Ročník 2020 electroscope Číslo I

Photovoltaic panel Spice modelling

A. Bouřa
Department of Microelectronics, Faculty of Electrical Engineering, Czech Technical University in Prague, Technická 2, Prague 6, 166 27, Czech Republic
E-mail : bouraa@fel.cvut.cz

Anotace:
Článek popisuje univerzální Spice model, který je vhodný pro modelování fotovoltaického panelu, a to především jeho stejnosměrných charakteristik. Model je založen na jednoduchém schématu, jehož parametry jsou přizpůsobeny tak, aby výsledná charakteristika kopírovala naměřené hodnoty. Jsou představeny dva typy model. Jsou to jednoduchý model a komplexní model. Jednoduchý model je platný pouze pro fotovoltaické panely s konstantní šířkou PN přechodu (volitelné jsou pouze délka a množství těchto přechodů). Komplexní model umožňuje měnit také šířku PN přechodu. Jednoduchý model vykazuje lepší přesnost, která je typicky lepší než 5 % zatímco komplexní model má přesnost typicky lepší jen 15 % (pro osvětlení v rozsahu 20 lx až 2000 lx).

Abstract:
The paper presents a universal Spice model that can be used for modelling the photovoltaic panels (PV) – namely the DC characteristics. The model was built on simplified schematic and fitting its parameters to the measured data. There were developed two types of the models – the simple model and the full model. The simple model is valid only for PVs with constant width of the PN junction strip (only the number and length are scalable). The full model allows user to set also the PN junction's width. The simple model exhibits better accuracy, typically better than 5 %, while the full model's accuracy is typically lower than 15 % (for illumination in the range from 20 lx to 2000 lx).

INTRODUCTION

Spice modelling is crucial part of the electronics design. It allows to predict behavior of the circuit at real conditions. The paper presents a universal Spice model that can be used for modelling the photovoltaic panels (PV) – namely its DC characteristics. The model was built on simplified schematic and fitting its parameters to the measured data. There were developed two types of the models – the simple model and the full model. The simple model is valid only for PVs with constant width of the PN junction strip (only the number and length are scalable). The parameters of the model were fitted to the measured characteristic. This is why the model's accuracy is affected by technological tolerances during PVs' fabrication process (two samples of the same PV can differ in the parameters) and also by precision of the measurement during the characterization. Namely the measurement of the illumination is affected by the spectral based error because the testing light spectrum changes during dimming. Another source of the errors can be non-homogeneity of the illumination on the test bench. The PVs were measured using the blue led to minimize the spectral change and results were recalculated to the white day light.

SPICE MODEL

The PV models are based on schematic that are presented in figures 1 and 2. User can define input parameters to simulate PVs with different configuration, dimensions and at different light conditions.

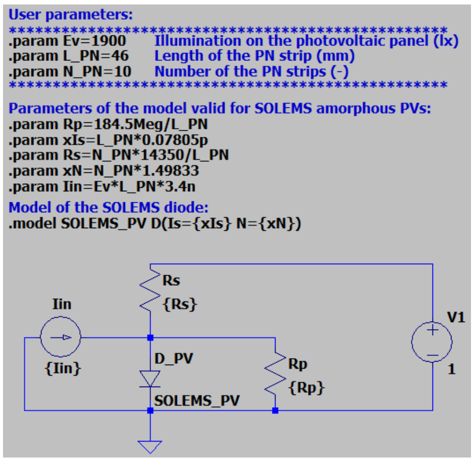

Fig. 1: Simple SPICE models of the PV in LTspice which is valid for panels SOLEMS with PN junction width 4.15 mm

It consists of diode D_PV that represents behavior of the PN junctions, serial resistor R_s, parallel shunt resistor R_p and current source I_{in}. The voltage source V1 serves only for the simulation purposes and it is not needed part of the model. Parameters of the devices in the modela are defined by equations that relate technological properties of the PV and parameters provided by users. The user can define parameter E_v (lx), that represents the Illumination applied on the PV. Parameter L_PN (mm) gives information about the pn-junction length, W_PN represents the pn-junction width and N_PN (-) represents number of the pn-junction strips in series (see figure 3).

