THE SOLAR PHOTOVOLTAIC PANEL SIMULATOR

SPASOJE MIRIĆ, MILOŠ NEDELIKOVič

Key words: Analog circuits, Current source inverter, Modeling, Renewable energy, Simulation.

In this paper, a simple photovoltaic (PV) simulator is presented. Based on the mathematical model of the PV cell, practical PV device simulator is designed. Without processor and memory, the simulator may generate arbitrary $I$-$V$ characteristic for different weather conditions. The idea for the realization PV simulator is supported by theory and confirmed with the simulation and laboratory results.

1. INTRODUCTION

Renewable energy sources have gained popularity around the world due to its high predictability and availability [1]. PV solar panels can supply different kind of loads. Sometimes, PV solar panels are connected directly to the load. More often, power converter between the panel and the load is used. PV solar panel has highly nonlinear $I$-$V$ characteristic. In one part of that characteristic it behaves like current source, and in another one like voltage source. Power converters with appropriate control algorithm are used to adjust that kind of characteristic to the needs of the load, and to maximize the efficiency of the module. Because of that, PV systems research are divided into two areas: array physics, design and optimization, and power conversion systems [2].

Conversion systems, like one proposed in [3], have to be tested on PV systems. Installing PV systems can be challenging and expensive. Even more, characteristics depend on illumination and temperature, which cannot be controlled. PV simulator can emulate desired characteristic of solar panel in every part of year, day, independently of illumination and temperature, which is very convenient for lab testing.

There are two methods to simulate real PV panel. First, requires use of the device which can simulate sunlight [4, 5]. The second method is to simulate equation of the PV device. Using the second method it is very convenient to build the circuit shown in the Fig. 1, which completely describes equation of PV device.

University of Belgrade, School of Electrical Engineering, Bulevar kralja Aleksandra 73, 11120 Belgrade, Serbia, E-mail: spasojemiric@gmail.com.

and makes simulation of PV devices much easier. When the second method is used, it is
important to consider the impact of irradiance and temperature [6].

The model of the PV device, used to build the simulator, is single-diode model. The
double-diode model is rejected, because the effect of the second diode can be noticed at low
irradiation and low voltage [7]. For single-diode model, five parameters has to be
determined: photo current, diode saturation current, diode ideality factor, parallel resistance
and series resistance. Some of the methods for determining parameters from real PV device
data can be used, for example [8], or procedure similar with described procedure in [9].

In Section 2 PV cell mathematic model and scheme are presented and explained.
Section 3 describes the practical realization of the simulator. Simulation and laboratory
results are presented in Section 4.

2. PV CELL MATHEMATIC MODEL

It is very important to understand the difference between PV cell and whole
solar panel. The solar panel can have more PV cells, connected in series or parallel,
which depends of desired output I-V characteristic. Single-diode model is used in
this paper. The PV cell model has ideal part, which is composed of current source
and one diode connected in parallel [8].

![Single-diode model of ideal PV cell and practical PV cell with serial and parallel resistance.](image)

In Fig. 1, PV cell is shown its ideal part and practical device. Ideal part of PV
cell can be described using basic semiconductor theory:

$$I = I_{pv,cell} - I_{0,cell} \exp \left( \frac{qV}{akT} \right) - 1,$$

(1)

where $I_{pv,cell}$ is the current generated by the incident light (it is directly proportional to
the sun irradiation), $I_{0,cell}$ is the reverse saturation or leakage current of the diode, $q$ is
the electron charge (1.60217646·10⁻¹⁹C), and $k$ is the Boltzmann constant
(1.3806503·10⁻²³J/K), $T$ (in Kelvin) is the temperature of the p-n junction, and $a$ is
the diode ideality factor [8].

Equation (1) is the equation for one PV cell. PV solar panels can have different
number of PV cells connected in series and/or parallel. The equation which determines
I-V characteristic of PV solar panel is:
The solar photovoltaic panel simulator

\[ I = I_{pv} - I_0 \left[ \exp \left( \frac{V + R_s I}{V_{t,a}} \right) - 1 \right] - \frac{V + R_s I}{R_p}, \]  

(2)

where \( I_{pv} \) and \( I_0 \) are the PV and saturation currents, respectively, of the array and \( V_t = N_s kT/q \) is the thermal voltage of the array with \( N_s \) cells connected in series, and \( R_s, R_p \) are the series and parallel resistance of the PV device, respectively.

