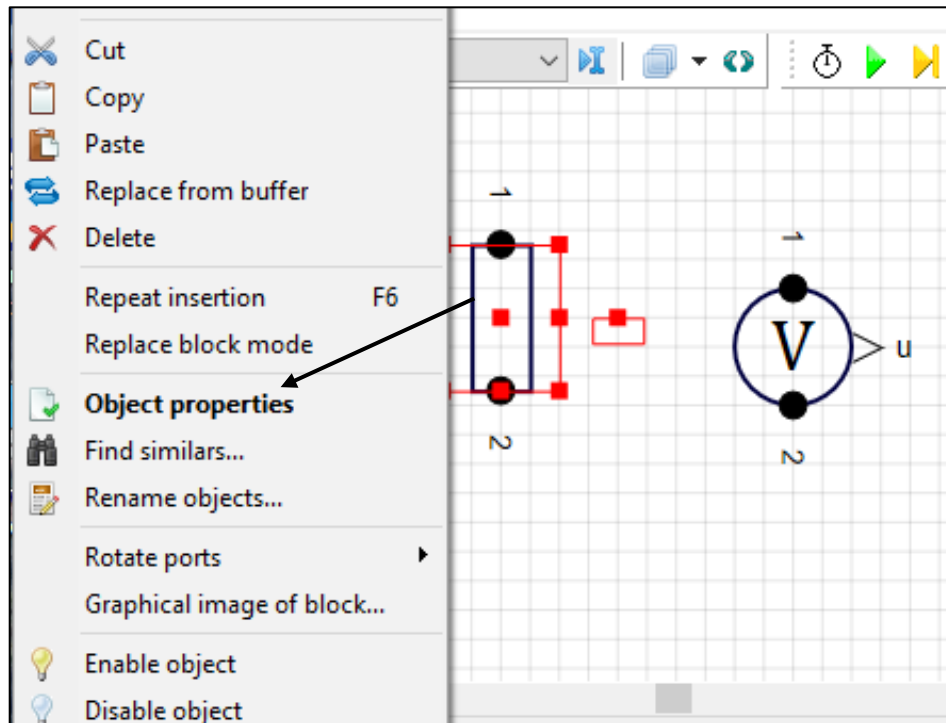


Figure 1.11

In the **Properties** window for the **RLC-circuit** block, on the **Properties** tab, set the values of its parameters shown in the Figure 12.

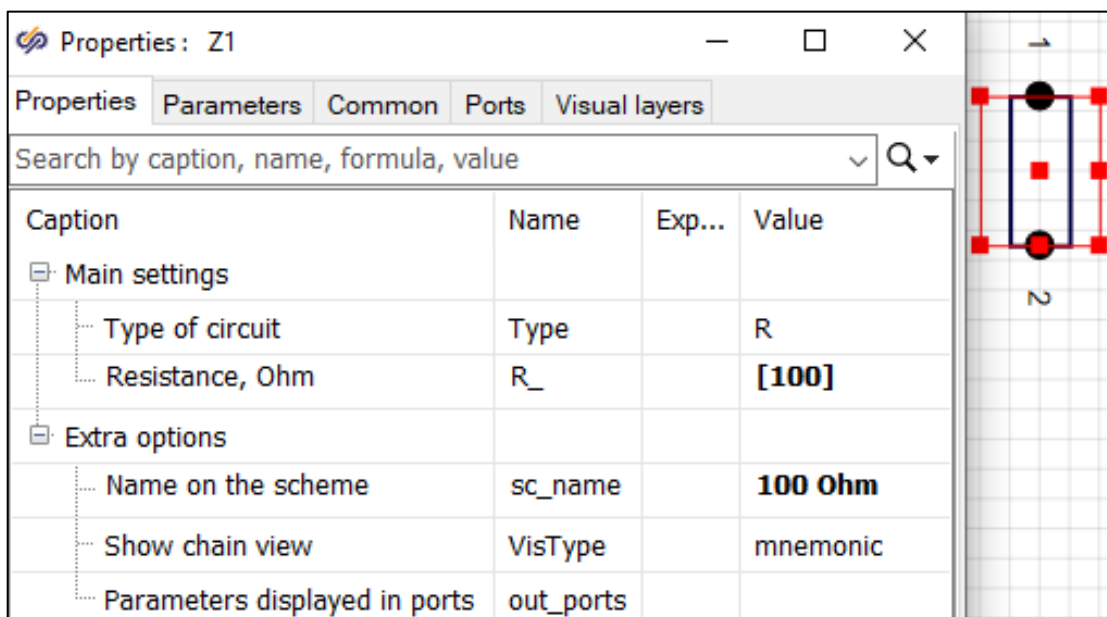


Figure 1.12

In the **Properties** window for the **Voltage source** block on the **Properties** tab, set the values of its parameters shown in the Figure 1.13.

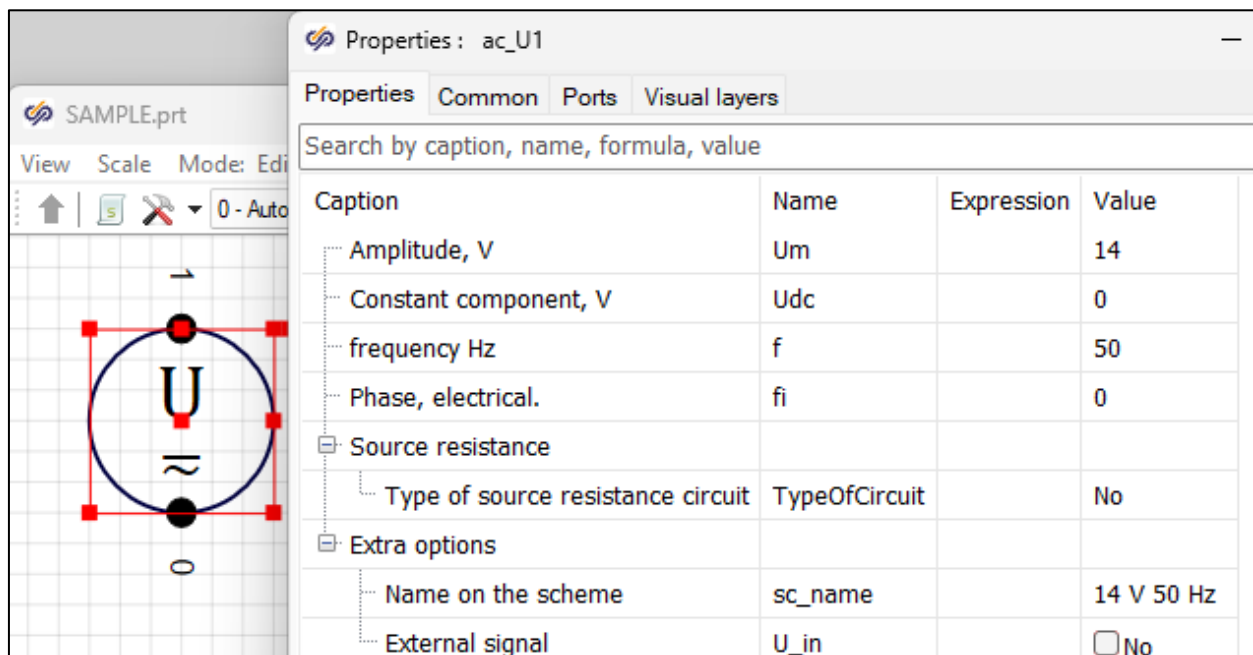


Figure 1.13

6. Add to the schematic window of the project:

- the second block of the **RLC-circuit**;
- the **Graph of time** block from the **Output of data** tab.

Set the parameter values shown in the Figure 1.12 in the **Properties** window for the second **RLC-circuit** block.

7. The blocks can have ports for forming both electrical connections of the design electrical circuit and mathematical connections for transmitting control signals.

To connect the two blocks located in the diagram, move the cursor to the output port of one block, click the LMB and, releasing the button (the cursor will turn into an arrow of the type $\uparrow$), extend the communication line to the input port of the other block, and then repeatedly click the LMB [2].

Connect the blocks together as shown in the Figure 1.14.

To make **wire** from the line, select it in the desired location, right-click and select the command «**Actions** → **Add wire** in the context menu that appears (Figure 1.15).

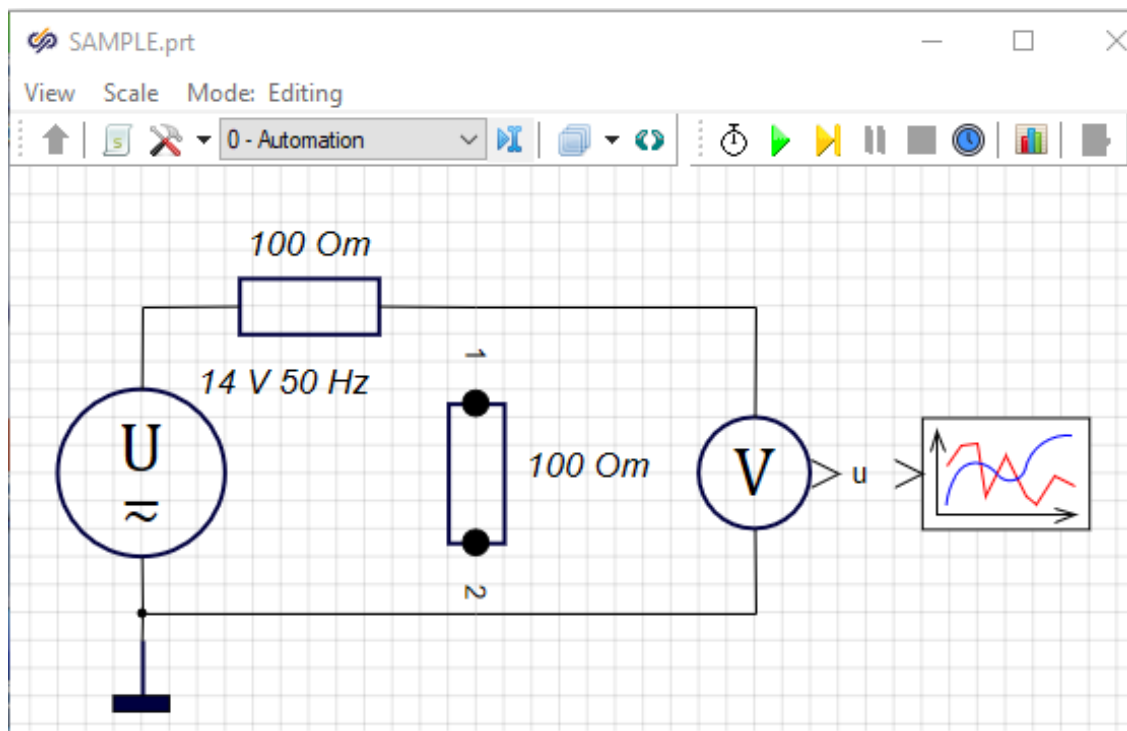


Figure 1.14

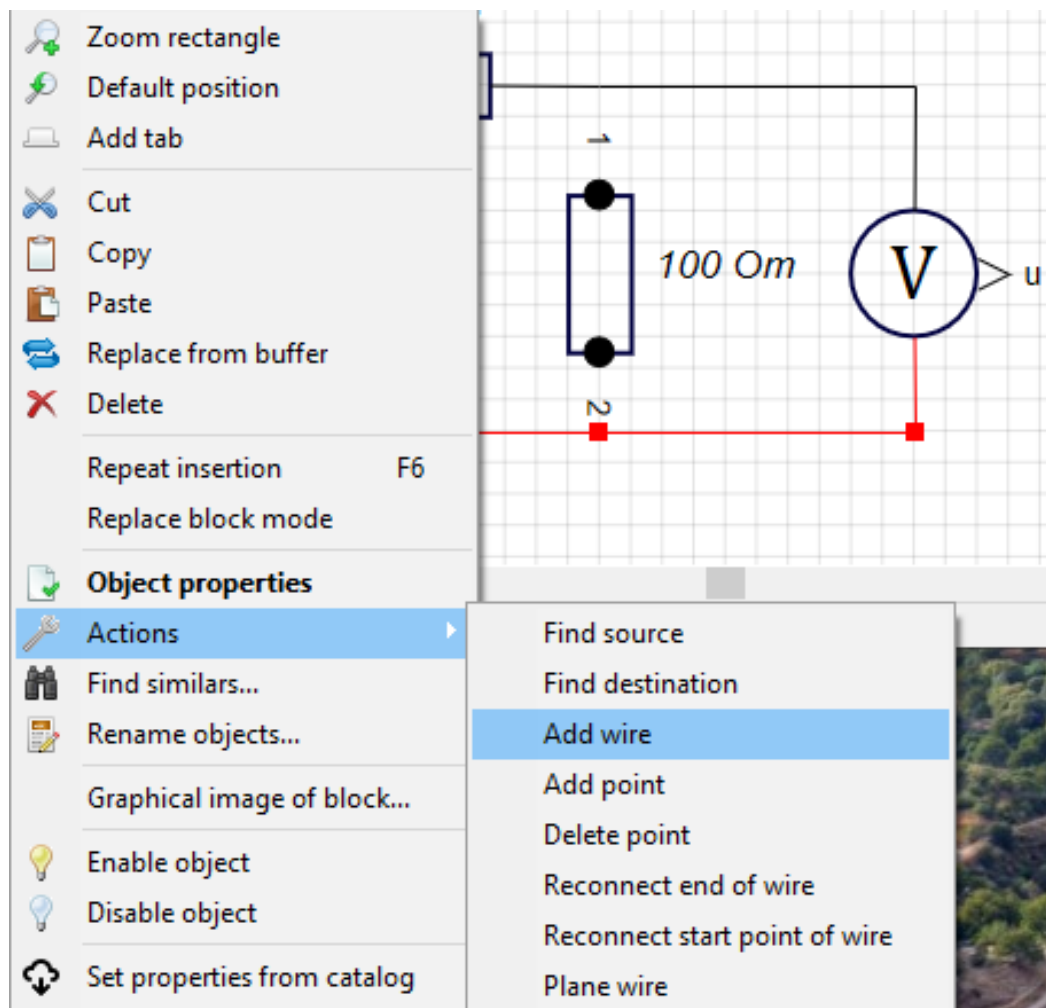


Figure 1.15

As a result, a new point will appear on the line, which must be connected to the input 2 of the **RLC-circuit** block. Similarly, connect pin 1 of the **RLC-circuit** block to the upper line. To delete a line, you need to select the paintwork and press the **Delete** key.

Add a caption to the block **Graph of time** by moving the cursor under it and finding the cursor position at which it will change its shape to an arrow with a question mark. After that, double-click the LMB. Move the cursor to the text input window that appears, click LMB and enter **Output**.

Connect the **Perfect voltmeter** and **Graph of time** blocks together as shown in the Figure 1.16.

8. Before starting the electric circuit model, it is required, based on the parameters of the electric circuit, to set the required values of the calculation parameters. To do this, click the **Simulation properties** button  of the schematic project window. As a result, the **Project properties** dialog box opens.

A new project was created in the SimInTech environment based on the calculation scheme **Control system diagram**.

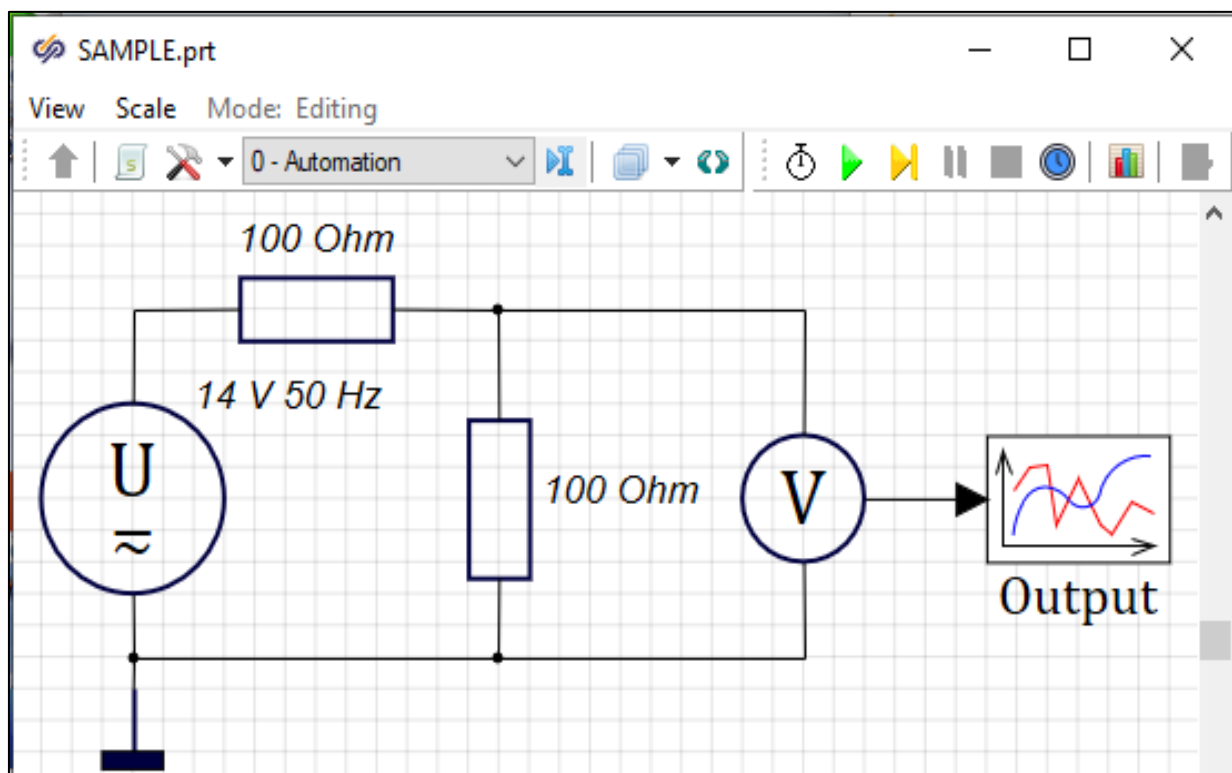


Figure 1.16

On the **Calculation Parameters** tab of the **Project properties** window, you need to set the values shown in the Figure 1.17.

