The Application of PSIM & Matlab/Simulink in Teaching of Power Electronics Courses

Sameer Khader[1] and Akram Abu-aisheh[2]

1Palestine PolytechnicUniversity, Hebron, Palestine
[2]University of Hartford, West Harford, CT, USA

Abstract—**This paper presents a comparison analysis between two engineering software platforms, Matlab/Simulink & PSIM, which are used as major educational tools in teaching of power electronics and electrical drive courses, in additional to conducted research in these fields. The comparison analysis is based on studying the design simplicity of the module, time consumed in building of the module, accuracy, functionality, simulation time, and the acceptability of obtained results. Various power electronic simulation circuits are illustrated and the results are processed and displayed.**

The simulation results states that Matlab/Simulink is a suitable platform for control and regulation of simulation processes, in additional to its dominant role in conducting research tasks. Conversely, PSIM is dedicated to power electronic circuits and machine simulation tasks with fast and robust algorithms and suitable for educational purposes. It is recommended that both packages can be used in teaching power electronics courses.

Keywords-- Computer simulation, Plecs, PSIM, Simulink.

I. INTRODUCTION

Several software packages and platforms are used in the building of simulation models for educational purposes describing power electronics circuits [1 & 2]. The major issue when designing a software package is to achieve simplicity in design and processing of the simulated electronic circuits, while maintaining a high level of accuracy and user friendly graphic interface. These software packages include Matlab, PSIM, Simplorer, Pspice, Multisim, PLECS, etc.

In the past decade, power electronics circuits found widespread applications in Energy Conversion Systems, Industrial Automation, Mechatronics, renewable energy systems, and Transport sector [4-7] due to rapid development of switching devices and control techniques. Therefore intensive progress in building and developing various software platforms [8-10] with a certain degree of simplicity, user friendliness and accuracy that can be used for educational and research purposes is essential.

Furthermore, teaching power electronics courses require introducing additional tools in the education process when explaining and analyzing circuit behaviors [11&12]. Class surveys and student interviews indicate that applying software packages in such courses resulted in better teaching outcomes and increased students professionalism in terms of implementing their class assignments, projects, and research tasks.

II. MODELLING

This paper describes modeling of various power electronics circuits by using Matlab/Simulink and PSIM with respect to output quantities, switch parameters and signal behavior for the following circuits:

- Single-Phase Uncontrolled Rectifier;
- Three-Phase Controlled Rectifier;

Other circuits such as Chopper circuits, Inverters, AC voltage controllers will be the subject of future work and will not be further discussed in present work.

A. Single-Phase Uncontrolled Rectifier

The circuit configurations for both simulation platforms are displayed in Fig.1 and Fig.2 for single-phase uncontrolled rectifier energizing resistive-inductive load with data: source voltage of Vs=110V; frequency of 50 Hz, load resistance and inductance respectively $R_L=10\Omega$, L=100mH), while simulation results in the form of main circuit waveforms are illustrated in fig. 3 for both simulation tools .
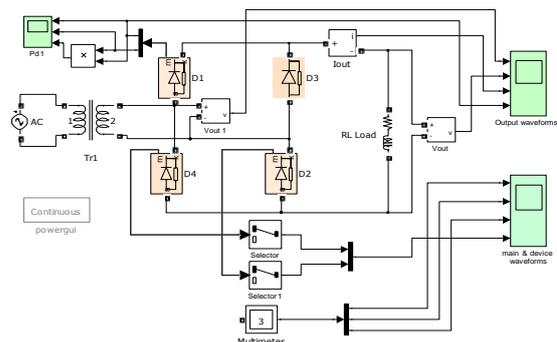

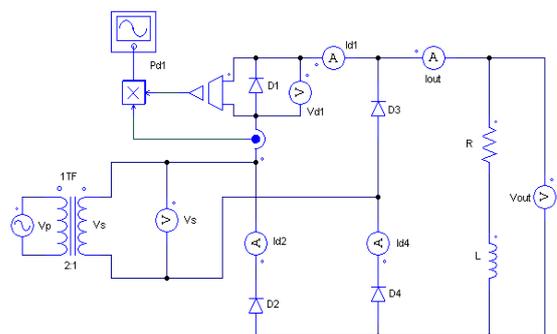Figure 1.   Simulink circuit of single-phase rectifier

Figure 2.   PSIM circuit of single-phase rectifier.

