IT'14
ŽABLJAK

XIX
naučno - stručni skup

INFORMACIONE TEHNOLOGIJE

SADAŠNJOST I BUDUĆNOST

Urednik
Božo Krstajić

IT'14

INFORMACIONE TEHNOLOGIJE

- SADAŠNJOST I BUDUĆNOST -

Urednik

Božo Krstajić

Zbornik radova sa XIX naučno - stručnog skupa
INFORMACIONE TEHNOLOGIJE - sadašnjost i budućnost
održanog na Žabljaku od 24. do 28. februara 2014. godine

Zbornik radova
INFORMACIONE TEHNOLOGIJE - sadašnjost i budućnost 2014

Glavni urednik
Prof.dr Božo Krstajić

Izdavač
Univerzitet Crne Gore
Elektrotehnički fakultet
Džordža Vašingtona bb., Podgorica
www.etf.ucg.ac.me

Tehnička obrada
Aleksandra Radulović
Centar Informacionog Sistema
Univerziteta Crne Gore

Tiraž
150

Podgorica 2014.

Organizator

Elektrotehnički fakultet Univerziteta Crne Gore

Skup su podržali:

* Ministarstvo za informaciono društvo i telekomunikacije
∗ Pošta Crne Gore
∗ Agencija za elektronske komunikacije i poštansku djelatnost
* Čikom

Programski odbor

Dr Novak Jauković, Elektrotehnički fakultet, Podgorica
Akademik Dr Ljubiša Stanković, CANU
Dr Zdravko Uskoković, Elektrotehnički fakultet, Podgorica
Dr Vujica Lazović, Ekonomski fakultet, Podgorica
Dr Branko Kovačević, Elektrotehnički fakultet, Beograd
Dr Milorad Božić, Elektrotehnički fakultet, Banja Luka
Dr Miroslav Bojović, Elektrotehnički fakultet, Beograd
Dr Zoran Jovanović, Elektrotehnički fakultet, Beograd
Dr Božidar Krstajić, Elektrotehnički fakultet, I. Sarajevo
Dr Milica Pejanović-Đurišić, Elektrotehnički fakultet, Podgorica
Dr Dejan Popović, Elektrotehnički fakultet, Beograd
Dr Božo Krstajić, Elektrotehnički fakultet, Podgorica
Dr Milovan Radulović, Elektrotehnički fakultet, Podgorica
Dr Budimir Lutovac, Elektrotehnički fakultet, Podgorica
Dr Igor Radusinović, Elektrotehnički fakultet, Podgorica
Dr Igor Đurović, Elektrotehnički fakultet, Podgorica
Dr Miloš Daković, Elektrotehnički fakultet, Podgorica
Dr Milutin Radonjić, Elektrotehnički fakultet, Podgorica
Dr Ramo Šendelj, Fakultet za Informacione Tehnologije, Podgorica
Dr Stevan Šćepanović, Prirodno-matematički fakultet, Podgorica
Dr Sašo Gelev, Elektrotehnički fakultet, Radoviš

Organizacioni odbor

Dr Božo Krstajić, Elektrotehnički fakultet, Podgorica
Dr Milovan Radulović, Elektrotehnički fakultet, Podgorica
Dr Ana Jovanović, Elektrotehnički fakultet, Podgorica
Dr Saša Mujović, Elektrotehnički fakultet, Podgorica
MSc Žarko Zečević, Elektrotehnički fakultet, Podgorica
Vladan Tabaš, dipl.ing., Čikom

Sekretarijat

Aleksandra Radulović, CIS Univerzitet Crne Gore

SADRŽAJ

LTspice IV KAO EDUKATIVNO SREDSTVO U NASTAVI ANALIZE ELEKTRICNIH KOLA
LTspice IV AS EDUCATIONAL TOOL FOR TEACHING ELECTRICAL CIRCUIT ANALYSIS

Biljana Chitkusheva Dimitrovska, Maja Kukusheva, Vlatko Chingoski, *Faculty of Electrical Engineering, University „Goce Delcev“ – Stip*

Sadržaj: *U ovom radu predlaže se uvođenje osnovnih koncepata računarskog program za analizu LTspice IV, kao neprocenjiv alat u nastavi studenata predmeta analize električnih kola. Za studente, korisćenje ovog softverskog alata pruža mogućnost da rade samostalne analize ili da provere dobivene resultate svojih analiza i time budu sigurni da su te analize napravljene na ispravan način. Kompjuterske simulacije u LTspice IV su vrlo značajne u procesu učenja omogućavajući studentu da eksperimentiše sa promenama i da vidi efekte koje te promene imaju na electricna kola. Posmatrajući rezultate svoje kompjuterske simulacije, brojčano i osobito grafički, student može da stekne intuitivno shvatanje ponašanja električnih kola bez zastrašivanja od upotrebe apstraktne matematičke analize.*

