

Simulation Software applied in Electronic Technology teaching

Dong Jie

School of Information Engineering
Shandong Youth University of Political Science
Jinan, China
dj@sdyu.edu.cn

Abstract—Multisim is a commercially available computer-based learning simulation software, that is often used as an educational tool for teaching electronics. This paper first introduces several simulation software's characteristic, then points out why chooses multisim to apply in electronic technology teaching. Besides, the author mainly shows that how to use multisim software to analyses the applied examples concretely and give the simulation figures, thus show good points of simulation software used in electronic technology teaching.

Keywords- simulation software; electronic technology; multisim

I. INTRODUCTION

Electronic technology is not only as the backbone of electronic specialty course, but also it is computers, communications, network management and other professional basic courses. It is a very important course which emphasize practice, a long time, mainly the electronics teaching is composed of three parts, theory teaching, experiments and practical training course teaching, in the traditional theory, experiment, practice three-stage teaching model, usually hard work of teachers teach method make students feel painful in learning, many students in learning course content are often feel very abstract and difficult to understand the concept. With the extensive application of computer multimedia technology, many universities use simulation software into the electronics classroom teaching process, thus, we can display abstract concepts and theories with specific graphics and sound. Using of software in the classroom simulation and presentation can enhance perceptions of students, also, in this way, students can learn both the basic use of various instruments and circuit parameters of the test methods to make teaching and learning in the classroom to form a good interaction. Teaching content is more vivid and intuitive, so that students easily understand the basic theory of the circuit, reaching the multiplier effect.

II. SIMULATION SOFTWARE SELECTION AND FEATURES ANALYSIS

Electronic simulation software is an advanced, fast and efficient electronic design automation tools with the rapid development of computer technology and integrated circuits.

It is based on computer hardware and software as the basic working platforms, set database, graphics, graph theory and topology, computational mathematics, optimization theory and other subjects from the development of general-purpose computer-aided design package[1]. The whole process of the circuit design, performance analysis, parameter optimization to the PCB (printed circuit board) and application specific integrated circuit design can be processed by computer, therefore, the educators and technology personnel, especially in the field of electronic technology are like to use it[2].

Commonly used electronic simulation softwares include view logic, orcad, pspice, pcad, protel, multisim, of which the pspice, protel, and multisim is most commonly used at home and abroad EDA simulation software. According to their distinctive features and the application, the simulation software should be chosen reasonably for different purposes and conditions. For PSpice, its function is very powerful, suitable for a comprehensive analysis of complex circuits and optimization; Protel owns comprehensive performance and a wide range of circuit simulation, achieve the integration of PCB design and circuit simulation. Multisim is intuitive to use and has a high cost performance, it is very suitable for the circuit, the theory of teaching electronic courses, experiments and curriculum design, have broad application prospects. Multisim is an distinctive electronic simulation software produced by Canada's IIT company, after the version 6.0, it has large-scale changes and can simulate more complex circuit. Also, it increases a large number of component models, in large measure to strengthen the high-frequency circuits ability and accuracy of the simulation. schematic drawing[3]. Compared with other general-purpose EDA software in the design file editing and printed circuit board design there is no advantage on multisim, its advantage is mainly focused on circuit simulation.

Multisim simulation is very powerful, we can get a real circuit simulation results without needing to manually connect the realistic circuit, if results are not ideal, just change the component parameters to complete. Multisim is also a more comprehensive analysis tools, including the circuit DC operating point analysis, communication frequency analysis, transient analysis, time domain and frequency domain analysis and equation circuit analysis method. Besides, simulation can set a variety of components in the circuit failure, so we can observe the situation in different conditions of the circuit state[4-5]. During the

simulation, the software can list all the components of in using, store the working status of test equipment, displays waveform and measurement data. In addition, multisim also has a better compatibility, its files can be exported and read by protel[6].

III. SIMULATION SOFTWARE IN ELECTRONIC TECHNOLOGY TEACHING ASSISTANT ROLE

University education is a quality education, the development of learning ability, practical ability, creativity are firstly important. Multisim provides a good practical tools to assist instruction, which enable teachers to going on the teaching process of the digital circuit simulation at any time, using simulation software to teach.

In the multimedia classroom teachers can easily comprehensible analysis of the characteristics of various logic circuits and explain the changes in various input signals or circuit parameter changing. Moreover, during the presentation, teachers can always change the parameters, numerical size or increase the number of logic devices to enable students to visually see the circuit from simple circuits to complex process of change, understanding the different performance of the circuit. Teachers can easily change the different test points to make the demonstration experimenbecomes very simple, the use of software assisted instruction show the contents of the simulated images in front of students which can not be demonstrated in class in real instruments. so the abstract visual knowledge changes to perceptual knowledge, students increase and deepen the understanding of theoretical knowledge and mastery.

Multisim is widely used not only in the teaching, but also in practical application of electronic product development, a lot of departments regard the mutisim as the software analysis tools for electrical systems, its application can significantly shorten the development cycle, saving funds and improve efficiency.

IV. SIMULATION EXAMPLES ANALYSIS

In electronics, the DC and AC analysis of the amplifier circuits has been a very important and basic issue, generally we should firstly do DC analyzation on the amplifier circuit, then do AC. Quiescent operating point analyzation is the key of the DC analysis, because the quiescent operating point is a carrier of the AC signal working. AC signal amplification must be going on in this carrier, leaving the quiescent operating point, AC signals can not work. What's more, the selection of the quiescent operating point need to be in an appropriate location. If location selection is inappropriate, AC signal amplification will produce distortion. The traditional method of amplifier circuits analysis is to calculate the value of the quiescent operating point, and then according to U_{ce} and I_c changing curve of transistors, we draw the curve of input and output AC signals. Using this method, a lot of data and complicated graphics often make people feel cumbersome and have no intuitive understanding of the AC signals chaning state on the quiescent operating point. We take the circuit in Figure 1 as the example and use mutisim to analyze. Figure.1 shows

a typical amplifier circuit, the input is a small AC signal, after amplification, it is output by capacitive coupling.

