A Virtual Photovoltaic Power Systems Laboratory

Javier Muñoz

Dpto. de Electrónica, Automática e Informática
Industrial
Universidad Politécnica de Madrid (UPM)
Madrid, Spain
javier.munoz@upm.es

Pablo Díaz

Dpto. de Teoría de la Señal y Comunicaciones
Universidad de Alcalá
Alcalá de Henares, Spain
pablo.diaz@uah.es

Abstract—**This paper describes the design and implementation of a virtual laboratory for photovoltaic power systems based on the Cadence PSpice circuit simulator, which is intended to be a complement to the traditional physical laboratory in undergraduate photovoltaic engineering courses.**

Keywords-component; photovoltaic power systems; laboratories; simulation; engineering education; pspice;

I. INTRODUCTION

Virtual laboratories are recognized as an attractive complement to traditional engineering laboratories. Among other advantages, they do not require expensive and dedicated equipment, reduces the required time for instruction, and promote student involvement and learning [1]-[2]. This paper describes the design and implementation of a virtual laboratory for photovoltaic (PV) power systems based on the Cadence PSpice circuit simulator. For this purpose, a library of PV components has been developed, which allows simulating the behavior of PV components and systems.

The use of PSpice is very common in industry, research, and also for training and teaching electronics [3]-[5]. Besides, despite PSpice is a circuit simulator, it has been also used for simulating other kind of systems [6]-[8]. Its application for the simulation of PV systems begins in the nineties [9]-[10], and there is even a textbook on this field [11]. The simulation tool described here was former designed for research purposes [10], but it has been updated as educational software that can be integrated into PV systems engineering courses, which are every day more frequent in the university curricula.

Regarding its implementation, the virtual laboratory is intended to be a complement to the traditional physical laboratory. In this approach, students can be firstly tasked with virtual pre-laboratory assignments, which provide adequate background and preparation before performing actual experiments. Secondly, just in the physical laboratory, students can compare measurements and simulations during the experiments. And finally, the virtual laboratory can also support post-laboratory tasks allowing the repetition of experiments or the completion of unfinished laboratory exercises. Besides, in any of these steps, students can gain a deeper understanding of the matter through the design, simulation and analyses of their own experiments.

Among the prerequisites that should have evaluated before the adoption of computers for instructional use [12], accessibility and student computer background have received special attention. Regarding the former, a free version of the PSpice simulator is readily accessible to students without the need of purchase the license. Besides, the full license software is usually available in departments that teach courses on PV power systems. Regarding the second prerequisite, students of electrical engineering usually have learned PSpice before boarding the PV laboratory. Anyway, the required technical skills for the students are basic and they could be taught at the beginning of the course.

II. LABORATORY DESIGN

This section presents an overview of the virtual laboratory design and the simulation procedure. Later, several examples of application are shown.

A. Symbol library

A symbol library of PV components, so-called Sisifo, has been created in PSpice using a two-level hierarchical design and following a bottom-up method [13]. At the bottom level, PV components are described by mathematical models. Next, a schematic is created for each mathematical model by means of equivalent electric circuits using the Analog Behavioral Modeling feature provided in PSpice [14].

For example, the current-voltage (I-V) curve of a PV generator can be expressed with a good approximation by a single exponential model [15]:

$$I = I_{SC}\left(1 - \exp\frac{V + IR_S - V_{OC}}{N_S V_t}\right) \qquad (1)$$

Where I_{SC}, V_{OC}, R_S and N_{SM} are, respectively, the short-circuit current, the open-circuit voltage, the series resistance, and the number of cells connected in series in the PV generator. The thermal voltage V_t is equal to mkT_C/q (where m, k, T_C and q are, respectively, an ideality factor, the Boltzmann's constant, the temperature of solar cells, and the electron charge).