Abstract: *This paper proposes the introduction of the basic concepts of computer analysis program LTspice IV, as an invaluable tool in teaching students subjects of electrical circuits analysis. For students, the use of this software tool provides the ability to do independent analysis or verification of the obtained results within their analysis and thus be sure that these analyzes are made in the right way. Computer simulations using LTspice IV are very important in the learning process allowing students to experiment with changes to see the effects that these changes have on an electrical circuits. Observing the results of their computer simulations, numerically and graphically in particular, the students can gain an intuitive understanding of the behavior electric circuits without intimidation from the use of abstract mathematical analysis.*

1. INTRODUCTION

Having understood the fundamental laws of circuit theory (***Ohm's law*** and ***Kirchhoff's laws***), we are now prepared to apply these laws to develop two techniques for circuit analysis: ***nodal analysis***, which is based on a systematic application of Kirchhoff's Current Law (**KCL**), and ***branch analysis*** which is based od Kirchhoff's Voltage Law(KVL).

With these two techniques, one can analyze any linear circuit by obtaining a set of simultaneous equations that are simultanously solved to obtain the required values of currents and/or voltages. One method of solving simultaneous equations involves Cramer's rule or method of replacing variables, which yealds the unknown variables. Another method of solving simultaneous equations is to use **LTspice**, or any other circuit simulation software, simulate the electric circuit and solve it for its parameters.

LTspice IV is a free SPICE (***Simulation Program for Integrated Circuit Emphasis***) simulator with schematic capture from *Linear Technology Ltd*. Linear Techology (**LT**) is one of the industry leaders in production of varius analog and digital integrated circuits [2]. LT provides a complete set of SPICE models mmostly using LT components. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltages and current sources, a few types of dependent or independent sources, transmission lines, switches, and several devices: including diodes, BJTs, JFETs, MOSFETs. Circuits with large number (*almost infinite*) number of components can be simulated. One can think of LTspice as a nodal network solver that outputs all node voltages and branch currents, or simulate their changes for prescribed time period. One node must be named "0" (*the ground node* or *the reference node*) and this node is used as a reference node for calculations of all other node voltages.

2. SHEMATIC ENTRY IN LTSPICE IV

Simulation of an electric circuit using LTspice IV can be done in two steps:

- drawing (*editing*) or entering circuit using schematic capture, and
- defining the desired type of simulation and execute the simulation or performing circuit analysis.

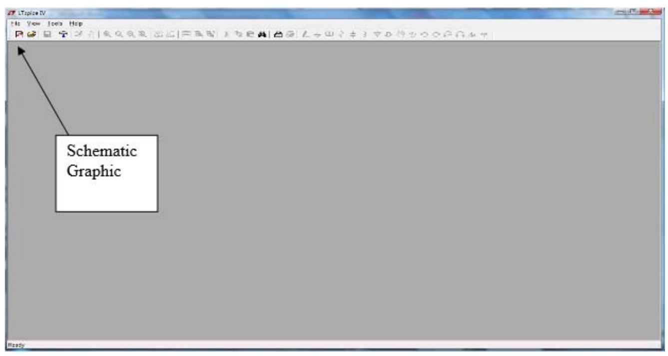


Fig. 1: Starting window in LTspice.

Drawing a circuit is easier when the grid mode is active. To activate the grid mode, click on ***View*** and select command ***Show Grid***.

There are many buttons on the main menu for selection of basic circuit elements such as resistors, capacitors, inductors, diodes, wires, as well as a ground node (GND). The Ltspice IV's ***Main toolbar*** is shown on Fig. 2.

3. CIRCUIT ANALYSIS WITH LTSPICE IV

As was already mentioned, the LTspice IV simulation software can be used as circuit analysis program which can easily determine all voltage and current values for any previously set circuit configuration and given numerical values for circuit components. We would demonstrate this, with several simple problems. These problems are usually solved by the students during their study of basic electrical circuit theory or any other basic or advance study subject that involves electric circuit analysis.

