

ELECTRONIC CIRCUIT SIMULATION

D. V. Riabchuk

Igor Sikorsky Kyiv Polytechnic Institute, Kyiv, Ukraine
dmytro.r.v@gmail.com

Electronic circuit simulation uses mathematical models to replicate the behavior of an actual electronic device or circuit. Simulation programs or simulation software allows for developing a circuit and discovering its properties without actually making a physical one. Today such softwares are widely used for different purposes. They took a perfect place in the education process. Many colleges and universities use this type of software for the teaching of electronics technician and electronics engineering programs. Electronic circuit simulator is a perfect choice for testing your knowledge in the real world. They help to understand the meaning of physical formulas.

As it was said before, simulating electronic circuits has a great amount of advantages. First of all, when talking about simulation software, should be mentioned the lack of need in realisation the device for more detailed research. It is also useful for students. With simulators students have an opportunity to explore the processes in the circuits by themselves.

At the same time, the simulators can twist the imaging of the real world as the mathematical modeling doesn't take into consideration a lot of factors that can affect the result in the physical world.

ADVANTAGES

- discovering the circuit without assembling the physical one;
- easy way to practise the skills after learning the theory;
- visualization of theoretical laws and formulas;

DISADVANTAGES

- receiving results from ideal environment;

Now it is time to move from overall view to more specific information. One of the most widely used simulator is a product by National Instruments Electronics Workbench Group called Multisim.

NI Multisim is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard.

Multisim was originally created by Electronics Workbench Group, which is now a division of National Instruments Corporation. Multisim can support the custom libraries as well as preinstalled libraries by developers. This fact makes Multisim one of the most flexible simulator on the market. Multisim is widely used in academia and industry for circuits education and electronic schematic design.

Multisim has a simple interface and that is one more reason why it is so common in different universities. In the majority of cases the user will not need any guides and manuals.

The program meets the user with the designing table (fig. 1), on which all of the circuits are usually built. The default view settings contains the Design Toolbox on the left, the spreadsheet at the bottom and status bar. The view can be easily changed at any time.

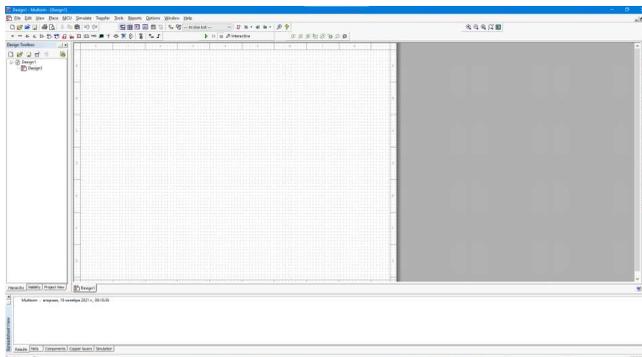
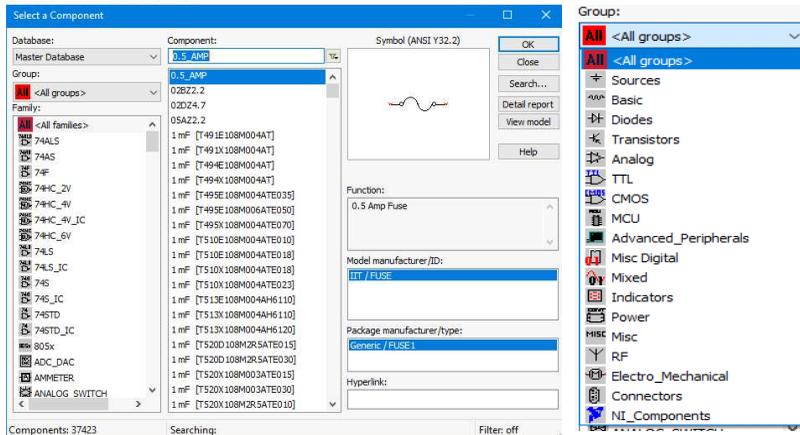


Figure 1. Starting screen of the program

The program has a wide choice of components. Any of them can be easily found as they are sorted by the types and libraries (fig. 2).

For example, in the group “Basic” can be found resistors, capacitors and inductors. Their types and properties can be chosen in the same window. More complicated components such as sources or transistors are located in separate groups. Custom components can also be found there.

How it was already mentioned before, the user assembles all of the circuits on the Design Table. The process of assembling is easy and intuitive. The leads of components are connected with wires. On any section of the wire the junction can be placed. The properties of the wires and junctions can be changed. Also on the Design Table can be added the text labels.



a)

b)

Figure 2. Component window and default groups

The values from the circuit can be received from the probes. There are some basic probes, which don't have specified properties and can be customized for different types of current (direct current or alternating current). On the other hand the user can pick

more complex probes. In such probes user can customize even the resistance of the probe. Such customizing will give more accurate results.