As illustrated in Fig.3, output voltage, current and device current to some extent have similar behaviors for both simulation tools under the same circumstances as shown in Fig.4 for the whole control range of the load impedance presented by the loading angle θ (0≤ θ ≤90°).

Therefore, looking for the differences between both systems requires determining the main circuit parameters as follows:

The output voltage can be expressed in Fourier series as follows:

$$Vo(\omega t) = \sum_{v=1}^{\infty} Vmv \cdot \sin(v \cdot \omega t) \quad (1)$$

Where V_{mv} is the magnitude of v-ti harmonic of the output voltage, and ω is the angular frequency.

The average and root mean square V_{av} and V_{rms} respectively of (1) can be expressed as :

$$Vav = \frac{1}{\pi} \int_{0}^{\pi+\vartheta} Vm \cdot \sin(\omega t) \cdot d(\omega t)$$

$$Vrms = \sqrt{\frac{1}{\pi} \int_{0}^{\pi+\theta} [Vm \cdot \sin(\omega t)]^2 \cdot d(\omega t)} \quad (2)$$

where V_m is the magnitude of supply voltage.

Additional statistical parameters such as the median, mean value and standard deviation [13] can be proposed in order to observe how the output voltage behaves as the load character changes in terms of varying the circuit inductance L, which in turn causes impedance angle to change.

The median of the output voltage is:

$$Median(V) = \arg_{Vout\min}\left(\sum_{Vout \in V} |V_{out} - V_{out\min}| \right) \quad (3)$$

where V_{outmin} and V_{out} are minimum and other greater than values of rms voltages respectively.

The mean value of the rms output voltage along the control range is:

$$Vmean = \frac{1}{N} \sum_{v}^{N} Vrms(v) \quad (4)$$

where N is the maximum number of simulation points.

The standard deviation σ of the output rms voltage is:

$$\sigma = \frac{\sqrt{\sum_{n}^{N} (V_{rms(n)} - V_{mean})^2}}{N} \quad (5)$$

Taking into account the illustrated results and obtained analytical results for both simulation platforms, comparison analysis is conducted with results presented in table 1, where it is clearly shown that Simulink has advanced features in simulation functionality and data tracking, while PSIM has advanced features in building the module, data extraction and graphical display.

a)　Simulink platform
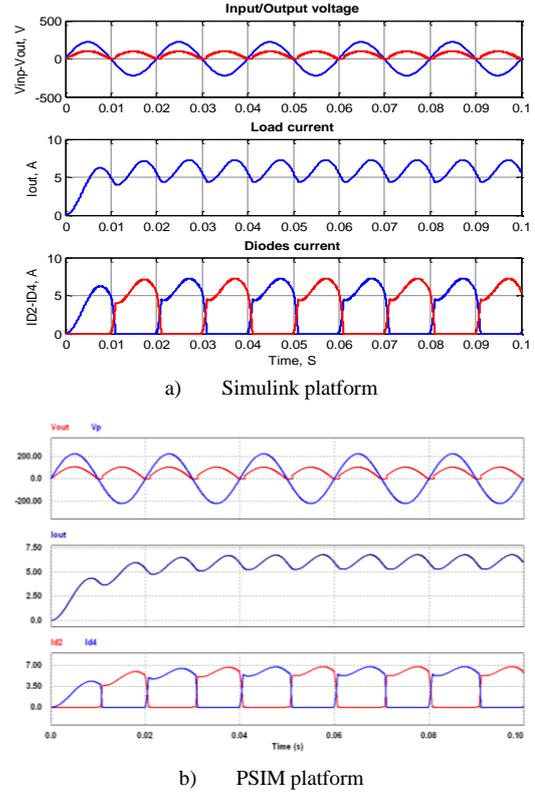

b)　PSIM platform

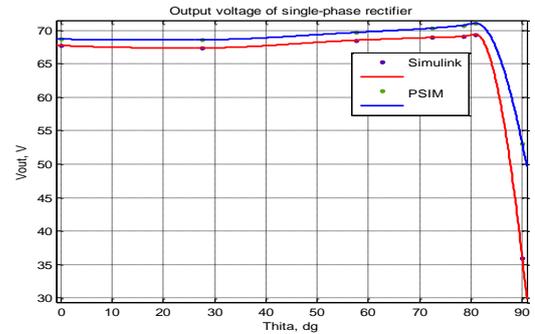Figure 3.　Main circuit waveforms of single-phase rectifier.

