International Journal of Informatics and Information Systems ISSN 2579-7069
Vol. 6, No. 3, September 2023, pp. 134-140

Research on Ripple Suppression of Switching Power Supply Based on Computer Simulation Software Multisim

Songmei Liu [1]

[1] Electrical Engineering Research Labs, Wanjiang University of Technology, Ma'anshan, Anhui, China, 243000
[1] liusm@wjut.edu.cn*
* corresponding author

(Received: June 25, 2023; Revised: July 19, 2023; Accepted: August 31, 2023; Available online: September 10, 2023)

Abstract

In an era of rapidly advancing computer software technology, its role is increasingly fundamental in addressing the challenges of ripple in electrical power sources originating from switching types of power supplies. This article aims to examine in detail the causes of ripple generation, the various methods used to overcome such ripple, as well as the relevant circuit designs in the Multisim simulation environment. In addition, this research will also explore various relevant simulation analysis methods. Ripple is a periodic fluctuation in voltage and current in power circuits, and is often a serious bottleneck in the reliability of electronic systems. In this context, this article will discuss the underlying causes of ripple, such as the characteristics of the components involved in switching power supplies and the role of impedance in circuits. We will dive into methods that have been developed and widely applied in industry to control ripple, such as the use of capacitors, inductors, and active filters. Furthermore, this research will provide insights into circuit design with the help of Multisim simulation software, which is a very useful tool for virtually testing and analysing power circuits before implementing them in the real world. We will discuss concrete steps as well as strategies applied in circuit design that are effective in reducing ripple in power supplies. Finally, this article will highlight various simulation analysis methods that can be used to evaluate circuit performance and the effectiveness of the ripple control measures taken. It is hoped that this research will make a valuable contribution to professionals who seek to maintain the stability and reliability of modern electronic systems by addressing the issue of ripple in switching type power supplies.

Keywords: Computer Software, Switching Power Supply, Ripple

1. Introduction

Switching power supplies have emerged as an emblematic technological marvel in the ever-evolving field of power electronics. These advanced systems harness the formidable capabilities of modern power electronic technology to meticulously govern the timing of switching on and off, facilitating precise regulation of the output voltage. This dynamic approach to voltage regulation has transformed the landscape of electrical power management, offering unparalleled advantages across a staggering array of applications, from consumer electronics to industrial machinery and beyond.

In contrast to their traditional linear power supply counterparts, switching power supplies often exhibit a relatively higher ripple coefficient. This seemingly inconspicuous ripple coefficient has, over the years, emerged as a critical parameter that fundamentally characterizes the performance and reliability of switching power supplies. To fully appreciate its significance, it is imperative to delve into the intricate dynamics of ripple within these systems.

Ripple, in the context of switching power supplies, refers to the periodic fluctuation in voltage or current levels, often manifesting as small but discernible variations from the ideal output. While high ripple levels might initially appear as a drawback, understanding and effectively managing ripple have become indispensable pursuits. Excessive ripple can profoundly impact the normal operation of electronic circuits, resulting in a litany of undesirable consequences.

These encompass a gamut of issues, ranging from signal source impurities that can distort data and communications to increased amplifier noise levels that degrade audio or sensor performance. Furthermore, in some cases, excessive ripple can precipitate overloads in critical components, risking system failure.

Hence, the meticulous analysis and control of ripple have become paramount in modern electronic design and engineering. Engineers and researchers have tirelessly worked to strike a delicate balance between the manifold advantages of switching power supplies, including improved efficiency and compact form factors, and the imperative to minimize ripple to ensure optimal circuit performance.

However, as we delve into the state of the art in ripple management, it becomes evident that there exists a research gap that demands further exploration. While significant strides have been made in understanding and mitigating ripple, the ever-increasing complexity of electronic systems and the burgeoning demand for high-performance, energy-efficient devices continually pose new challenges. These challenges include the need for innovative ripple reduction techniques that can accommodate the diverse requirements of emerging technologies such as electric vehicles, renewable energy systems, and advanced computing platforms.

