

---

# **PSCAD Simulation in a Power Electronics Application Course**

---

by

**Liping Guo**

**lguo@niu.edu**

**Department of Technology  
Northern Illinois University**

## **Abstract**

This paper introduces computer simulation studies of basic power electronics circuits using a Power System Computer Aided Design (PSCAD) software package in a junior/senior level power electronics course in a four-year electrical engineering technology institution. The Power Electronics Applications course mainly covers the following topics: Operation and characteristics of switching devices, rectifiers, dc-dc converter and inverters. Simulation of the power electronics circuits provides students an opportunity to observe the circuit parameters and signal waveforms before actual circuit wiring is completed. Case studies include a half-wave input-output rectifier, a buck converter and a full-bridge inverter.

## **I. Introduction**

In this paper, a Power System Computer Aided Design (PSCAD) software package is used to accompany the teaching of a power electronics application course. PSCAD is a software package for power system simulations developed by Manitoba HVDC Research Centre<sup>1</sup>. Free trial and student versions are offered on the web site at [www.pscad.com](http://www.pscad.com). This is especially attractive for students because they are able to download the software on their personal computer and work on the simulation before or after the laboratory. PSCAD results are solved as instantaneous values in time, but can be converted to phasor magnitudes and angles via built-in transducer and measurement functions. This is very similar to how real system measurements are performed, such as rms meters and Fast Fourier Transform (FFT) spectrum analyzers. The PSCAD simulation tool can therefore duplicate the response of power electronics circuits at all frequencies. Users are able to select time steps ranging from nanoseconds to seconds<sup>2</sup>. In comparison with other simulation software such as PSpice, PSCAD is specifically targeted to simulate power systems and power electronics circuits. On the other hand, PSpice is a general purpose analog and mixed-signal circuit simulator used to verify circuit designs and to predict circuit behavior.

## **II. Case Studies**

Case studies discussed in this paper include a half-wave rectifier, a buck converter and a full-bridge inverter. Before the case studies, students were given a tutorial of PSCAD in order to

become familiar with the software package. The tutorial includes the following topics: (1) How to locate components in the Master Library; (2) How to configure components; (3) How to assemble components and plot the voltage and current waveforms; (4) How to run the project file and obtain simulation results. A simple voltage divider circuit was built, and output voltage and current were plotted and observed. After the tutorial, students became familiar with the PSCAD software and were ready to proceed to simulate more advanced circuits.

### Case study I – Half-wave rectifiers

The first case study is an analysis of a half-wave rectifier circuit. During the lecture, students were introduced as to how a half-wave rectifier works with a pure resistive load and resistive-inductive load. However, without an observation of the voltage and current waveforms at AC and DC terminals, the topic can not be fully comprehended.

In a laboratory project, students were asked to simulate a half-wave rectifier with a pure resistive load. A PSCAD schematic of a half-wave rectifier with a resistive load is shown in Figure 1. The source is a sinusoid of 70.7 V rms at a frequency of 60 Hz. The load resistor is 1  $\Omega$ . Meters were connected at various points of the circuit, where  $E_s$  is the source voltage,  $E_d$  is the voltage across the diode,  $I_{load}$  is the current through the load resistor, and  $E_r$  is the output voltage as seen in Figure 1. After the schematic was built successfully, simulation was run. Simulation results were shown in Figure 2. From the simulation results, students observed that for the positive half cycle of the source, the diode is forward-biased and the output voltage is equal to the input voltage. While for the negative half cycle of the source, the diode is reverse-biased, making the output voltage and current equal to zero.