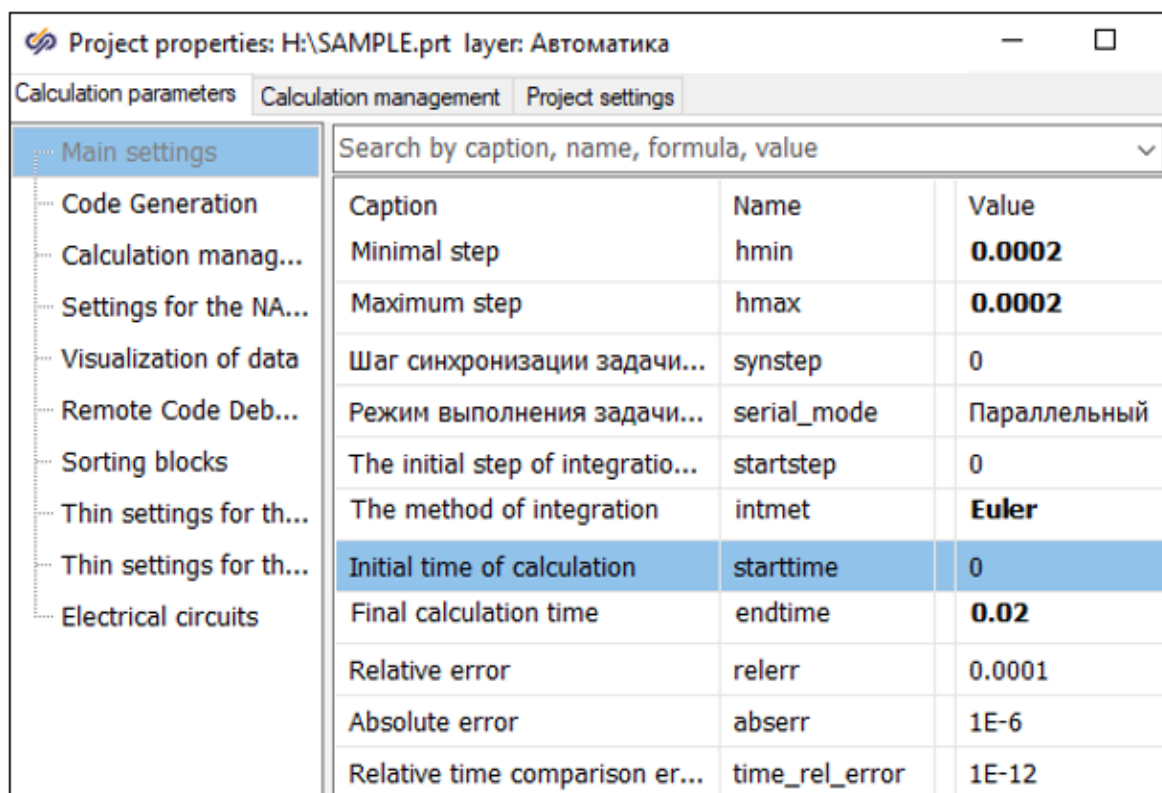


Figure 1.17

For the **Method of integration** parameter, leave the default value – the **Euler** method, which can only use a fixed step, and all other methods can be used both with a fixed step and with automatic step selection.

The **Final calculation time** should be set to **0.02**, which corresponds to one period of harmonic voltage in seconds of the **Voltage source** with a frequency of 50 Hz.

We will choose the **Minimal step**, which sets the restriction from below on the integration step, 100 times less than the **Final calculation time**.

Keep in mind that in the SimInTech system, the integer part of the number must be separated from the fractional part by a dot, so the **Minimum step** value in the **Calculation Parameters** window should be set in seconds as **0.0002** (see Figure 1.17).

The **Maximum step** for the **Euler** integration method should be equal to the **Minimal step** (see Figure 1.17).

The default **Relative error** (accuracy) is **0.0001**. Do not set this value to greater than 0.01 or less than **1e-10**.

The **Absolute error** default is **1e-6**. It has a significant impact on the choice of the step size only in cases where the values of some variables are close to zero.

The **Initial step of integration** is set to **0** by default, which means that it is set automatically.

9. To start the simulation process, click on the **Start** button  on the Toolbar or on the **F9** key.

At the end of the calculation, an inscription should appear in the line of the calculation information display of the Schematic window: **End time (time=0.02)**, otherwise an error has been made in the diagram.

The simulation results can be seen on the screen of the **Graph of time** block in the form of time dependence by double-clicking on it (Figure 1.18).

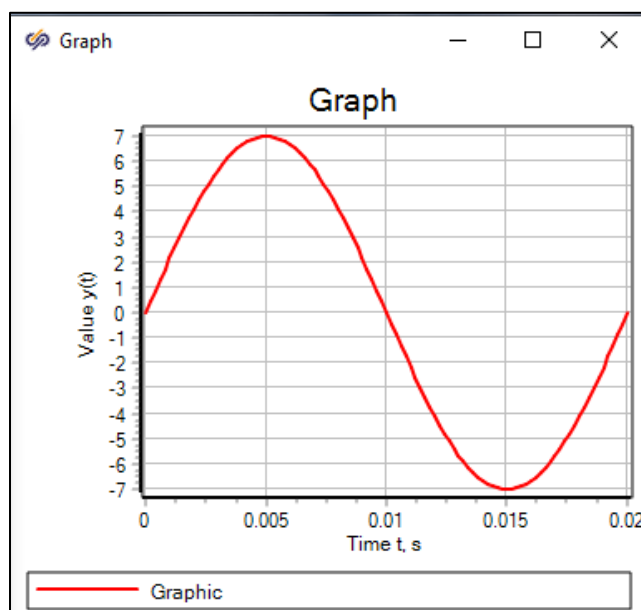


Figure 1.18

10. To zoom in on the desired part of the graph, hover the cursor over it and, while holding down the LMB, move the mouse in the «right-up» or «right-down» direction.

To restore the original graph view, it is required to repeat the same procedure, only to move the pressed LMB in the «left-up» or «left-down» direction. The visible part of the graph is moved by moving the mouse with the right mouse button pressed [2].

Use the graph to determine the amplitude value of the voltage by increasing the desired part of the graph and making sure that it is 7 V. Set the original graph view.

11. Move the cursor to the central part of the graph window and make a right-click. As a result, a context menu will appear (Figure 1.19), in which you can select commands for copying to the clipboard, exporting to a file, printing and saving graphics images to a text file, etc.

If you select **Cursor** in the context menu, a crosshair will appear next to the cursor, which can be moved to the point to calculate its coordinates (Figure 1.20). To cancel the command, open the context menu again and disable this command.

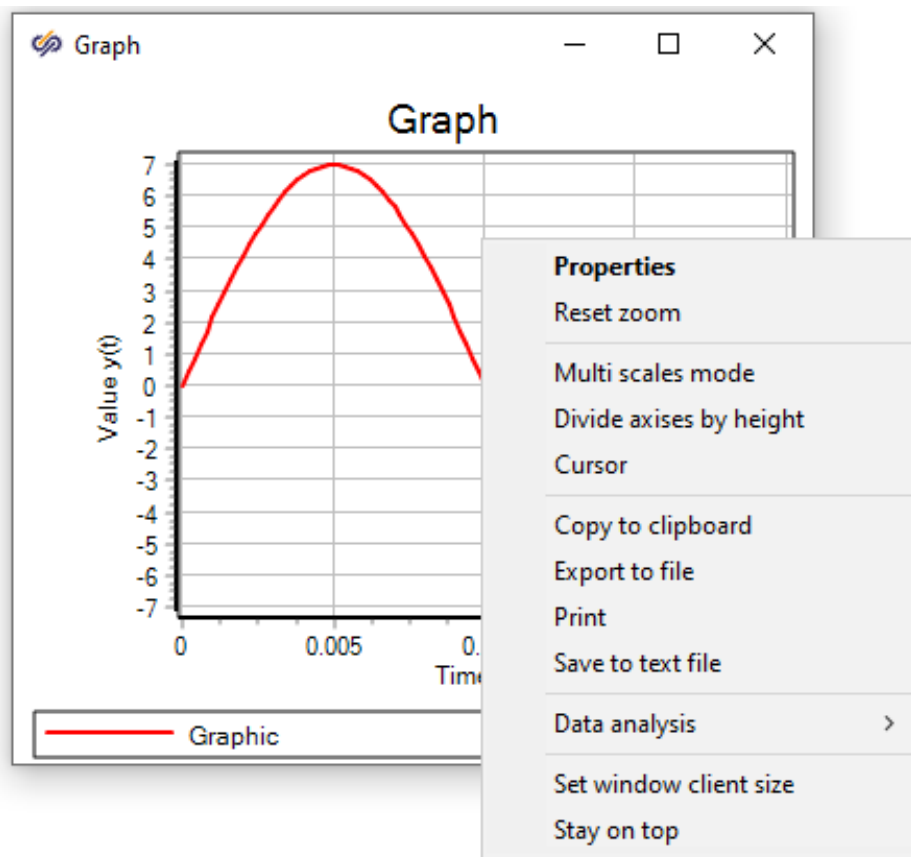


Figure 1.19

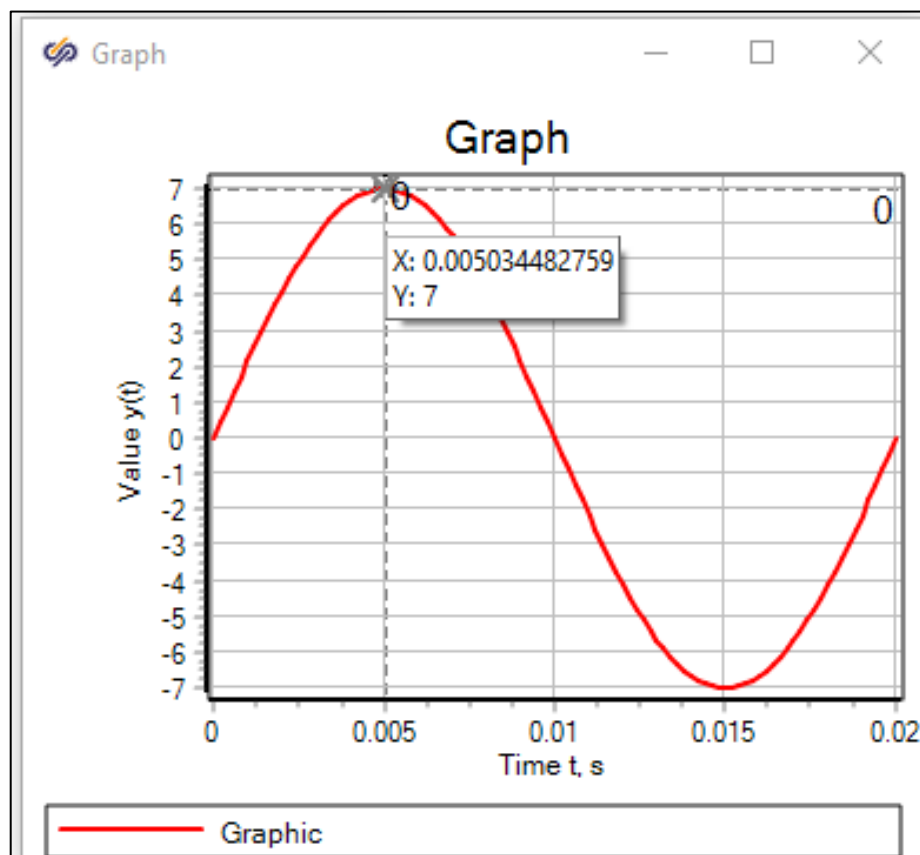


Figure 1.20

The **Properties** item in the context menu open a dialog box with the title **Plot properties** (Figure 1.21), which on the tab **Charts and axes** sets settings for such parameters as: graph name and type; line style and width; various settings on the **Axes X** and **Axes Y**, graph title, etc.

Change the names by **Axes Y** as shown in the Figure 1.21.

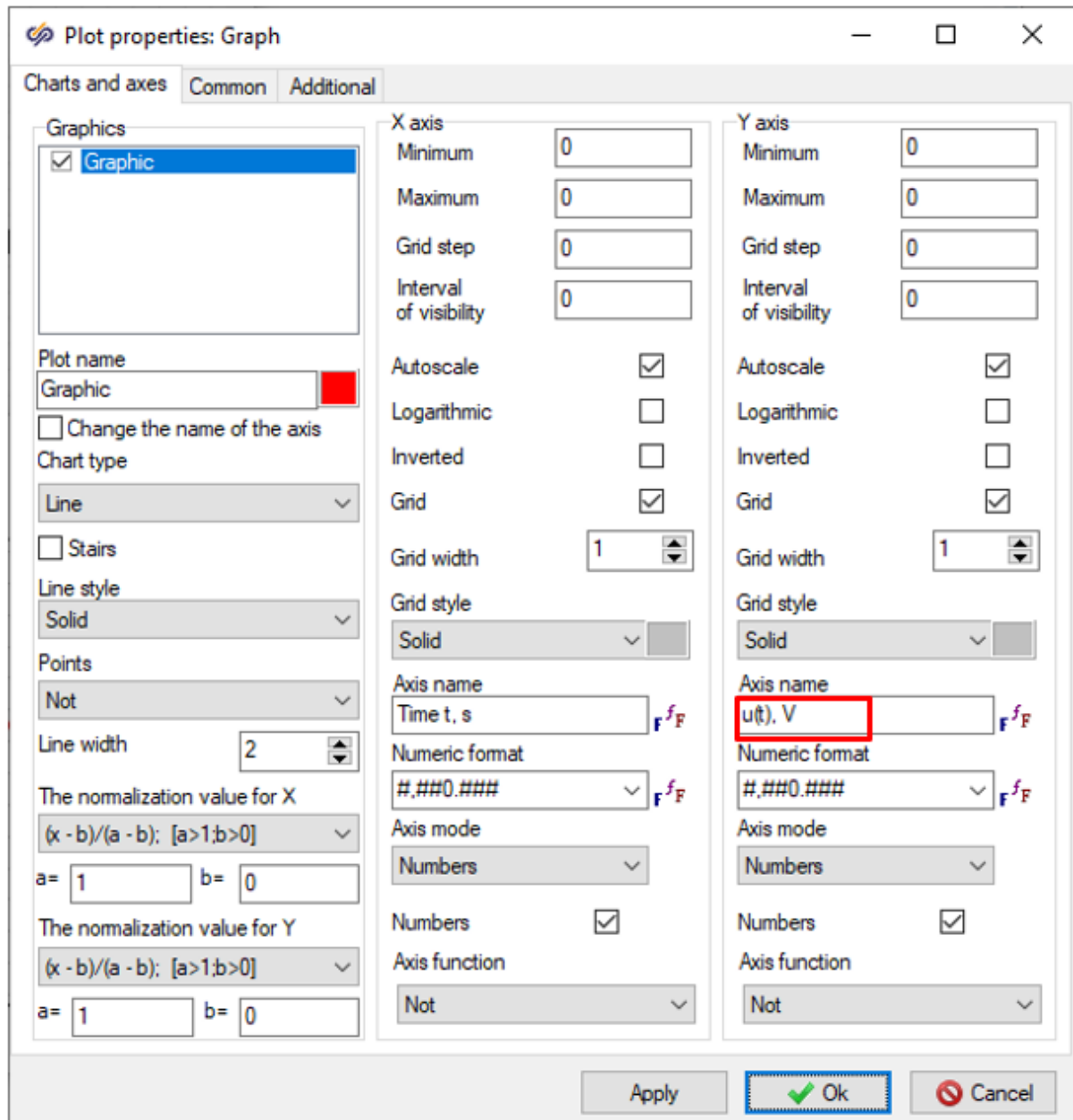


Figure 1.21

If the **Show legend** checkbox is selected on the **Common** tab of the **Plot properties** dialog box (Figure 1.22), then you can set its location (**Left**, **Right**, **Top**, **Bottom**). It is advisable to use a legend if more than one graph is displayed on the **Graph of time** block screen. In the Figure 1.22, the **Show legend** checkbox is unchecked.