State-of-the-art research is exploring novel materials, advanced control algorithms, and innovative circuit topologies to push the boundaries of ripple reduction. Moreover, the intersection of artificial intelligence and power electronics is opening up exciting possibilities for real-time ripple monitoring and adaptive ripple suppression. Addressing this research gap in the context of evolving technological landscapes is pivotal to ensuring the continued enhancement of switching power supplies and their seamless integration into modern electronic ecosystems.

In light of these multifaceted considerations, this article embarks on an expansive and comprehensive exploration of the intricate world of switching power supply ripple. We will traverse the spectrum of causes, consequences, and, most importantly, the vast array of methods and strategies employed by engineers and researchers to effectively suppress ripple, thereby optimizing the performance of switching power supplies. By delving deep into the multifarious aspects of ripple management, we aspire to provide not just insights but a holistic understanding, empowering engineers, researchers, and practitioners to unlock the full potential of switching power supplies while upholding the integrity and reliability of their electronic systems in today's dynamic and demanding technological landscape.

2. Analysis of the Causes of Ripple

Usually, the switching power supply converts the AC power provided by the grid into DC power through rectification and filtering. The high-speed switching on and off of the switching tube will cause fluctuations in the output voltage. The fast recovery diodes and inductors in the output loop will also cause fluctuations in the output voltage. The sum of these high-frequency and low-frequency fluctuations forms the output ripple, including voltage ripple and current ripple [1,2].

There are many reasons for the ripple in the switching power supply, and the ripple produced by the MOS tube turning on and off is one of the main reasons. When the switch tube is turned on and off, there will be a rise time and a fall time, which will cause a noise of the same frequency in the circuit. The inductance on the output circuit will also produce a noise with charging and discharging, and there will be leakage inductance. Between the wire and the wire, there will be various parasitic inductances between the pins of the components, and these parasitic inductances will follow the formula (1) to produce changes.

$$U=-L di/dt \qquad (1)$$

It can be seen from this formula that once the current at both ends of the inductor changes, the voltage at both ends of the inductor will change. Therefore, the layout of the components on the circuit board and the wiring method will affect the performance of the circuit. These influencing factors are the root causes of ripple. The ripple of the switching power supply has always been an important parameter, especially in the application of semiconductor lasers, the output voltage and current should be stabilized [3,4].

3. Ripple Suppression Method

3.1. Filtering Method

The filtering method is one of the easiest methods, because the output has ripples, then we have to design a suitable filter to filter out. There are active filters and passive filters. The filter is implemented in parallel or in series with several resistors and capacitors in the output loop. In this method, the ripple frequency characteristics must be obtained through detailed and rigorous calculations, so that accurate resistance and capacitance values can be selected. Although this method is simple, once the capacitor fails, the resistor fails or there is a slight inaccuracy, it is very likely that new ripple or noise will be mixed, which will increase the output ripple. And this method can be considered in low-power switching power supplies. If it is a large current of tens of amperes and a high-power power supply of tens of watts, the loss cannot be ignored, and the volume will increase accordingly [5,6]. The LC low- pass filter is shown in Figure 1.

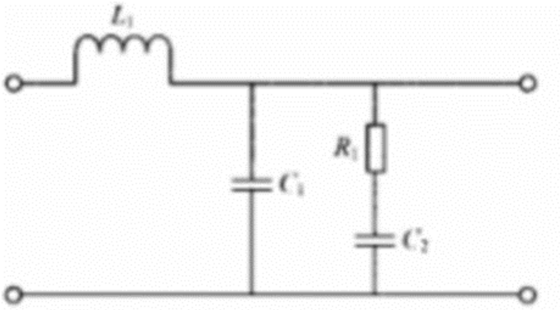

Figure 1. LC low-pass filter

One of the main factors in the generation of switching power supply ripple is the turn-on and turn- off of the MOS tube. Therefore, a circuit that absorbs switching spikes can be designed in the MOSFET part. There are many kinds of switching spike absorption circuits. Figure 2 shows an example of an LC absorption circuit. This method is suitable for the topological structure of external MOSFET, and it can't do anything for some integrated modules with built-in integrated MOSFET. And this kind of absorption circuit also needs accurate calculation [7,8].

