1st World Conference on Technology and Engineering Education
Kraków, Poland, 14-17 September 2010

Numerical simulation of circuits as an aid to better understanding of network theory by undergraduate students

E. Kurgan, S.A. Mitkowski, A.M. Dąbrowski & A. Porębska

AGH University of Science and Technology
Kraków, Poland

ABSTRACT: The influence of numerical simulation of circuits on students' ability to understand fundamental laws and analysis methods in network theory is presented in this paper. In using a program with a graphical interface, a student can model on the computer screen a given electrical example and then calculate the exact results. He/she can perform the calculations manually and compare the results with the numerical outcome. Time-dependent graphs also can be observed. Students can gain the ability to introduce a schematic diagram of passive and active circuits into computer memory using a graphical pre-processor. Any schematic errors can be corrected manually in real time. Students can develop the understanding that the graphical transformation of a given circuit does not influence the circuit behaviour for a given excitation. Both direct current and sinusoidal steady state solutions can be simulated. Output can be presented graphically and in the form of tables. The examples have to be simple enough for the student to calculate them by hand.

INTRODUCTION

Many areas of electrical engineering, such as high power systems, electric machines, control systems, power electronics, instrumentation and others are based on theorems contained in electric circuit theory. Therefore, a basic course in electric circuit theory is one of the most important factors for an electrical engineering student, and is always a good starting point for an undergraduate student in electrical engineering teaching. Circuit theory is also important to students who specialise in other areas of the polytechnic sciences because electrical circuits are good models for the study of any systems described by ordinary differential equations. These include pneumatic and hydraulic systems, mechanical structures, thermal systems - and even economic systems.

In general electrical engineering, transferring different kinds of energy from one place to another is a common requirement. This process can be accomplished by interconnecting different electrical devices. Such devices are referred to as electric circuits, and each component of the electrical circuit is known as an element. There are passive and active elements. Passive and active electric circuits are used in many electrical systems to carry out different tasks. Our objective in this article is not the study of various uses and applications of circuits. Rather our major interest is the fundamental analysis of circuits taking into account the fundamental circuit theorems. The *analysis of a circuit* is how it responds to a given input. That is, calculation of all or some currents and voltages when some excitations as independent current and voltage sources are given. The basic concepts that students must learn are charge, current, voltage, circuit elements, power and energy.

A course on circuit analysis for undergraduate students that are run in non-electrical engineering faculties is perhaps the first exposure students have to do with electrical engineering at all. Problem solving of electrical circuits is an essential part of the learning process. Solving as many problems as the time schedule allows, give the student an opportunity to understand practical application of circuit theory in real world problems. The laboratory exercises are carried out by two students. One of them introduces the circuit schematic into computer memory and conducts adequate numerical calculations and the second student solves the same problem manually. Next, both results are compared. Electric circuit theory is a fundamental theory widely used in all kinds of electrical engineering. For undergraduate non-electrical engineering students the concepts and theorems of great importance are the Ohm theorem, Kirchhoff's first and second laws, Thevenin's and Norton's theorems, mesh analysis, nodal analysis, operational amplifier circuits, sinusoidal steady state analysis and phasor method and three-phase circuits. All these methods and notions can be easily explained and consolidated with the PSpice (Simulation Program with Integrated Circuit Emphasis) circuit simulation program.

The main purpose of this paper is to present a teaching method based on the interaction between a student and a computer program. The interrelation of numerical and manual calculations is emphasised. Students can understand that efficient usage of the computer program requires a good understanding of main theorems and analysis methods in electrical circuit theory. Without a good understanding of the way system equations are built and then solved, it is impossible to correctly interpret computational results. Many teachers do not address this very important point in educating electrical engineering students.