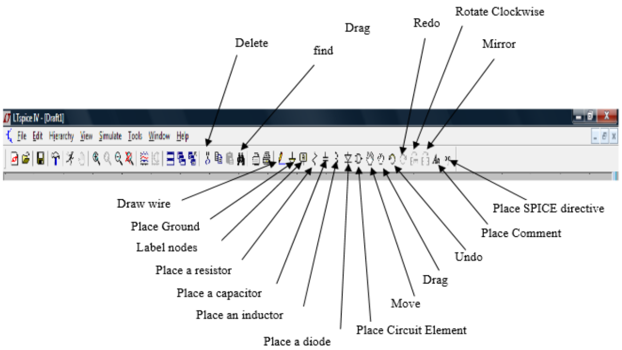


Fig. 2: Main toolbar of schematic editor for LTspice IV.

Example1: *Find the branch currents I_1, I_2, and I_3 for a simple electric circuit show on Fig. 3.*

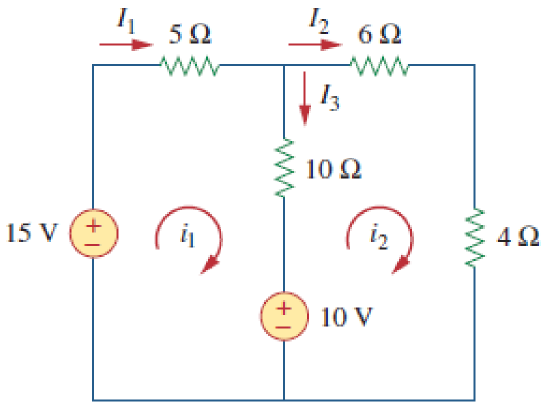


Fig. 3: Simple electric circuit as student task.

Usually students use some solving techniques such as contour currents or unknown node voltrage analysis to find the unknown circuit paramtars, voltages and/or currents.

Here, we could use the ***Method of contour currents*** to solve the above circuit for numerical values of all node voltages and branch currents. Consiqunetly, one can obtained the following general system of equations:

$$R_{11}\, i_1 \pm R_{12}\, i_2 = E_I$$
$$\pm R_{21}\, i_1 + R_{22}\, i_2 = E_{II}$$

Where, i_1 and i_2 are the unknown contour currents, R_{11}, R_{12}, R_{21} and R_{22}, are the equivalent contour resistances (*self and mutual contour resistance*), and E_I and E_{II} are the equvivalent voltage source per contour. If we replace the numerical values given as in Fig.3, we could obtain the followings:

$$15\, i_1 - 10\, i_2 = 15$$
$$-10\, i_1 + 20\, i_2 = 10$$

Using the metod of substitution to solve these system of two unknown currents with two equiations, we get the values for the contour currents $i_1 = i_2 = 1A$. Thus, for the values of the unknown branch current we have:

$$I_1 = i_1 = 1\ A, \quad I_2 = i_2 = 1\ A, \quad I_3 = 0\ A$$

Next, we could show, how easy one can solve this electric circuit and get the numerical values and, if it is requested the graphic representation of all branch currents using the electric circuit sover and simulator, LTspice IV.

First, we open the LTspice program and start drawing (*editing*) the electrical circuit shown in Fig. 4. The analized electrical circuit contains two voltage generators and four resistors. After entering all circuit elements by clicking on the designated icons and setting their numerical values as parameters (*click on each component and set the desired parameters and values*), we place the circuit elements inside the program's workspace and finally connect the entire electric circuit with wires (*wiring*). The obtained electric circuit with its parameters shown inside the LTspice IV program's workspace is shown on Fig. 4.

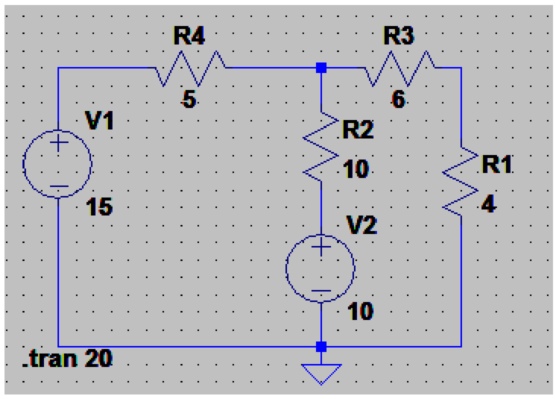


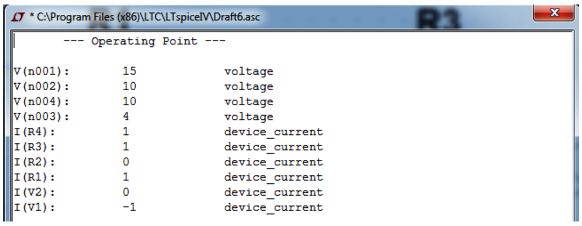
Fig. 4: Electric circuit built in LTspice IV.