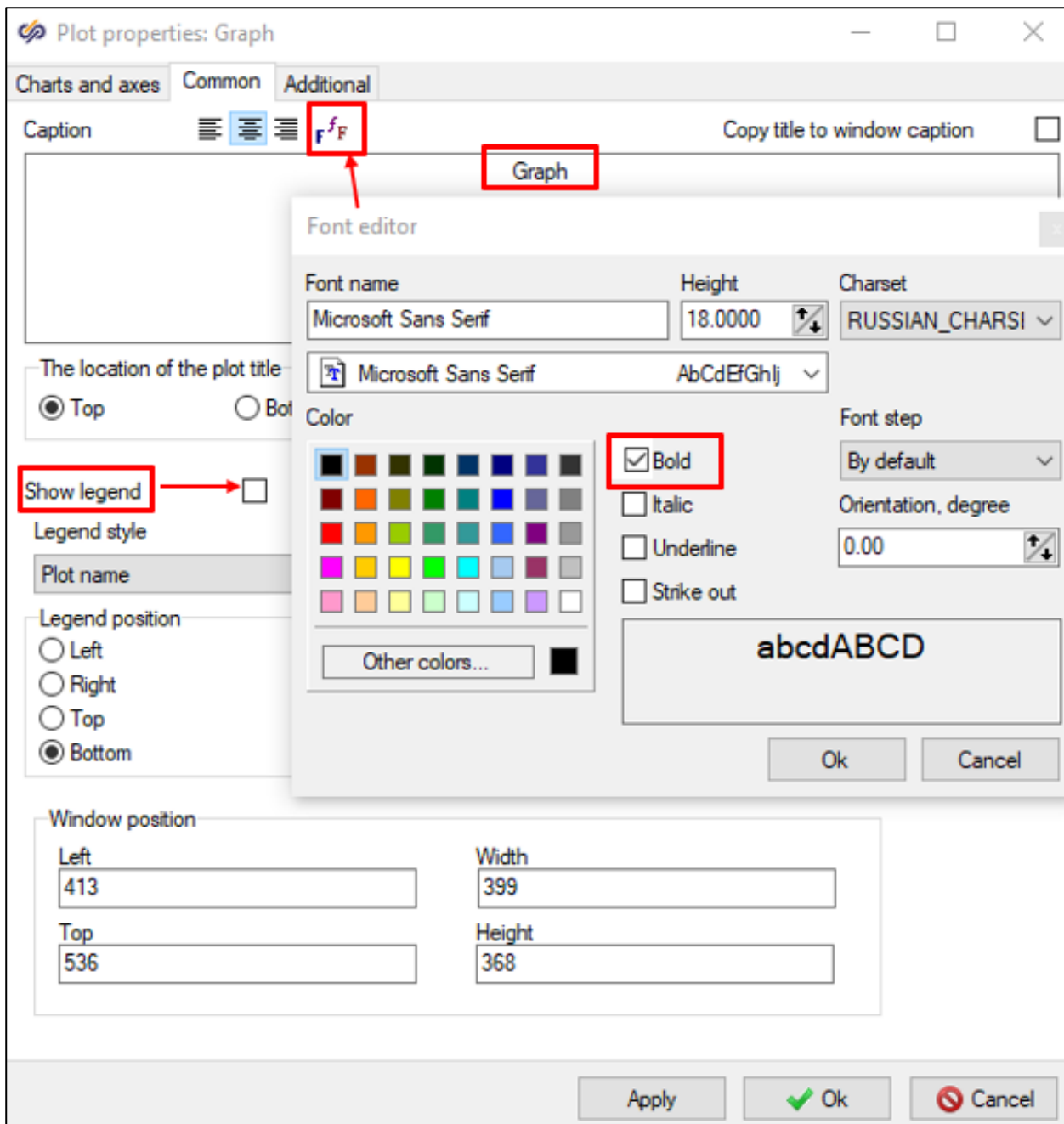


Figure 1.22

In addition, on the **Common** tab of the **Plot properties** dialog box, you can change the title of the graph. Change the heading of the graph by selecting the LMB **Graph** area and entering $u(t) = 7 \sin(314 t)$ from the keyboard.

Open the **Font editor** window by selecting the LMB area  and format the title by setting the font **Bold**.

Figure 1.23 shows how the screen of the **Graph of time** block will change after the changes are made.

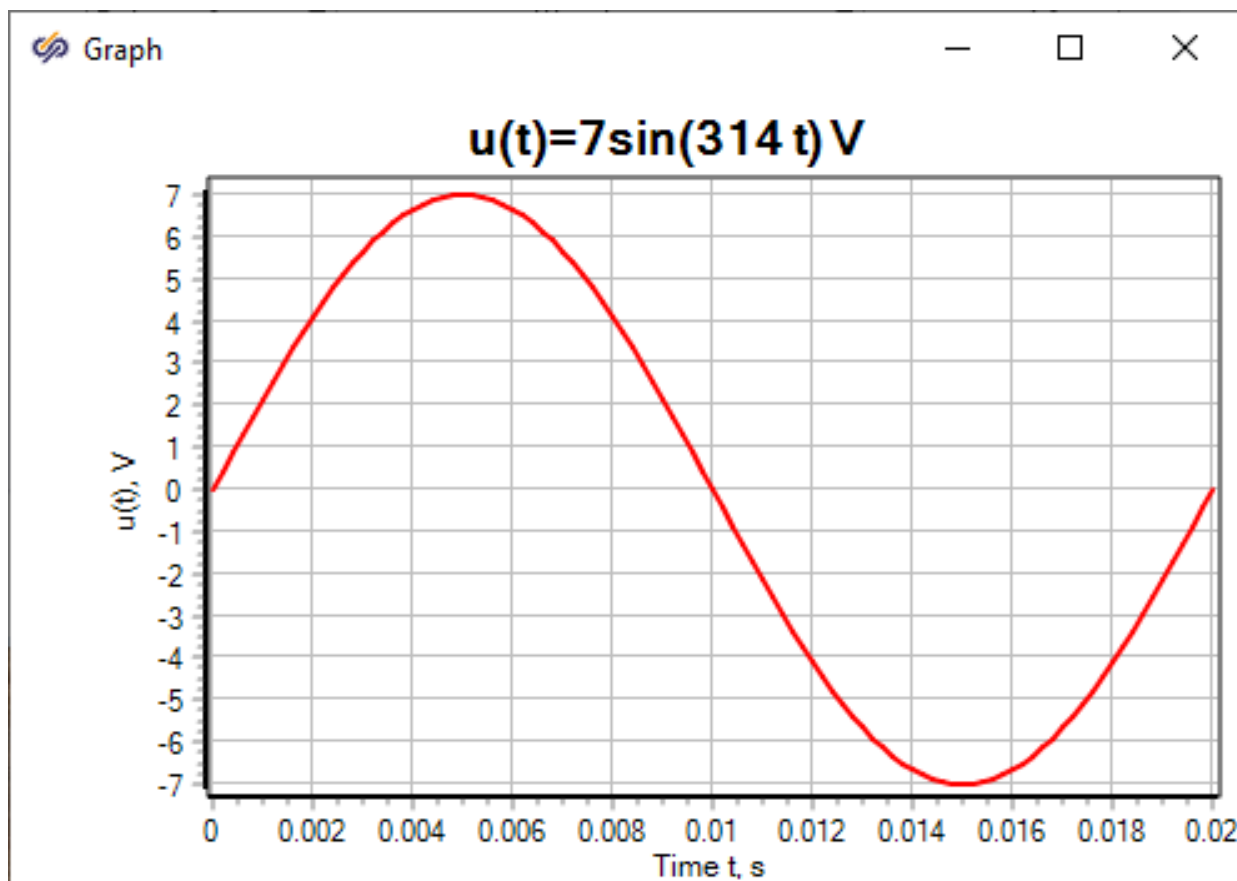


Figure 1.23

Report

Set the amplitude and frequency values of the **Voltage source** block and the resistance of the **RLC circuit** blocks equal to the variant number for the model in the Figure 1.16.

Set the final calculation time to one period of the harmonic signal at the output of the **Voltage source** block. Run the simulation process and see what changes on the dependency graph on the **Graph of time** block screen.

Insert the resulting project window into the laboratory report, indicating the parameter values for your version, as well as the resulting dependency graph on the screen of the **Graph of time** block.

Control questions

1. Which libraries in the SimInTech system are used for modeling electrical circuits and how do they differ?
2. What is the sequence of building a model in the projector window in the SimInTech system?
3. How are blocks inserted into the project window?
4. How are the blocks rotated in the project window?
5. How are the block properties set in the project window?

6. How are the blocks connected in the project window?
7. How do you make the same signal reach several blocks?
8. How can you delete a block or a link between blocks?
9. What calculation parameters are set to perform the simulation?
10. For what reasons is the simulation time chosen?
11. How is the step value selected when choosing a fixed-step simulation?
12. How is the simulation run performed in the SimInTech system?
13. What is the **Graph of time** block for?
14. How can you enlarge a graph fragment?
15. How can you copy to the clipboard, export to a file, print and save images of the **Graph of time** block in a text file?
16. How can you determine the coordinates of the specified points on the graph?
17. What settings exist in the **Plot properties: Graph** window?

Laboratory work № 2

Simulation of Direct Current Circuits in the SimInTech System

The purpose of the work: familiarization with the main features of the SimInTech system for modeling DC electrical circuits.

Work assignment

1. Assemble a DC circuit in the SimInTech system project window (Figure 2.1). *In*, A

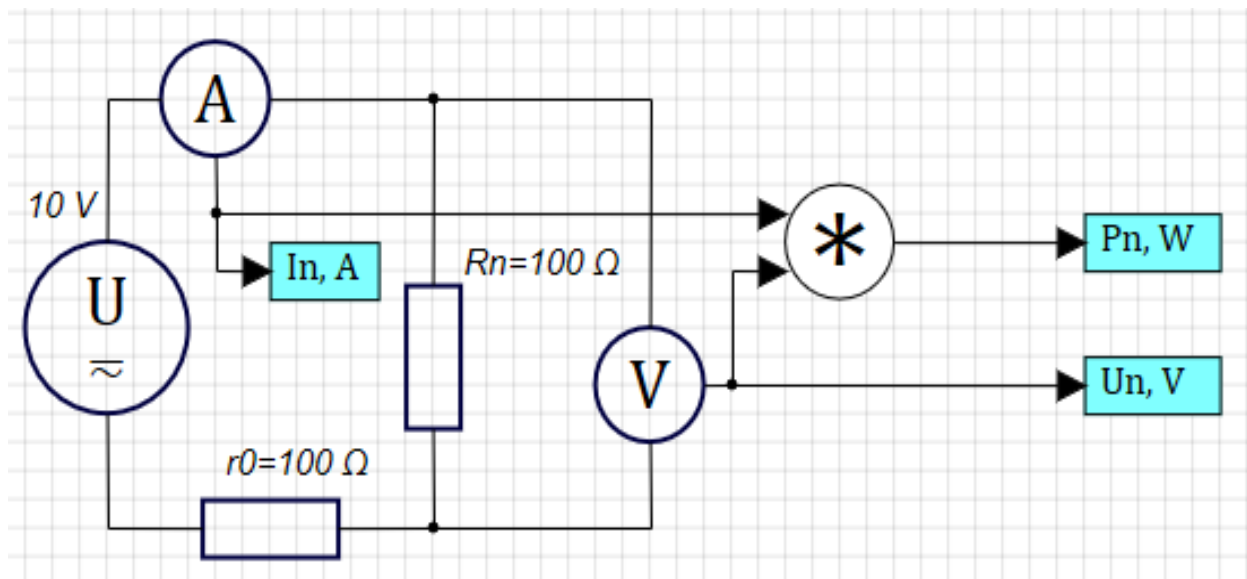


Figure 2.1

To do this, select the following blocks from the **EC – Dynamics** library:

- **Voltage source** block from the **Sources** tab;
- two **RLC-circuit** blocks from the **Electrical elements** tab;
- The **Ideal Voltmeter** and **Ideal Ammeter** blocks are from the **Meters** tab.

Add **The Multiplier** block from the **Operators** library and three **Output Port** blocks from the **Substructure** library.

2. Set the parameter values shown in the Figures 2.2–2.4 in the properties window for the **Voltage source** and **RLC circuit** blocks.

For the **Output Port** blocks, set the **Values of the submodel port** names in accordance with Figure 2.1 by double-clicking on them.

Properties : ac_U1

Properties Common Ports Visual layers

Search by caption, name, formula, value

Caption	Name	Expression	Value
Amplitude, V	Um		0
Constant component, V	Udc		10
frequency Hz	f		0
Phase, electrical.	fi		0
Source resistance			
Type of source resistance circuit	TypeOfCircuit		R
Active resistance, Ohm	R		0.01
Extra options			
Name on the scheme	sc_name		10 V
External signal	U_in		☐ No

Figure 2.2

Properties : Z1

Properties Parameters Common Ports Visual layers

Search by caption, name, formula, value

Caption	Name	Expression	Value
Main settings			
Type of circuit	Type		R
Resistance, Ohm	R_		[100]
Extra options			
Name on the scheme	sc_name		r0=100 Ohm
Show chain view	VisType		mnemonic
Parameters displayed in ports	out_ports		

Figure 2.3

Caption	Name	Expression	Value
Main settings			
Type of circuit	Type		R
Resistance, Ohm	R_		[100]
Extra options			
Name on the scheme	sc_name		Rn=100 Ohm
Show chain view	VisType		mnemonic
Parameters displayed in ports	out_ports		

Figure 2.4

3. Click the **Simulation properties** button  of the schematic project window and set the **Final calculation time** in seconds to **1**, and set the **Minimum step** and **Maximum step** to **0.01**.

Start the simulation process by clicking on the **Start** button.

After the following message appears in the calculation information display line of the Schematic window: **End time (time=0.01)**, press the **Show values on wires** key .

Figure 2.5 shows the simulation results for the resistance value of the right block of the **RLC-circuit** – $R_n = 100 \text{ Ohm}$.

4. Conduct a simulation for the circuit Figure 2.5, setting the constant component of the source voltage in volts equal to the number of the variant, and the value of the internal resistance r_0 and the load resistance R_n in ohms equal to the number of the variant plus 50.

Report

Insert the contents of the received project window into the laboratory report, indicating the values of all parameters for your option on the diagram.

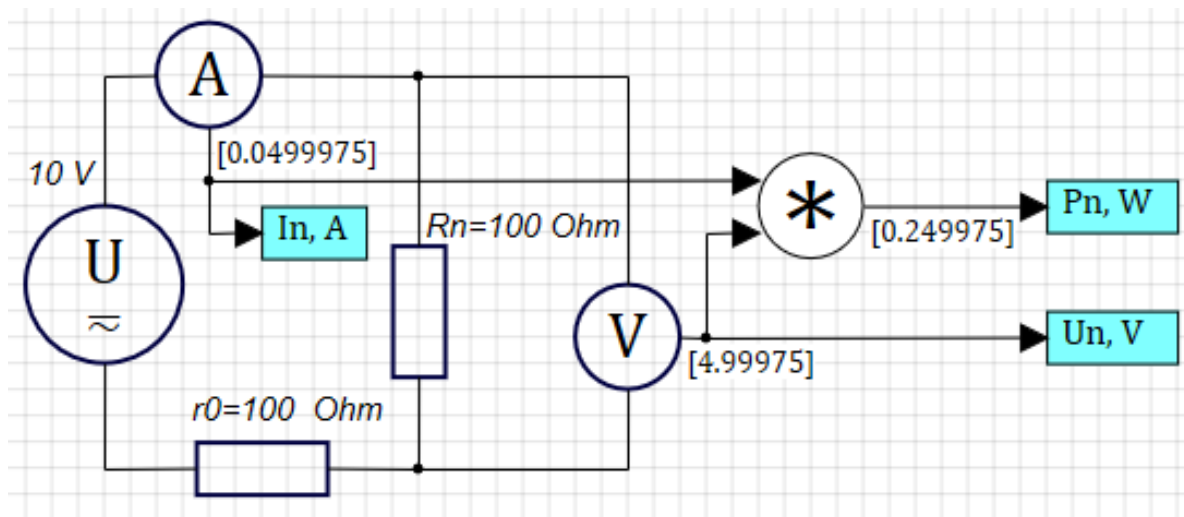


Figure 2.5

5. Assemble DC circuit in the project window of the SimInTech system to remove the volt-ampere characteristic (VAC) (Figure 2.6).

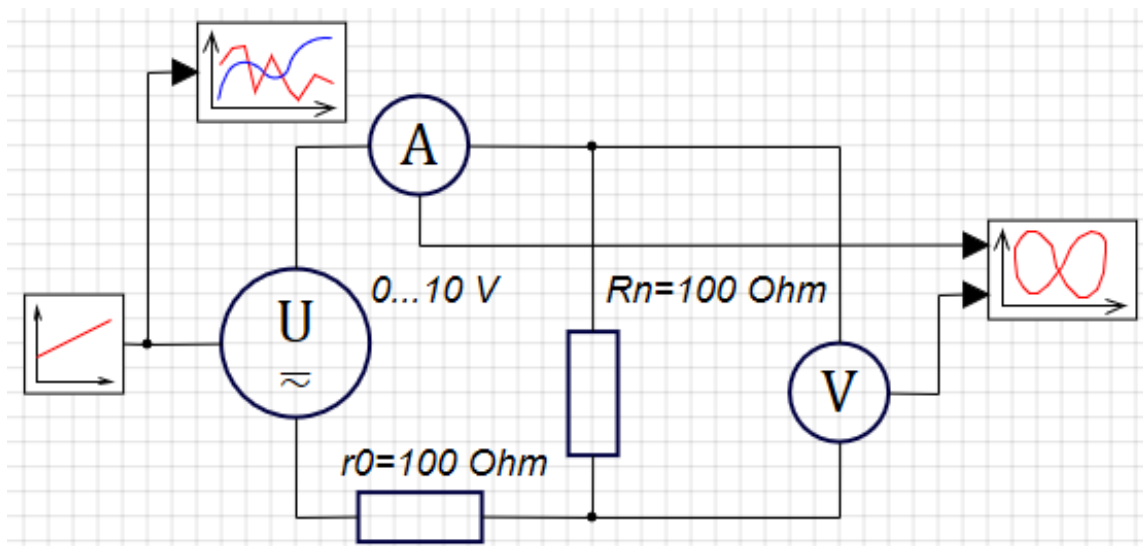
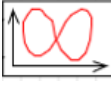
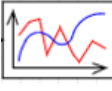


Figure 2.6

To do this, select from the **EC – Dynamics** library:

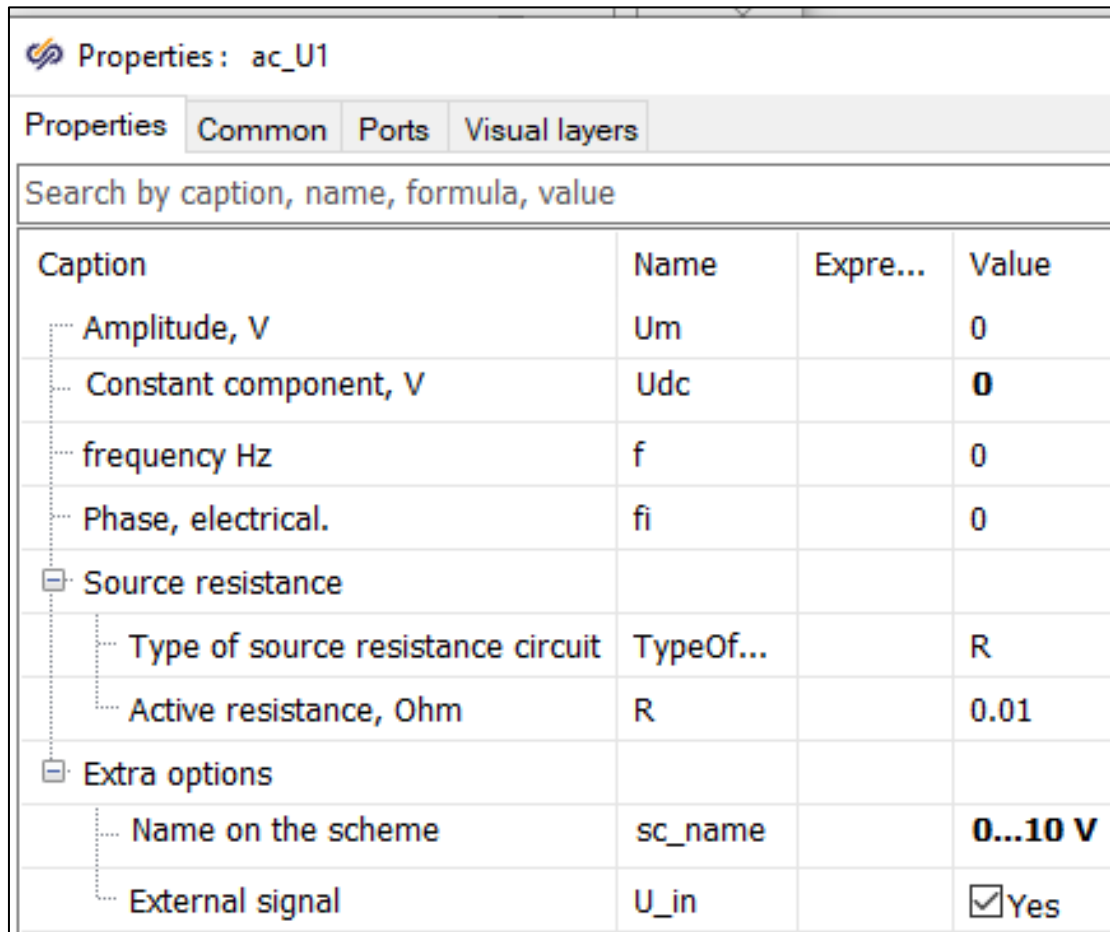
- the **Voltage source** block from the **Sources** tab;
- two **RLC-circuit** blocks from the **Electrical component** tab;
- the **Perfect voltmeter** and **Ammeter perfect** blocks are from the **Sensors** tab.