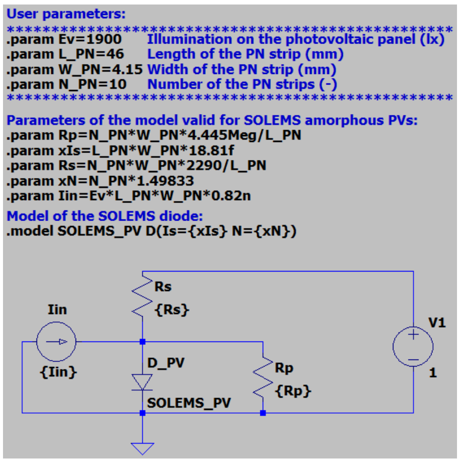

Fig. 2: Fully scalable SPICE model PV in LTspice

Simple Spice model

Each pn-junction strip has a constant width (e.g. W_PN = 4.15 mm) in the simple model. The total area of the PV is given by N_PN × W_PN × L_PN (see figure 3). The input current I_{in} is proportional to the input illumination E_v and area of single pn-junction strip. Other parameters such as saturation current I_s, diode technological parameter n, serial and parallel resistances are given by PV's technology and geometrical properties.

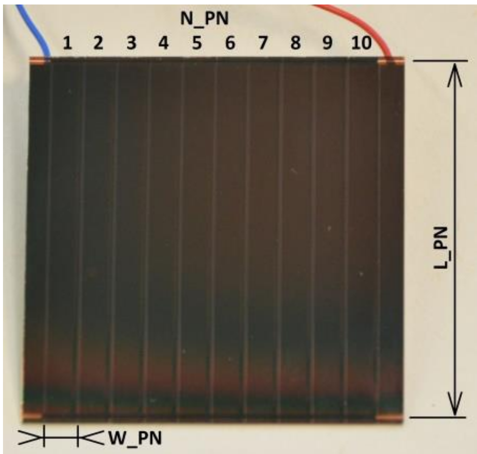

Fig. 3: Solar panel SOLEMS 5×5 cm^2 with its basic properties

The model was built and optimized according the measurement of the PVs. There were measured the loading characteristic of the panels and then the model parameters were fitted to the measured data. Figure 3 shows representative PV with its dimensions as the input parameters to the spice model. Its basic loading characteristic are presented in figure 4.

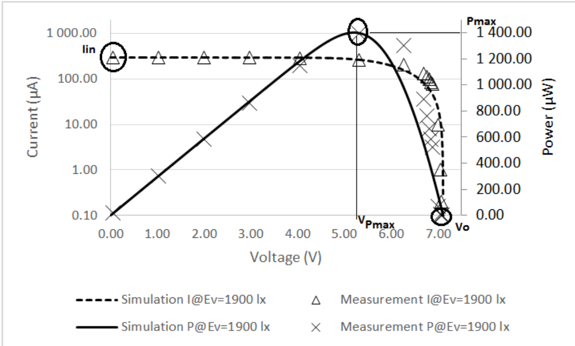

Fig. 4: Typical loading characteristic of PV with important points in the graphs

Typical parameters of the loading characteristics are I_{in}, V_o, P_{max} and V_{Pmax}. I_{in} represents short-circuit current of the panel. V_o is the open-circuit voltage on the panel when it is unloaded. P_{max} is the maximal power that can be harvested from the panel when it is loaded optimally and voltage V_{Pmax} is the optimal voltage for the maximal power. All the parameters are related to given illumination level E_v.

Full Spice model

The full model of the PV is presented in figure 2, right. User can also change the pn-junction strip width. This model exhibits lower accuracy compared to the simple model. The most critical parameter is the serial resistivity R_s that affects the maximum power. This parameter is scaled linearly in the model while the real behavior across different sizes is more complex. This is why the accuracy of the model is worse – typically around 10 % but it can be worse (see the comparison in following chapter).

SPICE MODELLING AND MEASUREMENT COMPARISON

Simple model comparison

Figures 5 to 8 show simulation results and measurement results for the PV from figure 3. The typical error is lower than 5 %. Few exceptions, where the error is above 10 %, are caused by the measurement tolerances during the characterization. The simulation can give good overall information about the PV performance.

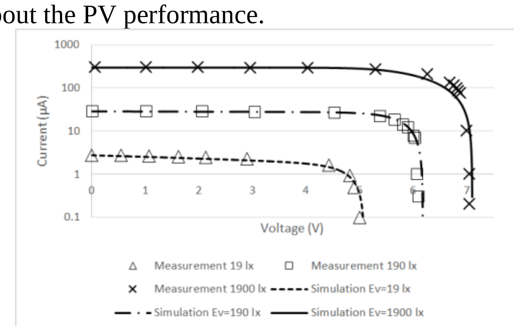

Fig. 5: Comparison of simulated and measured loading characteristics using the simple model and for PV SOLEMS 5×5 cm^2.

