

Research on the Application of Multisim in Electronic Design

Yuanzi He and Renbo Xu

(NanChang Institute of Science and Technology, Nanchang,330108)

Keywords: Multisim; Electronic design; Application

Abstract. In this paper, the application of Multisim in electronic design is studied through examples, and the specific steps of simulation analysis are discussed. The experimental results show that the traditional design method combined with the current EDA technology will form a newer electronic design method, which has important significance for guiding future product development.

No matter at what time, the research and development of electronic products or electronic technology cannot be separated from continuous simulation exploration and experimental exploration. When electronic design is actually carried out, various influencing factors should be considered. Especially for college students at this stage, it is obviously unrealistic to start designing a large-scale integrated circuit just after the recent contact with electronic design. Due to the complexity of the internal structure of many inherited circuits, it is difficult to have a comprehensive grasp of the relevant treatment alone. Electronic design operations must be carried out according to actual conditions and specific link arrangements. For electronic designers, when designing electronically, it is necessary to begin with the concept of electronic design, as well as master the relevant protocols and algorithms for electronic design. These can all be done through computer operations^[1].

Functions and Features of Multisim

Multisim is a tool software for electronic circuit simulation and design, which occupies an important position in China's electronic virtual platform. This software provides more detailed information data analysis methods, including targeted analysis, dynamic analysis, and transient analysis of circuit static operating points. Users often refer to this operation when they customize it.^[2]

Its software mainly works by using two-digit decimal counting. Under normal circumstances, the basic steps of its work include: creation of simulation circuit schematics, setting of circuit diagram options, use of simulation instrument analysis, and startup simulation. The schematic drawing of the American life is more complex. Multisim provides rich component support for the design and drawing of principles and drawings, making it much easier to draw schematics.^[3] First of all, to get the ability of logic analysis from the virtual instrument, this is mainly done by using a logic analyzer. There are 16 access segments such as 1~F, and different access segments have their own control functions. Press the F5 key to turn on the emulation switch for simulation operation. When the waveform is obtained, the simulation operation must be ended immediately. Click "OK" to end the operation. The use of a logic analyzer is similar to that of the logic analyzer. The simulation of the advanced nature will be closed after the waveform is obtained, so that a more intuitive layout window pattern can be obtained.

Practical Application of Simulation Software

When it is applied, it often involves the use of large and medium-sized integrated circuits. In the operation of the integrated circuit, it is necessary to ensure that the function of the analog circuit is consistent with the function of the digital circuit, and the external resistance and the capacitance can be greatly facilitated to constitute a monostable trigger. When it is applied again, it has the advantages of complete function output, large output current, flexibility, and so on. Therefore, once it is produced, it has received wide attention and application in the industry.

Designing Waveform Generator Using 555 Timer. You can find the 555 timer from the hybrid integration library and use the oscilloscope to measure the components and build the circuit (as shown in Fig. 1). When the circuit is designed, a self-oscillating multivibrator is mainly used to perform simulation analysis on the circuit, which mainly involves the following steps: first, to activate the circuit first, the motor icon, you can see the output waveform on the self-oscillation multivibrator, and its shape is roughly triangular. In general, channel A is the channel that mainly outputs rectangular pulse waves. Second, to double-click the frequency counter shown in the figure, the resistance can be adjusted when the displayed value shows 1.05 kHz. ^[4] Third, by calculating the parameters in Fig. 1, it was found that the calculated results are in good agreement with the simulation results.

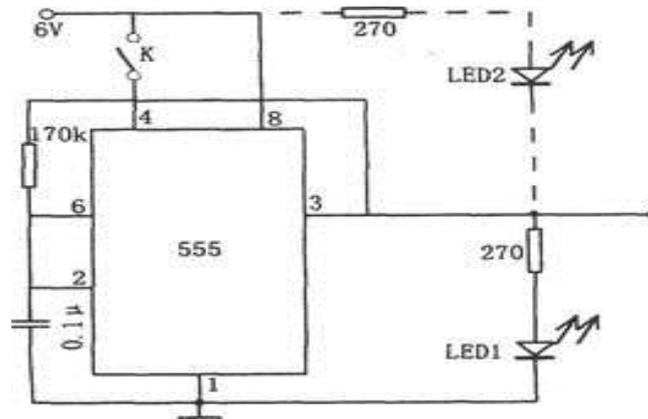


Figure 1. Finite 555 Timer Circuit Diagram

Designing a Complete Circuit Using the 555 Timer. Under normal circumstances, in the design of the circuit, use the 555 timer to complete, monostable starter is mainly used for waveform shaping, timing and delay functions, you can use it to create a complete waveform diagram (as shown in Fig. 2). Specific simulation steps can be divided into setting function generator output values, activating circuits, and measuring the results. First, set the output of the function generator to a square wave. Normally, the amplitude is 5 volts. It is formed by a capacitor to form a sharp pulse, and then it is subjected to DC bias voltage processing through the power supply. Under normal circumstances, the DC bias voltage should be 12 volts. Finally, the input waveform of the 555 timer is a positive pulse waveform. Second, when the circuit is activated, it is necessary to pay attention to the accuracy of each step. When the B channel is an output rectangular pulse, pay attention to control the irregular sharp pulse waveform to achieve the best operation effect. When the input signal of the pulse is less than 1 volt, its potential must be adjusted to adjust the pulse width to 2.168 ms. Finally, the simulation result is calculated according to the relevant calculation formula, and the calculated result is compared with the measured result. Then the shape of the input spike is adjusted so that the effect of changing R2 can be achieved. ^[5]