PSPICE PROGRAM

PSpice helps the student to simulate the electrical circuit scheme before he/she can build it. This lets the student decide if changes are needed, without having to build any hardware [4]. PSpice also helps the student to check the completed design. It allows comparison of real world measurements and numerical computational results to reveal any abnormal behaviour. It allows the user to check that the designed circuit will work correctly after assembly, in the real world, and if it has the properties expected. Therefore, PSpice is a simulation program on which the student creates test circuits and makes measurements. It is important to understand that PSpice will not design the circuit for the user. The most practical way to check, if any electrical circuit behaves correctly, is to build it. Circuits assembled from components in a laboratory may not give correct test results. The electrical circuit must be either physically built, which is expensive and time consuming or simulated using a computer analysis program. The most important features of the PSpice program are:

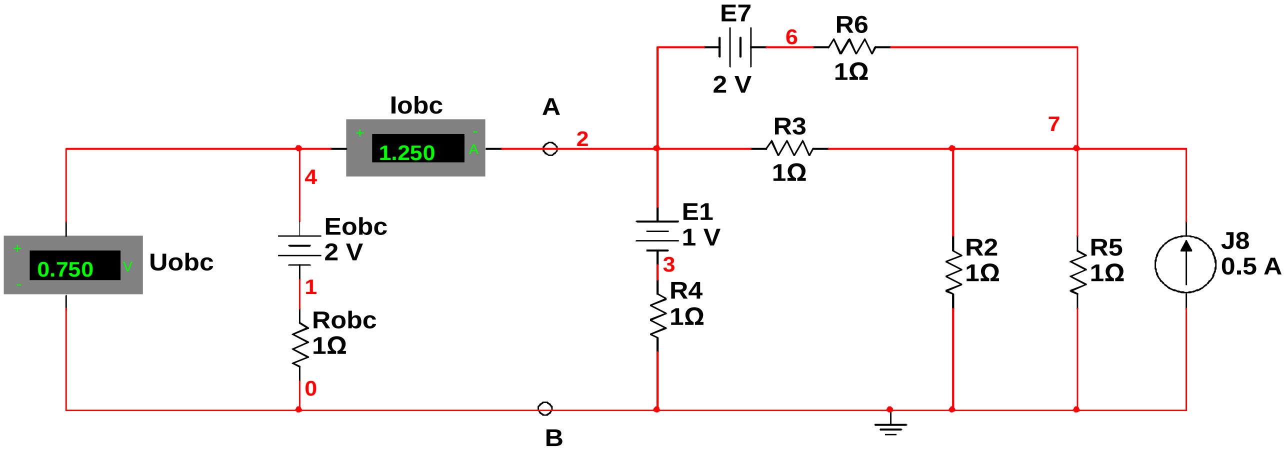
- Most PSpice concepts are represented visually in the schematic;
- It uses a wizard to set PSpice simulation defaults to match your circuit;
- It defines, names and saves unlimited numbers of analyses, each with a corresponding graph;
- It has an indexed library of Spice models and sub-circuits;
- The sub-circuit schematic symbols auto-adjust the number of pins;
- The opamp and generic sub-circuit schematic symbols adjust the number of pins to match the number of nodes in the user-selected sub-circuit so that users can enter names for the pins which are then saved in the library;
- There is no need to keep track of circuit node numbers when graphing;
- The Program Help tells how to perform tasks;
- A notes section in the schematic and one in each analysis allows users to document their work;
- It has Noise, AC and DC Sensitivity analysis;
- It has FFT and large signal harmonic Distortion analysis (Fourier Transform based);
- It allows export of tabular data from a graph/table to file for import into another program.

The circuit theory program mostly has two types of simulation: DC (direct current) and AC (alternating current). PSpice is a simulation program that models the behaviour of a circuit containing analogue devices. Used with *Capture* properties for design entry, PSpice is a software-based circuit breadboard that can be used to test and refine a design before having to touch a piece of hardware. PSpice can perform:

- DC, AC, and transient analyses, for testing the response of a circuit to different inputs.
- Parametric, Monte Carlo, and sensitivity/worst-case analyses for seeing circuit behaviour.

NORTON'S THEOREM

The main goal of this simulation is numerical and analytical verification of Norton's theorem. The whole task is composed in four steps. In the first step the student has to calculate the equivalent circuit from nodes A-B (Figure 1).



$U_{obc} = 0.75V$ $\qquad$ $I_{obc} = 1.250A$

Figure 1: Circuit with two-terminal load on the left from nodes A-B.