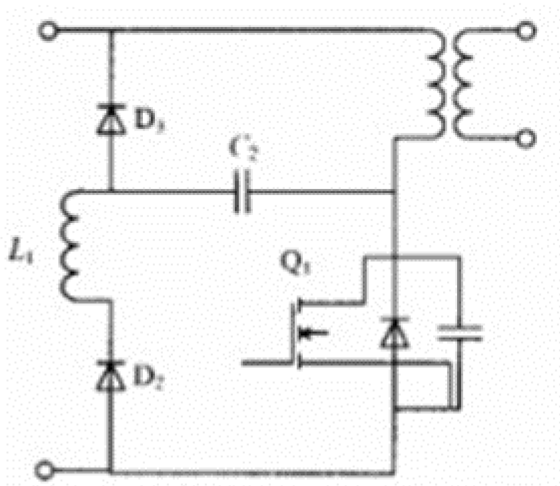

Figure 2. MOSFET spike absorption circuit

3.2. Two-Way Parallel Superposition Method and Improvement Ideas

The ripple elimination of high-current, high-power switching power supplies can be achieved by adjusting the frequency of the PWM on the upper control terminal of the MOSFET, or by using the idea of multi-channel superposition. The output ripple can also be controlled by adjusting the PWM frequency of the control terminal of the switching tube. The basic idea of dual parallel is also achieved by fine-tuning the frequency and duty cycle of PWM. The switching power supply adopts dual parallel, and both channels provide output at the same time. Power is conceptually divided into main power supply and auxiliary power supply [9,10]. The main power supply has ripples, and the secondary power supply also has ripples, but if the duty cycle of the two power supplies is 50% and the phase difference is 180°, superimposing the two at the output terminal will greatly reduce the ripple and improve performance,as shown in Figure 3 and Figure 4.

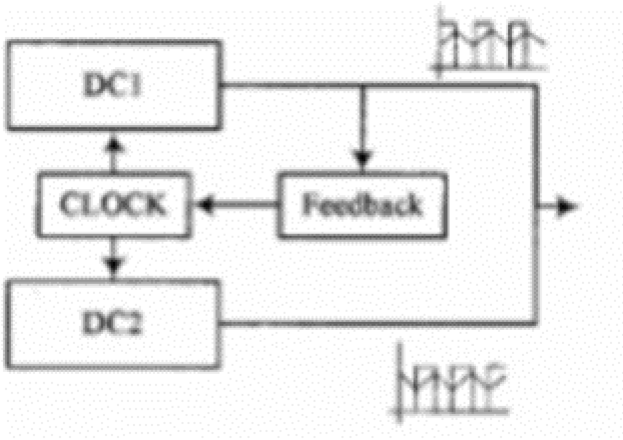

Figure 3. Basic structure of dual power supplies in parallel

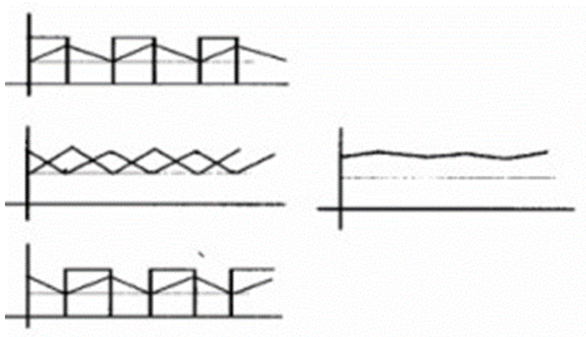

Figure 4. Waveform superposition of dual power supplies in parallel

DC1 is the main power source, and DC2 is the secondary power source. When DC1 is turned on, the voltage and current rise, and DC2 is turned off at this time. When DC2 turns on, the voltage and current rise, and DC1 turns off. The output phase difference between the two is 180°, and the output results are superimposed on each other, just to make the output ripples cancel each other, this is the idea of dual parallel connection. However, due to load fluctuations or external noise factors, the phase of the main and auxiliary power sources changes. When the phase difference is no longer 180°, the amplitude and frequency of the wave will increase. Therefore, the improvement method proposed is to take out the ripple phase signal from the output of the main power supply, and feed this signal back to the secondary power supply, so that the secondary power supply can correct the phase difference in time,this also can keep the phase difference between the two power supplies at 180° [11,12].