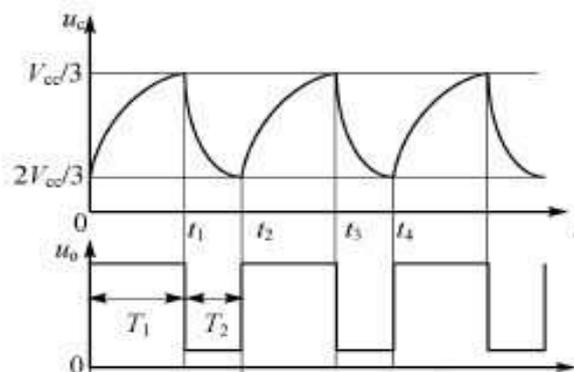


Figure 2. Finite Complete Waveform Diagram

Static Operating Point's Effect on Output Waveform. As shown in Fig. 3, when deleting the digital universal meter, it is necessary to apply the series connection in the collection of electrons of the triode and guarantee the effect of the application and the quality of the application. When measuring I, pay attention to the amount of triode conversion, and the digital display of the digital multimeter connected between triodes T and C. According to the value of the amount of change in the multimeter adjustment, supporting the simulation of the waveform data, the data are in saturated distortion, distortion amplification, and as of distortion. After analyzing and discussing the above simulation results, it is not difficult to see that the results follow the following rules: When the bias resistance R is changed, the quiescent current and the voltage will change accordingly, and in the course of the change, the triode will be caused. The work area has changed significantly. When the bias resistance, voltage, and current are within the respective values, the area of the transistor is amplified to some extent. If the bias resistance becomes smaller, the current becomes larger and the voltage is smaller, and the triode is in a saturated state. In contrast, the triode is in the work cut-off zone.

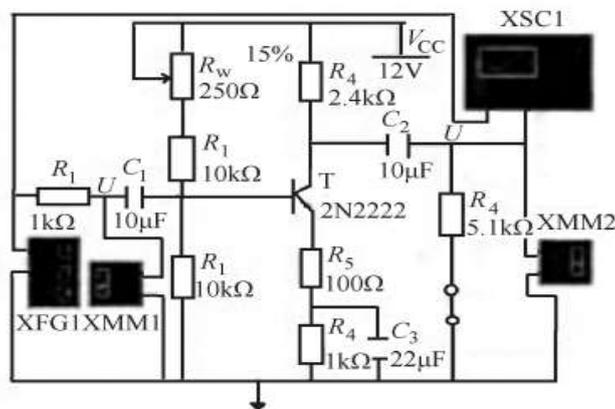


Figure 3. Finite Transistor Integrated Electronic Diagram

Conclusion

When using this tool software for circuit design, expensive experimental equipment is not required. The computer can provide the required technology and data support throughout the process, and it can provide a quiet design environment that is conducive to the completion of the design operation. At the present stage, due to the continuous development of electronic design technology, once problems occur in the actual design operation, they can be modified in a timely manner, and the modification process is simple and can be performed more systematically at any time. Traditional electronic design requires a large number of electronic components in the laboratory to cooperate with each other to complete, and the birth of Multisim tool software makes the development of electronic design work easier, and also saves operating time and operation cost to a great extent.

Project Funding

Nanchang DME Photoelectric Engineering Key Laboratory (No.NCZDSY -004)

References

- [1]Li Xiang. Application of Multisim12.0 Software in Electronic Circuit Course Design [J]. 16 or 17 in Life, 2017,10(17):52.
- [2] Wu Xiaohua, Shao Zhongliang, Li Shujun. Multisim and Its Application in Electronic Design [J]. Electromechanical Product Development and Innovation, 2014,21(03):137-138+146.
- [3]Lian Lifang. Application of Multisim10.0 Software in Electronic Circuit Course Design [J]. Journal of Putian University, 2015,36(05):68-71.

- [4]Xian Kaiyi. Application of Multisim2001 Simulation Software in Electronic Technology Circuit Design [J]. Instrumentation and Analysis Monitoring,2013,21(01):18-20.
- [5]Liang Li. Application of Multisim Simulation Software in Electronic Circuit Design [J]. China Educational Technology Equipment,2015,29(10):35-37.
- [6] Simulation and Analysis of Electronic Circuit Based on Multisim10 [J]. Yang Yulan. Henan Science and Technology. 2013(17)