At this point, our circuit is fully defined and we could continue to the next step, to solve or simultate the circuit behavior. The only relevant simulation for this electric circuit is a simple DC analysis, known in LTspice IV as ***DC operating point***. This analysis we set by clicking on the icon ***Simulate>Edit Simulation Cmd*** from the top toolbar and select **DC op pnt**. This is called an **LTspice IV directive**.

To run a circuit (*execute the solver*), we can select from the file menu icon ***Simulate*** and ***Run,*** or we can simply click on the ***Run*** button from the main toolbar. Next, we could see a new window appearing, containing the simulation results, or in our case solution of the electric circuit in a simple table manner as shown in Fig. 5. As can be seen, the solution is exactly the same as in previous solution method using contour current method as a circuit solution method.

We must have in mind, that this was only a simple example, only two currents to solve for. However, if we have evan a little more complex electric circuit, e.g. four or five contour currents, than the advantages of using LTspice IV for obtaining the unknown currents instead of solving by hand the same circuit by means of any other solution method are obviously large.

Next, we would present the advantages of utilizing LTspice IV softwere in the eduction of young students by means of another example very often used in the classwork – the solution of an electric circuit using the **Thevenin's theorem**. Thevenin's theorem provides a solution technique by witch one, usually called *the fixed part of circuit*, is replaced by an equivalent circuit [1]. Our major concern using this technique is to find the ***Thevenin equivalent voltage*** $\mathbf{V_{TH}}$ and the ***Thevenin equivalent resistance*** $\mathbf{R_{TH}}$.



```
* C:\Program Files (x86)\LTC\LTspiceIV\Draft6.asc
       --- Operating Point ---

V(n001):      15          voltage
V(n002):      10          voltage
V(n004):      10          voltage
V(n003):      4           voltage
I(R4):        1           device_current
I(R3):        1           device_current
I(R2):        0           device_current
I(R1):        1           device_current
I(V2):        0           device_current
I(V1):        -1          device_current
```

Fig. 5: Simulation (solution) results file.

__**Example 2:**__ *Find the equivalent Thevenin circuit of the circuit shown in Fig. 6 to the left of the terminals* ***a-b****. Then, find the current through the resistance* $R_L = 6,16\ \Omega$.

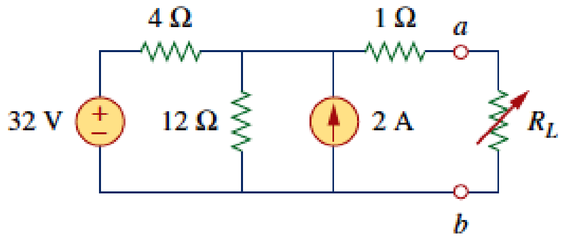


Fig. 6: Electric circuit for the Example 2.

We find equivalent resistance R_{TH} by turning off the 32V Voltage source (*replacing it with the short circuit*), and the Current source of 2A (*replacing it with an open circuit*).

Thus,

$$R_{TH} = 4 \,||\, 12 + \frac{4 \cdot 12}{4+12} + 1 = 4\Omega$$

To find the equivalent voltage V_{TH}, we consider the electric circuit given on Fig. 7.

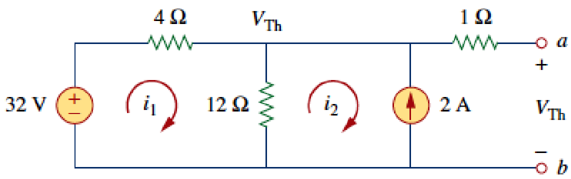


Fig. 7: Modified circuit to obtain the equivalent voltage V_{Th}.

Applying mesh analysis for the two loops, we obtain:

$$-32 + 4\, i_1 + 12\, (i_1 - i_2) = 0.$$

Solving for i_1, we get $i_1 = 0,5\ A$. Thus,

$$V_{Th} = 12\ (0,5 + 2,0) = 30\ V.$$

Respectively, the Thevenin equivalent circuit for the original electric circuit given on Fig. 6, get the configuration shown on Fig. 8.

