Teaching Practice of Analog Electronic Technology Theory Course Based on Multisim

Ya-nan TIAN*, Ji-hong LIU, Jing-hong LI, Wei-hong ZHANG and Hong-wei LEI

School of Information Science & Engineering, Northeastern University, Shenyang, China

*Corresponding author

Keywords: Analog electronic technology, Oscillating circuit, Multisim.

Abstract. In view of the problems existing in the teaching of analog electronic technology theory course, the simulation software Multisim is applied in the teaching of theory course. The application of Multisim in the teaching of RC sine wave oscillating circuit is taken as an example. And Multisim simulation shows a positive effect on improving the teaching quality of analog electronic technology theory course.

Introduction

Analog electronic technology course is a compulsory basic course for the students majored in automation, electrical engineering, communication engineering, electronic information and other electrical specialties in colleges and universities. It is the foundation of the students' follow-up professional courses and occupies a very important position in the whole curriculum system. Analog electronics is a subject researching analog signals' processing and computing. It uses semiconductor diode, transistor and field effect tube as the core electronic devices. The contents include voltage amplification circuit, power amplifier circuit, operation amplifier circuit, feedback amplifier circuit, signal operation and processing circuit, signal generating circuit, DC regulated power supply and so on. Analog electronic technology is a theoretical and practical course. It is also a course that is generally considered not easy to learn and hard to teach. In the traditional teaching mode of analog electronic technology, the tendency of emphasizing theory and despising practice is widespread. In addition, the contradiction of less time and more contents in the course makes the teaching effect not good. Some students do not understand, do not love learning, and can not use the knowledge to design the circuit. In order to solve this problem, we must improve and innovate the traditional teaching mode of theory courses.

In recent years, with the rapid development of electronic computer technology, all kinds of circuit simulation software are coming out continuously, which provides new means and methods for the teaching activities of college teachers. In order to motivate the students' interest in learning and improve the teaching quality and teaching effect, we combine the classroom theory teaching with the Multisim simulation and make the boring circuit run, so that the students can easily accept and understand the theoretical knowledge. In this paper, the application and practice of simulation software in the teaching of analog electronic technology theory course are introduced by a specific teaching case, that is the application of Multisim in the teaching of RC sine wave oscillating circuit.

Teaching of RC Sine Wave Oscillating Circuit Based on Multisim

Sine wave oscillating circuit is widely used in scientific research, industrial production, medicine, communication and other fields. Oscillating circuit is a closed loop system that realizes positive feedback in the middle frequency band, without input signal and comparison. Its block diagram is shown in Figure 1. In order to maintain stable output, there is \( \dot{x}_d = \dot{x}_f \). Because of \( \dot{x}_f = A \hat{F} \dot{x}_s \), the equilibrium condition of the oscillating circuit is written as
\[ \dot{A}F = 1. \] (1)

It contains two meanings, one is its module, that is
\[ |\dot{A}F| = 1. \] (2)

The other is its argument, that is
\[ \arg \dot{A}F = \pm 2n\pi \quad n = 0, 1, 2 \cdots. \] (3)

The former is called the amplitude balance condition, and the latter is called the phase equilibrium condition. These two conditions keep the sine wave oscillating circuit oscillatory. Oscillating circuits usually include amplifiers, feedback networks, selecting frequency networks and stabilizing amplitude links. Where does the initial signal in the oscillating circuit come from? In fact, small disturbances in the circuit, such as the current impact generated by the power supply, or the noise voltage produced within the electronic components, can be the initial signals of the oscillating circuit. These disturbances contain a very wide frequency component, but only this frequency \( f = f_0 \) satisfies the phase equilibrium condition. This requires the oscillating circuit have the function of “selecting frequency”. Usually, the feedback network of the oscillating circuit contains a suitable reactance element, so the feedback network can not only complete the function of the feedback signal, but also realize the function of “frequency selection” [1,2].

Figure 1. The block diagram of oscillating circuit.  
Figure 2. RC network.

RC sine wave oscillator circuit is composed of amplifying circuit and RC network. RC network is shown in Figure 2. The equation (4) is obtained by analyzing RC network.

\[
\frac{\dot{U}_2}{\dot{U}_1} = \frac{\frac{R_2}{j\omega C_1}}{R_1 + \frac{R_2}{j\omega C_1} + \frac{\frac{R_2}{j\omega C_1} + 1}{R_1 + \frac{\frac{R_2}{j\omega C_1}}{R_2 + \frac{R_2}{j\omega C_1}}}}
\]

(4)

Usually set \( R_1 = R_2 = R \), \( C_1 = C_2 = C \) and \( \omega_0 = \frac{1}{RC} \). And the equation (4) is written as

\[
\frac{\dot{U}_2}{\dot{U}_1} = \frac{1}{3 + j\left(\frac{\omega}{\omega_0} - \frac{\omega_0}{\omega}\right)}.
\]