The characteristic of the solar panel is shown on the Fig. 2, where typical important points of the characteristic are specified. \( I_{sc} \) is the short-circuit current and \( V_{oc} \) is the open-circuit voltage. MPP is the maximum power point, \( I_{mp}, V_{mp} \) are the current and voltage for the MPP, respectively.

During electrical design, different number of PV cells are connected in series \( N_s \) and parallel \( N_p \). Detailed procedure of solar panel electrical design is described in [10]. Parameters in (2) depend of \( N_s \) and \( N_p \). If \( N_p \) PV cells are connected in parallel, then the currents from (2) are: \( I_{pv} = N_p I_{pv,cell} \) and \( I_0 = N_p I_{0,cell} \) [8].

If the larger \( I_{sc} \) is needed, more PV cells are connected in parallel. And, when the larger \( V_{oc} \) is needed, more PV cells are connected in series. This is very important fact during the realization of solar panel simulator, which is the main subject of this paper.

3. PRACTICAL REALIZATION OF THE SIMULATOR

The solar panel simulator realized in this paper is cheap and simple, without processor, memory, etc. It has a current-controlled power converter which is controlled by analog simulator. Current source, diode and current through the resistor \( R_p \) from the model, shown in the Fig. 1, are the parts of the control topology. The output of the control topology is a current reference for converter. Resistor \( R_s \) is connected in series with the load resistor. Different \( I-V \) characteristics have different parameters in mathematic models, described by (2). Different models mean that different diode characteristics have to be set. The circuit for setting different diode characteristics is shown in the Fig. 3 [11]. To analyze this circuit, Ebers-Moll model...
of a bipolar transistor is used [12]. Ebers-Moll model of transistor gives relation between the base-emitter voltage \( V_{BE} \) and collector current \( I_C \):

\[
I_E = I_{ES} \cdot \left[ \exp \left( \frac{V_{BE}}{V_T} \right) - 1 \right] \approx I_{ES} \cdot \exp \left( \frac{V_{BE}}{V_T} \right),
\]

(3)

where \( V_T = kT / q \) is thermal voltage and \( I_{ES} \) base-emitter reverse saturation current. From the semiconductor theory it is very well known the relation between collector and emitter current: \( I_C = \alpha I_E \), where \( \alpha \) is common-base current gain.

![Electrical circuit used for arbitrary setting of the diode characteristic.](image)

In Fig. 3 there are two transistors \( Q_1 \) and \( Q_2 \), so there are two collector currents: \( I_{C1} \) and \( I_{C2} \). If we divide these two currents, we will have:

\[
\frac{I_{C2}}{I_{C1}} = \exp \left( \frac{V_{BE2} - V_{BE1}}{V_T} \right).
\]

(4)

From Fig. 3 voltages \( V_{BE1} \) and \( V_{BE2} \) can be found as:

\[
V_{BE1} = V_{B1} - V_{E1} \approx \frac{R_2}{R_1 + R_2} \cdot V_{in} - V_{E1},
\]

(5)

and

\[
V_{BE2} = V_{B2} - V_{E2} = -V_{E2} = -V_{E1},
\]

(6)

where \( V_B \) and \( V_E \) are electrical potentials of the transistors’ pins of the base and emitter, respectively. If we subtract Eq. (5) and Eq. (6), we get:

\[
V_{BE2} - V_{BE1} = -\frac{R_2}{R_1 + R_2} \cdot V_{in}.
\]

(7)
From Fig. 3, collector currents of the $Q_1$ and $Q_2$ are equal: $I_{C1} = +15\text{V}/1k\Omega$ and $I_{C2} = \frac{V_{out}}{R_{out}}$. Now, from Eq. 4 and Eq. 7 we have analytical expression which relates $V_{in}$ and $V_{out}$ from the Fig. 3:

$$V_{out} = R_{out} \frac{15\text{V}}{1k\Omega} \cdot \exp\left(\frac{-R_{1}}{R_{1} + R_{2}} \frac{V_{in}}{V_{r}}\right),$$

(8)

from which can be concluded that the diode ideality factor is $a = 1 + \frac{R_{1}}{R_{2}}$. Thermal voltage, $V_t$ in this equation is thermal voltage of the transistors, and it is not multiplied by $N_s$. Exponent in (8) has negative sign, so the input signal has to be inverted. The voltage in (8), $V_{out}$, is the current reference. So, the value of the diode saturation current from the model fits to the value of expression $R_{out} \cdot 15\text{V}/1k\Omega$, and it should be somewhere about $10^{-8}$, depends on the type of solar panel. This requires a very low value of the resistor $R_{out}$. To solve this, small offset voltage (0.6V) is supplied to the $V_{in}$, and now the value of the saturation current is:

$$I_0 = 15 \cdot 10^{-3} \cdot R_{out} \exp\left(\frac{-0.6\text{V}}{aV_{r}}\right),$$

(9)

where $\exp(-0.6V/aV_{r})$ is somewhere about $2 \cdot 10^{-8}$. Offset voltage is supplied to the $V_{in}$ through the resistor of 25 kΩ shown in Fig. 4.