The values of all elements should be relatively simple and have simple analytical calculations [1]. After the numerical solution, values of the voltage U_{obc} and current I_{obc} can be read from measuring instruments or from a table in *Simulate*

→ *Analyses* → *DC Operating Point* → *Simulate* sequence after transferring all variables to the window *Selected variables for analysis*.

In the second step, the student has to measure or calculate, on the basis of node potentials, the short-circuit current I0 of the two-terminal network on the right from points A-B when load E_{obc} and R_{obc} are removed from the circuit.

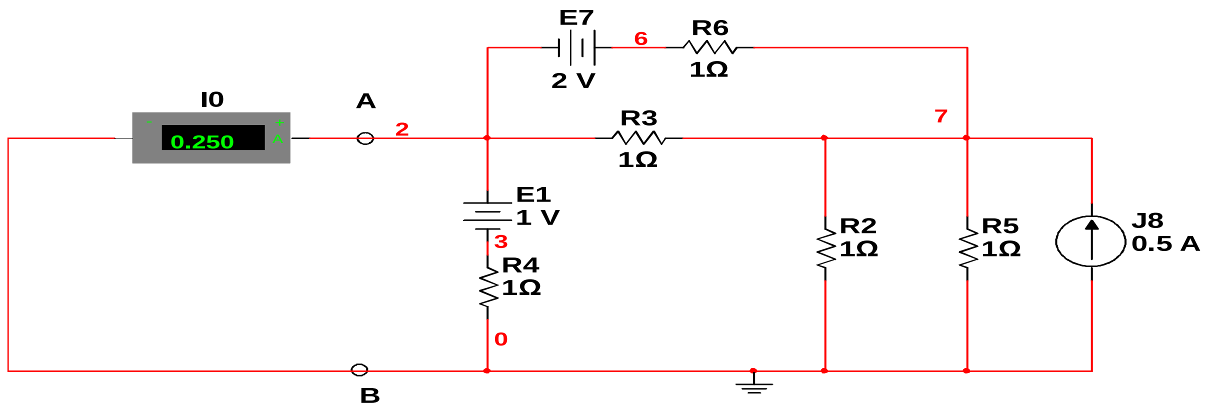


Figure 2: Circuit with removed load E_{obc} and R_{obc}. Measurement of short-circuit current.

In the third step, the student has to measure resistance seen from points A-B, as in Figure 3. Exciting voltage source is connected to the circuit input.

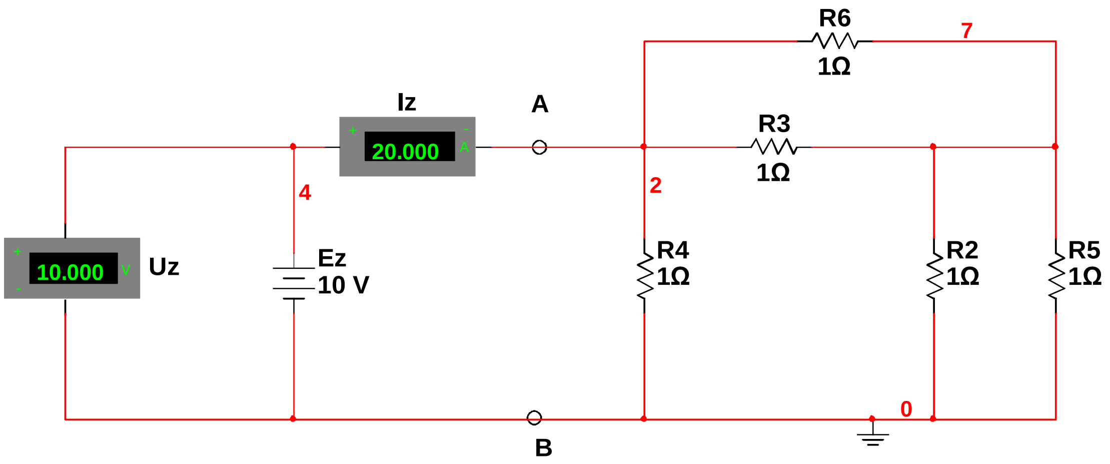


Figure 3: Measurement of equivalent resistance.