Figure 4.　Rectified output voltage vs. load angle θ .

TABLE I.
COMPARISON ANALYSIS FOR SINGLE-PHASE RECTIFIER.

Parameter	Simulink	PSIM
Building the model	Complicated	Simple
Simulation time for 20 cycles)	Large time	Negligible time
Graph display	Needs additional interference, but with large features	Simple and directly displayed with less features.
Functionality	Multifunctional	Less functional
Data flow and extraction	Complicated	Simple
Research capability	Advanced capabilities	Less capabilities
Median of the voltage	66.52 V	69.69V
Mean value of the voltage	63.86 V	67.46V
Standard deviation	12.31V	6.442V

The obtained analytical results states that PSIM has standard deviation of σ=6.422 which indicates that the obtained results are clustered around the mean value.

While Simulink results have better Gaussian distribution σ=12.31 as θ varies form 0 to 90°, which means better control range. These conclusions are still valid for three-phase uncontrolled rectifiers.

The occurred transient behaviors during the simulation processes depends on the load character and circuit components that are frequently observed in more complicated power electronic circuits such as Three-phase controlled rectifiers, Inverters, and DC Choppers. Controlled rectifier circuit configuration will be discussed in hereinafter.

B. Three-phase controlled rectifier

Figure 5 illustrates Three-phase controlled rectifier with six Silicon Controlled Rectifiers (SCRs) connected in such a way to form three-phase full bridge rectifier. Figure 6 illustrates the same circuit built in PSIM environment. Close loop control is applied in order to regulate the circuit performances by controlling the SCR's firing instants.

Comparison analysis between both circuits is conducted in with respect control characteristic (normalizing performance) presenting how the output voltage varies by regulating the firing angle α of controlled SCR.
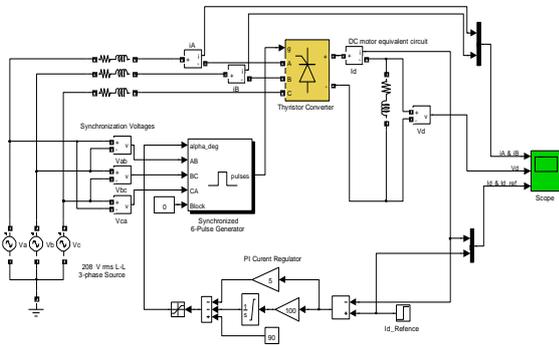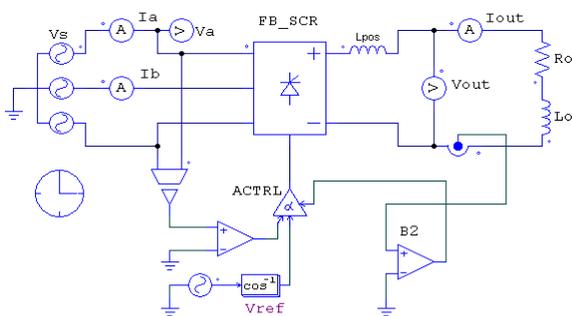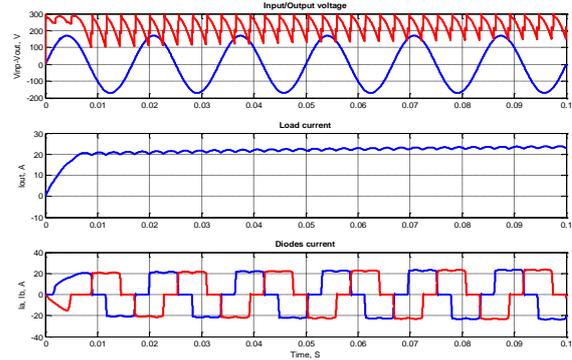
Figure 5.   Simulink circuit of Three-phase rectifier.