The current through the R_L is:

$$I_L = \frac{V_{Th}}{R_{Th} + R_L} = \frac{30}{4 + R_L}$$

When $R_L = 6\Omega$, $I_L = \frac{30}{10} = 3A$. When $R_L = 16\Omega$, $I_L = 1,5A$.

If we built this electrical circuit in LTspice IV, the current through the R_L is determining in a very simple way without any calculations and complex methods.

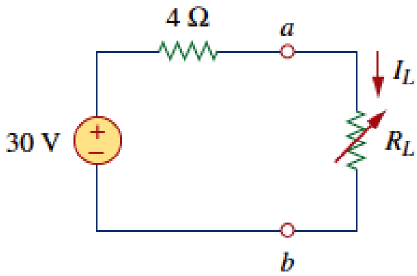


Fig. 8: Thevenin equivalent circuit for Example 2.

Using the same methodology described for the previous Example 1, one can draw the electric circuit od Example 2 in LTspice IV (see Fig. 9 – above).

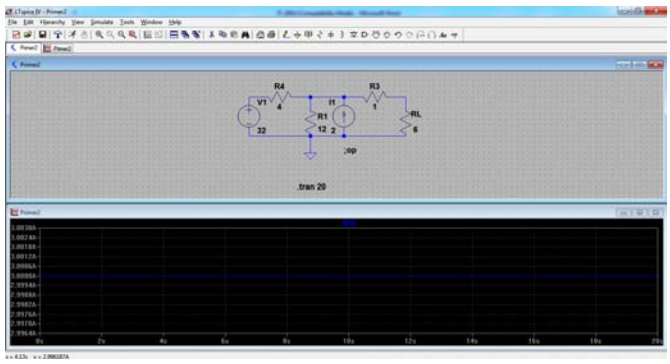
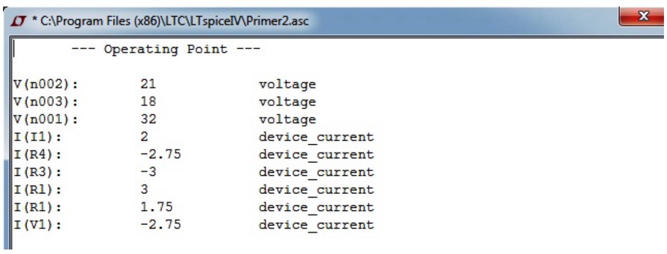


Fig. 9: Electric circuit of Example 2, built in LTspice IV (*above*), and the solutiuon for the unknown current (*below*).

First, we could obtain the values of the current through resistor $R_L = 6\Omega$. For this value of R_L, we simulate the electric circuit shown in Fig. 6. After simulation, we get the results in output text file (.op file).



```
* C:\Program Files (x86)\LTC\LTspiceIV\Primer2.asc
       --- Operating Point ---

V(n002):      21          voltage
V(n003):      18          voltage
V(n001):      32          voltage
I(I1):        2           device_current
I(R4):        -2.75       device_current
I(R3):        -3          device_current
I(RL):        3           device_current
I(R1):        1.75        device_current
I(V1):        -2.75       device_current
```

Fig. 10: Obtained results for $R_L = 6\Omega$.

LTspice has a user-friendly interface to show the results of simulations in two ways. One way is to display the results in a text file (.op file) (see Fig. 10), and the other way is using graphic display, as shown in Fig. 9 – below.

The third example of the advantages that simulation program LTspice IV provides for students and teachers in solving the subjects of electric circuit analysis, concerns about time-varying electric circuit analysis.

Example 3: *Determine the time variation of the $V_0(t)$ in the circuit given on Fig. 11.*

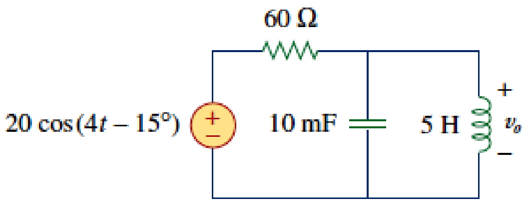


Fig. 11: Time-varying electric circuit.

