CONCEPTION OF THE COURSE PROJECT IN THE THEORETICAL FOUNDATIONS OF ELECTRICAL ENGINEERING

Snejana Dimitrova Terzieva, Simeon Stoianov Vladov, Valeri Markov Mladenov

Department of Theoretical Electrical Engineering, Technical University - Sofia, 8 Kl. Ohridski bul., 1000 Sofia, Bulgaria, phone: +359 2 9652394, e-mail: ster19@tu-sofia.bg, svladov@tu-sofia.bg, valerim@tu-sofia.bg

The importance of course projects in basic sciences and general engineering subjects is recognized by many university teachers, including the authors of this paper. In order to improve the quality of training the students of the Faculty of Electronic Engineering and Technology in the subject of the Theoretical Foundations of Electrical Engineering a change in the syllabus has been proposed in the paper, namely: removing the course project from third to the fourth semester. In this way the course project may include additionally important and interesting topics taught in both semesters.

Keywords: circuit theory, course project, PSpice, syllabus.

1. INTRODUCTION

Teaching the Theoretical Foundations of Electrical Engineering aims basic concepts, laws, methods and theorems for circuits and fields analysis to be given in systematized form [1, 2]. This subject is taught to the Faculty of Electronic Engineering and Technology (FEET) second year students. The third semester syllabus includes a course project using the Student version of ORCAD PSpice.

Traditionally, the course project consists of several parts that are topically connected with the lectures and seminars [3]. In this way, however, the range of problems considered is too limited. The limitation is due to the number of topics included in lectures of the third semester. In practice the topics could be the following:

1. DC analysis in a complex electric circuit;
2. AC analysis in a complex electric circuit, mutual inductance included;
3. Resonance phenomena.

In such a manner, many interesting from practical point of view topics, such as Periodic Nonsine Steady-State, General Two-Ports Networks, Transients, are not included in the course project.

In order to improve the quality of education a suitable change in the syllabus for the FEET students could be very useful, namely: removing the course project from third to the forth semester. Thus, the significance and the usefulness of the course project will be much increased [4].

2. THE COURSE PROJECT NOW

2.1 DC analysis

Students get accustomed to the PSpice together with the first assignment of the course project, which is a DC analysis of a complex electric circuit as the one of

Fig.1. The assignment usually includes finding all branch currents through General Nodal Analysis (GNA) or Loop Analysis (LA) and then, for the sake of verification the results, consumed and delivered powers are compared. All the students are given a circuit with the same graph but different values of resistors and voltage sources.

To analytically solve the circuit of Fig. 1 the following set of equations based on the LA are written:

$$\begin{cases} (R_1+R_4)i_1 - R_4 i_6 = E_1 + E_2 \\ (R_3 + R_5 + R_7)i_3 + R_5 i_6 = E_2 + E_3 \\ -R_4 i_1 + R_5 i_3 + (R_4 + R_5 + R_6)i_6 = E_6. \end{cases}$$

Then, applying the GNA results in the two equations for the nodal voltages

$$\begin{cases} (1/R_1 + 1/R_4 + 1/R_6)V_1 - V_2/R_4 - V_3/R_6 = -E_1/R_1 + E_6/R_6 \\ -V_1/R_6 - V_2/R_5 + (1/(R_3+R_7) + 1/R_5 + 1/R_6)V_3 = -E_3/R_3 - E_6/R_6, \end{cases}$$

where $V_2 = E_2$.

Next, the **Shematics** is used to introduce the given circuit to a PSpice simulation. Numerical results for the branch currents and for the nodal voltages are directly portrayed on the scheme (Fig. 2) after **V** and **I** keys activated. The direction of a given current can be determined after clicking on its numerical value.