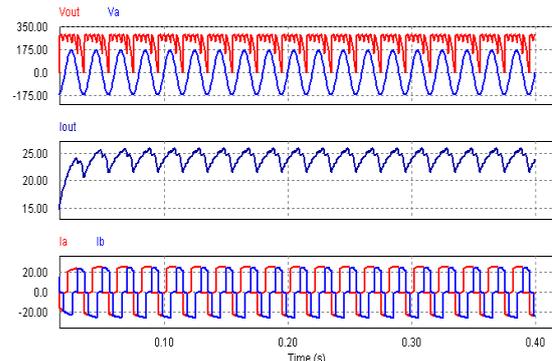Figure 6.   PSIM circuit of three-phase rectifier.

Figure 7 illustrates the instantaneous circuit performances for both platforms at g iven loading character . The obtained results are displayed in Fig.8 for various values of load inductance (L=0, 50 & 150mH), while the firing angle α is varied within the range 0≤ α ≤ 120°.

These performances are built with open loop control circuit in order to study the natural behaviors of proposed simulation platforms. The simulation results indicate that standard deviation of PSIM varies within the range of 3.865 ≤ σ ≤ 4.50.

While Simulink platform permits a larger standard deviation change within the range of 51.6 ≤ σ ≤ 100, which is closed to normal Gaussian distribution, and permits wide control range and better output voltage regulation.

a)   Simulink platform.

b)   PSIM platform.

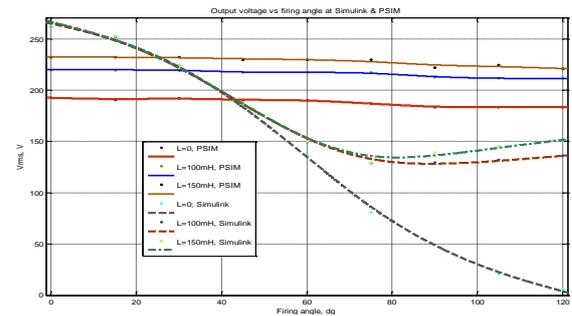Figure 7.   Main circuit waveforms of Three-phase rectifier.

Figure 8.   Output voltage vs. impedance angle at both simulation platforms.

III.   CO-SIMULATION PRINCIPLES

The conducted analysis in previous sections indicates that each of described platforms has its advantages and disadvantages. Therefore, a co-simulation platform capable of developing new functionalities for coupling PSIM software and Matlab/Simulink that combines their advantages will be designed for control and regulation of simulation processes with friendly used graphic interfaces.

PSIM is dedicated for power circuits and machine simulations with a fast and robust algorithm. The Sim-Coupler Module is an add-on module to the basic PSIM software. It provides interface between PSIM and Matlab/

Simulink for co-simulation. With the Sim-Coupler Module, part of power electronics circuit can be implemented and simulated in PSIM, and the rest of the system is processed in Matlab/ in Simulink is shown in Fig.9.
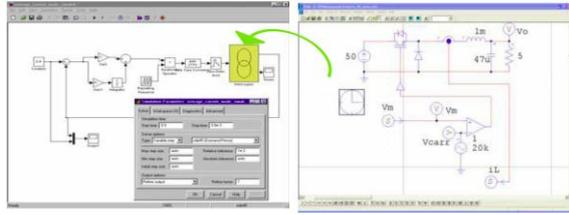

Figure 9.   Principle circuit of Sim-Coupler module.

Another practical example that found widespread application in power electronics courses is the use of PLECS circuit simulator, which combines power electronics circuits built in Simulink platform as shown in Fig.10.
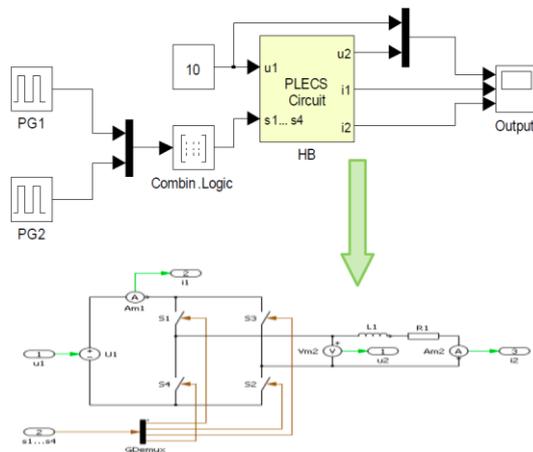

Figure 10. Combined circuit with Simulink and PLECS simulator.