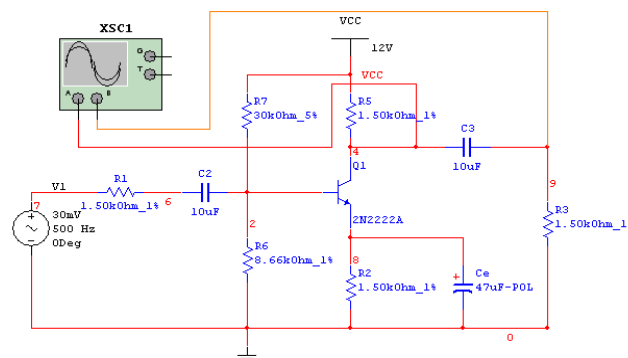


Figure 1. Typical amplifier circuit

By estimating, at the quiescent operating point we get $U_b = 2.6v$, $U_c = 10v$.

Through simulation, we obtain wave figure at node 4 and 9, which is shown in figure.2. From the figure.2 we can see that B channel circuit final output of the sine wave peak is about 734mv, the amplification value is more than 20 times of the input signal The test signal by channel A is the AC signal which stack on the quiescent operating point. Two peaks value of the wave was reduced to 10v, just about equal to the U_c values at quiescent point.

through the figure, it is seen that we use a external DC power supply so that transistors can work at a suitable quiescent point, then the AC signal can be normal amplified, through the capacitor C3, the amplified signal isolates DC signal to get the final wave, from the simulation figure.2, DC and AC analysis intuitively clear.

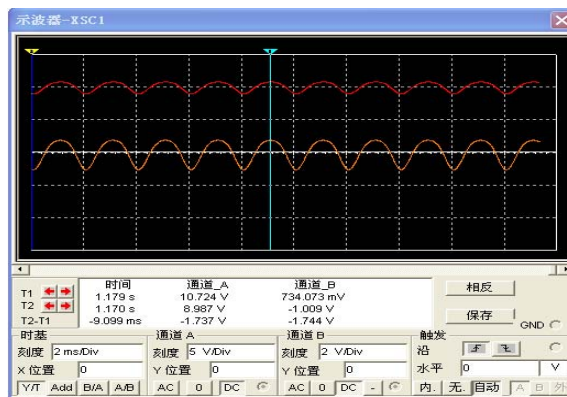


Figure 2. Simulation waveform

Another example, the use of simulation software can be easily completed the design of a specific issue and achieve with the actual circuit simulation. Such as there is a motor, a total of three switches to control it at the same time, no matter which switch changes state, the motor's state will be changed, we use the multiplexer to design. According to this practical problem, we choose 74LS153 as the main

component. We list the truth table, three input switch variable that we set A, B, C, out variable we set Y. As long as the three variables in an odd number of 1, the output Y = 1, otherwise, Y=0. According to the truth table, we write the standards AND/OR type of Y,

$$Y = \overline{A}\overline{B}C + \overline{A}B\overline{C} + A\overline{B}\overline{C} + ABC \quad (1)$$

The output expression of 74LS153 is

$$F = \overline{A1A0D0} + \overline{A1A0D1} + \overline{A1A0D2} + A1A0D3 \quad (2)$$

Compared equations (1) and (2), we get the equations as follows,

$$A1=A, A0=B, D0=D3=C, D1=D2=\overline{C}$$

Then, we use simulation software to draw the actual circuit figure.3.

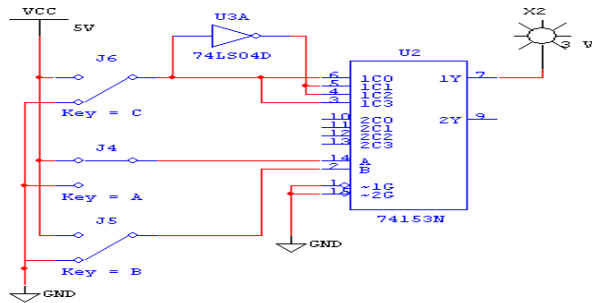


Figure 3. Simulation Circuit

In figure.3, we can switch J4, J5, J6 to change the input state to see changes in motor output, through simulation, the circuit functions are correctly.

V. CONCLUSION

Multisim is a circuit simulation software particularly suitable for teaching electronics. Students experiment in use of multisim can extend the experiment content, but also to expand their thinking space. Study electronics through multisim, we can improve greatly both in theory and in practice.

REFERENCES

- [1] Wang Ping. "Electronic Experiment Guide," China Machine Press, 2009.
- [2] Lianying.Wang. "Based on Multisim 10 Electron Simulation Experiment and Design," Beijing University of Posts and Telecommunications Press, 2009.
- [3] Zhou Hui-chao; Sun Xiao-feng. "Common electronic devices and typical application," Beijing Electronics Industry Press, 2007.
- [4] X.J. Wang. "An application of multisim to the testing experiment of impulse width with electronic control injection engine," UEST of China, vol.135, PP:43-46, Feb 2006.
- [5] Z. Y. Deng. "Analysis of electromagnetic compatibility test technologies," Electric Power Automation Equipment, vol.335, PP:92-95, Aug 2005.

- [6] Z. M. Qian; H. L. Chen, "State of art of electromagnetic compatibility research on power electronic equipment," Transactions of China Electrotechnical Society, vol.198PP:1-11. July 2007.