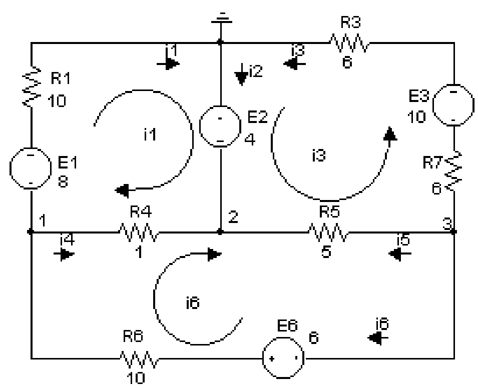

Fig. 1

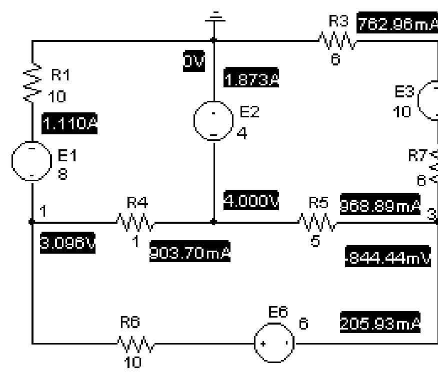

Fig. 2

The output file after simulation contains a lot of information that is explained in detail to the students. The consumed power is given at the end of the output file `TOTAL POWER DISSIPATION   2.52E+01  WATTS`. The results for the currents and nodal voltages obtained analytically are not given since they completely coincide with the ones obtained through the PSpice.

This example is very useful for practicing the two basic methods for circuit analysis, generalized Ohm's law, Joule's law on one hand and one type of PSpice analysis (**Bias Point Detail**) on the other hand.

2.2 AC analysis for one particular frequency

The second assignment is an AC analysis of an electric circuit with one sinusoidal voltage source (Fig. 3) with frequency $\omega = 1000 rad/s$. The unknowns are current phasors and complex power. The latter is necessary for confirmation the equality of delivered and consumed power.

To analytically find the solution it is necessary for the students to know how to simplify a given circuit and how to find currents in a two-node circuit [1]. Using the phasor concept the following numerical results are obtained:

$$\dot{I}_1 = 4.196 + j10.959A; \quad \dot{I}_2 = \dot{I}_5 = 1.414 + j11.42A; \quad \dot{I}_3 = -1.488 + j7.149A;$$

$$\dot{I}_4 = 2.902 + j4.271A; \quad \dot{I}_E = 5.61 + j11.42A.$$

Delivered complex power is $\dot{S}_{gen} = \dot{E}\overset{*}{I} = 2237.89 + j561VA$. This result is sufficient to be compared with the PSpice one.

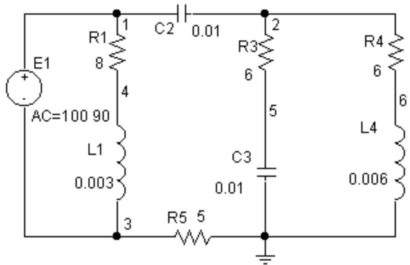

Fig. 3

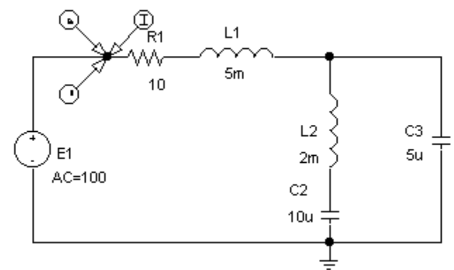

Fig. 4

In order to analyze this circuit for one frequency an independent voltage source (**VSRC**-type) should be introduced and the **AC Sweep** analysis for one particular frequency ($f = 159.155Hz$) should be implemented. Built-in functions **R()** and **IMG()** to find a real and an imaginary part of a complex number are used. The results concerning the currents $\dot{I}_1$ and $\dot{I}_3$ are: `R(I(R1)) = 4.10959` `IMG(I(R1)) = 10.9589`; `R(I(R3)) = -1.48805` `IMG(I(R3)) = 7.14877`.

The equality of delivered and consumed power is checked up with a creation of specific "macroses" for the average power P and the reactive power Q according to the formulae $P = UI\cos\varphi, Q = UI\sin\varphi$. The average power "macros", for example, is `actp(a, b) = abs(a)*abs(b)*cos((p(a)-p(b))/180*pi)` . Using both "macroses" the delivered power is found: `actp((V(3)-V(1)),I(E1)) = 2237.89` `reactp((V(3)-V(1)), I(E1)) = 552.387`.