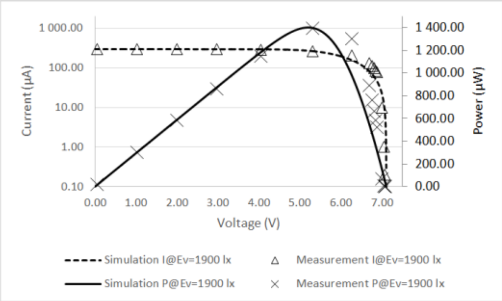

Fig. 6: Comparison of simulated and measured loading characteristics using the simple model and for PV SOLEMS 5×5 cm^2, detail for illumination 1900 lx.

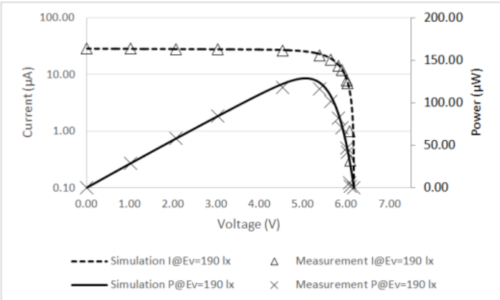

Fig. 7: Comparison of simulated and measured loading characteristics using the simple model and for PV SOLEMS 5×5 cm^2, detail for illumination 190 lx.

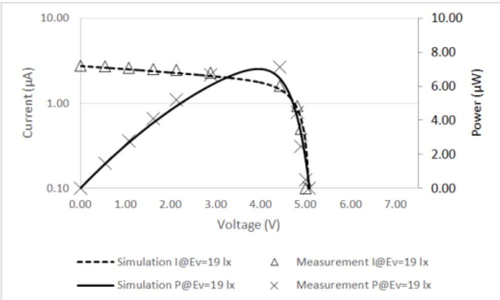

Fig. 8: Comparison of simulated and measured loading characteristics using the simple model and for PV SOLEMS 5×5 cm^2, detail for illumination 19 lx.

Full model comparison

The full model differs from the simple by possibility of changing the pn-junction strip's width. Problem is to define the serial resistivity parameter because it does not behave strictly linearly with the parameter W_PN. The value in the model is tradeoff between several measured samples. Another parameter is the parallel shunt resistance that depends on the technology. The full Spice model is defined as follows.

User parameters (parameters given by dimensions of the panel and applied illumination):
Ev*=… – Illumination on the PV (lx)*
L_PN*= … – Length of the pn junction strip (mm)*
W_PN*= … – Width of the pn junction strip (mm)*
N_PN*= … – Number of the pn junction strips in series (-)*

Parameters of the model (parameters are given by dimensions and PV's technology):
Rp*=N_PN*W_PN****CRp****/L_PN – Parallel shunt resistance*
xIs*=L_PN*W_PN****CxIs*** *– Saturation current in the Shockley equation*
Rs*=N_PN*W_PN****CRs****/L_PN – Serial resistance*
xN*=N_PN****CxN*** *– Emission coefficient in the Shockley equation*
Iin*=Ev*L_PN*W_PN****CIin*** *– Current generated by the panel*

Parameters CRp, CxIs, CRs, CxN, CIin are characteristic for the PV's fabrication technology. Comparing the simulated and measured data, the typical error is lower than 15 %.

ACKNOWLEDGMENTS

Research described in the paper has been supported by the project EnSO, Electronic Components and Systems For European Leadership Joint Undertaking in collaboration with the European Union's H2020 Framework Programme (H2020/2014-2020) and National Authorities, under grant agreement n° 692482 and partially by Smart Access Control for Smart Buildings (SACON), No. 12708.

REFERENCES

[1] G. Dennler, C. Lungenschmied, A. Labouret, H. Neugebauer, and N. S. Sariciftci, "Flexible, conjugated polymer-fullerene-based bulk-heterojunction solar cells: Basics, encapsulation and integration," J. Mater.Res. 20(12), pp. 3224 – 3233, 2005

[2] Labouret, A., Villoz, M.: Solar Photovoltaic Energy (Let Renewable Energy), The Institution of Engineering and Technology, UK, 2010, 4th edition, ISBN: 978-1-84919-154-8

[3] Siyu Guo,Fa-Jun Ma,Bram Hoex,Armin G. Aberle,Marius Peters: Analysing Solar Cells by Circuit Modelling, Energy Procedia, Elsevier, 2012, doi.org/10.1016/j.egypro.2012.07.004

[4] Gurrola, E. I., Dongaonkar, S.: SPICE Simulation of Thin-Film Solar Cell Modules, User's Guide, Online 2017, URL: https://nanohub.org/resources/12259/download/ Users_Guide.pdf

[5] Sarkar, Md., N., I.: Effect of various model parameters on solar photovoltaic cell simulation: a SPICE analysis, Renewables: Wind, Water, and Solar, 2016, https://doi.org/10.1186/s40807-016-0035-3