Add the **Phase Portrait** block  and the **Graph of time** block  to the schematic window of the project from the **Output of data** palette.

The mathematical communication ports for the output of the instantaneous current value i of the **Ammeter perfect** block and the voltage u of the **Perfect voltmeter** block are connected to the inputs of the **Phase Portrait** block as shown in the Figure 2.6.

6. Set the property values in the properties window on the **Properties** tab for **RLC-circuit** blocks in accordance with Figures 2.3, 2.4.

7. In the properties window on the **Properties** tab for the **Voltage source** block, set the properties value in accordance with Figures 2.7.



Properties: ac_U1			
Properties	Common	Ports	Visual layers
Search by caption, name, formula, value			
Caption	Name	Expre...	Value
Amplitude, V	Um		0
Constant component, V	Udc		0
frequency Hz	f		0
Phase, electrical.	fi		0
Source resistance			
Type of source resistance circuit	TypeOf...		R
Active resistance, Ohm	R		0.01
Extra options			
Name on the scheme	sc_name		0...10 V
External signal	U_in		☒ Yes

Figure 2.7

As a result of setting the parameter value, **Extra options** → **External signal** → **Yes**, the **us** mathematical communication port will appear on the left of the **Voltage source** block, designed to control the voltage value of the source.

As an external signal for the **Voltage source** block, use the **Linear Source**  block from the **Sources** library, the output signal of which is also applied to the **Graph of time** block.

The **Linear source** block generates the value of the output signal as a function of time as

$$y(t) = a + b \cdot t.$$

Set the following property values for it on the **Properties** tab:

- **Free member** – **a = 0**;
- **Coefficient at t** – **b = 10**.

8. Click the **Simulation properties** button  of the schematic project window and set the **Final calculation time** in seconds to **1**, and set the **Minimum step** and **Maximum step** to **0.2**, so that the number of points for modeling is equal to five, not counting the starting point;

Run the resulting model for simulation, press the **Show values on wires** button . As a result, the mathematical communication ports of the ammeter and voltmeter will display the values of current and voltage at the load resistance R_h (Figure 2.8).

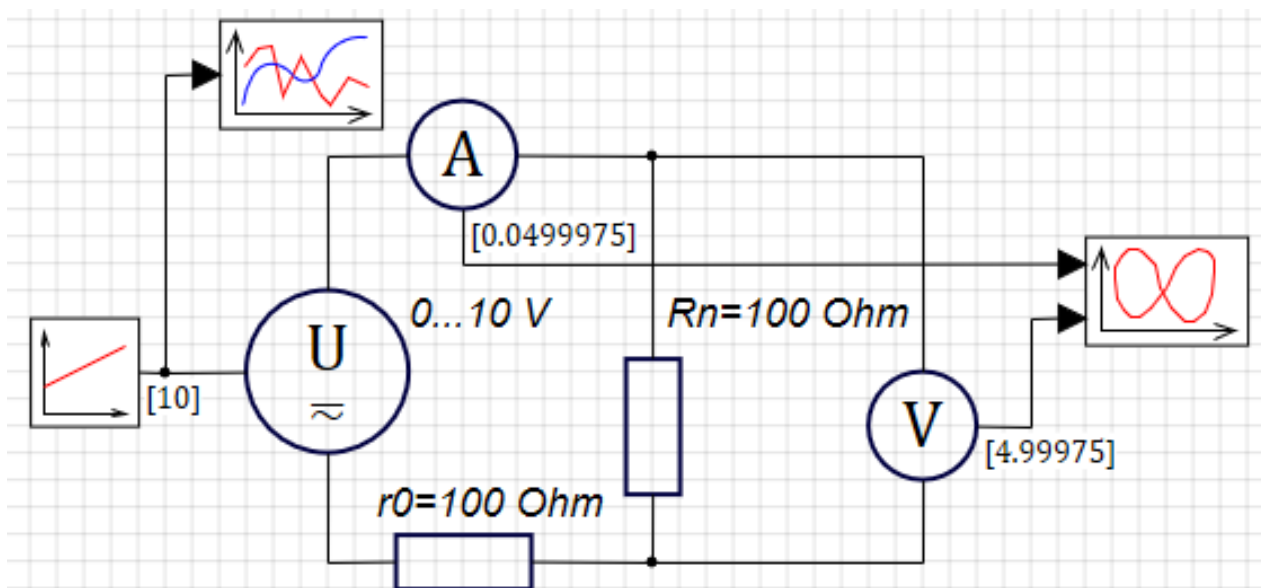


Figure 2.8

Double-click the graphical windows of the **Phase Portrait** and **Graph of time** blocks and view the modeling results in them.

Set the values of the following parameters in the properties window for graphical windows: line style and thickness, type of points on the graph, the name of the axes on the **Graphics and axes** tab, as well as the name of the graph on the **General** tab in accordance with the Figures 2.9, 2.10.

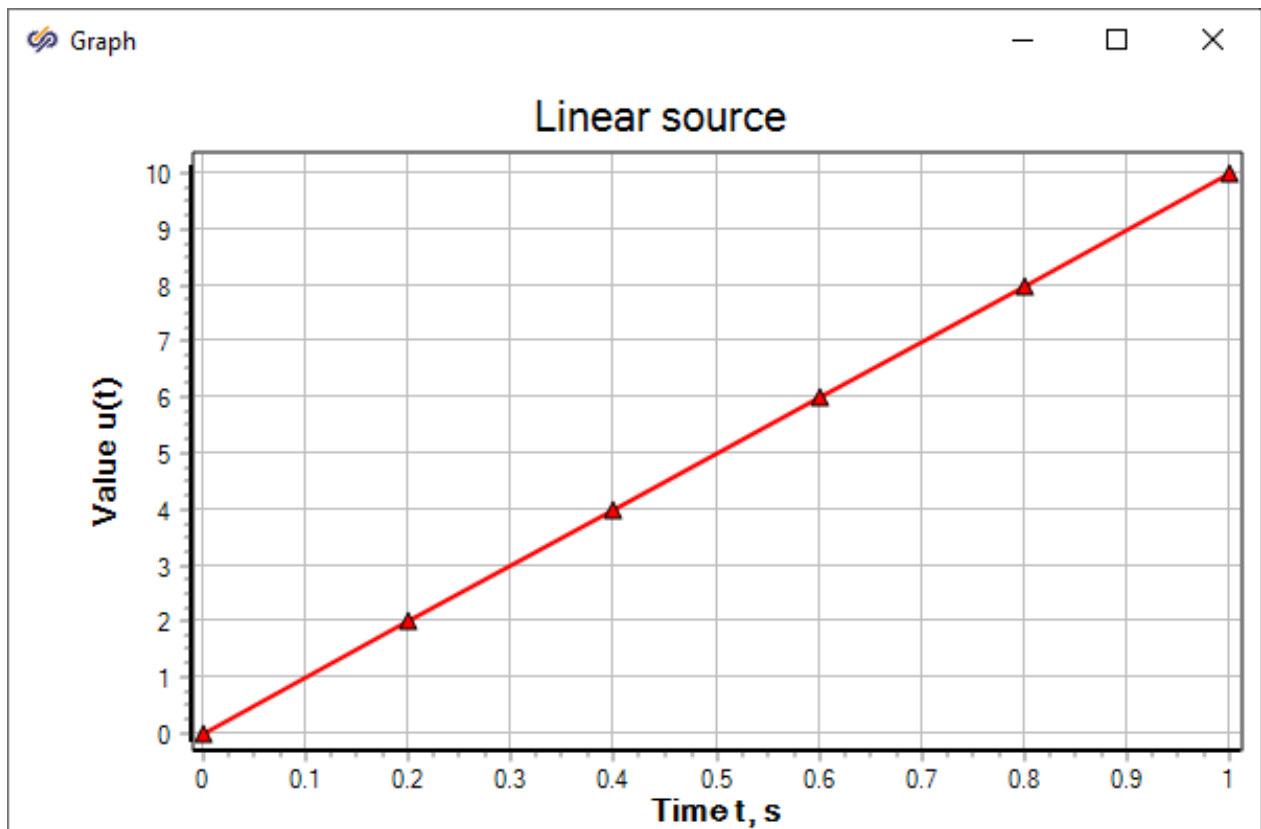
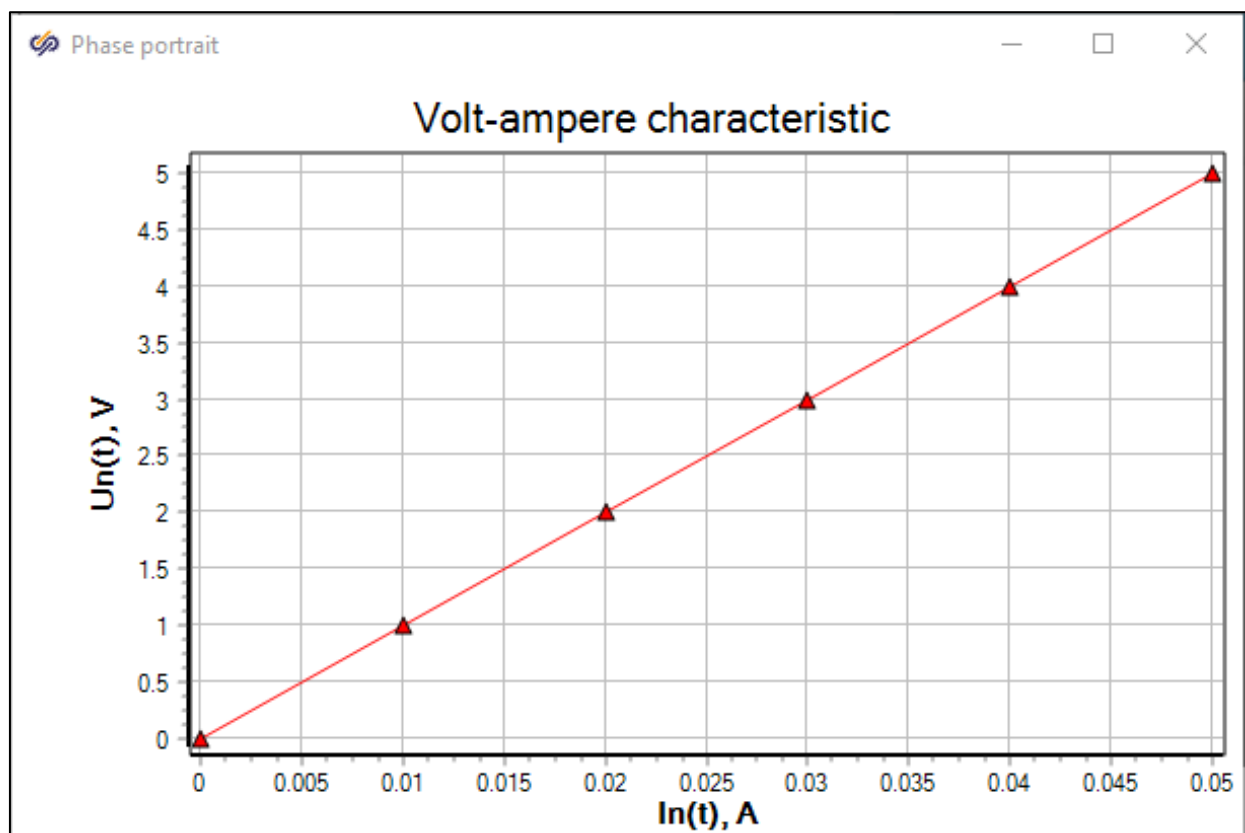


Figure 2.9



Figures 2.10

9. Conduct a simulation for the circuit Figure 2.8, setting the resistance values r_0 and R_n in ohms equal to the option number plus 50.

Report

Insert the contents of the resulting project window into the laboratory report, indicating on the diagrams the values of all parameters for your variant, as well as the contents of the graphical windows **Graph of time** and **Phase portrait** with the current modeling results for your variant.

Analyze the obtained graphs and draw conclusions about the linearity of the VAC.

Based on the results of the experiment, determine their coordinates for any two points on the graph and conclude that Ohm's law is fulfilled.

10. In the SimInTech project window, assemble a DC circuit to obtain the dependences U_n and P_n for the given R_n from the current I_n flowing through it.

The scheme includes the blocks shown in the Figure 2.11.

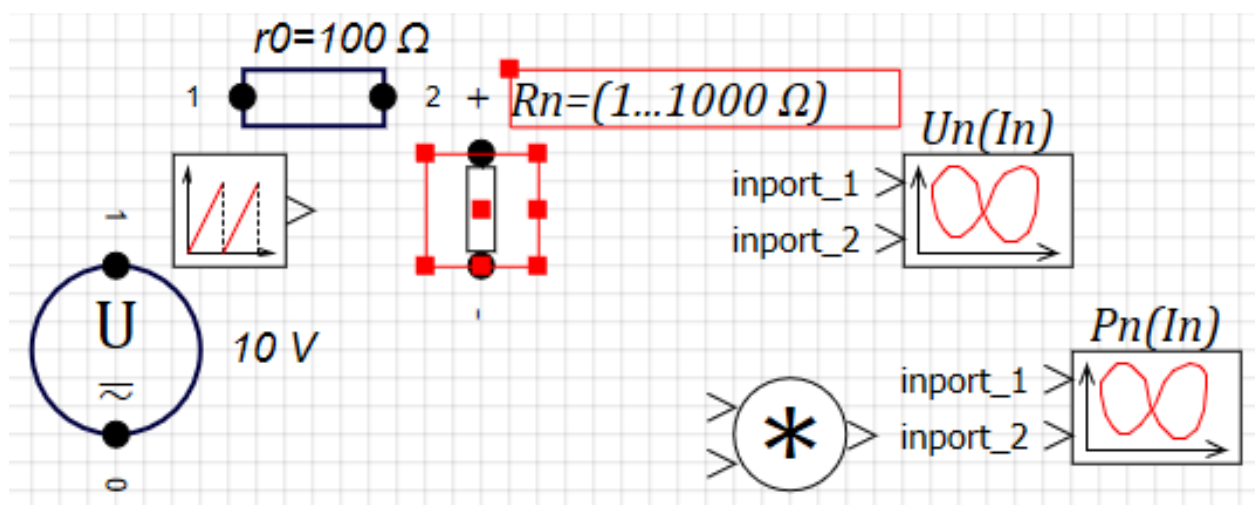
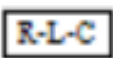


Figure 2.11

Select from the **EC – Dynamics** library:

- the **Voltage source** block from the **Sources** tab;
- the **RLC-circuit** block from the **Electrical component** tab;
- the **Generalized chain** block from the **Developer Items** tab (in the

palette , which is used as a variable load resistance R_n (highlighted in the Figure 2.11).

Add a **Multiplier** block from the **Operators** library, a **Saw** block from the **Sources** library, and two **Phase Portrait** blocks from the **Data of Output** library.

11. Set in the properties window:
 - for the **Voltage source** block, the property values are in accordance with the Figure 2.2;
 - for the **RLC circuit** block, the property values are in accordance with the Figure 2.4;
 - for the **Phase Portrait** and **Generalized chain** blocks, the Block **Signature** property values on the **General** tab in accordance with the Figure 2.11.

In the properties window for the **Generalized chain** block, set the following property values shown in Figure 2.12.

For the **Type of chain element** property, select the value **R** from the drop-down list. For the **Element resistance, Ohms** property set the value in ohms to **1000**.

To select the required **Parameters set from ports** in the properties window of the **Generalized circuit** block, hover the cursor in this row over the right side of the **Value** column before the button  appears and click on it (Figure 2.13).

As a result, the **Select objects from the list** window opens, in which you can select the required **Sources** (in the Figure 2.14 **Resistance** is selected).

Similarly, you can select the required **Parameters displayed in ports** (Figure 2.15).

This opens the **Select Objects from the list** window, where you can select the required **Sources** (**Current** and **Voltage** are selected in the Figure 2.16).