A simplified bottom schematic that describes the electric behavior of (1) is displayed in Fig. 1, which is composed of a current source (IL) equal to I_{SC}, a voltage-controlled current source (IDIODE), and a series resistor (RS). The schematic has two output terminals (VG+ and VG-), and two inputs (irradiance, G, and ambient temperature, T) that define the operating conditions of the PV generator, of which depend on the variables I_{SC}, V_{OC}, and V_t that appear in Eq. (1).
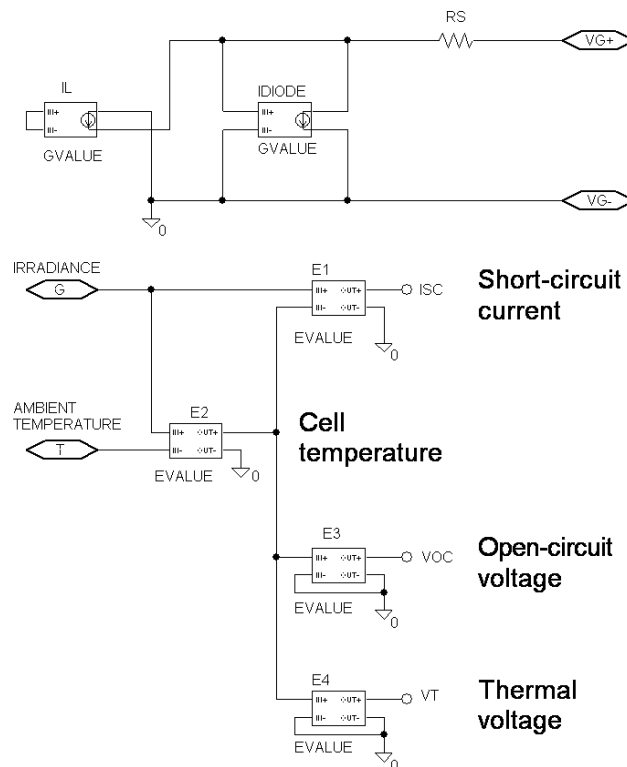

Figure 1. Bottom schematic for the PV generator.

At the top level, each schematic is associated with a hierarchical symbol that can be electrically connected in a photovoltaic system. For example, Fig. 2 displays the symbol of the PV generator associated with the schematic of Fig. 1. Each symbol has a set of attributes that define some parameters of the mathematical model, which can be modified by the students. For example, table 1 shows the attributes of the PV generator, which can be directly obtained from manufacturer's datasheet.

Besides the PV generator, other system components have been created from mathematical models using the Analog Behavioral Modeling: ON/OFF series charge regulators [15], lead-acid batteries [16] and DC constant power loads [17], whose detailed description can be found elsewhere [10]. For example, Fig. 3 shows all the Sisifo symbols arranged for simulating a Solar Home System.

The two-level hierarchical design of Sisifo allows using the simulation tool in beginning, intermediate and advanced laboratories. In the first ones, students can simulate experiments previously designed by teachers, as in Fig. 3. In intermediate laboratories, students can simulate their own experiments only at the system level considering each component as a "black-box" represented by its hierarchical symbol, without the necessity for a deep knowledge of the underlying mathematical model. In advances laboratories, students can move down in the hierarchical design in order to analyze the models or even fit model parameters as a function of the experimental measurements. Besides, advanced students can also create new models and symbols for enlarging Sisifo.
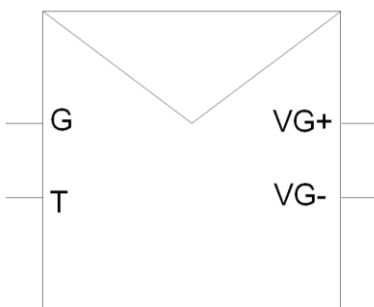
Figure 2. Hierarchical symbol for the PV generator.

TABLE I. ATTRIBUTES OF THE PV GENERATOR SYMBOL

Attribute	Definition	Units
ALFA	Short-circuit current temperature coefficient	$°C^{-1}$
BETA	Open-circuit voltage temperature coefficient	$°C^{-1}$
ISCM	Module short-circuit current	A
NOCT	Nominal Operation Cell Temperature	°C
NPC	Number of cells in parallel (per module)	
NPM	Number of modules in parallel	
NSC	Number of cells in series (per module)	
NSM	Number of modules in series	
PMM	Module maximum power	W
VOCM	Module open-circuit voltage	V

B. Simulation procedure

Once Sisifo has been created, it is added to the list of symbol libraries available in the Cadence Schematics environment, which allows designing and drawing PV circuits, as in Fig. 3, simulating them using PSpice, and analyzing the waveform of simulated data using a graphical interface called Probe [13].

Regarding the simulations, PSpice supports three different types of analysis that are very useful for the virtual PV systems laboratory:

- DC sweep analysis. It allows varying the value of a source (voltage or current) or a parameter through a specified range, which can be used for plotting the current-voltage curve of a PV component, for depicting the variation of a model variable as a function of a given parameter (e.g., I_{SC} versus G), and so on.