Resistance of the two-port A-B is given thus by:

$$R_0 = \frac{E_z}{I_z} = \frac{10.0}{20.0} = 0.5\Omega \qquad (1)$$

In the fourth step, the student has to substitute the two-port A-B with its equivalent circuit, as in Figure 4.

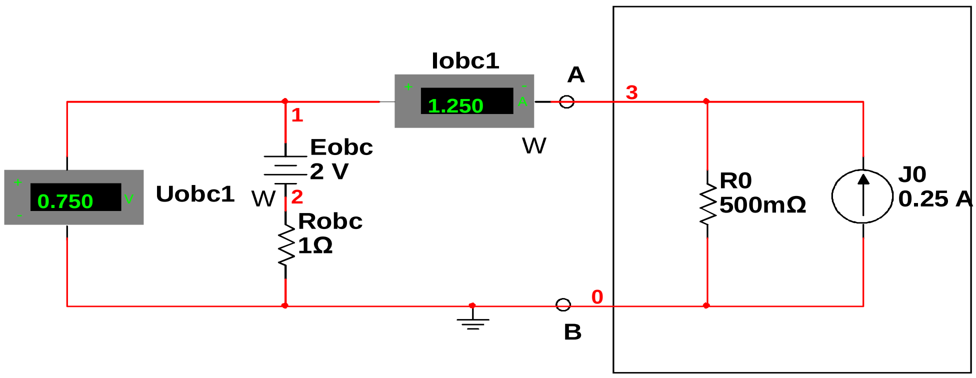


Figure 4: The Circuit on the right, from nodes A-B, is substituted with its equivalent circuit.

The student has to compare the measured results obtained at this point with results obtained in the circuit in Figure 1. The student should reach a conclusion from these measurements. Next, the student must solve this problem manually and compare the results with the numerical simulations.

OPERATIONAL AMPLIFIERS

Differential or difference amplifiers are fundamental elements in the design of most signal systems used in different areas of electrical engineering applications, as well as in general instrumentation. The operational amplifiers are so crucial because of their inherent ability to reject unwanted DC levels, interference, and noise voltages common to both inputs. An ideal operational amplifier responds only to the so-called difference-mode signal at its two inputs [2].

The first difficulty for students concerns one-stage voltage amplifiers. With these, students must learn how to analyse circuits with operational amplifiers. It is assumed that each operational amplifier is *ideal* in that it has infinite input and zero output resistances. Moreover, the difference in voltage between plus and minus terminals is equal to zero. Students have to assemble a circuit (as in Figure 5) and then calculate numerically and manually the input resistance and voltage gain.

The input resistance and voltage gain are calculated on the basis of measurement, as follows [3]:

$$R_{we} = \frac{U_{we}}{I_1} = \frac{1}{0.666} = \frac{3}{2}[\Omega] \tag{2}$$

The output can be calculated from measurements, as follows:

$$k_u = \frac{V_{wy}}{U_{we}} = \frac{-0.332}{1} = -0.332\left[\frac{V}{V}\right] \tag{3}$$