(5)

The amplitude-frequency and phase-frequency characteristics are described by the equation (6) and (7).
\[
\left| \frac{\dot{U}_2}{\dot{U}_1} \right| = \frac{1}{\sqrt{\omega^2 + \frac{\omega_0^2 - \omega_0}{\omega}}} 
\]
(6)

\[
\arg \frac{\dot{U}_2}{\dot{U}_1} = -\arctan \frac{\omega_0 - \omega}{3} 
\]
(7)

It is known from the phase-frequency characteristic that only in the condition of \( \omega = \omega_0 \), the argument \( \arg \frac{\dot{U}_2}{\dot{U}_1} \) is \( \pm 2n\pi \), \( n = 0, 1, 2, \ldots \). That is to say, \( \dot{U}_2 \) and \( \dot{U}_1 \) have the same phase. And the amplitude \( \frac{\dot{U}_2}{\dot{U}_1} = \frac{1}{3} \).

The sine wave with a frequency of \( f_0 = \frac{\omega_0}{2\pi} = \frac{1}{2\pi RC} \) is used in the RC network in Figure 2. The phase of input signal and output signal remains the same, shown in Figure 3.

A RC sine wave oscillating circuit consisting of the non-inverting amplifier and RC network is shown in Figure 4. The signal with frequency of \( f_0 = \frac{\omega_0}{2\pi} = \frac{1}{2\pi RC} \) is input into the non-inverting terminal of integrated operational amplifier. And the output is in the same phase with the input signal. When the output is transmitted through the RC network to the non-inverting terminal, the phase is not changed, and the phase equilibrium condition is satisfied. The other frequency signals can not satisfy the phase equilibrium condition. It is known that feedback coefficient of the feedback network \( F = \frac{1}{3} \).

To satisfy the amplitude balance condition \( |A| = 1 \), the voltage gain of the non-inverting amplifier \( A = 1 + \frac{R_1}{R_2} = 3 \). But can this circuit oscillate? The simulation result shown in Figure 5 indicates that the circuit can not oscillate. Because the initial signal of the oscillation comes from the tiny disturbance in the circuit, the amplitude of the signal is very small, and it can not form sine wave output of a certain amplitude. So when the circuit is closed, setting \( |A| > 1 \), that is \( A = 1 + \frac{R_1}{R_2} > 3 \), will ensure that the circuit starts to oscillate reliably [3].
In order to start to oscillate reliably, the RC oscillating circuit is modified, as shown in Figure 6. By adjusting the resistance of the potentiometer, let $|\dot{\phi}| > 1$ when the circuit is closed. In this way, the tiny disturbance signal satisfying the phase equilibrium condition is amplified continuously in the feedback loop, and finally the sine wave with a certain frequency is output. The starting process of the oscillation can be seen through the simulation of the circuit, shown in Figure 7. But after starting of oscillation, the maintain of $|\dot{\phi}| > 1$ makes the waveform distorted [4, 5].

In order to output sine wave without distortion, it is necessary to use automatic stabilizing amplitude links. The oscillating circuit is modified, as shown in Figure 8. When two anti-parallel diodes D1 and D2 are parallel connected to the resistance R6 in the feedback branch, there is always a diode in the conduction state, whether in the positive half period or the negative half period. Due to the small amplitude of the output signal in the starting of oscillation, the voltage added to the diode is small and the equivalent resistance of the diode becomes larger. Then the voltage gain of the circuit becomes larger and the circuit start to oscillate reliably. As the amplitude of the output signal becomes larger, the voltage added to the diode becomes larger. The equivalent resistance of the diode becomes smaller, and the voltage gain of the circuit becomes smaller. Thus the automatic amplitude stabilization is realized, and the output waveform is shown in Figure 9.

![Figure 7. Simulation result of the circuit (2).](image1)

![Figure 8. RC sine wave oscillating circuit (3).](image2)

![Figure 9. Simulation result of the circuit (3).](image3)

**Summary**

Practice shows that the application of simulation software Multisim in the teaching of analog electronic technology theory has played a positive role in promoting the teaching effect. First of all, Multisim simulation enriches classroom teaching contents, vividly demonstrates the results of circuit operation, and deepens students' understanding and cognition of theory. Secondly, Multisim is very suitable for demonstrating the influence of circuit parameters on circuit performance and deepening students' understanding of the importance of parameter selection. Finally, the application of Multisim...
strengthens the students’ understanding of the design process of electronic circuit, namely, “design - simulation - production”, and improves the students' ability of circuit design, analysis and debugging.

Acknowledgement
This work is supported by the Fundamental Research Funds for the Central Universities (N140403002).

References