- Transient analysis. It allows the simulation of a PV circuit response as a function of time, from an initial state (time=0) to a specified time.

- Parametric analysis. It repeats one of the two previous analysis (DC sweep or transient) several times varying the value of a source, or a parameter, in each iteration.

Next section shows some examples of simulation with PSpice using these types of analysis.
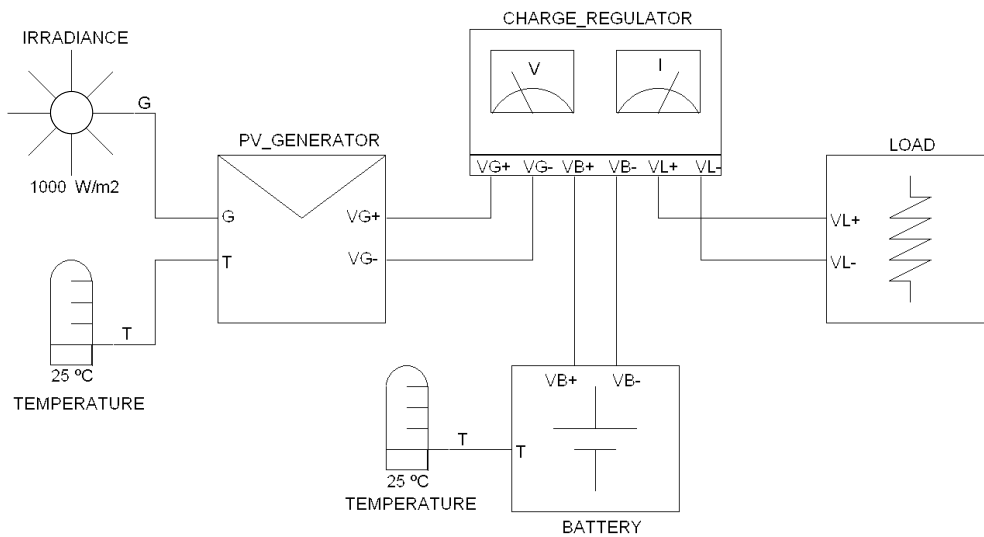

Figure 3. All Sisifo symbols arranged for simulating a Solar Home System.
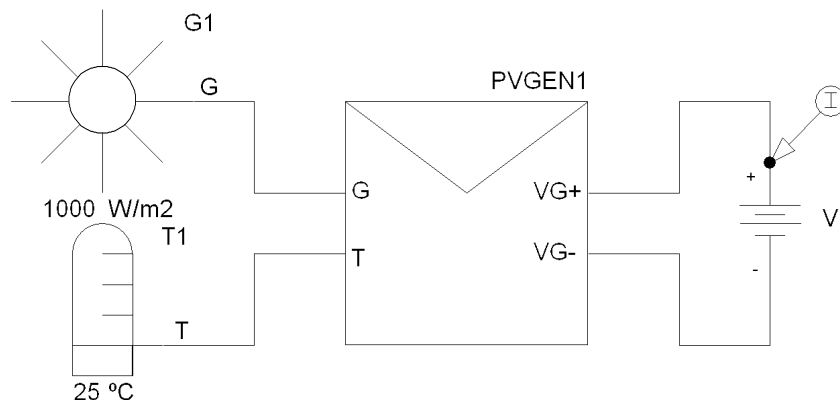

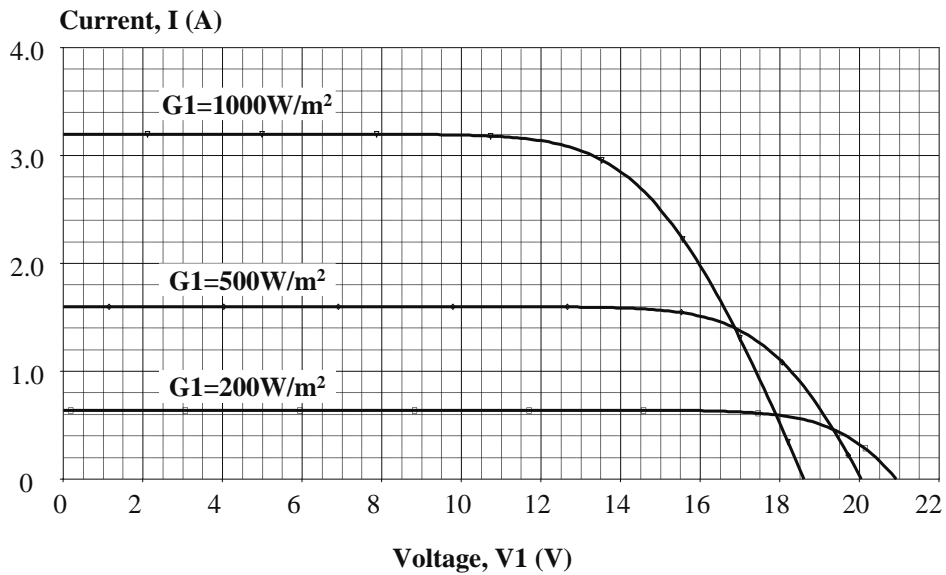Figure 4. Circuit for simulating current-voltage curves of a PV generator.

