Application of Multisim Simulation Software in Teaching of Analog Electronic Technology

Zhengdong Li¹, Xiuling Li²*, Decai Jiang³, Xingzong Bao¹, Yan He¹

¹Track Electrical Department, Chongqing Vocational College of Public Transportation, Chongqing, 402247, China
²Communication NCO Academy, Army Engineering University of PLA, Chongqing, 400035, China
³AInnovation Limited Company of Science and Technology, Chongqing, 400000, China
*Corresponding author’s e-mail: gusi@foxmail.com

Abstract. Analog electronic technology is a core basic course of electronic specialty in colleges and universities. It has the characteristics of complex theory, abstract content, difficult to explain by teachers and difficult to understand by students. This article mainly discusses some typical applications of Multisim virtual simulation software in the teaching of analog electronic technology courses, and analyzes the impact on classroom teaching methods. The practice shows that the application of simulation software in the teaching of simulation electronic technology can solve the problems of insufficient experimental teaching site and potential safety hazards. It can help students to understand abstract theory and master related knowledge. It can help to organize lively and efficient classroom teaching and promote the quality and efficiency of classroom teaching.

1. Introduction

Analog electronic technology is a core basic course offered by electronics-related majors in colleges and universities. Through the study of this course, students need to master the basic theory, basic knowledge and basic skills of electronic circuits, and lay a good foundation for learning more in-depth professional knowledge. In the teaching of analog electronic technology courses, teachers explain the basic knowledge, theory, and methods, and more importantly, let students master the method of analyzing problems and the ability to solve practical problems. Due to the strong practicality of this course, the experimental platform must be used in the teaching process, but there are some problems in the experimental teaching, such as the variety of devices required for some experiments, which is not easy to implement; sometimes the components are damaged and the human body is damaged due to experimental operation errors Injury, etc. Using virtual simulation software to simulate the circuit can not only be free from the influence of the experimental site and the instrument, but also avoid the risk of component damage and personal injury, and improve safety. At the same time, through virtual simulation instruments, students can observe the signal waveforms of various positions of electronic circuits in real time, deepen their understanding of the principles, and thus greatly improving the effect of classroom teaching [1-3].
2. Multisim simulation software
Multisim is an electronic design automation software launched by National Instruments, which is used to simulate analog and digital circuits [4-8]. Multisim software has an intuitive and easy-to-use operation interface, convenient component callout, intuitive component labeling, strong simulation reality, and high similarity to the actual experimental platform. The software has a full range of component libraries, a variety of test instruments, such as multimeters, oscilloscopes, signal generators, logic converters, logic analyzers, comprehensive simulation analysis methods, and rich simulation capabilities. Multisim software circuit input methods are diverse, both graphical input of circuit schematics, and hardware description language input, with rich simulation analysis capabilities. Compared with other simulation software, Multisim software has an intuitive interface image, complete instrument components, simple operation and easy to learn. The effect of analysis and simulation in analog electronic circuits is also better. It is currently widely used in circuit analysis.

3. Application of Multisim in the teaching of analog electronic technology——Taking 555 timer as an example
The 555 timer is a very useful and very accurate timer that can be used both as a timer and as an oscillator. As timers, the most common are monostable and Schmitt triggers. A monostable flip-flop circuit can easily generate a single pulse. The Schmitt trigger can be used to complete the waveform shaping. As an oscillator, the most common is the non-steady-state mode. The 555 integrated circuit can generate a square wave output, and the waveform can be adjusted by an external RC charge/dischARGE circuit. The internal structure of 555 timer is shown in Figure 1.

![Figure 1. Architecture diagram of the 555 timer](image)

### 3.1. Mode of the monostable trigger
In the monostable working mode, the monostable trigger has only one steady state and one transient steady state. Under the action of an applied pulse, a monostable flip-flop can flip from a stable state to a temporary steady state. Due to the role of the RC delay link in the circuit, the transient state is maintained for a period of time and then returns to the original steady state. The time for which the transient state is maintained depends on the parameter value of the RC.

The 555 integrated circuit constitutes a monostable flip-flop simulation circuit as shown in Figure 2. The power supply voltage is 5v, and the input trigger voltage Vin is a square wave signal with a duty cycle of 95 and a frequency of 0.02KHz instead. In monostable circuits, the initial output of the 555 integrated circuit is low, and the discharge transistor is turned on and discharged at the same time. When a negative trigger pulse is applied to pin 2, comparator 2 outputs a low level, the trigger is set, and a high level is output. At the same time, the discharge transistor is turned off, the power source charges the capacitor $C_1$, and the charging voltage increases from 0v to $V_{CC}$. When the $C_1$ capacitor charging
voltage reaches $2/3 V_{cc}$, Comparator 1 outputs a low level and the trigger is reset to a low level. At the same time, the discharge transistor is turned on, the capacitor is quickly discharged to 0v, and the output will remain in a low steady state until another trigger pulse is applied.

Figure 2. Simulation circuit of monostable flip-flop

An analog Tektronix oscilloscope is used to display the square wave input signal on pin 2, the voltage signal on capacitor $C_1$ at pin 6, and the output signal on pin 3. The waveforms of the three signals are shown in Figure 3. The monostable circuit has only one steady state. The output is reset at 0v. When a negative phase trigger pulse is input to pin 2, the output will be set to a high level, and its duration is determined by the $R_1C_1$ network. The width of the high-level output pulse is:

$$t_{width} = 1.1R_1C_1$$

Figure 3. Input and output signal waveforms of monostable trigger

3.2. Mode of the Schmitt trigger

Schmitt trigger is a bistable multivibrator. Its working principle is: when the input voltage is higher than the positive threshold voltage, the output is low; when the input voltage is lower than the negative threshold voltage, the output is high; when the input voltage is between the positive and negative threshold voltage, the output status is unchanged. Schmitt triggers are widely used in waveform shaping circuits, which can shape analog signal waveforms into square wave waveforms that digital circuits can
handle. In addition, due to the hysteresis characteristics of Schmitt triggers, they are often used in anti-jamming applications. The Schmitt trigger simulation circuit with 555 timer is shown in Figure 4.