This example shows quite clear the one-to-one correspondence between sine quantities and their phasors. Students learn how to use phasors and how to turn them to their instantaneous values, how to find average and reactive power and finally how to use one new analysis in PSpice environment (**AC Sweep** for one frequency).

2.3 Resonance phenomena

The third assignment is for analyzing resonance phenomena in a particular circuit (Fig. 4) and visualizing some frequency responses. The frequency of the voltage source varies in a stated interval. For this example $e(t) = 100\sqrt{2}\sin\omega t\, V$. In order to find out series and parallel resonance frequencies one should determine the equivalent impedance (Z_e) and the equivalent admittance (Y_e) of the circuit and nullify their imaginary parts.

The equivalent impedance Z_e is $Z_e = R_1 + j\dfrac{L_1L_2C_2C_3\omega^4 - (L_1C_2 + L_1C_3 + L_2C_2)\omega^2 + 1}{\omega(L_2C_2C_3\omega^2 - C_2 - C_3)}$, and the two series resonance frequencies are $f_{r1} = 2128.16Hz$, $f_{r2} = 532.63Hz$. The equivalent admittance Y_e is the reciprocal of Z_e and we find the parallel resonance frequency to be $f_{r3} = 1949.23Hz$.

PSpice is used for visualizing the amplitude-frequency response (AFR) and phase-frequency response (PFR) of the current. Visualized is also the imaginary part of the current in the frequency interval of *10 Hz* to *10 kHz*. The AC Sweep analysis is used with 100 points for a decade for better image of the graphs. The results are presented in Fig. 5.

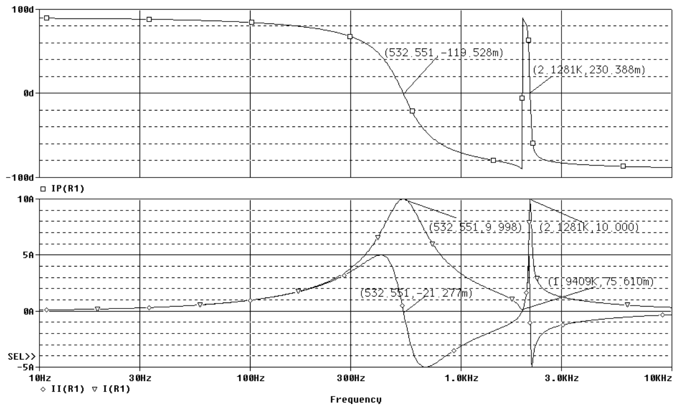

Fig. 5

3. The Change in the Course Project – Useful and Necessary

3.1 Frequency responses of a two-port

It is proposed a new assignment that requires determination both AFR $K_U(\omega)$ and PFR $\psi_U(\omega)$ of a first-order electric circuit analytically and with the PSpice. The RL-circuit of Fig. 6 is considered as an example. Two cases are studied: first the circuit is open and second – the circuit is loaded with a capacitor ($C_2 = 50\mu F$). Another variant is to consider an RC-circuit – open and loaded with an inductance.

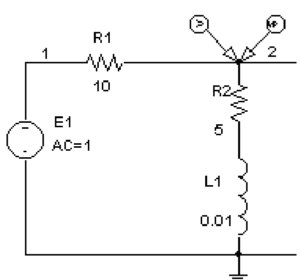

Fig. 6

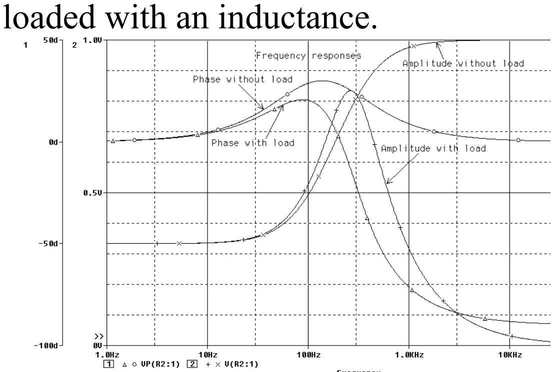

Fig. 7

In order to analytically find the frequency responses one should find the expression for an open-circuit complex voltage gain $\dot{K}_U(\omega) = \frac{\dot{U}_2}{\dot{U}_1} = \frac{R_2 + j\omega L_2}{R_1 + R_2 + j\omega L_2}$.