Current, I (A)

G1=1000W/m²

G1=500W/m²

G1=200W/m²

Voltage, V1 (V)

Figure 5. Current-voltage curves from the circuit displayed in Fig. 4.

C. Simulation examples

This section illustrates some of the possibilities of the virtual PV power system laboratory through several examples of simulation.

1) PV generator current-voltage curves

Figure 4 shows the circuit for simulating the current-voltage curve of a PV generator. Figure 5 displays the simulation results after performing a DC sweep analysis of the voltage source V1 (from 0 to 22V) and a parametric analysis for three irradiance values of G1 (200, 500 and 1000W/m²).

2) Battery voltage-time discharge curve

Figure 6-a shows the circuit for plotting the voltage-time curve of a 2V/100A·h lead-acid cell, which has been simulated for a constant discharge current of 10A using a transient analysis. Figure 6-b displays the simulated curve.
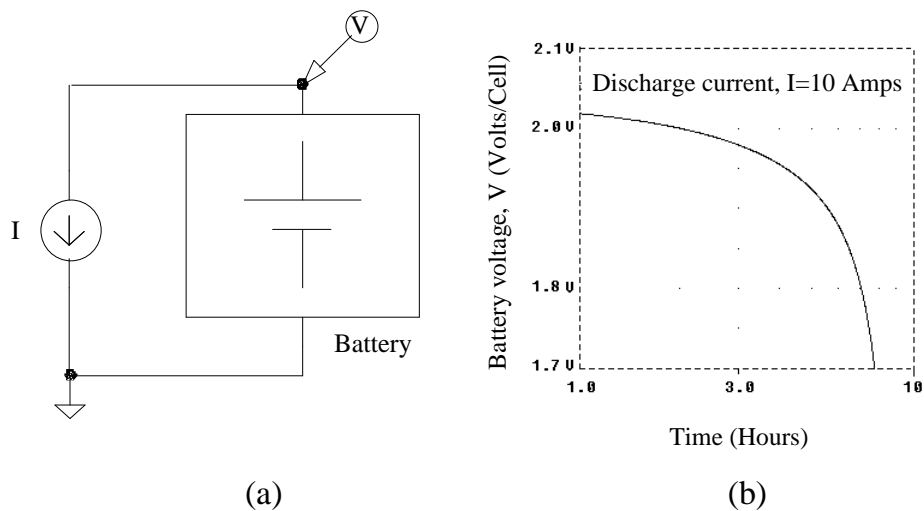

(a)

(b)