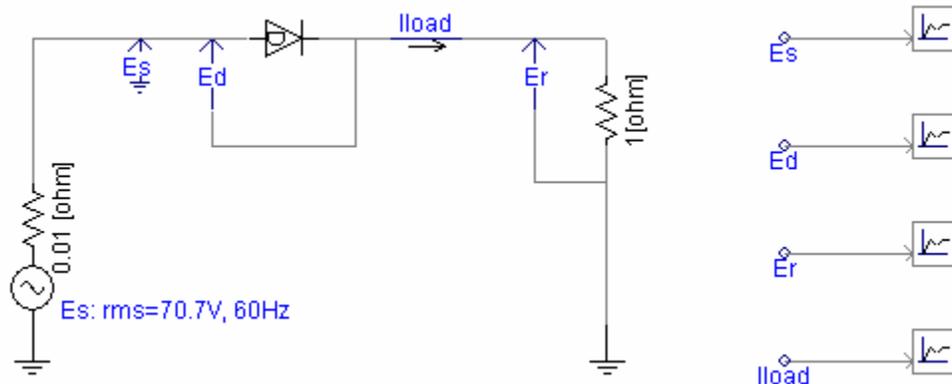


Fig. 1 Schematics of a half-wave rectifier with resistive load

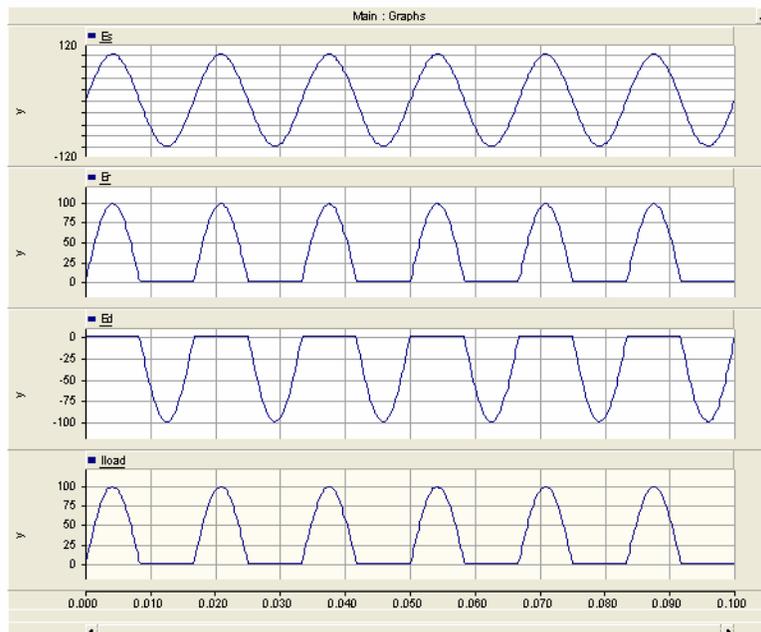


Fig. 2 Simulation results of half-wave rectifier with resistive load

After successfully simulating the half-wave rectifier with resistive load, an inductor of 2 mH was added in series with the load resistor to represent a simple motor load. Simulation results of the input voltage  $E_s$  and the output voltage  $E_r$  are plotted in Figure 3. From this result, students observed that after adding inductance at the load, the voltage across the load resistor lagged the input voltage. This is because the load inductance makes the output current lag the output voltage. Students were asked to compare the difference of the voltage and current waveforms of the half-wave rectifier with resistive load and resistive-inductive load, and explain their justifications orally and in their written laboratory reports.

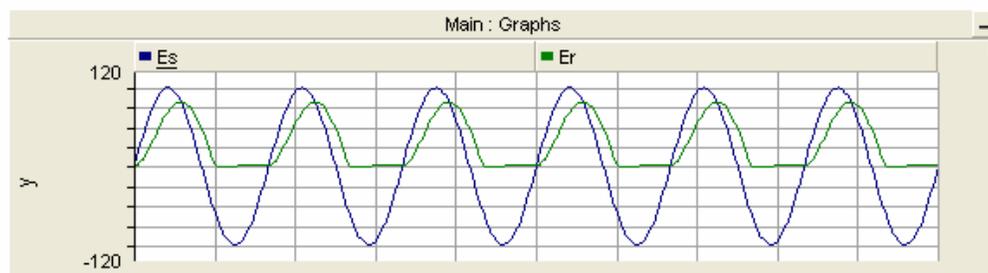


Fig. 3 Simulation results of half-wave rectifier with resistive-inductive load

Next, the inductor value was decreased from 2 mH to 0.5 mH, and increased to 10 mH, and the simulation was repeated with different values of inductance. Students observed that by decreasing the inductance value, the voltage across the load resistor lags the input voltage less, while by increasing the inductance value, the lagging effect is more obvious. In the lecture, in order to show quantitative results of a half-wave rectifier with resistive-inductive load, the solution needs to be obtained by expressing the output current and voltage as the sum of the forced response and natural response. Some students may have difficulty understanding the mathematical analysis. From this simulation in the laboratory, students were able to obtain direct

observation of the waveform while changing the value of the components. This greatly complements the lecture. The concept of lagging effect of the inductor is well understood from the PSCAD simulation.