PLECS circuit simulator [10] realizes such purposes when simulating power electronic circuits is required to be conducted in Simulink platform, in addition to its existing research features.

IV.   CONCLUSION

The following conclusion may be derived when using Simulink and PSIM platforms in teaching power electronics courses :

- Both software platforms have their strengths and weaknesses when being used as simulation tools in power electronics courses.
- With respect to the educational point of view, PSIM has the simplest approach on which to build and on which to extract the results rather than Matlab/ Simulink.
- With respect to the research point of view, Simulink has advanced tools with rich system functionality and simulation capability. The Simulink platform permits detailed study of the circuit behaviors and transient processes.
- It is recommended that both these two platforms be used in the teaching process, in order to give students the ability to use both simulation tools in building on and implementing their learning and research tasks.
- Applying Co-simulation tools by using Sim-Coupler enhances the course efficiency and outcomes.

REFERENCES

[1] Randall Shafer, "Fundamentals of Power Electronics with Matlab", Charles River Media, 1st ed., 2007.

[2] Armata J.M, Besrest B., Pauly R.,"PSIM/MATLAB co-simulation for electrical architectures global models", AES 2005

[3] Bülent Ertan H., "A Report on the Evaluation of PSIM and Some Other Possibilities for Power Electronics Circuit Simulation", 2007.

[4] Dmitry B., Raul R. Sam Y., "Simulation of Thyristor Operated Induction Generator by Simulink, Psim and Plecs", Proceedings of the 2008 International Conference on Electrical Machines.

[5] Derec A., "Induction Motor Speed Control by Stator Voltage Control", IEEE Transactions on Power Apparatus and Systems, vol. PAS-87, pp. 585-590, February 1968. doi:10.1109/ TPAS.1968.292056

[6] Abolfazl J., Mohammad R., Sohrab Kh., "Fuzzy Control of a Three-Phase Thyristorized Induction Motor", Transaction on Engineering, Computing and Technology, vol.VI, pp.274-277, December 2004.

[7] Gurkan Z ,Isic C.i, Muammer E. and Cuneyt B., "Performance Optimization of Induction Motors During Voltage-Controlled Soft Starting", IEEE Trans. on Energy Conversion, vol.19, pp. 278-288, June 2004. doi:10.1109/TEC.2003.822292

[8] Matlab/Simulink User's Guide , 2010b, www.mathwork.com

[9] PSIM, User;'s Guide, 2009, www.powersim.com

[10] PLECS user's guide, 2009: www.plexim.com

[11] Rashid M., "Power Electronics: Circuits, Devices and Applications", Prentice-Hall, 2004

[12] Daniel W. Hart, 2010 " Power Electronics", Mc Graw Hill, pp. 196-230.

[13] Oliver S., Carol G.,"Statistical Methods for Spatial Data Analysis", Chapman & Hall/ CRC Press Company, 2009.

AUTHORS

SAMEER KHADER is an Associate Professor of Electrical and Computer Engineering at Palestine Polytechnic University (PPU) – Palestine. He is a director of Power Electronics & Signal Processing Research Unit at PPU. Before that he has served for ten years as university academic provost, dean of College of Engineering & Technology at PPU. His research interests include Electrical machines, Power Electronic Converters, Renewable Energy Sources and Smart Grids, in addition to engineering education. Dr. Khader may be contacted at sameer@ppu.edu

AKRAM ABU-AISHEH is an Associate Professor of Electrical and Computer Engineering at the University of Hartford where he has served as the assistant chair of the Electrical and Computer Engineering Department and director of the electronic and computer engineering technology program for two years. Dr. Abu-aisheh has a doctorate in Optical Communications from the Florida Institute of Technology and Master of Science and Bachelor of Science degrees in Electrical Engineering from the University of Florida. Dr. Abu-aisheh may be contacted at ABUAISHEH@HARTFORD.EDU.