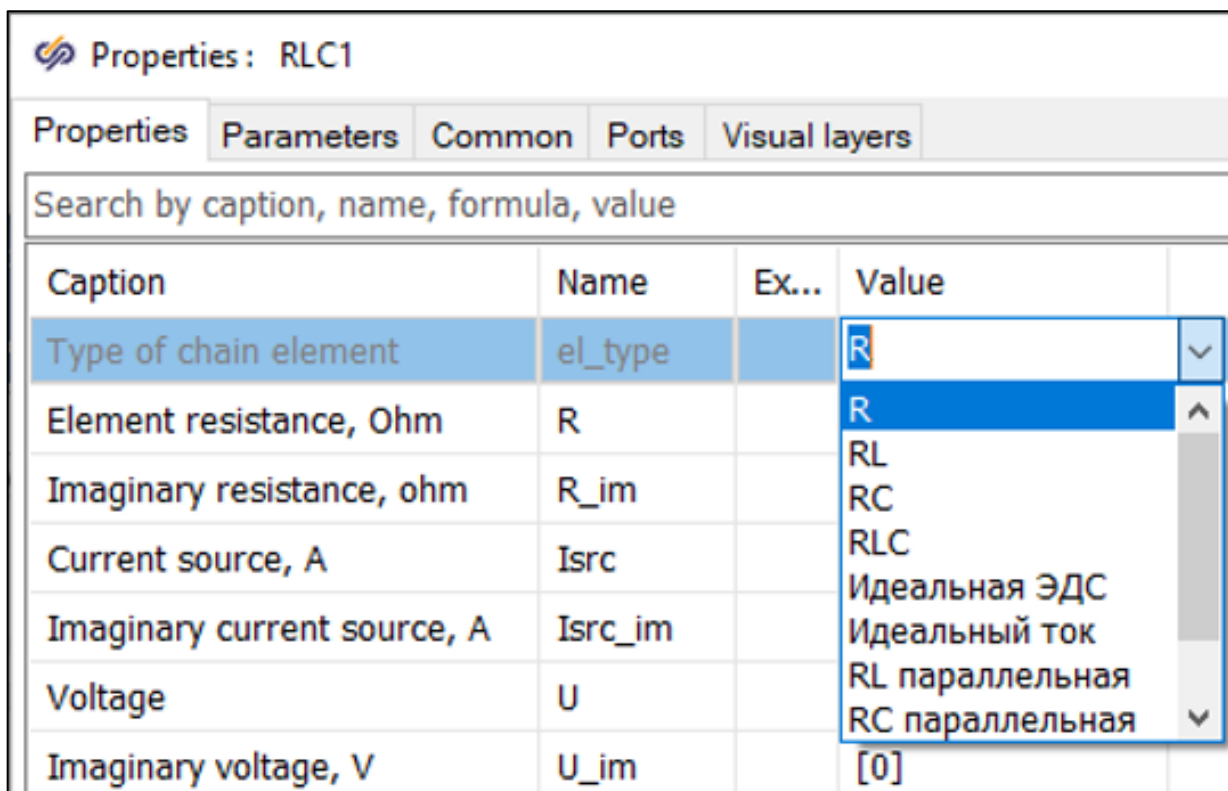


Figure 2.12

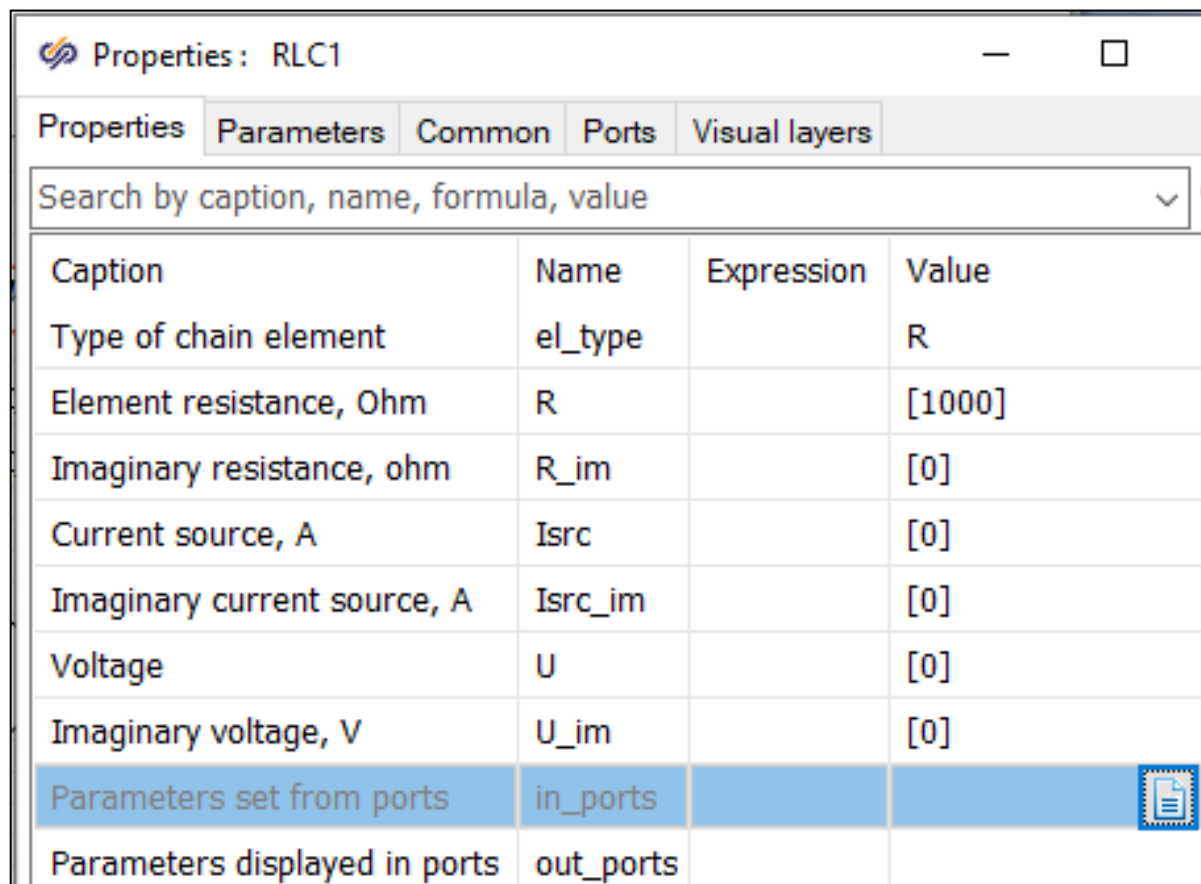


Figure 2.13

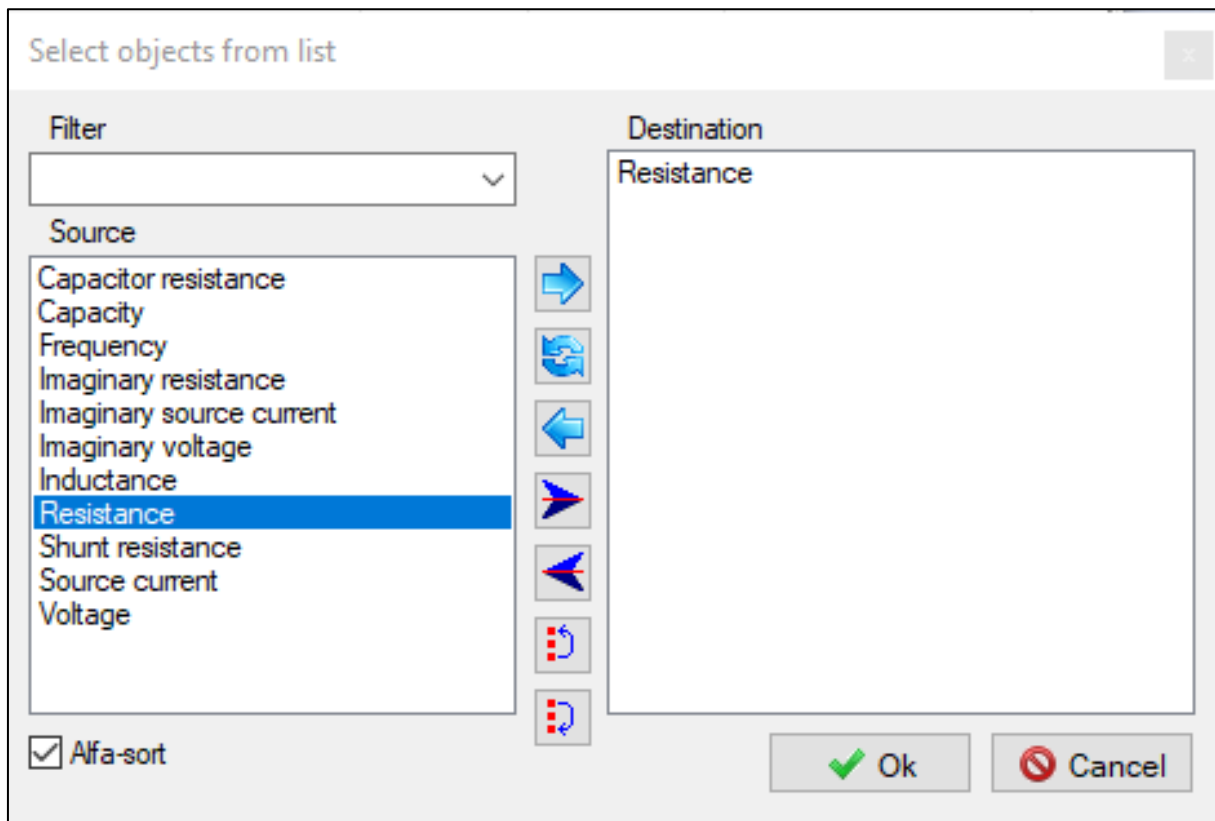


Figure 2.14

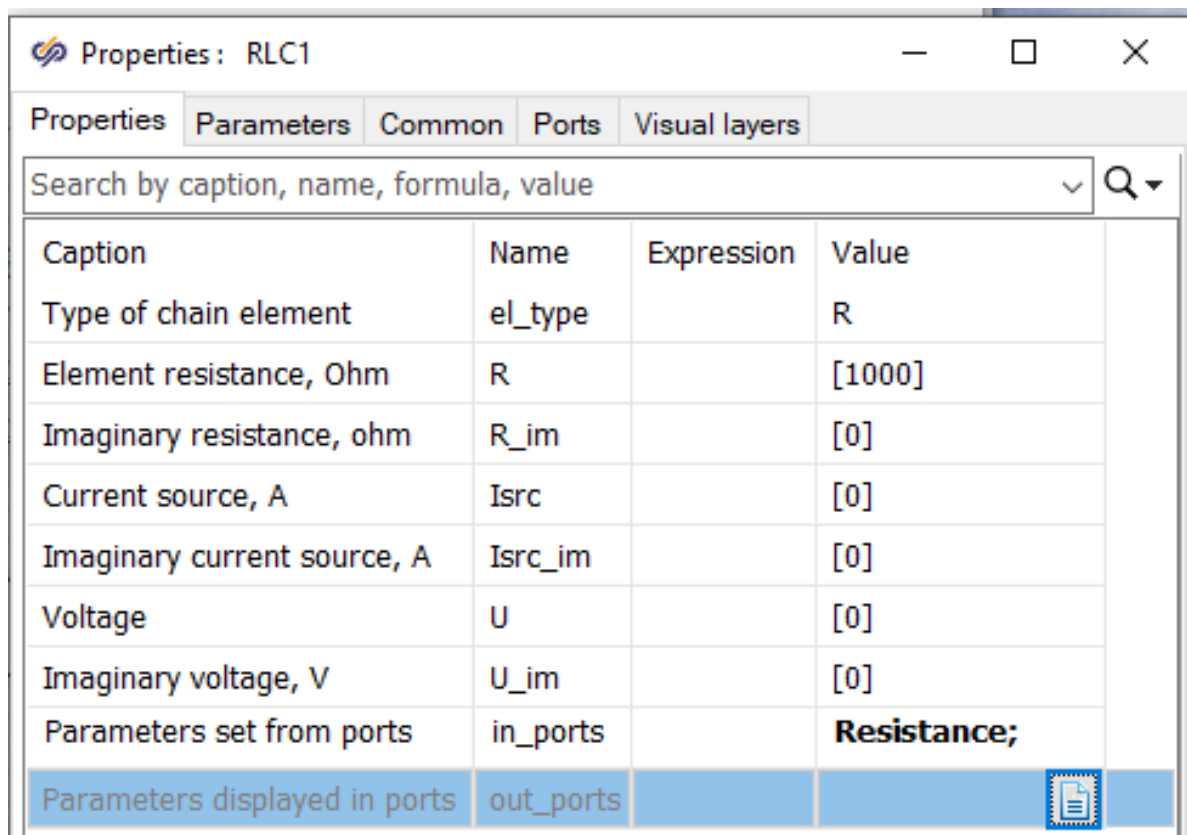


Figure 2.15

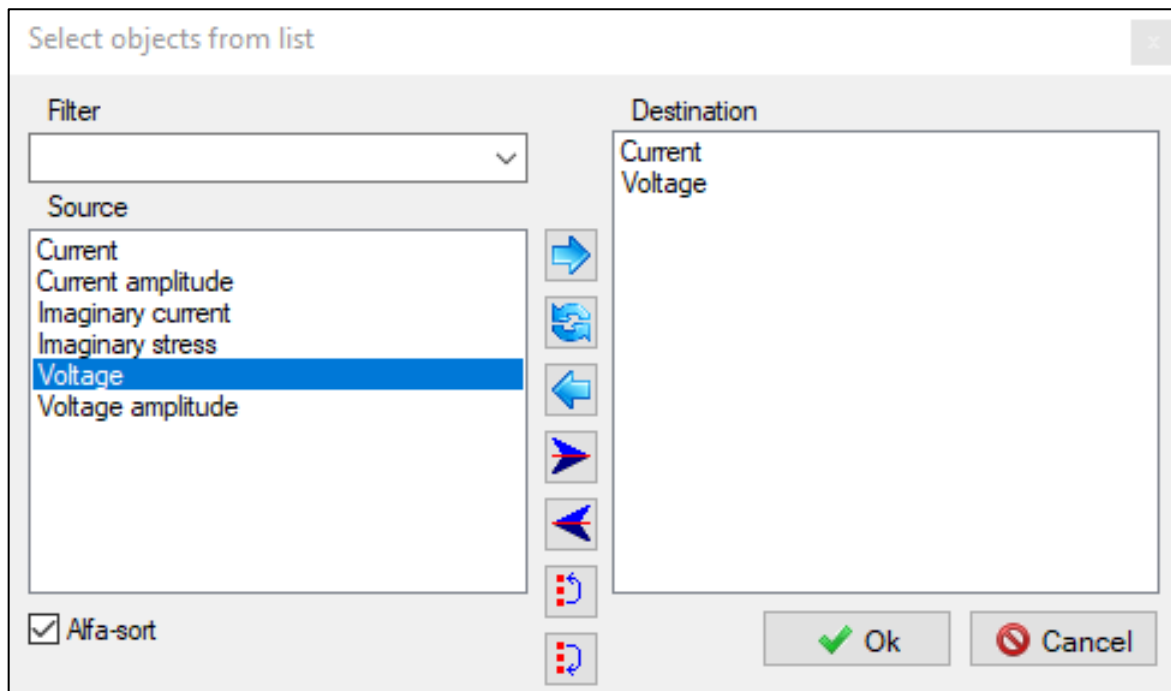


Figure 2.16

As a result, three mathematical ports will appear to the left and right of this block:

- **R** – for resistance control in ohms;
- **Ire** – to output the real value of the current flowing through the unit;
- **Ure** – to output the real value of the voltage applied to the block.

Set the parameter values shown in the Figure 2.17 in the properties window for the **Saw** block.

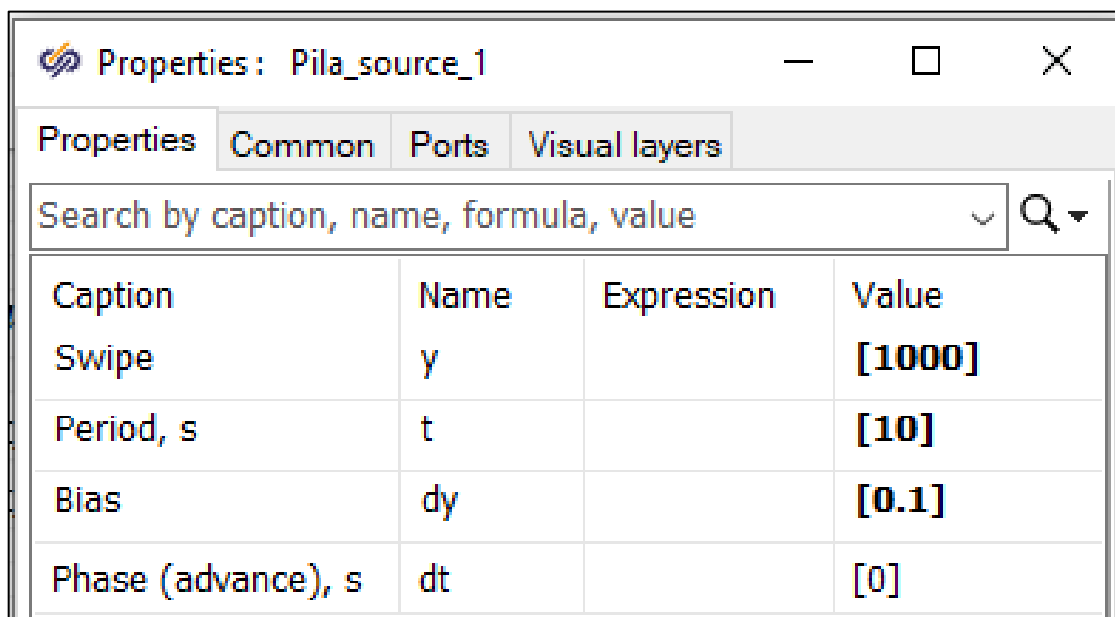


Figure 2.17

12. Connect the blocks together in accordance with the Figure 2.18.

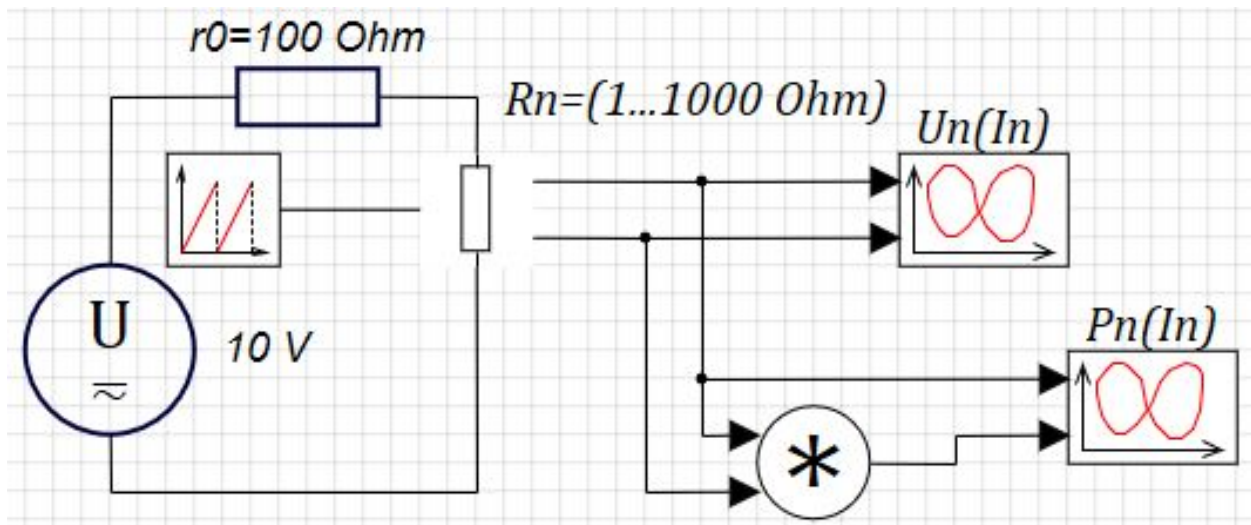


Figure 2.18

The **R** port of the **Generalized chain** block is connected to the output of the **Saw** block.