### Case study II – Buck converters

The second case study is a buck converter that is one of the basic topologies of dc-dc converters. It is very important for students to understand how a buck converter works before proceeding to other topologies of dc-dc converters. The objective of this laboratory is to observe operation of a buck converter, verify the ratio of output voltage over input voltage, and understand the boundary between continuous and discontinuous conduction modes. In this case study, students simulated a buck converter with the following circuit parameters:

Input voltage:  $V_s = 20$  V,  
Output voltage:  $V_o = 12$  V,  
Switching frequency:  $f_s = 20$  kHz,  
Inductance:  $L = 800$   $\mu$ H,  
Capacitance:  $C = 50$   $\mu$ F,  
Load resistance:  $R = 10$   $\Omega$ .

The circuit diagram of a buck converter to be studied is shown in Figure 4. The control signal for the switch is generated by comparing a triangle waveform and a constant input set value. The duty cycle of the PWM signal is manually adjusted by changing the constant value. Simulation results at steady state when the duty cycle is 60% is shown in Figure 5, where  $V_o$  is the output voltage, and  $i_L$  is the inductor current. From the waveform of the inductor current, students observed that the buck converter operated in the continuous conduction mode.

Next, students reduce the duty cycle to 40%, and observe that the average (dc) output voltage is reduced to 8 V. Students then change the duty cycle back to 60%, and reduce the inductor value from 800  $\mu$ H to 50  $\mu$ H. The simulation result is depicted in Figure 6, where it is seen that the average (dc) output voltage has risen from 12 V to 14.4 V. Students observed that the buck converter operates in the discontinuous conduction mode when the inductance is 50  $\mu$ H. The minimum inductance required for continuous current operation of the buck converter is given as follows:

$$L_{\min} = \frac{(1-D)R}{2f} = 100 \mu\text{H}$$
, where  $f$  is the switching frequency,  $D$  is the duty cycle, and  $R$  is the load resistance.