4. Main Functions and Features of Multisim 10.0

Using Multisim software to design electronic circuits is like lapped circuits on a laboratory breadboard, and it is not limited by the types, quantities and test instruments of components. Multisim software can achieve circuit building, simulation analysis, design and experiment synchronization, design while simulation, debugging and modification is convenient; the simulation experiment does not consume actual test instruments and components, the experiment cost is low, and the experiment speed is fast. The software can complete the whole process from circuit simulation design to circuit layout generation, which provides a brand-new means and convenient way for the design of electronic systems and the development of electronic products. It includes graphic input of circuit schematic diagram and circuit hardware description language input mode, and it has rich simulation and analysis capabilities [13,14].

Multisim is a complete integrated design environment. It has the following characteristics:

1) It has an intuitive graphical interface: the entire operation interface is like an electronic experimental workbench, and the test instruments and components required for drawing circuit simulation can be directly dragged and dropped onto the screen,tap the mouse to connect them with wires. The operation mode and control panel of the software instrument are similar to the real object. The measured waveform, data and characteristic curve are exactly the same as those seen on the real instrument.

2) It has a huge component library. It has a variety of components, such as basic components, signal sources, analog integrated circuits, digital integrated circuits, control components, indicating components, etc [15,16].

3) It has powerful simulation capabilities: it can simulate digital circuits or analog circuits separately, as well as digital/analog hybrid simulations, especially with the addition of radio frequency (RF) circuit simulation functions. When the simulation fails, an error message will be displayed, prompting the possible cause of the error, and the simulation result can be printed and stored at any time.

4) It has powerful analysis functions. It provides 14 simulation analysis methods. Such as DC operating point analysis, AC analysis, distortion analysis, transient analysis, noise analysis, DC sweep analysis, parameter sweep analysis, temperature sweep analysis, pole-zero analysis, transfer function analysis, post-processing analysis, Fourier analysis, etc.

5) It has a powerful virtual instrument function. Such as multimeters, oscilloscopes, word signal generators, wattmeters, scanners, distortion meters, network analyzers, logic converters, etc.

6) VHDL/Verilog design input and simulation. Multisim software includes VHDL/Verilog design and simulation (option). This integrates simulation with the design of analog circuits, digital circuits, and the design and simulation of large-scale programmable logic devices. This breaks through the original bottleneck that large-scale programmable logic devices cannot be integrated with ordinary circuits for simulation.

7) It can be seamlessly connected with circuit board design software. The design results of Multisim software can be easily exported to the circuit board design software for circuit board routing.

8) Remote control function. Multisim software supports remote control function. It can not only share the interface of Multisim software with other people, but it can also make others see the controller's operation on their own computer, and it can also transfer control to other people. Let it operate the software, so that interactive teaching can be realized, which is especially suitable for the teaching of electronic circuits [17,18].

5. Circuit Design and Simulation Analysis Method Under Multisim Environment

Circuit design uses Multisim 10.0 for circuit design. We don't need to learn computer control language and various input and output instructions. We don't need to write electronic circuit diagram programs. It is close to a real integrated experimental environment, like a circuit in a laboratory. We only need to place the designed virtual electronic components and virtual instruments in the Multisim 10.0 circuit design window, and then use wires and nodes to connect the corresponding virtual components and the measurement interface of the instrument. In this way, the results of various simulation waveforms and parameters can be observed from the virtual instrumentation, which is very intuitive and convenient. The author has studied the use of electronics simulation teaching to complete theoretical teaching and part of practical teaching in the teaching in recent years. It is found that simulation teaching can get twice the result with half the effort, and a large amount of knowledge can be imparted to students in a limited number of hours. Through the use of simulation software, students can carry out experiments and problem discussions independently, which can effectively cultivate students' self-learning ability and consolidate the theoretical foundation [19,20].