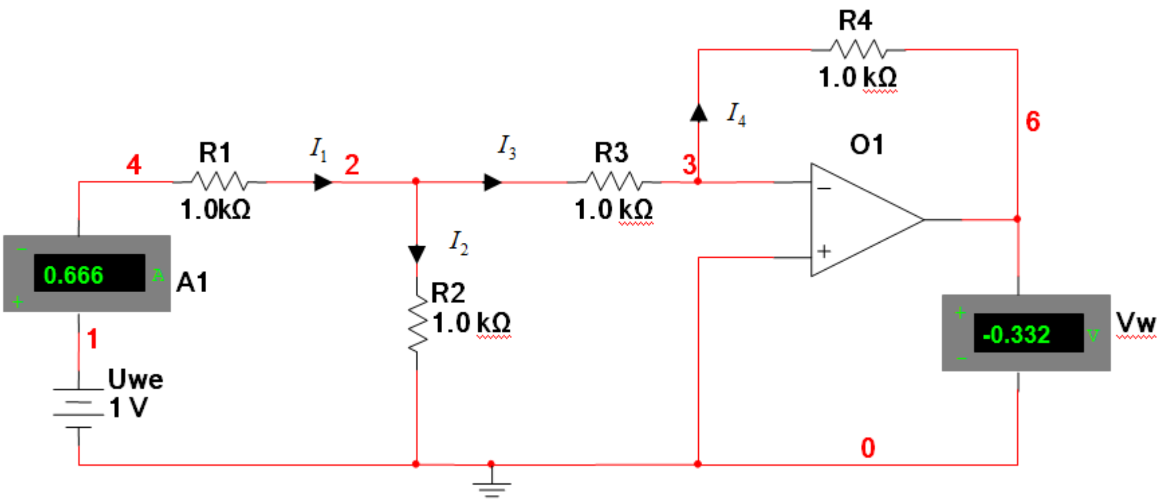


Figure 5: Voltage amplifier with one operational amplifier.

Next, the student has to calculate these two values manually. First he/she has to write the Kirchhoff voltage and current laws for all the fundamental loops:

$$U_{we} - R_1 I_1 - R_2 I_2 = 0 \tag{4}$$

$$R_2 I_2 - R_3 I_3 + U_{+-} = 0 \tag{5}$$

$$R_4 I_4 + V_{wy} = 0 \tag{6}$$

$$-I_1 + I_2 + I_3 = 0 \tag{7}$$

Taking into account that $U_{+-} = 0$ and $I_4 = I_3$, we get:

$$R_{we} = \frac{U_{we}}{I_1} = \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_2 + R_3} \tag{8}$$

and total amplification:

$$k_u = \frac{V_{wy}}{U_{we}} = -\frac{R_2 R_4}{R_1 R_2 + R_2 R_3 + R_3 R_1} \qquad (9)$$

After introducing adequate values for resistors, the same values are derived as from measurements.

THE SYMBOLIC METHOD

Numerical simulations also are useful when teaching solutions to networks in sinusoidal steady states. As an example, consider where students have to mount a circuit as follows below. As a computation task, students must calculate voltage values and the phase difference between voltages on resistor R_2 and capacitor C_1. The current flowing in the circuit is given by:

$$I = \frac{V_1}{R_1 + j\omega L_1 + R_2 + \frac{1}{j\omega C_1}} \qquad (10)$$

Where, $V_1 = 230 \exp(j0)$ [V] and $\omega = 2\pi 50 = 314$[rd/s]. Voltages on resistor R_2 and capacitor C_1 are given by:

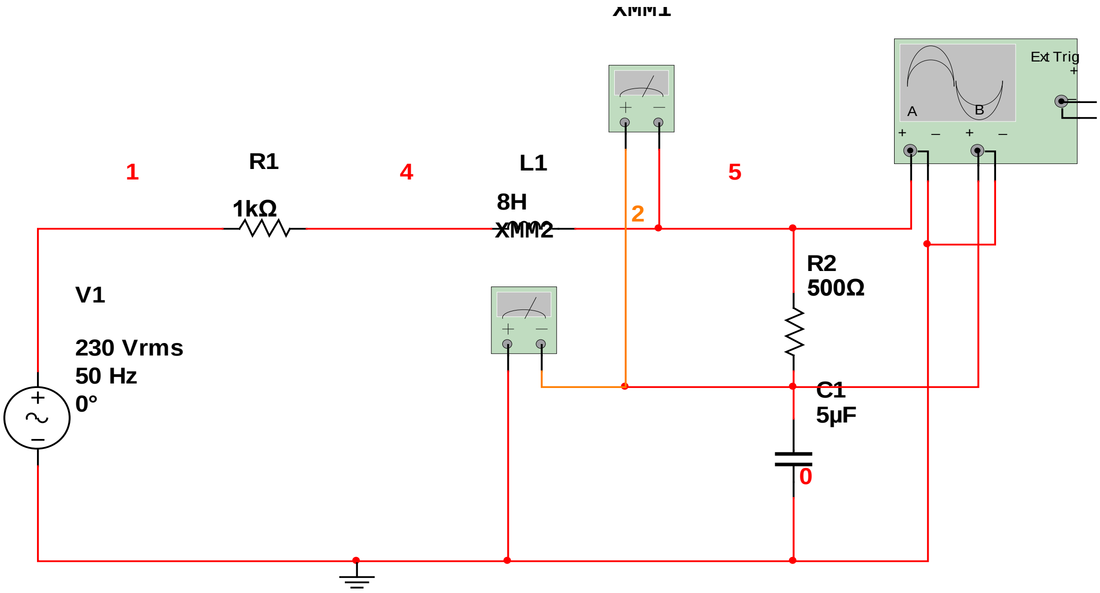


Figure 6: Example for teaching the symbolic method.