The Multisim also contains different types of scopes, generators and analyzers such as Function generator, Word generator, Bode Plotter, Oscilloscope, Spectrum analyzer, Distortion analyzer, Logic analyzer, etc. Some of them are shown in Fig. 3.

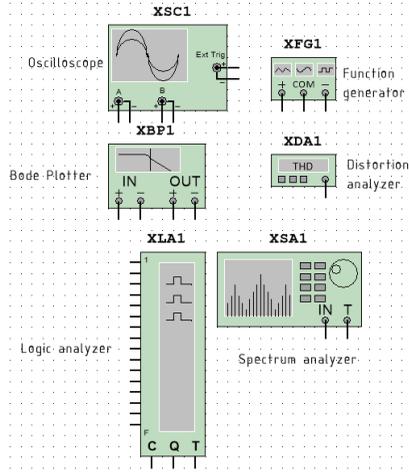
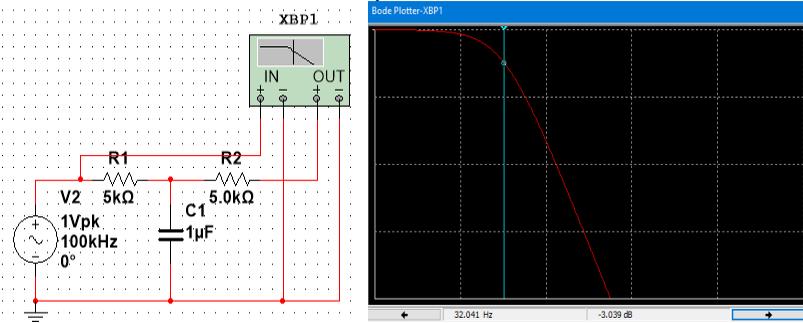


Figure 3. Examples of analyzers and generators

In Fig. 4 there is an example of a circuit which was made by me and some of my groupmates in university. We started using Multisim during learning the frequency filters in the second semester of our first year of education.



a)

b)

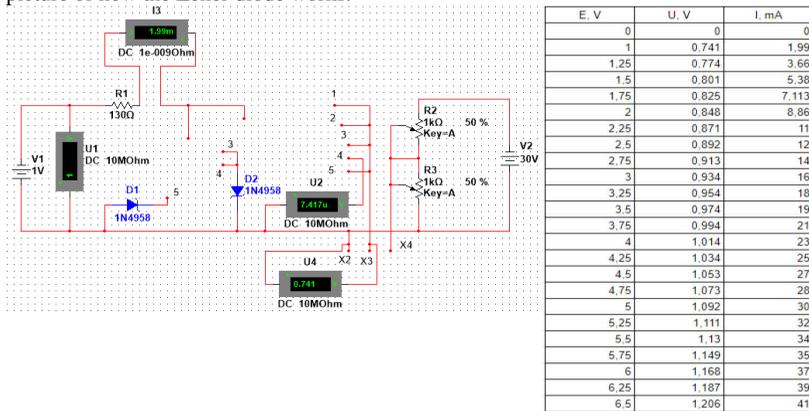
Figure 4. a - circuit of the low-pass filter; b - Frequency response of filter

The passive part of the circuit contains two resistors and capacitors. Combination such this is able to pass only signals with a frequency lower than a selected. That frequency is called cutoff frequency. The scheme attenuates signals with the frequencies higher than the cutoff frequency.

Cutoff frequency can be found in an analytical way (by using the formula) or by using the graph such as in the example. From the graph it is clear that the cutoff frequency is approximately 32 Hz.

Today me and my groupmates also work with Multisim. It is quite helpful during learning the components of circuits. We have already learned the properties of Zener diode as a task for the subject. The properties of component were firstly modulated and then tested in the laboratory.

In Fig 5. there are parts of my own work and some results. The value of voltage in column “U” is almost constant (Fig 5, b). After receiving results the user can form a full picture of how the Zener diode works.



a) b)
 Figure 5. a - general scheme for researching the Zener diode; b - some results from Multisim

And now it is time to conclude everything said before. It is clear which outlook has electronic circuit softwares. They have been already commonly used all around the world as a professional tool for technicians and engineers. And today such softwares is gaining more and more relevance because of the conditions of the world. Today, when students don't have an opportunity to visit colleges and universities, modulating software becomes a perfect solution.

Sources:

1. Моделирование электронных схем [Електронний ресурс] / Матеріал из Википедии — свободной энциклопедии – Режим доступу до ресурсу: https://en.wikipedia.org/wiki/Electronic_circuit_simulation
2. NI Multisim [Електронний ресурс] / Матеріал из Википедии – Режим доступу до ресурсу: https://en.wikipedia.org/wiki/NI_Multisim
3. “Getting Started with NI Circuit Design Suite (Bilingual)” - National Instruments