Taking the input circuit design of the actual circuit voice amplifier as an example, the design steps are as follows:

1) Entering the Multisim 10.0 working environment interface, we can conveniently place components and instruments: call out resistors, variable resistors, electrolytic capacitors, variable capacitors and other components from the "Basic" component box, and adjust them from the "Source" component box Output the DC voltage source and the grounding symbol, call out the transistors from the "Transistor" component box, and call out the "dual trace oscilloscope", "function generator", "digital multimeter", etc. from the instrument box on the right side of the working interface. Arrange the components according to the schematic diagram of the circuit.
2) Circuit wiring: Place the mouse on the component pin or the instrument interface, and after the mouse pointer changes to a "+" shape, move the mouse to another component pin to complete the connection between the two.
3) Setting parameters: Double-click the edited component with the mouse, and set the component parameters in the pop-up dialog box. If the typical value of the transistor is β=100, we need to modify it to β=80. Then just double-click the transistor, click the EditModel button on the Value page, and the EditModel dialog box will appear. Change the BF=100 to BF=80 to complete the modification of the β value of the transistor.

6. Conclusion

In summary, as we witness the relentless march of progress in the field of computer technology, its pervasive influence extends to the domain of switching power supply design. This integration of computer technology into power supply systems has yielded significant benefits, foremost among them being the substantial reduction in the undesirable ripple that power supplies often generate. This reduction in ripple not only enhances the overall performance and stability of electronic devices but also contributes to their longevity and reliability, thereby enhancing user experiences across a myriad of applications.

Nonetheless, as is often the case with technological advancements, the adoption of new technologies brings with it an array of challenges and novel issues to grapple with. While integrating computer technology into switching power supply design has offered solutions to many long-standing problems, it simultaneously presents us with new complexities and nuances that demand our attention and ingenuity. These emerging challenges necessitate a relentless pursuit of innovation and a continuous drive to refine and optimize switching power supply systems.

The trajectory of power supply technology is far from static; rather, it is marked by an ever-evolving landscape characterized by constant innovation and adaptation. To stay at the forefront of this dynamic field, it is imperative that we embark on an ongoing journey of exploration and discovery, delving deeper into the intricacies of switching power supply systems. With each challenge encountered, we have the opportunity to cultivate solutions that not only resolve the immediate issues at hand but also lay the groundwork for the development of power supply technology with even higher levels of performance and efficiency.

In this quest for advancement, interdisciplinary collaboration and cross-pollination of ideas will undoubtedly play a pivotal role. The convergence of computer technology, electrical engineering, materials science, and other disciplines holds the promise of unlocking unprecedented possibilities in power supply design. Through such collaboration and a steadfast commitment to innovation, we can navigate the evolving landscape of switching power supply technology, surmount the obstacles that arise, and usher in an era of power supplies that not only meet but exceed our ever-expanding needs and expectations.

References

[1] Li Hongtao, Ren Weide. Research on suppression methods of resonance and harmonics of mine special power grid based on computer simulation [J]. *Coal Technology*, vol. 1, no. 1, pp. 56-57, 2014.

[2] P. Doungmala and T. H. Thai, “Investigation into the Application of Image Modeling Technology in the Field of Computer Graphics”, Int. J. Appl. Inf. Manag., vol. 3, no. 2, pp. 82–90, Jul. 2023.

[3] Kang Lixue, Liu Yang, Dong Zhenghong. Design and simulation of switching power supply based on Multisim [J]. *Civil Aviation Management,* vol. 1, no. 1, pp. 68-70, 2013.

[4] Dianingrum, M., Hermanto, N., & Rifa'i, M. (2019). Expert System for Simulation of Pest and Disease Diagnosis in Onion Plant Using Putty Shafer Method and Rule-Based Approach. International Journal of Informatics and Information Systems, 2(1), 1-8. doi:https://doi.org/10.47738/ijiis.v2i1.8

[5] Li Lin. Calculation and simulation of switching power supply ripple-steady-state ripple [J]. *Today Electronics,* vol. 1, no. 1, pp. 9-12, 2014.