![Fig. 4 – Whole control logic for PV simulator, where block PV cell is circuit for setting diode characteristic from Fig. 3.](image)

It is simpler to use the circuit from Fig. 3 only to simulate one PV cell. Thus, $V_{in}$ is then measured voltage divided by the number of series connected PV cells, $N_s$. That division is realized with the resistor $R_{Voc}$, where its value in kΩ is equal to the $N_s$, (Fig. 4). The current $I_{pv}$ is set by the two resistors from Fig. 4, $R_{temp}$ and $R_{bvc}$. With $R_{temp}$, temperature of PV device is set. With higher values of the $R_{temp}$, the
higher temperature of the PV device is set. $I_{sc}$ is set with the resistor $R_{isc}$. The resistor $R_p$ from Fig. 4 is 1,000 times greater than the real value of the parallel resistance. $I_{ref}$ is the reference for the current controlled converter, which has hysteresis current controller.

Fig. 5 – Whole simulator.

The resistor $R_s$ is not part of the control topology. Through it passes current $I$, so $R_s$ has to be designed for the current $I_{sc}$ of the simulated PV solar panel. Its value is changeable due to the estimated circuit parameters from Fig. 1. The block diagram of the whole system is shown on the Fig. 5. The current source from Fig. 5 (CCS) is a current controlled power converter. It is equivalent of the currents $I_{pv}$, $I_d$ and the current trough the resistor $R_p$ from Fig. 1.

## 4. SIMULATION RESULTS

The algorithm explained in Section 3 is simulated and compared with the PV equation of the device. Five important parameters of the model, given in Tab. 1, are set with the correct tuning of the resistors in the scheme shown in Fig. 3. The parameters are derived using method in [8].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_{pv}$</td>
<td>1.5; 1.3; 1 A</td>
</tr>
<tr>
<td>$I_{0,n}$</td>
<td>$9.825 \times 10^{-8}$ A</td>
</tr>
<tr>
<td>$a$</td>
<td>1.3</td>
</tr>
<tr>
<td>$R_p$</td>
<td>500 $\Omega$</td>
</tr>
<tr>
<td>$R_s$</td>
<td>0.28 $\Omega$</td>
</tr>
</tbody>
</table>

The values of five the parameters used in PV circuit model for simulation are shown in Table 1. $I_{pv}$ depend on the light and this value in Tab. 1 is chosen arbitrary. $I_{0,n}$ is the value of the parameter on the nominal temperature. The parameters $a$, $R_p$ and $R_s$ are determined using the procedure given in [7]. The simulation is performed in PSpice.
The results of the simulation are shown in the Fig. 6. It can be noted that the two curves are perfectly matched. The curves without dots are the characteristics of the PV model algorithm used for the simulation of the solar panel, explained in Section 3. The simulation is performed for the three values of the short-circuit current: 1.0A, 1.3A and 1.5A. These results are the good base for the practical realization, and good laboratory results.

5. LABORATORY RESULTS

Practical realization of the solar panel simulator is shown on the Fig. 7. The usage of the resistors $R_{\text{temp}}$, $R_{\text{ioc}}$, and $R_{\text{ Voc}}$ is explained in the Section 3. The resistor $R_2$ is the external resistor used for setting diode ideality factor $a$. This resistor is shown in Fig. 3.
Five parameters are set, like in the simulation. For different values of the load resistor, voltage and current at the power output of the simulator are measured. The measured results are shown in the Fig. 8. The dotted curves are measured, and the curves without dots are simulated curves. A good match of the laboratory and simulation results can be noticed.

Fig. 8 – Laboratory and simulation results.

6. CONCLUSIONS

One way of the realization of the PV simulator is presented in this paper. The idea is covered by the theory and confirmed with the simulation and laboratory results. It is shown how on easy and cheap way, PV simulator can be realized. Arbitrary PV device characteristics can be performed with this simulator. It can be used for different kind of laboratory testing where PV solar panel characteristic is needed.

Received on December 6, 2014

REFERENCES