The AFR $K_U(\omega)$ is the $\mathrm{mod}\,\dot{K}_U(\omega)$ and PFR $\psi_U(\omega)$ is the $\arg \dot{K}_U(\omega)$. The complex voltage gain under load is $\dot{K}_U(\omega) = \frac{R_2 + j\omega L_2}{R_1 + R_2 - \omega^2 R_1 L_2 C_2 + j\omega(L_2 + R_1 R_2 C_2)}$.

An independent voltage VSRC and **AC Sweep** analysis are introduced in order to fulfill a simulation with the PSpice. The results for both cases are given in Fig. 7.

3.2 Characteristic parameters of a two-port

Following the example considered above it is naturally to find the input impedance of two cascaded RL-circuits (Fig. 8). The input impedance is found for $\omega = 1000 rad/s$. A possible problem could be finding the propagation coefficient $\dot{\gamma} = \alpha + j\beta$.

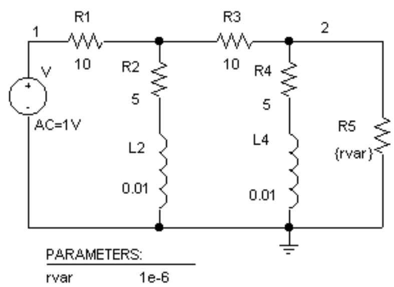

Fig. 8

```
A = V(1)@2/V(2)@2
B = V(1)@1/I(R5)@1
C = -I(V)@2/V(2)@2
DE = -I(V)@1/I(R5)@1
pi = 3.14159265
Rc1 = R(Zc1)
Rc2 = R(Zc2)
Xc1 = IMG(Zc1)
Xc2 = IMG(Zc2)
Zc1 = SQRT(A*B/(C*DE))
Zc2 = SQRT(DE*B/(C*A))
```

Fig. 9

```
Rin1 = R(Zin1)
Xin1 = IMG(Zin1)
Zin1 = -V(1)/I(V)
```

Fig. 11

In accordance with the two-port theory [1] transmission parameters A, B, C and D are found. Since the two-ports are identical it follows that

$$A_{1-2} = 1.4 - j0.8; \quad B_{1-2} = 10\Omega; \quad C_{1-2} = 0.04 - j0.08S; \quad D_{1-2} = 1.$$

The equivalent A-matrix for the combined circuit is given by the matrix product

$$[A] = [A_1][A_2] = \begin{bmatrix} 1.72 - j3.04 & 24 - j8 \\ 0.032 - j0.224 & 1.4 - j0.8 \end{bmatrix}.$$

A two-port is said to be match loaded if the output impedance Z_{load2} is equal to its characteristic impedance Z_{c2}, i.e. $Z_{load2} = Z_{c2} = \sqrt{\frac{DB}{CA}} = 4.891 + j5.2623\Omega$ and its input impedance Z_{in1} equals to Z_{c1}, i.e. $Z_{in1} = Z_{c1} = \sqrt{\frac{AB}{CD}} = 14.937 + j4.3783\Omega$.