The **Ire** port of the **Generalized chain** block is connected to the upper input of the **Phase Portrait** block and to the input of the **Multiplier** block.

The **Ure** port of the **Generalized chain** block is connected to the lower input of the **Phase Portrait** block ($Un(In)$) and to the input of the **Multiplier** block.

The output of the **Multiplier** block is connected to the lower input of the **Phase portrait** block ($Pn(In)$).

13. Click the **Simulation properties** button  of the schematic project window and set the **Final calculation time** in seconds to **10**, and set the **Minimum step** and **Maximum step** to **0.001**.

Run the resulting model for modeling and view the simulation results in graphical windows.

Set parameter values in the properties window for graphical windows: line style, axis name, and graph name in accordance with the Figures 2.19, 2.20.

14. Perform a simulation for the circuit in the Figure 2.18 by setting the value of the constant voltage component for the **Voltage source** block in volts equal to the option number plus 10.

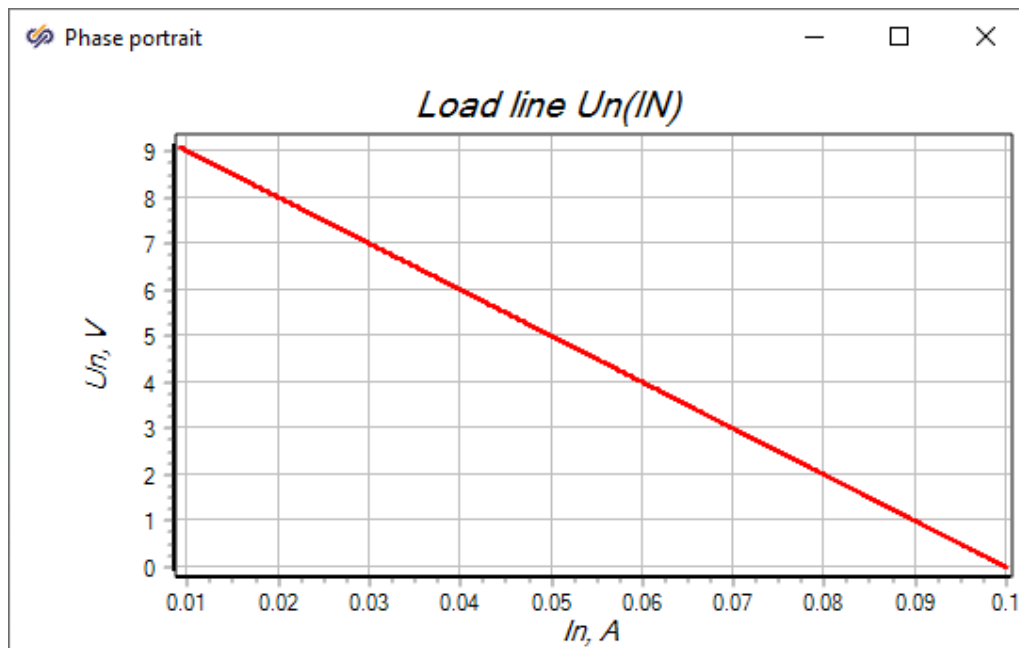


Figure 2.19

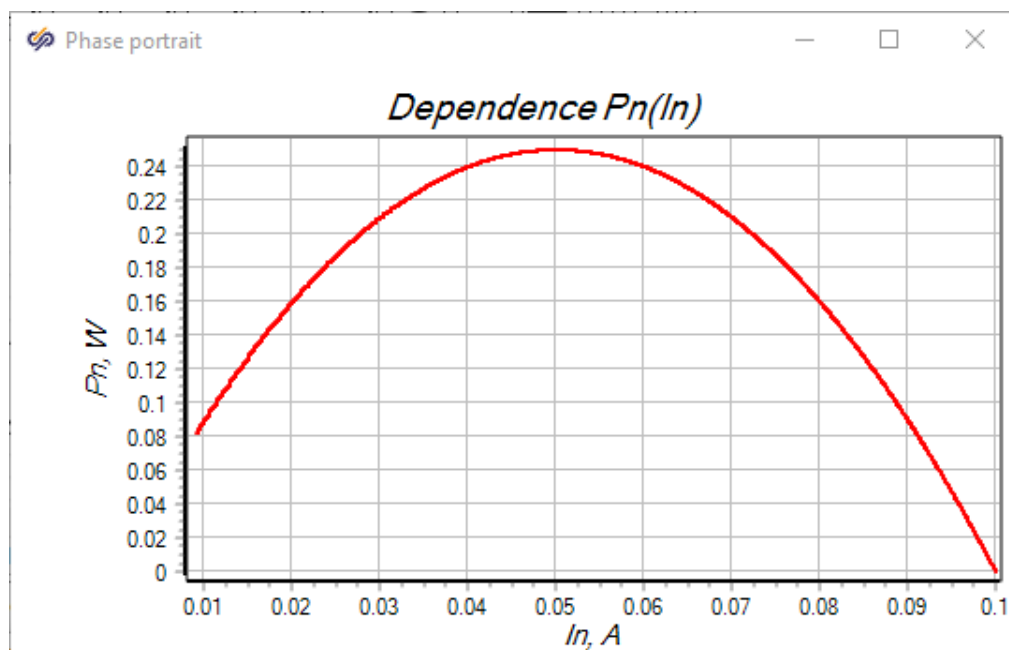


Figure 2.20

Report

Insert the contents of the resulting project window into the laboratory report, indicating the values of all parameters for your variant on the diagrams, as well as the contents of the **Phase Portrait** graphical windows with the current modeling results for your variant.

Analyze the graph of the load line and explain the decrease in voltage U_n with an increase in current I_n , taking the resistance r_0 for the internal resistance of the Voltage source.

Analyze the $P_n(I_n)$ graph. Determine at what resistance R_n value the maximum power from the Voltage source is transmitted to it.

15. Replace the **Saw** block in the model in the Figure 2.18 with a **Piecewise Constant** block from the **Sources** library (Figure 2.21), which generates an arbitrary multi-stage output signal.

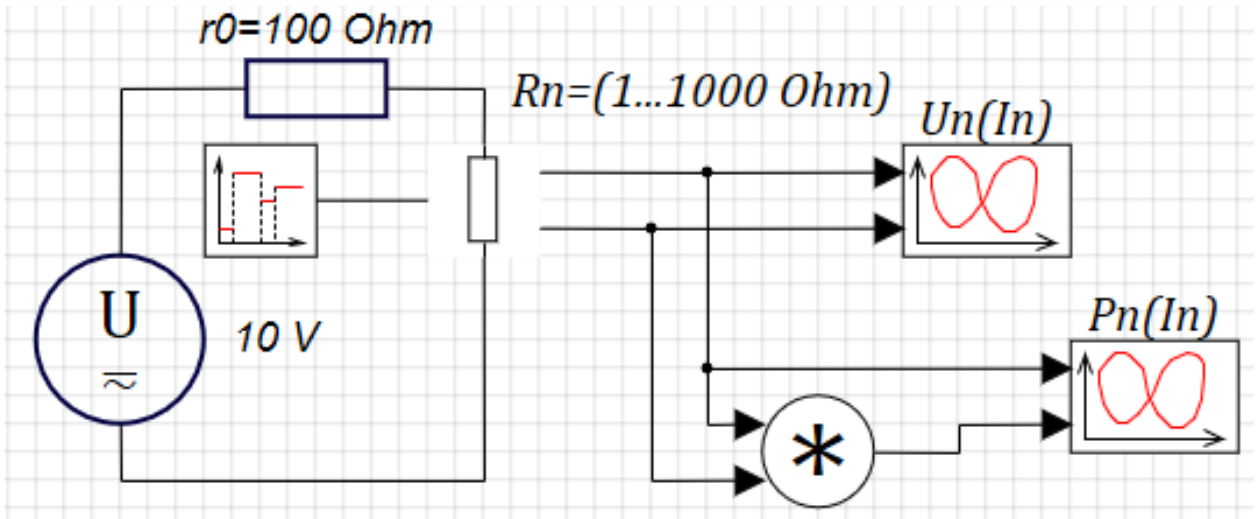


Figure 2.21

This will allow you to set a number of discrete values of the resistances R_h and plot graphs for these values Figures 2.19, 2.20.

Set the eight parameter values shown in the Figure 2.22 in the properties window for the **Piecewise constant** block.

16. Click the **Simulation properties** button of the schematic project window and set the **Final calculation time** in seconds to **10**, and set the **Minimum step** and **Maximum step** to **1**.

Run the resulting model for simulation and view the simulation results in graphical windows (Figures 2.23, 2.24).

Properties: PolyStep_source_1		
Properties Common Ports Visual layers		
Search by caption, name, formula, value		
Caption	Name	Value
Array of time intervals	t	[[1 , 1 , 1 , 1 , 1 , 1 , 1 , 1]]
Array of Values	y	[[0.1 , 25 , 50 , 100 , 200 , 400 , 700 , 1000]]

Figure 2.22

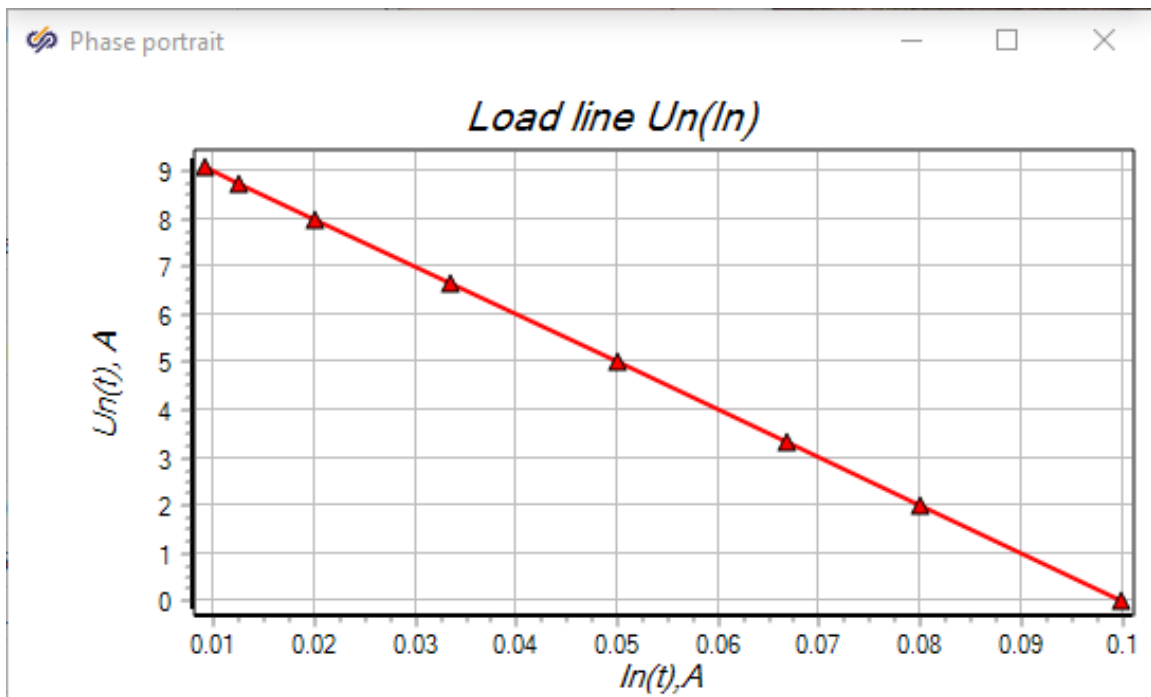


Figure 2.23

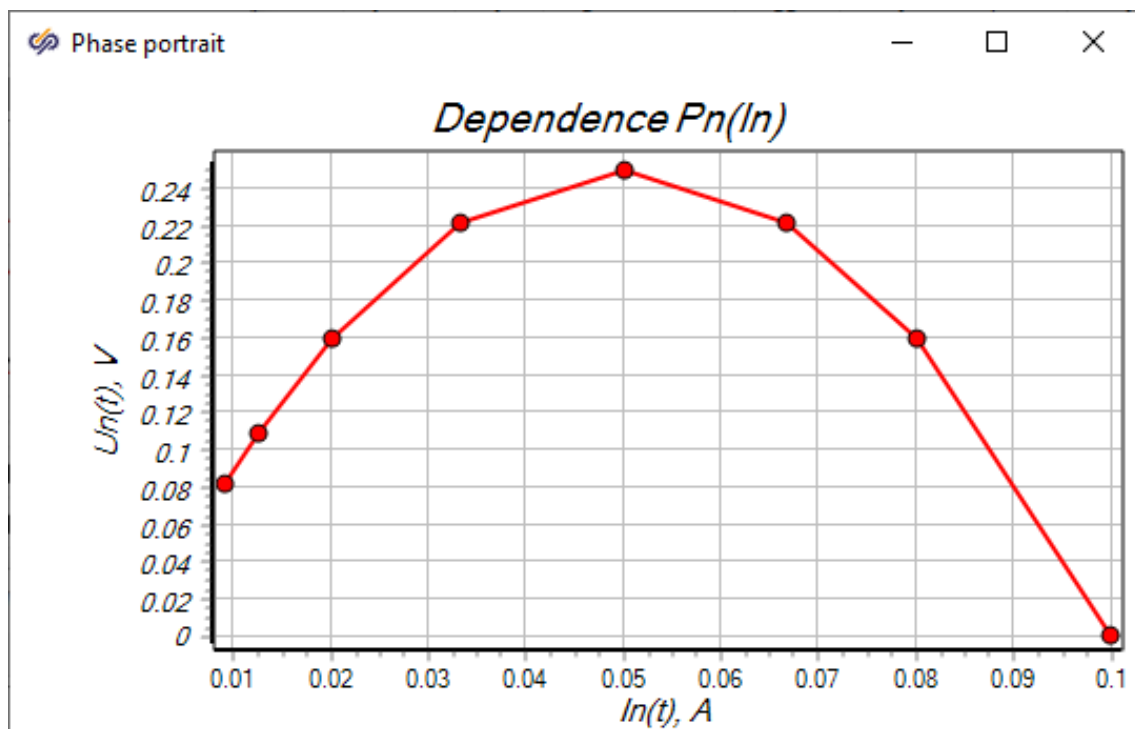


Figure 2.24

17. Perform a simulation for the circuit Figure 2.22 by setting the values of the internal resistance r_0 in ohms equal to the option number plus 50.

In the properties window for the block, set the **Piecewise constant** eight parameter values to an **Array of – y** values equal to ohms:

0,1· r_0 ; 0,25· r_0 ; 0,5· r_0 ; r_0 ; 2· r_0 ; 4· r_0 ; 7· r_0 ; 10· r_0 .

Report

Insert the contents of the resulting project window into the laboratory report, indicating the values of all parameters for your variant on the diagrams, as well as the contents of the **Phase Portrait** graphical windows with the current modeling results for your variant. Compare the graphs obtained with similar graphs obtained when performing clause 14. Draw conclusions.

Control questions

1. In which library is it located, what are the main properties and the purpose of **The Multiplier** block?
2. In which library is it located, what are the main properties and the purpose of the **Output Port** block?
3. In which library is it located, what are the main properties and the purpose of the **Linear Source** block?
4. What is the **Phase Portrait** block for?
5. In which library is it located, what are the main properties and the purpose of the **Saw** block?
6. In which library is it located, what are the main properties and the purpose of the **Piecewise Constant** block?
7. In which library is it located, what are the main properties and the purpose of the **Generalized Chain** block?
8. What is the purpose of the **External signal** port in the **Voltage source** block?
9. How can you set the block signature in the required format?

Laboratory work № 3

Simulation of Alternate Current Circuits in the SimInTech System

The purpose of the work: familiarization with the main features of the SimInTech system for modeling AC electrical circuits.

Work assignment

1. Assemble a dynamic model of the 1st-order inertial link in the project window of the SimInTech system when a harmonic signal is applied to its input (Figure 3.1).