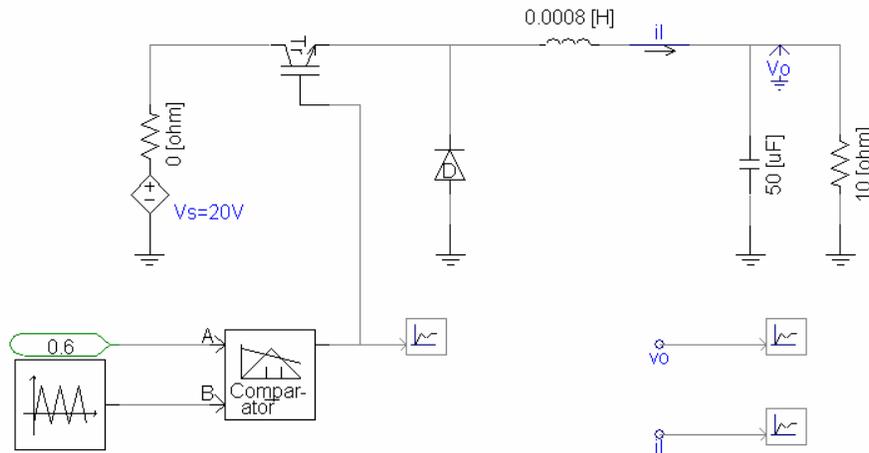


Fig. 4 Schematic of a buck converter

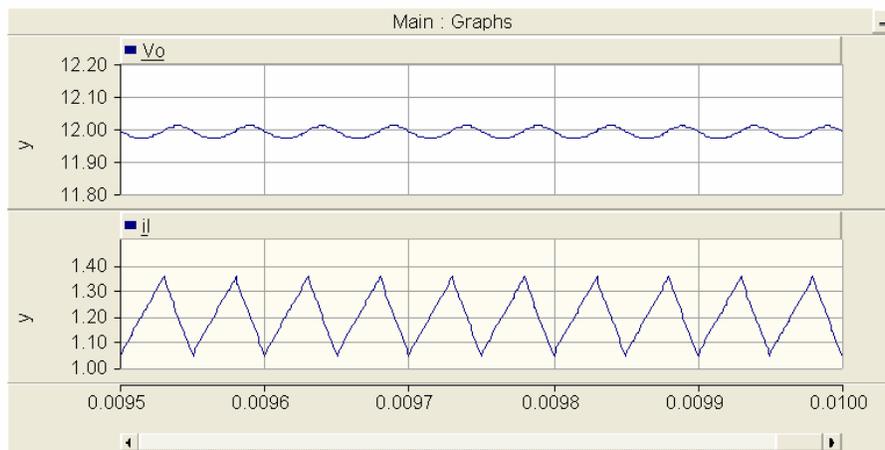


Fig. 5 Simulation results of a buck converter at steady state in continuous conduction mode

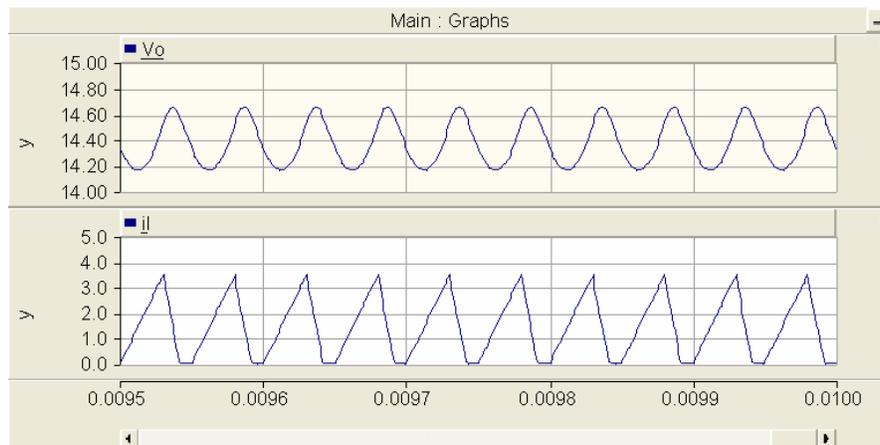


Fig. 6 Simulation results of a buck converter at steady state in discontinuous conduction mode

Since the inductance of 50  $\mu\text{H}$  is less than the minimum inductance, the buck converter operates in discontinuous conduction mode. This experiment verifies that dc-dc converter will operate in discontinuous conduction mode when the inductor value is less than the critical inductance<sup>4</sup>.

In the laboratory, each student designed a buck converter with a specific input voltage, output voltage, load, and voltage and current ripples. At the end of this laboratory, students proceeded to verify their own buck converter design.

### Case study III – Full-bridge inverters

Full-bridge inverter is a basic circuit to study DC to AC conversions<sup>5</sup>. In this laboratory, a full-bridge inverter is controlled by a set of square wave inputs. The objective of this case study is for students to understand how to control a full-bridge inverter using square waves, and observe Fourier series and Total Harmonic Distortion (THD) of the output voltage and current<sup>6</sup>. The PSCAD schematic of the circuit is shown in Figure 7. The circuit has the following parameters:

Input voltage:  $V_s=100\text{ V}$ .

Load:  $R=10\ \Omega$ ,  $L=2.5\text{mH}$ .

Switching frequency:  $f_s=10\text{ kHz}$ .

Transistors S1 and S2 will be turned ON for the first half of the switching period, and turned OFF for the second half of the switching period. On the other hand, transistors S3 and S4 will be turned OFF for the first half of the switching period, and turned ON for the second half of the switching period. C1 and C2 are square waves with opposite phase. Transistors S1 and S2 are controlled by C1, and transistors S3 and S4 are controlled by C2. After observing circuit operation, students connect the output voltage and current to Fast Fourier Transform (FFT) blocks.

The FFT block determines the harmonic magnitude and phase of the input signal as a function of time. The frequency spectrums of the output voltage and current are shown in Figures 8 and 9, respectively, where the magnitudes of their first seven harmonics are displayed.

The output of the FFT block is connected to a THD block which determines the THD of the load voltage and current. From simulation, the THD of the output voltage is determined as 41.69%, and the THD of the output current is found as 11.88%. Student discussed the observed THD values with allowed limits of THD values determined by the IEEE 519 Standards<sup>7</sup>.