Figure 6.   (a) Circuit for simulating the discharge curve of a battery. (b) Voltage-time curve at a constant discharge current of 10A.
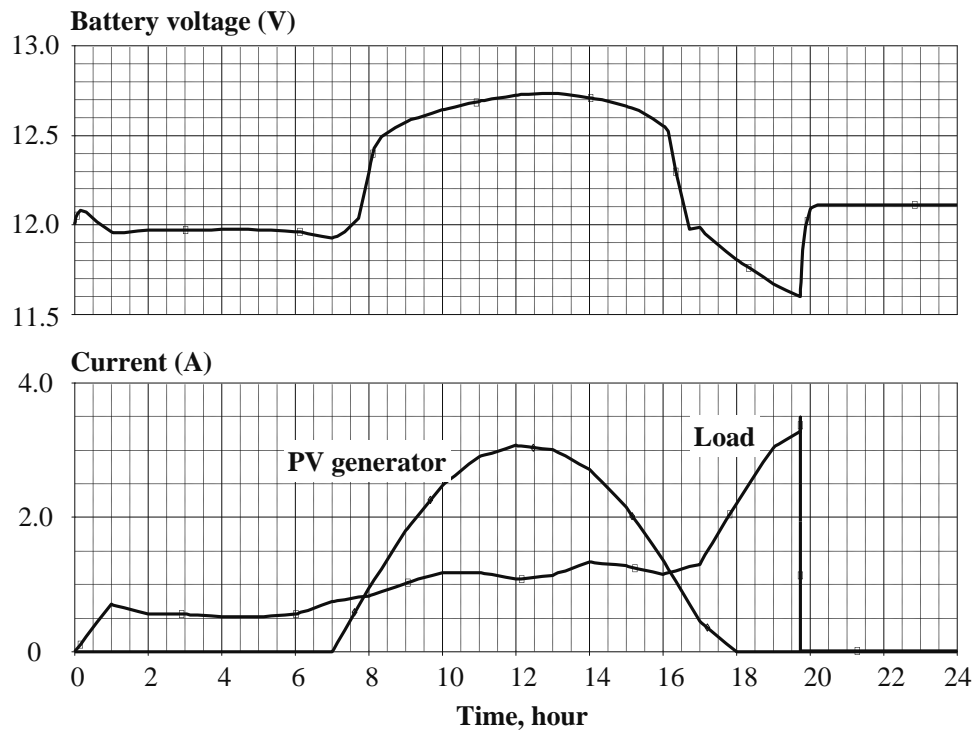
Figure 7. Evolution of the battery voltage, PV generator and load currents, after performing a transient analysis of the circuit displayed in Fig. 3.

3) Transient response of a photovoltaic system

Figure 7 displays the evolution of the battery voltage, as well as PV generator and load currents, after performing a transient analysis of the circuit shown in Fig. 3.

III. IMPLEMENTATION

In general, it is agreed that computer simulations can not completely replace physical experiments [18]. Following this approach, the virtual laboratory has been constructed as a complement to the traditional laboratory. This section describes the application of the virtual laboratory in supporting pre-laboratory and post-laboratory assignments. Despite the virtual laboratory could be also used during the physical laboratory session for comparing measurements and simulations, this application has been not considered here because the available time for hands-on experiments is usually scarce.

A. *Pre-laboratory assignments*

Exercises of simulation within the PV virtual laboratory can be introduced in pre-laboratory assignments required to students. Such exercises may provide a theoretical background to the behavior of PV power systems before performing hands-on experiments.

Pre-laboratory assignments may include not only a set of simulation tasks but also quizzes that should be given to the instructor at the start, or before, the physical laboratory session, which can be employed for identifying points of confusion or individual student problems.

Obviously, the designed of the set of simulations should be adapted to the student level. In their basic form, simulations may be performed on straightforward circuits previously designed by instructors, as in Fig. 4 and Fig. 6-a. Students should just push a button and the results of simulation appear in the screen, as in Fig. 5 and 4-b. Students can also modify some parameters of the circuit, for example changing the discharge current in the circuit of the Fig. 6-a, repeat the simulation, and analyze the results. A simple, but essential,

application of these circuits is analysing the current-voltage curve of a given component, for example, as in Fig. 5.

Pre-lab simulations can also reproduce the behavior of hands-on experiments, which may provide an adequate preparation to students before starting the physical laboratory. Anyway, students must be aware that simulations are based on simplified mathematical models, which have their limitations in terms of accuracy, and therefore they only provide an approximation to the behavior of real PV components and systems.

B. Post-laboratory assignments

The virtual laboratory can also support post-lab tasks allowing the completion or the repetition of hands-on experiments, which is one of the main limitations of the physical laboratory.

Simulations also allow performing experiments that are very difficult of carrying out in the physical laboratory because they require a lot of time. For example, the measurement of one battery discharge curve, as in Fig. 6-b, lasts several hours or days, depending on the battery capacity and on the discharge current. Other experiments are simply impossible of performing in the physical laboratory. For example, determining the variation of the battery capacity as a function of the discharge current requires the measurement of several discharge curves, which would last weeks or even months.

Besides, in both pre and post-laboratory assignments, students can gain a deeper understanding of the matter through the design, simulation and analyses of their own experiments. Finally, it is worth mentioning that simulations can also play the role of assistant for answering pre and post-laboratory quizzes, which can be linked to simulations for this purpose.