$$U_2 = \frac{R_2 V_1}{R_1 + R_2 + j\left(\omega L_1 - \frac{1}{\omega C_1}\right)} \qquad (11)$$

$$U_1 = \frac{\frac{1}{j\omega C_1} V_1}{R_1 + R_2 + j\left(\omega L_1 - \frac{1}{\omega C_1}\right)} = \frac{V_1}{\left(1 - \omega^2 C_1 L_1\right) + j\omega C_1 \left(R_1 + R_2\right)} \qquad (12)$$

The phase difference between voltages on series connection of resistor R_2 and capacitor C_1 and voltage on resistor R_2 alone can be evaluated as

$$\varphi = \arg[U_2] - \arg[U_1] = \arg[I] - \arg\left[IR_2 + I\frac{1}{j\omega C_1}\right] = \arg\left[R_2 + \frac{1}{j\omega C_1}\right] = -\arctan\left[\frac{1}{\omega R_2 C_1}\right] \qquad (13)$$

Set adequate value of the timebase *I* for channel *A* and channel *B* on the oscilloscope. First, set a *Trigger* on *Nor* and after several seconds, set *Trigger* on *Sing*. Set the vertical coloured pointers to zeros for both graphs. Phase difference is then given in this example by:

$$\varphi = \frac{T_2 - T_1}{T_{okres}} 360 = \frac{2.138\text{ms}}{20\text{ms}} 360^{\circ} = 38.48^{\circ} \qquad (14)$$

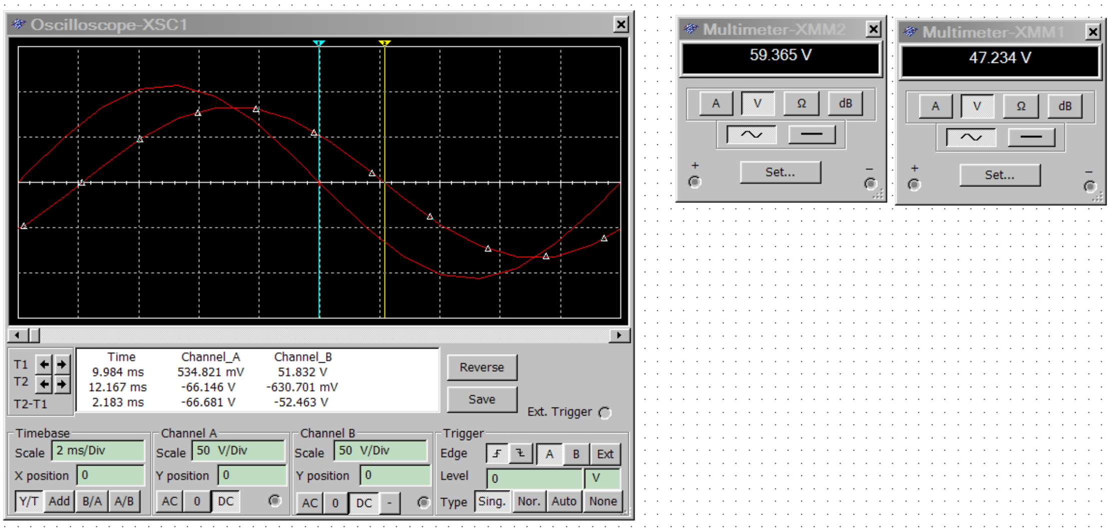


Figure 7: Phase measurements.

CIRCUITS WITH TRANSFORMERS

From a practical viewpoint, it is important to understand the functionality of one phase transformers. Students must connect the circuit as in figure below and measure indications of all instruments. The transformer can be taken from the library by the instruction *Place→Component…→Basic→TRANSFORMER→TS_IDEAL*.