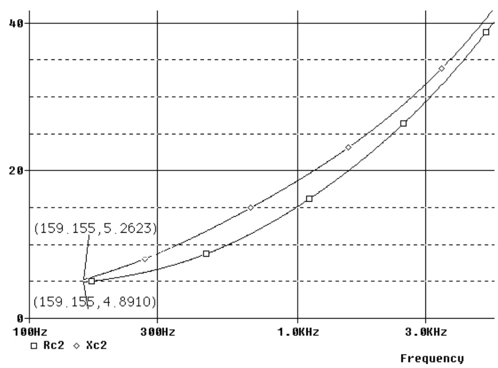

Fig. 10

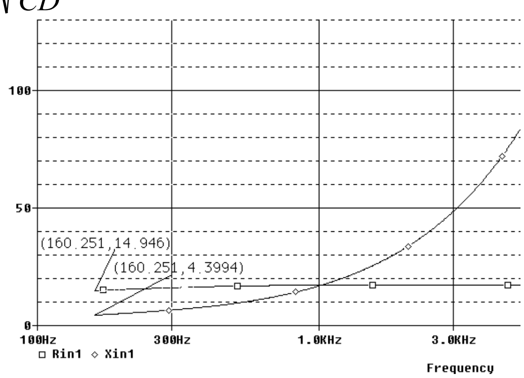

Fig. 12

Transmission parameters of the equivalent two-port are found with PSpice using the open-circuit output voltage and the short-circuit output current [3]. The open-circuit output voltage is found when a load resistor with a very high value is connected to the output ($R_{5\infty} = 100M\Omega$) and for the short-circuit output current – the value is very small: $R_{50} = 1\mu\Omega$ (Fig. 8). In order to find Z_{c1} and Z_{c2} the following "macroses" are created (Fig. 9). The graphs of the real R_{c2} and the imaginary part X_{c2}

of the impedance Z_{c2} are given in Fig. 10. It is clear that the Z_{c2} has resistive-inductive character that corresponds to an inductor with an inductance $L_5 = 5.2623mH$.

To check equality $Z_{in1} = Z_{c1}$ a resistor with resistance $R_5 = R_{c2} = 4.891\Omega$ and an inductance of $L_5 = L_{c2} = 5.2623mH$ are connected in series with the two-port output. The "macroses" in Fig.11 are defined. The graphs of R_{in1} and X_{in1} are given in Fig.12.

3.3 Transients

A basic theme considered in a project is transients in a first-order circuit activated with different voltages. As an example the input voltage of the circuit of Fig. 13 is constant and the switch closes at $t = 0$. A non-homogenous equation holds for the circuit [2] and its solution for currents $i_{R2}(t)$ and $i_{R3}(t)$ is

$$i_{R2}(t) = 4(1 - e^{-833t})\,A, \qquad i_{R3}(t) = (4 + 2.67e^{-833t})\,A.$$

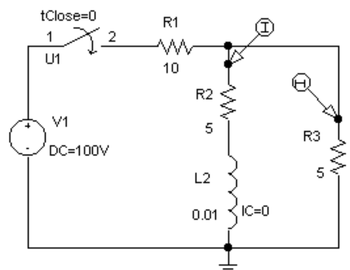

Fig. 13

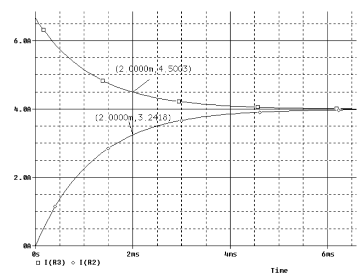

Fig. 14

The results of the PSpice simulation are given in Fig. 14. Comparison between analytical and numerical solution is made at $t = 2ms$. The analytical results are $i_{R2} = 3.244A, i_{R3} = 4.5046A$ and the PSpice results are: $i_{R2} = 3.2418A, i_{R3} = 4.5003A$. It is seen that there is a negligible difference.

4. CONCLUSION

The importance of the course project for the students may be generalized as follows:

- the students get accustomed to software that is used during next semesters;
- the teaching topics become easily understandable through visualization the processes in electric circuits;
- a given problem is solved for a very short time;
- receiving family characteristics through variation the parameters of electric circuits.

5. REFERENCES

[1] K. G. Brandiski, J. D. Georgiev, V. M. Mladenov and R. D. Stancheva, *Textbook on Theory of Electrical Engineering – Part I*, IK KING, ISBN 954-9518-28-0, Sofia, 2004 (in Bulgarian).

[2] K. G. Brandiski, J. D. Georgiev, V. M. Mladenov and R. D. Stancheva, *Textbook on Theory of Electrical Engineering – PartII*, IK KING, ISBN 954-9518-29-9, Sofia, 2005 (in Bulgarian).

[3] K. G. Brandiski, V. M. Mladenov and K. P. Stanchev, *Manual fors Solving Problems on Theoretical Electrical Engineering using PSpice*, CIELA, Sofia, 2002 (in Bulgarian).

[4] Sn. D. Terzieva, S. S. Vladov, G. A. Toshev and V. M. Mladenov, *Significance of Course Project in Teaching Electrical Engineering Theory*, III-rd International Conference CHER21"05, Sozopol, Bulgaria, June 2005.