[6] B. Hu and J. Yu, “Elevator Group scheduling by improved Dayan Particle Swarm algorithm in Computer Cloud Computing Environment,” Journal of Applied Data Sciences, vol. 3, no. 2, pp. 72–81, 2022. doi:10.47738/jads.v3i2.60

[7] Huang Heying. Circuit design and research based on Multisim electronic simulation software [J]. *Modern Electronic Technology,* vol. 1, no. 1, pp. 41-44, 2012.

[8] Y. Xu, X. Wei, and C. Dong, “The mechanical and system design of finger training rehabilitation device based on speech recognition,” Journal of Applied Data Sciences, vol. 3, no. 2, pp. 60–65, 2022. doi:10.47738/jads.v3i2.58

[9] Fang Yujie, Su Binghua, Hang Lingxia. Research on Ripple Suppression of Switching Power Supply [J]. *Modern Electronic Technology,* vol. 1, no. 1, pp. 144-146, 2012.

[10] I. Yusuf, D. Purwana, and A. D. Buchdadi, “The influence of interpersonal communication, universal-diverse orientation (Udo), and self-efficacy on the quality of administrative services at State University of Jakarta,” International Journal for Applied Information Management, vol. 2, no. 2, pp. 97–105, 2021. doi:10.47738/ijaim.v2i2.32

[11] Yang Jingtao. Multisim software in passive filter device design and simulation research application [J]. *Science and Technology Innovation and Application,* vol. 1, no. 1, pp. 85-85, 2017.

[12] Q. Yuan and W. Yin, "Torque Ripple Suppression of High-Speed Permanent Magnet Synchronous Motors with the Harmonic Injection," 2018 Asia-Pacific Magnetic Recording Conference (APMRC), Shanghai, China, 2018, pp. 1-2, doi: 10.1109/APMRC.2018.8601063.

[13] Zhang Bing, snow study switching power supply ripple suppression [J]. *Yibin Technology,* vol. 1, no. 1, pp. 29-31, 2011.

[14] X. Zhao, Y. Yu, G. Zhao and S. Wei, "The Application of Harmonic Suppression Method Based on Ripple Prediction Model in Underwater Propulsion System," 2019 Chinese Control And Decision Conference (CCDC), Nanchang, China, 2019, pp. 242-246, doi: 10.1109/CCDC.2019.8833215.

[15] Wang Yan. Optimizing the classroom teaching of "Computer Circuit" based on Multisim simulation software [J]. *Western China Science and Technology,* vol. 1, no. 1, pp. 70+84-85, 2012.

[16] H. Yansong, Y. Dejun and P. Yunhao, "Research on Torque Ripple Suppression Strategy of PMSM under Variable Speed Condition," 2021 6th Asia Conference on Power and Electrical Engineering (ACPEE), Chongqing, China, 2021, pp. 898-904, doi: 10.1109/ACPEE51499.2021.9437005.

[17] Zheng Wei, Huang Ping. Simulation research of electronic circuit design based on Multisim [J]. *China Electronic Commerce,* vol. 1, no. 1, pp. 95, 2013.

[18] M. Huang, Y. Deng and H. Li, "Fractional-Order Based Resonant Controller for Torque Ripple Suppression of Permanent Magnet Synchronous Motors," 2021 IEEE International Conference on Mechatronics (ICM), Kashiwa, Japan, 2021, pp. 1-6, doi: 10.1109/ICM46511.2021.9385650.

[19] Chen Qi Hong Based on VC ++ simulation software of computer-based interlocking system [J]. *Automation and Instrumentation,* vol. 1, no. 1, pp. 182-185, 2016.

[20] J. Li, X. Li, Y. Zhang and X. Gui, "Torque Ripple Suppression Based Fast Harmonics Decomposition," 2019 22nd International Conference on Electrical Machines and Systems (ICEMS), Harbin, China, 2019, pp. 1-6, doi: 10.1109/ICEMS.2019.8922499.