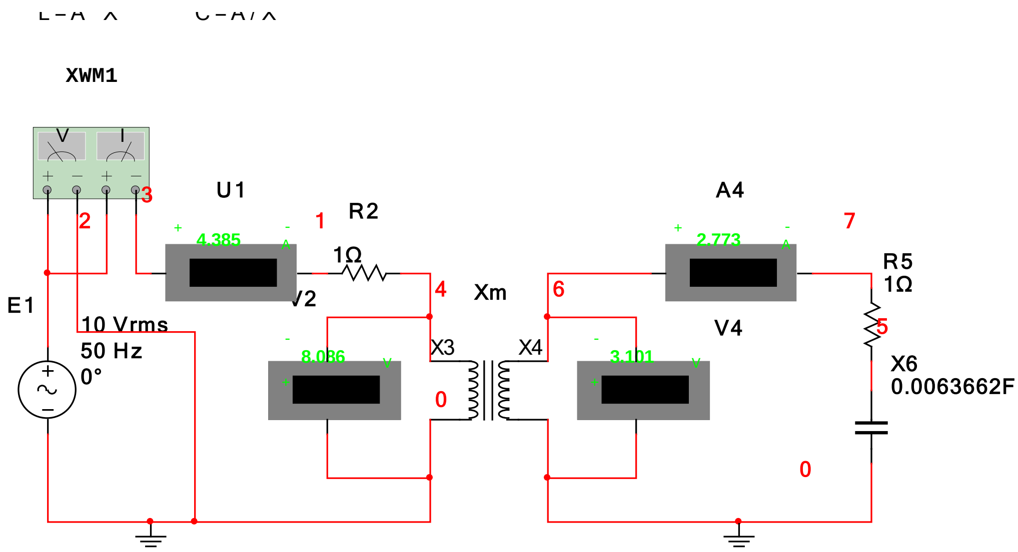


Figure 8: Circuit with single phase transformer.

In the transformer user field, the values of *Primary Coil Inductance*, *Secondary Coil Inductance* and mutual inductances must be introduced; compute these as:

$$M = k\sqrt{L_1 L_2} \tag{15}$$

where k is given coefficient coupling. Reactances of reactive elements have the values:

$$X_L = \omega L = 2\pi f L \qquad X_C = \frac{1}{\omega C} = \frac{1}{2\pi f\, C} \tag{16}$$

Because source frequency has value $f = 50$Hz therefore assuming, for example, inductance $L3 = 2$H, a large value can be reached for reactance $X3 = 628.31\Omega$, and this can be inconvenient in manual calculations. In order to get reactances convenient in manual calculations, it is better to use the following relations:

$$L = \frac{X_L}{2\pi f} = A X_L \qquad C = \frac{1}{2\pi f\ X_C} = \frac{A}{X_C} \quad \text{where} \quad A = \frac{1}{2\pi f} = 0.0031831 \tag{17}$$

For example, to get reactance for inductance $X_1 = 3\Omega$, the inductance should have the value $L_1 = A \cdot X_1 = 0.0031831 \cdot 3 = 0.0095493$ H. Similarly, capacitance gives $C_2 = A/X_2 = 0.0031831/0.4 = 0.00127324$ F for $X_2 = 0.4\ \Omega$. Students should manually calculate all measured values together with power and power factor, then compare with numerical results.

CONCLUSION

Numerical simulation of electric circuits with PSpice (or even Matlab) can be a valuable tool in circuit theory education. Students can gain expertise in numerical circuit simulation, which is of great practical value, as well as understanding how the program works. This is highly relevant, because without an accurate understanding of the equations the program uses to simulate problems, it is impossible properly to interpret computational results.

REFERENCES

1. O'Malley, J., *Basic circuit analysis*. NY: McGraw-Hill Co. (1992).
2. Grimbleby, J.B., *Computer-Aided Analysis and Design of Electronic Networks*. London: Pitman Publishing (1990).
3. Paul, C.R., *Analysis of linear circuits*. NY: McGraw-Hill Co. (1989).
4. Tuinenga, P.W., *Spice. A Guide to Circuit Simulation and Analysis Using Pspice*. New Jersey: Prentice Hall (1992).