To do the circuit analysis in frequency domain, we must first transformthe time domain circuit, and its parameters into the phasor domain equvivalent circuit. This transformation produces:

$$Vs(t) = 20\cos(4t-15^0) \Rightarrow V_s = 20\angle -15^0,\ \omega = 4$$

$Z_1=60\Omega$ is a impedance of the 60 Ω resistor and $Z_2=\frac{1}{j\omega C} \| j\omega L =$ j100 Ω is a impedance of the parallel combinationof the 10 mF capacitor and the 5 H inductor. Using the voltage-division principle, one can obtain:

$$V_0=\frac{Z_2}{Z_1+Z_2}V_s = 17.15\angle 15.96^0 \text{ V}$$

Now, we can convert this back to the time domain and obtain the final time-varying voltage value:

$$V_0(t) = 17.15\cos(4t + 15.96^0)\text{V}.$$

Let us now, solve the same problem using LTspice IV. First, we built the electrical circuit shown in Fig. 1. The electrical circuit contains one sinusoidal voltage source, resistor, capacitor and inductor, all of them with their respective numerical values. For the passive components (R, L, C), point at the component and do a right click. A dialogue box will appear and we could type the numerical value of the component. For the voltage supply, we do a right click on it and then click on the button ***Advanced***. This enables us to set a variety of different voltage source waveforms. The one that we need for this analysis is on the left side of the dialogue box, as shown in Fig. 12.

Next, we do simulation of the circuit by clicking ***Simulate>Edit Simulation Cmd*** and type the ***Stop time***. Finally, as a result we get the time changes of the unknown voltage $V_0(t)$ shown on Fig. 13. Later, various analysis can be perform of the obtained results such as reading of the frequency, max values, min values, attenuation factor, time constant, etc.

4. CONCLUSION

In this paper, we present how electric simulation program LTspice IV can be utilized to support teachers in eduction of students on the subject of electric circuit analysis.

We showed that LTspice is easy to use, very flexible and highly fast and accurate softwere for teaching electric circuit analysis. Three simple examples are given in order to provide step-by-step insight how easy and resourcefull could be this program for eduction of future electrical engineers. In addition to its solver capabilities, the program provides advanced graphic routines to visually grasp the nature of the process that happens inside simple, and especially very complex time and frequency varying electric circuits.

In such cases, use of LTspice simulation software allows for modeling of electric circuits and is a invaluable analysis tool. LTspicve simulation software engages the user by integrating them into the learning experience. These kinds of interactions actively engage learners to analyze, synthesize, organize, and evaluate content and result in learners constructing their own knowledge.

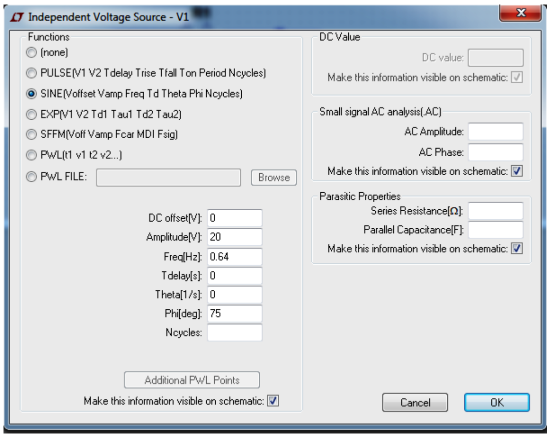


Fig. 12: Input screen for sinusoidal voltage source.

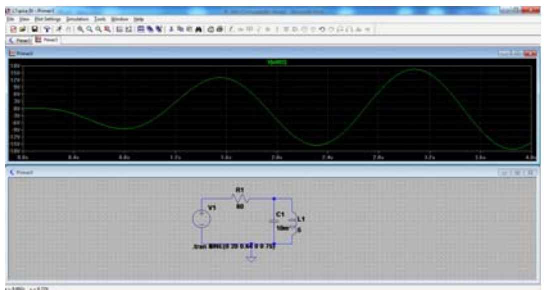


Fig. 13: Obtained time-varying function of voltage $V_0(t)$.

REFERENCES

[1] Charles K. Alexander, Matthew N. O. Sadiku, ***"Fundamentals of Electrical Circuits,"*** *5th ed., The McGraw - Hill Companies, New York, 2013.*

[2] ***LTspice IV tutorial***, www.linear.com/ltspice

[3] Влатко Чингоски, ***„Основи на CAD/CAM во електротехника,"*** *University „Goce Delcev", Stip, 2013.*

[4] Susan A. Riedel and James W. Nilsson, ***"Introduction to Pspice for electric circuits,"*** *Pearson Eduction, Inc. 2007.*

CIP - Каталогизација у публикацији
Национална библиотека Црне Горе, Цетиње

ISBN 978-86-85775-15-4
COBISS.CG-ID 24749840

ISBN 978-86-85775-15-4

9 788685 775154 >