![Schmitt trigger simulation circuit](image)

**Figure 4. Simulation circuit of Schmitt trigger**

In this circuit, pins 6 and 2 are connected in parallel. The input signal is connected to pins 6 and 2 using a triangle wave. When the triangle wave voltage signal is greater than $2/3 V_c$, pin 3 outputs a low level. When the triangle wave voltage signal is less than $1/3 V_c$, pin 3 outputs high level. Figure 5 shows the input and output waveforms using an analog Tektronix oscilloscope.

![Input and output signal waveforms of Schmitt trigger](image)

**Figure 5. Input and output signal waveforms of Schmitt trigger**

### 3.3. Mode of the multivibrator

The multivibrator is a self-excited oscillator capable of generating a rectangular wave, also called a rectangular wave generator. Multivibrators have no steady state, only two transient steady states. During operation, the state of the circuit is automatically alternated between these two transient stable states, thereby generating a rectangular wave pulse signal, which is often used as a pulse signal source and a clock signal in a sequential circuit. The multivibrator simulation circuit with 555 timer is shown in Figure 6.
Figure 6. Output signal waveforms of multivibrator

When the system was powered on, the capacitor was not charged yet. The initial voltage of pin 2 was 0v, which was less than 2/3 Vcc, so the 555 timer output was high. At this time, $\bar{Q}$ is at a low level, the discharge transistor is turned off, and VCC charges the capacitor C through R1 and R2. When the capacitor voltage exceeds 2/3 Vcc, Comparator 1 outputs a low level and the 555 timer outputs a low level. At this time, $\bar{Q}$ is high, the discharge transistor is turned on, and the capacitor C starts to discharge through R2. When the voltage on the capacitor drops to 1/3 Vcc, the 555 timer outputs a high level again, puts it to the end of the transistor, and the capacitor starts to charge again. In this way, the cycle repeats and the output signal is a square wave signal. Figure 7 shows the output waveform of a multivibrator using an Tektronix oscilloscope.

Figure 7. Output waveform of multivibrator

Through the simulation analysis of the monostable flip-flop, Schmitt trigger, and multivibrator circuit composed of the 555 timer, the relationship between the input waveform and the output waveform is very clear and clear. Using Multisim simulation software for simulation teaching can enable students to see experimental phenomena very intuitively, so as to deepen their understanding of theoretical knowledge, and thus effectively improve the classroom teaching effect of the course.

4. Conclusions

Multisim simulation software has a clear interface, simple operation and vivid image. The introduction of Multisim simulation software into the teaching of analog electronic technology can enable students to combine theoretical knowledge and simulation results, thereby deepening the understanding of theoretical knowledge and solving pure theoretical inconveniences. At the same time, the use of
simulation software is not limited by teaching hours. Students can use the spare time to conduct simulation exercises on the knowledge they have learned. This has a positive effect on conducting flipped classroom teaching, cultivating students' autonomous learning ability, and improving the effectiveness of course teaching Meaning.

Acknowledgments
This research was financially supported by the Research Project on Teaching Reform of Chongqing Education Committee in 2019 (Grant: 193561), the Key Project of “13th five” Chongqing Education Science in 2019 (Grant: 2019-GX-185), the Research Project on Science and Technology of Chongqing Education Committee in 2019 (Grant: KJQN201905803), the Scientific Research Project of Chongqing Vocational College of Public Transportation (Grant: ysky2018-08).

References
[1] Xian F., Lai X.Z.(2019) Based on Multisim Simulation Digital Logic Experiment Teaching Reform. Research and Exploration in laboratory, 38(9): 228-232, 297.
[2] Ye C.H., Hua C.H., Yan J.(2017) Exploration on cultivation of practical and innovative ability of analog electronic technology. Experimental Technology and Management, 34(1): 29-32.
[3] Zhang L., Deng T.P., Peng L.(2018) The Flipped Classroom Teaching Practice for Basis of Analog Electronics Technology Course. Journal of Electrical & Electronic Education, 40(2): 57-61.
[4] Hou Y.Y., Chen B., Li T.L.(2018) Teaching Research and Practice on "Analog Electronic Technology Basis" Based on Hybrid Teaching Mode. Education Teaching Forum, 33: 172-173.
[5] Zhang X.W., Si Y.Q.(2019) Application of Multisim9 in small bulb series-parallel circuits. Journal of Hubei Normal University(Natural Science), 39(4): 84-88
[6] Zhang J.L., Li K.R.(2019) Simulation Experiment Study on Inductance Filtering of Half-Wave Rectification Based on Multisim 10. Physical Experiment of College, 32(6): 104-107
[7] Zhu J.N., Guo X.F., Lyu Y., et al(2018). Circuit Design of Flickerless Direct-current LED Lamp Based on Multisim China. Illuminating Engineering Journal, 29(5): 120-123
[8] Li Y., Li X.H., Guo W.L.(2019) Design and implementation of low-frequency virtual laboratory based on LabVIEW-Multisim. Modern Electronics Technique , 42(6): 72-75