During the lecture, some students had difficulty understanding Fourier series and FFT because it is a relatively abstract concept. From this experiment, students visualized amplitude of different harmonics and understood the concept of Fourier series. Similarly a full-bridge inverter circuit with bipolar switching was also simulated. FFT frequency spectrum of output voltage and current waveforms, and THD for voltage and current were also observed for the full-bridge inverter circuit studied.

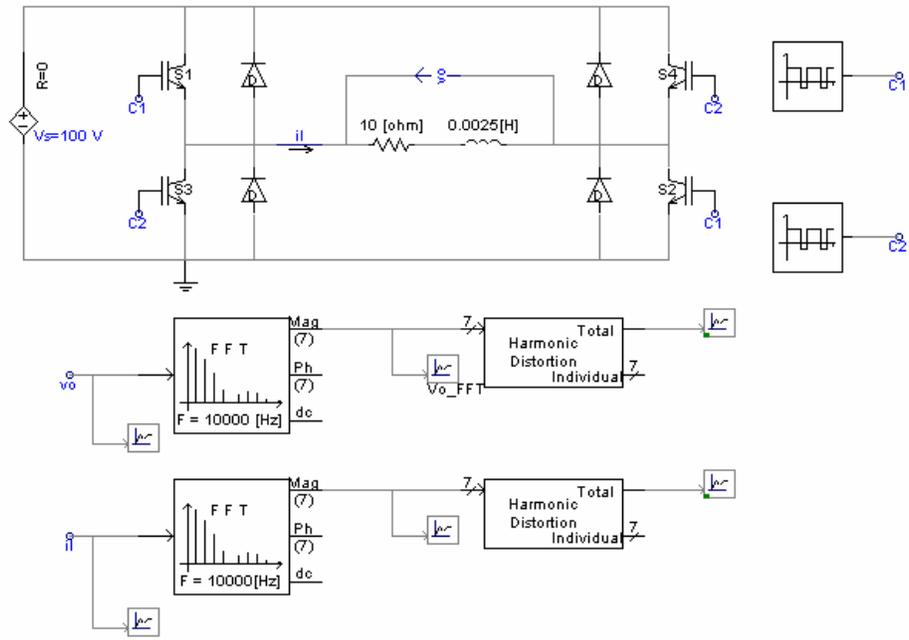


Fig. 7 Schematic of a full-bridge inverter

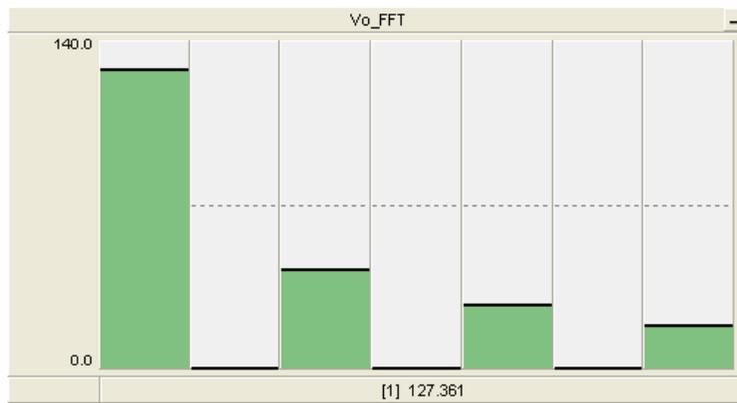


Fig. 8 Frequency spectrum of output voltage

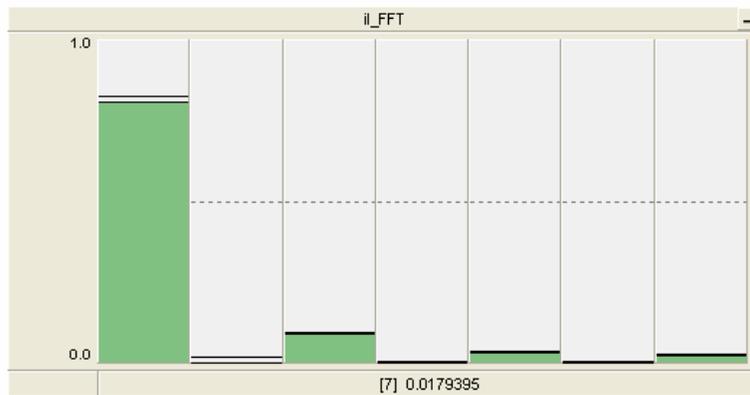


Fig. 9 Frequency spectrum of output current

### III. Course Outcome Assessment

Results from the University of Northern Iowa student evaluation of teaching at the end of the power electronics course indicate the following assessment: 71.4% of the students state that the instructor always creates an atmosphere conducive to learning and 28.6% of the students state that the instructor frequently creates an atmosphere conducive to learning. 71.4% of the students state that the instructor always explains the material appropriately, and 28.6% of the students state that the instructor frequently explains the material appropriately. 71.4% of the students state that the instructor always sets high but reasonable standards for students, and 28.6% of the students state that the instructor frequently sets high but reasonable standards for students. 71.4% of the students state that the instructor always communicates effectively, and 28.6% of the students state that the instructor frequently communicates effectively. Some comments of students in the assessment include “The instructor knows the material very well and she can teach it to us effectively.”, “The instructor explains thing well and goes at a pace that everyone can follow.”, and “The instructor is very willing to answer our questions and work with us.”

Overall, 71.4% of the students are highly satisfied with their learning in the course, and 28.6% of the students are satisfied with their learning in the course. 71.4% of the students rate the overall teaching effectiveness of the instructor in the course to be highly effective, and 28.6% of the students rate the overall teaching effectiveness of the instructor in the course to be effective.