IV. CONCLUSIONS

A virtual photovoltaic power systems laboratory based on PSpice has been developed and presented in this paper with the aim of promoting and improving student learning in this engineering field. For this purpose, a PSpice symbol library of PV components, called Sisifo, has been constructed, which nowadays includes the following components: PV generators, ON/OFF charge regulators, lead-acid batteries and DC constant power loads. The current version of Sisifo allows the simulation of these individual components as well as stand-alone PV systems created with them, and it is free available in Internet [19].

The virtual laboratory is intended to be a complement to hands-on experiments and can be used in supporting both pre-laboratory and post-laboratory assignments. Pre-lab exercises may provide theoretical background and preparation before performing hands-on experiment, and they can also be employed for identifying points of confusion or individual student problems. Post-lab tasks allow the repetition of hands-on experiments or performing those that are very difficult of making in the physical laboratory.

An evaluation of this virtual laboratory will be implemented in order to assess the students perception of its effectiveness on their learning process, whose results will be presented in a future work.

REFERENCES

[1] J. O. Campbell, J. Bourne, P. Mosterman, and A. J. Brodersen, "The effectiveness of learning simulations for electronic laboratories," Journal of Engineering Education, vol. 91, pp. 81–87, January 2002.

[2]   D. Koretsky, D. Amatore, C. Barnes, and S. Kimura, "Enhancement of Student Learning in Experimental Design Using a Virtual Laboratory," IEEE Transactions on Education, vol. 51, pp. 76-85, February 2008.

[3]   N.R. Desai, K.V. Hoang, G.J. Sonek, "Applications of PSPICE simulation software to the study of optoelectronic integrated circuits and devices," IEEE Transactions on Education, vol. 36, pp. 357-362, November 1993.

[4]   I. Chamas, M.A.E. Nokali, "Automated PSpice simulation as an effective design tool in teaching power electronics," IEEE Transactions on Education, vol. 47, pp. 415-421, August 2004.

[5]   W. G. Hurley and C. K. Lee, "Development, implementation, and assessment of a web-based power electronics laboratory," IEEE Transactions on Education, vol. 48, pp. 567–573, November 2005.

[6]   M. Sun, B.L. Hesterman, "PSpice high-frequency dynamic fluorescent lamp model," IEEE Transactions on Power Electronics, vol. 13, pp. 261-272, March 1998.

[7]   E. Maione, P. Tortoli, G. Lypacewicz, A. Nowicki, J.M. Reid, "PSpice modelling of ultrasound transducers: comparison of software models to experiment," IEEE Transactions on Ultrasonics, Ferroelectrics and Frequency Control, vol. 46, pp. 399-406, March 1999.

[8]   Zhiyu Chen, "PSpice simulation of one atmosphere uniform glow discharge plasma (OAUGDP) reactor systems," IEEE Transactions on Plasma Science, vol. 31, pp. 511-520, August 2003.

[9]   L. Castañer, R. Aloy, and D. Carles, "Photovoltaic System Simulation Using a Standard Electric Circuit Simulator," Progress in Photovoltaics: Research and applications, vol 3, pp. 239-252, 1995.

[10]   J. Muñoz, "Simulación de Sistemas Fotovoltaicos con PSpice," Proyecto Final de Carrera, ETSI de Telecomunicación, Universidad Politécnica de Madrid, 1998.

[11]   L. Castañer, and S. Silvestre, Modelling Photovoltaic Systems using PSpice®. John Wiley & Sons Ltd, 2002.

[12]   P. C. Wankat and F. S. Oreovicz, Teaching Engineering. McGraw-Hill, 1993.

[13]   PSpice Schematics - Schematic Capture Software. User's Guide. Cadence Design Systems, 1999.

[14]   PSpice© A/D - Schematics version. User's Guide. Cadence Design Systems, 1999.

[15]   E. Lorenzo et al., Solar electricity: Engineering of photovoltaic systems, Progensa, 1994.

[16]   J.B. Copetti, E. Lorenzo, and F. Chenlo, "A General Battery Model for PV System Simulation," Progress in Photovoltaics: Research and Applications, vol 1, pp. 283-292, 1993.

[17]   MicroSim Application Notes, Version 8.0, MicroSim Corporation, 1997.

[18]   L. D. Feisel and A. J. Rosa, "The role of the laboratory in undergraduate engineering education," Journal of Engineering Education, vol. 94, pp. 121–130, January 2005.

[19]   https://blogs.upm.es/javiermunoz/