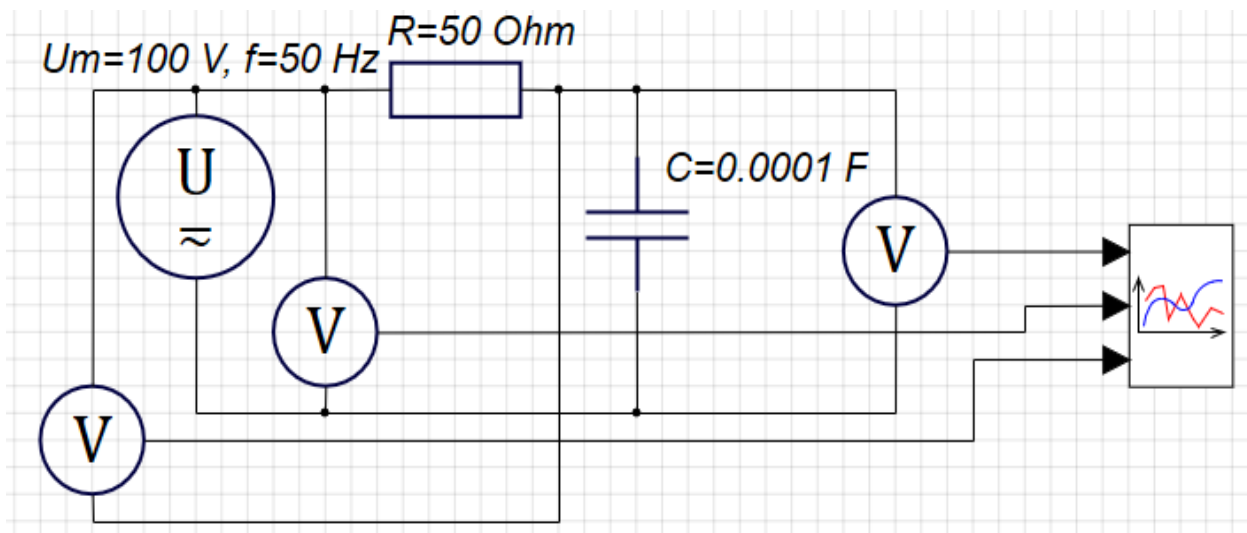


Figure 3.1

Select the following blocks from the **EC – Dynamics** library:

- the **Voltage source** block from the **Sources** tab;
- two **RLC-circuit** blocks from the **Electrical elements** tab;
- Three **Perfect voltmeter** blocks from the **Sensors** tab.

Add the **Graph of time** block from the **Output of data** library.

2. Use the **Properties** window to set the following properties:

- for the **Voltage source** block, the value of the property **Amplitude, V**, is equal to the option number, the value of the property **frequency, Hz**, is equal to the option number multiplied by 10, and the value of the property **Active resistance, ohm** is **0.01**;

- for the upper block of the **RLC-circuit**, the value of the property **Resistance, ohm**, equal to 100 is divided by the option number;

– for the right block of the **RLC-circuit**, the value of the property **Capacity, F**, is equal to **0.0001**.

3. Click the **Simulation properties** button  of the schematic project window and set:

– the **Final calculation time** equal to three input voltage periods (for the diagram in Figure 3.1, this value is 0.06 s);

– the **Minimum step (Maximum step)** is 100 times less than the final simulation time (for the scheme in the Figure 3.1, this value is 0.0006 s)

Leave the other settings as default.

4. Run the scheme for simulation. At the end of the calculation, when you click on the **Show values on wires** button , the instantaneous voltage values at the end of the simulation will be indicated on the mathematical outputs of the voltmeters in volts (Figure 3.2):

– at the input of the circuit, equal to – **0.0089993859 V**;

– at the resistance, equal to **44.99693 V**;

– at a capacity equal to –**45.005929 V**.

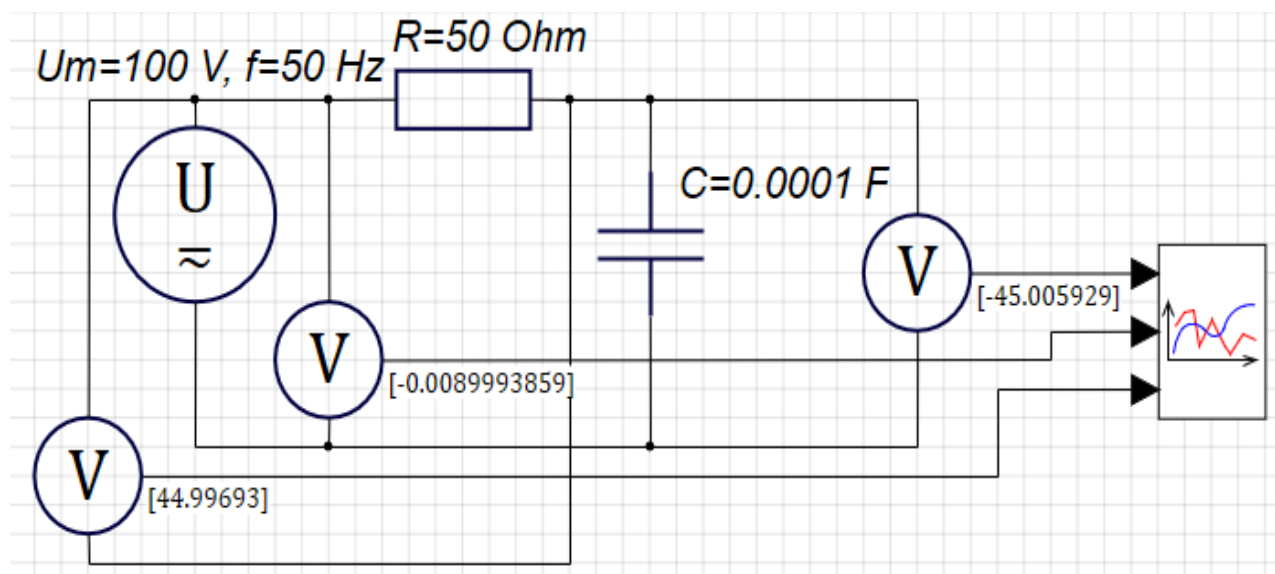


Figure 3.2

Double-clicking on the **Graph of time** block opens a window with time diagrams of the voltages entering its three inputs. Set the necessary settings in the **Plot properties: Graph** window to get graphs as shown in the Figure 3.3.

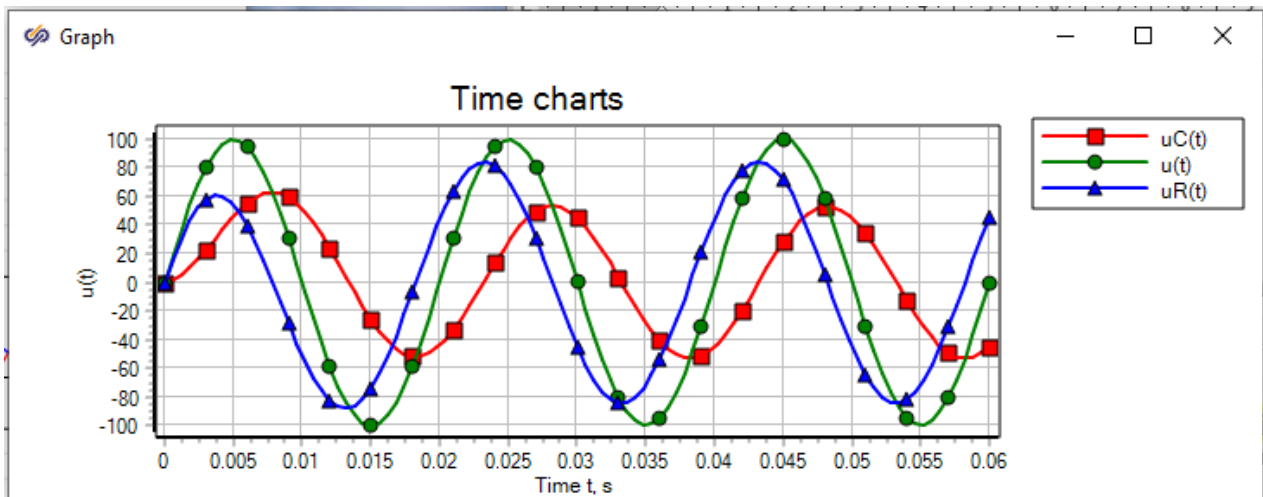


Figure 3.3

Report

Insert a simulated diagram into the laboratory report with the values of all parameters and the modeling results for your variant, as well as a graphical window with the corresponding time diagrams in the **Graph of time** block.

5. Set the **Minimum** and **Maximum** parameter values in the **X axis** section of the **Plot properties: Graph** window so that the graphic window displays one last period of the harmonic signal for your variant. For the diagram in Figure 3.1, these values are **0.04** and **0.06**, respectively. Also, uncheck the checkbox for configuring **Autoscale** in the **X axis** section.

After pressing the **Apply** or **Ok** key, one period of harmonic signals will be displayed in the window of the **Time graph** block (Figure 3.4).

It can be seen from the graphs that the voltage across the resistance outstrips the voltage across the capacitance by 90° , which corresponds to the theory.

Report

Insert the graph of the Figure 3.4 for your option number into the lab report. Analyze the graphs obtained and draw conclusions about the phase shifts between the resistance and capacitance voltages.

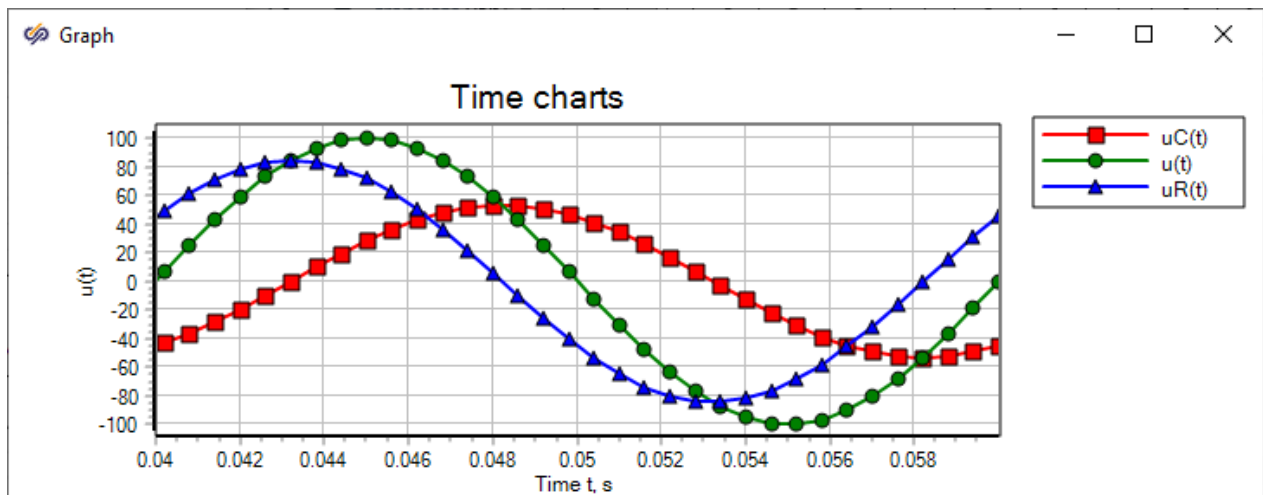


Figure 3.4

6. Assemble a dynamic model of a 1st-order inertial link in the project window of the SimInTech system when a harmonic signal is applied to its input and measure the total and active power in it (Figure 3.5).

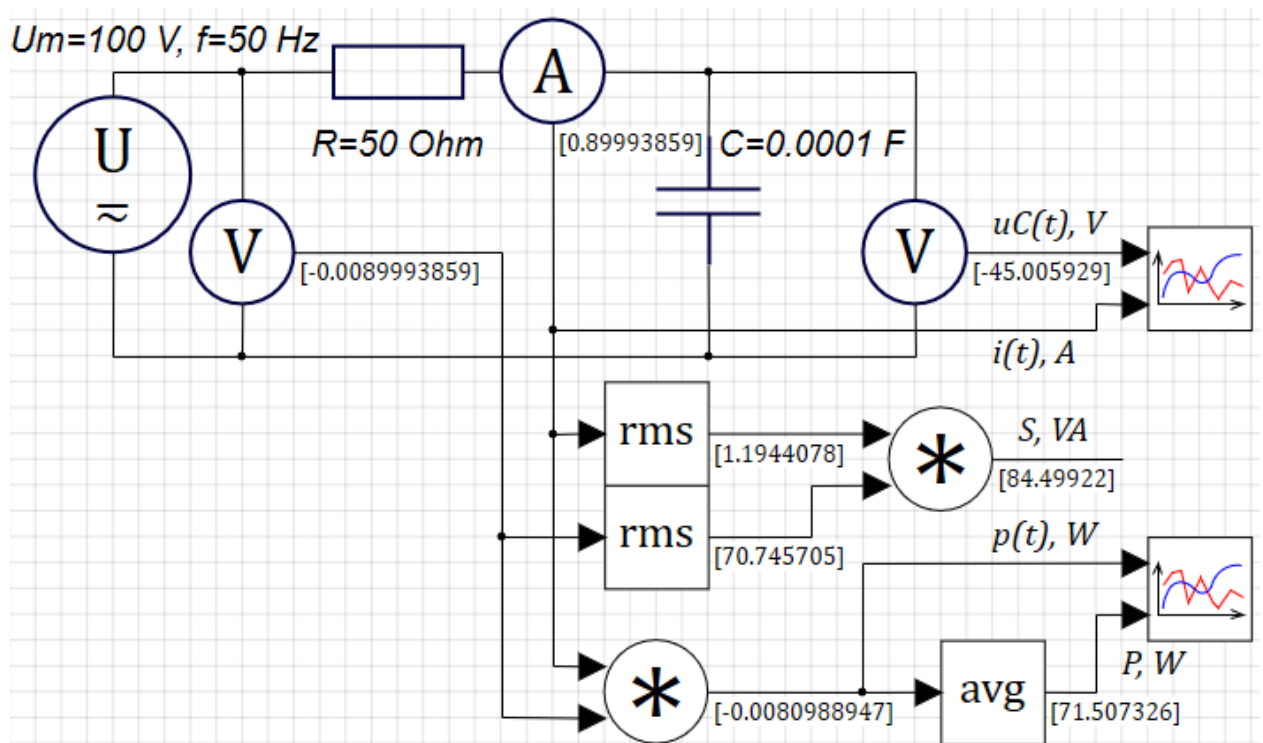


Figure 3.5

Select the following blocks from the **EC – Dynamics** library:

- the **Voltage source** block from the **Sources** tab;
- two **RLC-circuit** blocks from the **Electrical components** tab;
- two **Perfect voltmeter** blocks from the **Sensors** tab;
- the **Ammeter perfect** block from the **Sensors** tab;

– two blocks **RMS value for the period (rms)** and block **Period average (avg)** from the **Controls and signal processing** tab.

Add the two **Multiplier** block from the **Operators** library and two **Graph of time** block from the **Output of data** library.

7. Set the following properties using the Properties window:

– for the **Voltage source** and **RLC circuit** blocks, the same parameter values as in the diagram of Figure 3.1;

– for blocks **RMS value for the period** and **Period average**:

– the value of the properties **Period, s**, is equal to the period of the input signal (for the scheme of Figure 3.5 – **0.02 s**);

– the value of the properties **Discretization step, s**, when integrating with a constant step, it is recommended to set a multiple of the integration step (for the scheme of Figure 3.5 – **0.0006 s**).

8. Click the **Simulation properties** button  of the schematic project window and set:

– the **Final calculation time** equal to three input voltage periods (for the diagram in the Figure 3.1, this value is 0.06 s);

– the **Minimum step (Maximum step)** is 100 times less than the final simulation time (for the scheme in the Figure 3.1, this value is 0.0006 s)

Leave the other settings as default.

9. Run the simulation scheme and click on the button . The simulation results corresponding to the time of 0.06 s are shown in the Figure 3.6.

At the output of the upper **Multiplier** unit, a value of **84.49922** is formed, equal to the total power in volt amperes.

At the output of the block, the **Period average** is **71.507326**, equal to the active power consumed by the circuit from the Voltage source in watts.

Figure 3.6 shows the time diagrams **$uC(t)$** and **$i(t)$** on the screen of the **Graph of time** block for the model from Figure 3.5. It can be seen from the graphs that since there is a reactive element (capacitor) in the circuit, a transient process takes place at the beginning of the simulation, which is almost completed in about one period.

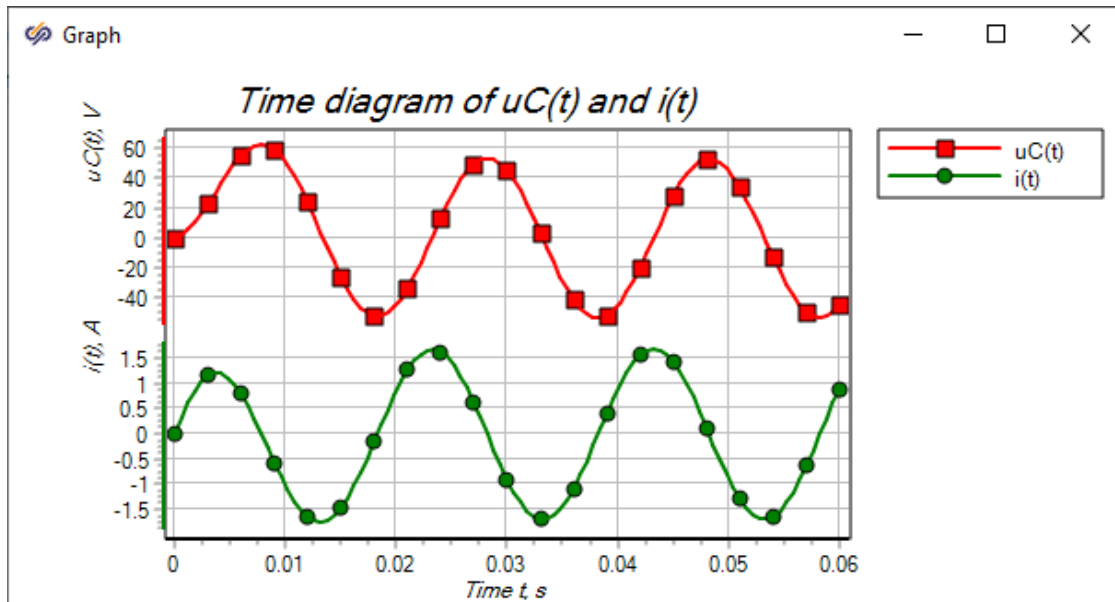


Figure 3.6

Figure 3.7 shows a time diagram on the screen of the **Graph of time** block for the instantaneous power $p(t)$ and the active power P consumed by the circuit from the Voltage source, which is equal to the average power over the period. It can be seen from the time diagram that the active power is approximately equal to its calculated value shown in the Figure 3.5.