### IV. Conclusions

The PSCAD simulation software tool provides students in the undergraduate power electronics classes an opportunity to study expensive power electronics circuits in a cost-effective manner<sup>8,11,12</sup>. Case studies presented in this paper include a half-wave rectifier, a buck converter and a full-bridge inverter. After successful simulation, students built a prototype circuit in the laboratory. Feedback from students is very encouraging. In the future, students may be able to use the software to investigate new power electronics circuits and move towards control concepts. Instructor of this class is developing laboratory projects in conjunction with the simulated power electronics circuits discussed in this paper.

### References

1. PSCAD/EMTDC User's Manual: Ver.4.2, Manitoba HVDC Research Centre, 2005.
2. F. Jurado, N. Acero, et. al., “Using Various Computer Tools In Electrical Transients Studies”, 30<sup>th</sup> ASEE/IEEE Frontiers in Education Conference, October 2000, Kansas City, MO.
3. T. S. Sidhu, M.S. Sachdev, “Laboratory Setup for Teaching and Research in Computer-based Power System Protection”, 1995 Int. Conference on Energy Management and Power Delivery, Nov 1995, Vol. 2, pp. 474 – 479.
4. D. W. Hart, *Introduction to Power Electronics*, Prentice Hall, New Jersey, 1997.
5. N. Mohan, *First Course on Power Electronics*, MNPERE, Minneapolis, 2005.
6. H. L. Hess, “Power Electronics Instruction: Topics, Curricula, and Trends”, Proceedings of American Society of Engineering Education Annual Conference, 1997.
7. IEEE Std 519 Working Group Website, <http://grouper.ieee.org/groups/519/index.html>
8. O. Nayak, S. Santoso, and P. Buchanan, “Power electronics spark new simulation challenges”, IEEE Computer Applications in Power, Oct 2002, Vol. 15, Issue 4, pp. 37-44.
9. S. Krishnamurthy, V.V.Sastry, V.Ajjarapu, “An Innovative Rapid Prototyping Tool for Power Electronic Circuits”, Proceedings of the 2003 American Society for Engineering Education Annual Conference & Exposition.

10. O. Anaya-Lara, E. Acha, "Modeling and analysis of custom power systems by PSCAD/EMTDC", IEEE Transaction of Power Delivery, Jan 2002, Vol. 17, Issue 1, pp. 266-272.
11. D. W. Hart, "Circuit simulation as an aid in teaching the principles of power electronics", IEEE Transaction on Education, Feb. 1993, Vol. 36, Issue 1, pp. 10-16.
12. I. Chamas, M. A. E. Nokali, "Automated PSpice simulation as an effective design tool in teaching power electronics", IEEE Transactions on Education, Aug. 2004, Vol. 47, Issue 3, pp. 415-421.