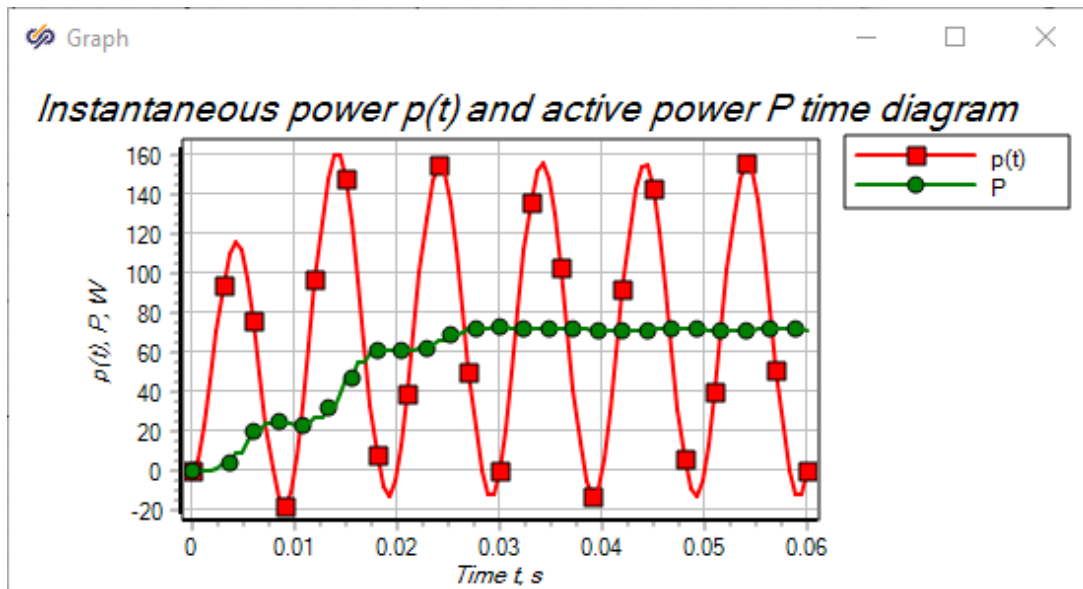


Figure 3.7

10. On the tab, determine the **Parameters** of the **Properties** window of the right block of the **RLC-circuit**, the voltage across the capacitance at the end of the simulation is **Voltage, V**. For the circuit, Figure 3.5, this value in volts is **-45.00592903** (Figure 3.8).

Properties : Z2		
Properties	Parameters	Common
Search by caption, name, formula, value		
Caption	Name	Value
Voltage, V	Ure_out	[-45.00592903]
Current, A	Ire_out	[0.8999385928]

Figure 3.8

11. Set the value of the **Initial voltage across the capacitor, V**, equal to the voltage across the capacitance in steady state for the right block of the **RLC circuit** on the **Properties** tab of the **Properties** window (Figure 3.9).

Properties : Z2		
Properties	Parameters	Common
Search by caption, name, formula, value		
Caption	Name	Value
[-] Main settings		
Type of circuit	Type	C
Capacity, F	C	[0.0001]
[-] Extra options		
Name on the scheme	sc_name	C=0.0001 F
Show chain view	VisType	mnemonic
Parameters displayed in ports	out_ports	
[-] Initial conditions		
Начальное напряжение конденсатора, В	UC_0	[-45.00592903]

Figure 3.9

12. Run the scheme for simulation.

Figure 3.10 shows the time diagrams $u_C(t)$ and $i(t)$ on the screen of the **Time graph** blocks for non-zero initial conditions. It can be seen from the graphs that there is no transition process in the scheme.

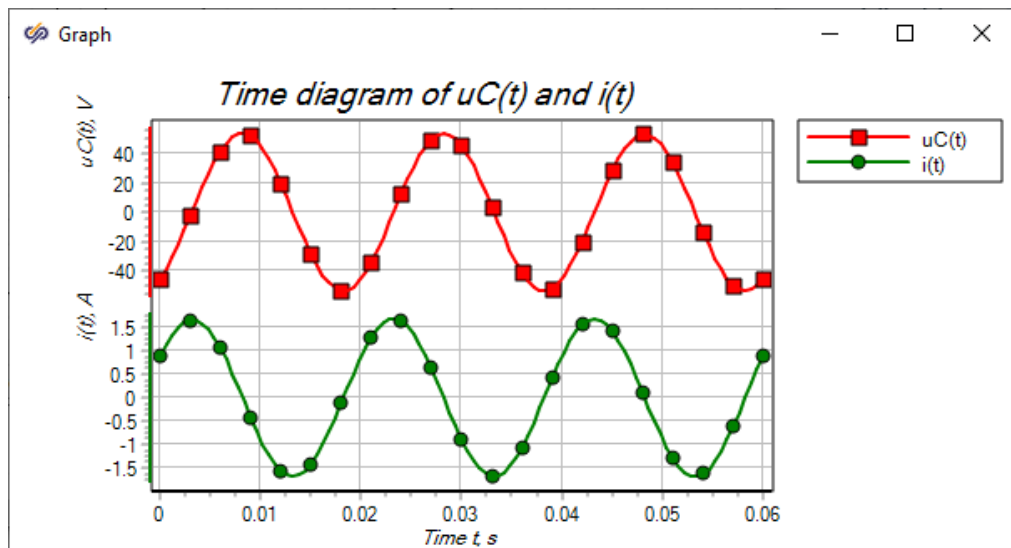


Figure 3.10

Report

Insert into the laboratory report a simulated scheme with simulation results for your variant, as well as graphs with simulation results in the **Graph of time** blocks for paragraphs 6–12. Analyze the graphs obtained and draw conclusions.

13. Build a branched harmonic current circuit in the SimInTech project window (Figure 3.11) using blocks from the **EC – Statics v.2.0** library as electrical elements.

Select the following blocks from the **EC – Statics v.2.0** library:

- the **Voltage source** block from the **Sources** tab;
- three **RLC-circuit** blocks from the **Electrical elements** tab;
- the **Zero potential** block from the **Topologies Elements** tab;
- the **Outline Parameters** block  from the **Topology Elements** tab.

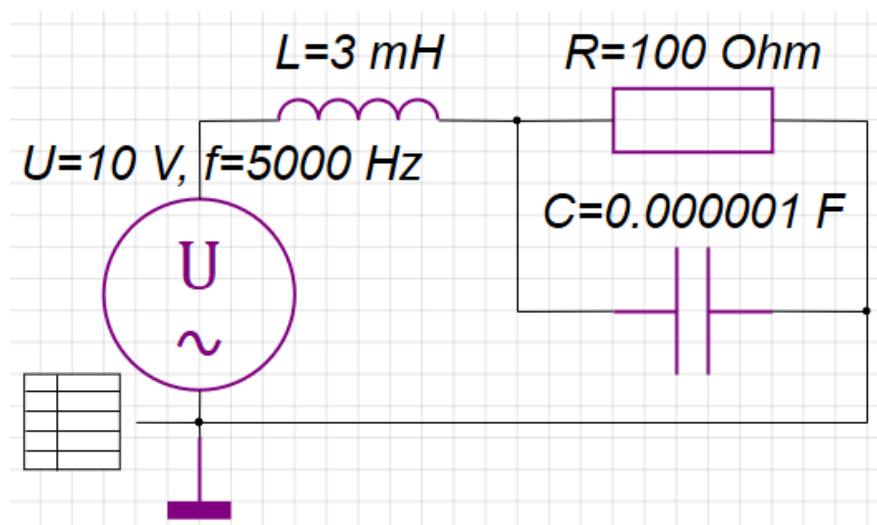
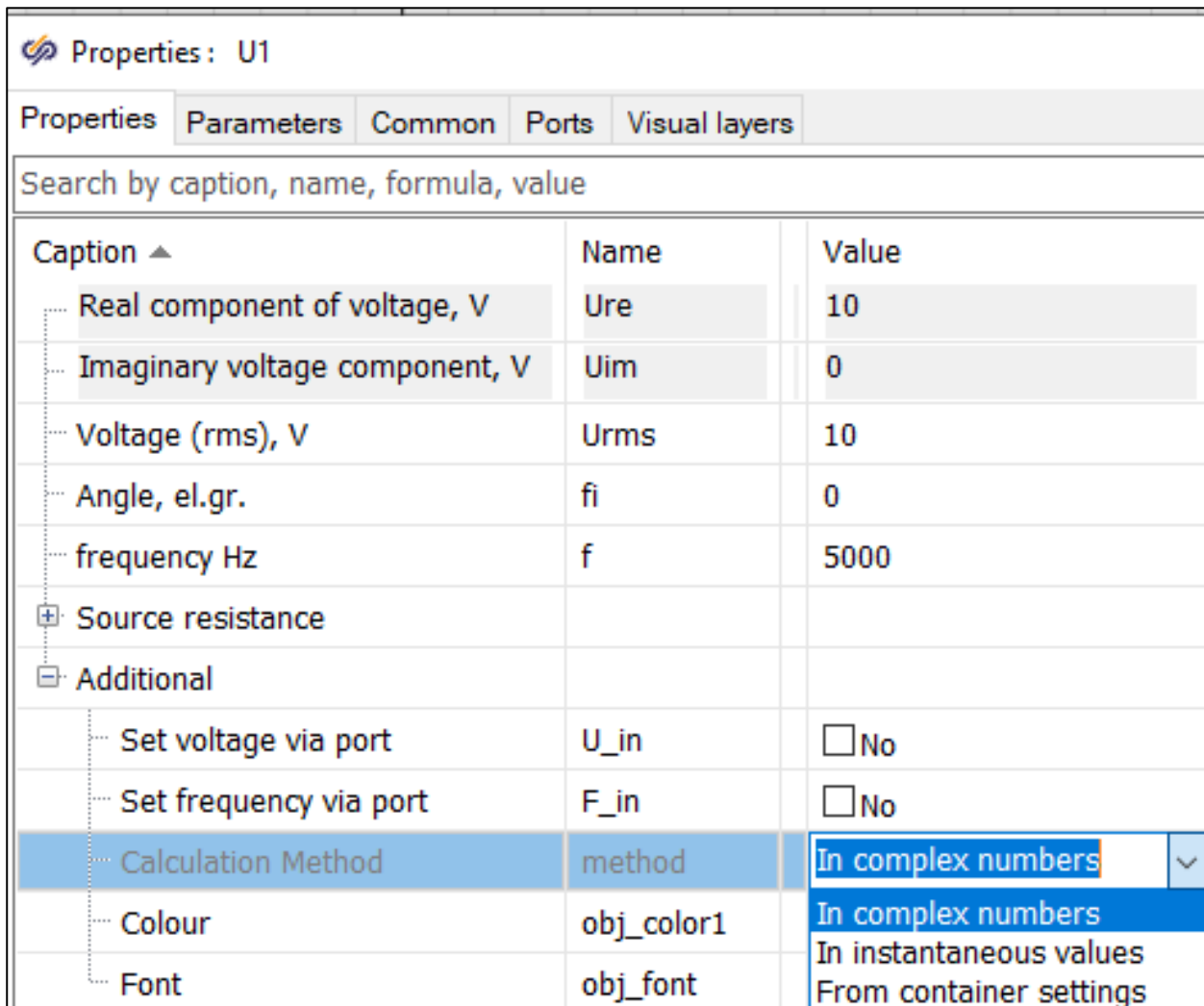


Figure 3.11

14. Use the **Properties** window to set for the **Voltage source** block properties shown in the Figure 3.12.



Properties : U1

Properties Parameters Common Ports Visual layers

Search by caption, name, formula, value

Caption ▲	Name	Value
Real component of voltage, V	Ure	10
Imaginary voltage component, V	Uim	0
Voltage (rms), V	Urms	10
Angle, el.gr.	fi	0
frequency Hz	f	5000
Source resistance		
Additional		
Set voltage via port	U_in	☐ No
Set frequency via port	F_in	☐ No
Calculation Method	method	In complex numbers ▼
Colour	obj_color1	In complex numbers
Font	obj_font	In instantaneous values
		From container settings

Figure 3.12

Please note that the **Calculation Method** in the Figure 3.12 is **In complex numbers**.

Use the **Properties** window to set the parameter values shown in the Figure 3.13 for the **Outline Parameters** block.

Properties : CPROPS6		
Properties	Parameters	Common
Ports	Visual layers	
Search by caption, name, formula, value		
Caption	Name	Value ▲
Parameters set from ports	in_ports	
Number of skipped integration steps	KofStep	0
Relative error	ConturRelError	0.0001
Absolute error	ConturAbsError	0.001
Integration step for calculating the initial state	h_InSt	1E-14
Maximum number of contour iterations	iter_count	5
The method of calculating the initial state	init_mode	Asked n.u.;
The method of integration	int_method	Implicit trapezoid;
Basic calculation method	top_method	In complex numbers;
Execute on preliminary steps	execallsteps	☐ No
Enable debug printing	enable_debug_print	☐ No
Calculation on a good step	F_calcGoodStep	☐ No
Solution debounce	F_Y2	☐ No
Analysis of the need for LU decomposition	F_LU	☐ No
Step selection based on iteration error	F_SolveErr	☐ No
Integration step control	fstepcontrol	☒ Yes
All circuits are autonomous	F_Avtonom	☒ Yes
Take errors from general settings	F_ErrorCommon	☒ Yes
Nominal frequency of the circuit, Hz	f_nom	[5000]

Figure 3.13

Use the **Properties** window to set the parameter values shown in Figure 3.11 for **RLC-circuit** blocks.

15. Click the **Simulation properties** button  of the schematic project window and set:

- The **Final calculation time** is **0.1 s**;
- The **Maximum (Minimum) step** is equal to **0.001 s**.

Leave the other settings as default.

Run the scheme for simulation.

The simulation results in the form of currents in the branches can be viewed in the corresponding **Properties** window for the **RLC-circuit** blocks on the **Parameters** tab (see Figures 3.14–3.16) where:

– **Ток комплексный (rms), A** – displays the value in complex form when calculated in complex numbers; the parameter value is zero when calculated in instantaneous values;

– **Ток (rms/inst), A** – displays the effective value when calculated in complex numbers; instantaneous value when calculated in instantaneous values.

Properties : Z1		
Properties	Parameters	Common Ports Visual layers
Search by caption, name, formula, value		
Caption	Name	Value ▲
Ток комплексный (rms), A	I_out	0.02112682031-0.1500589063i
Ток (rms/inst), A	I_rms	0.151538833

Figure 3.14

Properties : Z2		
Properties	Parameters	Common Ports Visual layers
Search by caption, name, formula, value		
Caption	Name	Value ▲
Ток комплексный (rms), A	I_out	-0.04142718729-0.01991155904i
Ток (rms/inst), A	I_rms	0.04596392097

Figure 3.15

Properties : Z3		
Properties	Parameters	Common Ports Visual layers
Search by caption, name, formula, value		
Caption	Name	Value ▲
Ток комплексный (rms), A	I_out	0.0625540076-0.1301473473i
Ток (rms/inst), A	I_rms	0.1443999164

Figure 3.16

16. Use the **Properties** window for the **RLC-circuit** blocks to set the values of the parameters C, L shown in the Figure 3.11, and the resistance value in ohms equal to 100 minus the option number.

Report

Insert the simulated circuit into the laboratory report, indicating the parameter values for your option, as well as the simulation results in the form of branch currents.

Control questions

1. Which libraries in the SimInTech system are used to simulate instantaneous values of currents and voltages of AC electrical circuits?
2. What properties should be set for the **Voltage source** block when modeling AC electrical circuits?
3. For what reasons is the **Final calculation time** selected in the **Project Parameters** window, as well as the **Minimum (Maximum)** the integration step?
4. How are the simulation results displayed on the communication lines?
5. How is the required time interval of the schedule displayed in the **Graph of time** blocks?
6. Which blocks are used to obtain the RMS value and the average value over a period for a harmonic signal, and where are they located?
7. How can you display the instantaneous power value in the **Graph of time** blocks?
8. How can you set the initial values of the voltage across the capacitance and current through the inductance?
9. How can you simulate an extensive harmonic current circuit to obtain simulation results in the form of a complex number?
10. What settings of the **Circuit Parameters** block should be set when modeling a harmonic current circuit?
11. On which tab of the **Properties** window for the **RLC-circuit** blocks can you view the simulation results and in what form?

REFERENCES

1. Website SimInTech.ru. URL: <http://simintech.ru/>
2. SimInTech Help System. URL: https://help.simintech.ru/#o_simintech/browsers.html
3. SimInTech technology modeling, development and creation of ACS. URL: <https://miem.hse.ru/data/2018/10/29/1141996123/SimInTech.pdf>

CONTENTS

Introduction	3
Laboratory work № 1. Fundamentals of SimInTech Modeling	5
Laboratory work № 2. Simulation of Direct Current Circuits in the SimInTech System	22
Laboratory work № 3. Simulation of Alternate Current Circuits in the SimInTech System	39
References	51

Учебное издание

**Регеда Владимир Викторович,
Регеда Ольга Николаевна**

**Моделирование электрических цепей
в среде SimInTech =
SimInTech simulation of electrical circuits**

На английском языке

Редактор *М. Н. Кузнецова*
Технический редактор *Ю. В. Анурова*
Компьютерная верстка *Ю. В. Ануровой*

Подписано в печать 16.10.2025.
Формат 60×84¹/₁₆. Усл. печ. л. 3,02.
Заказ № 532. Тираж 15.

Издательство ПГУ
440026, г. Пенза, ул. Красная, 40
Тел.: (8412) 66-60-49, 66-67-77; e-mail: iic@pnzgu.ru

