

PVspice

A PSpice symbol library of photovoltaic components

User's Guide

Javier Muñoz Cano
Instituto de Energía Solar
Universidad Politécnica de Madrid

Copyright © 2010 Javier Muñoz Cano. All rights reserved.

Redistribution and use of PVspice as well as this user's guide, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions must retain the above copyright notice, this list of conditions and the following disclaimer.
2. Redistributions must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THE COPYRIGHT HOLDER PROVIDES THIS SOFTWARE "AS IS" AND DISCLAIMS ANY RESPONSABILITY FOR THEIR USE.

Trademarks

OrCAD, PSpice, PSpice Schematics and Probe are registered trademarks of Cadence Design Systems, Inc.

First online edition: November 2010

<https://blogs.upm.es/javiermunoz/>

1 Getting Started

1-1 Overview

PVspice¹ is a symbol library (.slb) of photovoltaic (PV) components for PSpice, which allows the simulation of individual components as well complete stand-alone Solar Home Systems (SHS). PVspice has been developed mainly for educational purposes [1] .

The previous reference, whose preprint can be downloaded from <https://blogs.upm.es/javiermunoz/>, describes the design of the PVspice library, the mathematical models that have been used for developing the PV components, their symbols, and their electrical parameters.

The present version of PVspice includes the following components: PV generators, ON/OFF series charge regulators, lead-acid batteries and constant and variable power DC loads.

1-2 Software requirements

PVspice has been tested up to OrCAD 16.2 using PSpice Schematics, which allows designing and drawing PV systems, setting up and running simulations in PSpice, and evaluating simulation results using Probe. It is assumed that the reader knows Schematics and Probe as basic user [3] , [4] .

It is worth mentioning that the development and support for PSpice Schematics has been discontinued with the OrCAD 10.0 release. The user can download a former free version of this simulator (PSpice Student 9.1) from

¹ The original name of this library was Sisifo [1] which is the acronym of Photovoltaic System Simulation in Spanish (Simulación de Sistemas Fotovoltaicos). But this former name was later used for a web simulator of PV systems [2] . Hence, to avoid confusion between both simulators, the name of the PSpice library has been renamed to PVspice.

<https://blogs.upm.es/javiermunoz/>, which allows the use the PVspice library, but with a limitation in the number of components.

1-3 Structure of this document

Chapter 2 describes the installation of PVspice. Chapter 3 describes the modelling of PV components using the Analog Behavioural Modelling (ABM) feature of PSpice. Finally, chapter 4 illustrates several applications of PVspice through different examples of simulation. The annex of this document displays the circuit models of each PV component.

2 PSpice installation

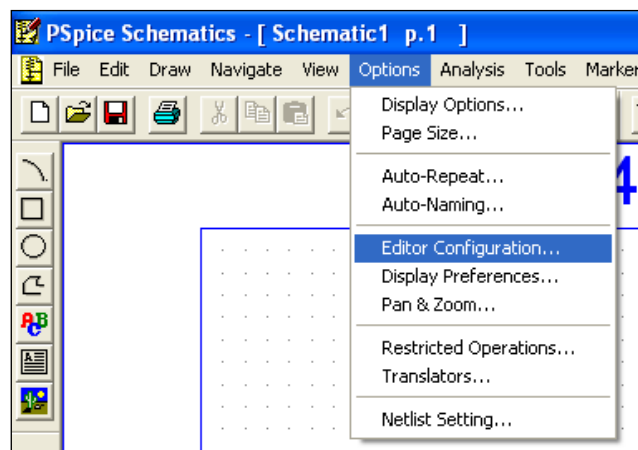
2-1 Installation steps

1. Download the executable file “PSpice 9.1 Student version”, double click on it and unzip all the files in a folder.
2. Run the executable file Setup.exe and install the program².
3. Download and unzip **pvspice.zip** to a folder.
4. Start the schematic editor of the program by clicking on the



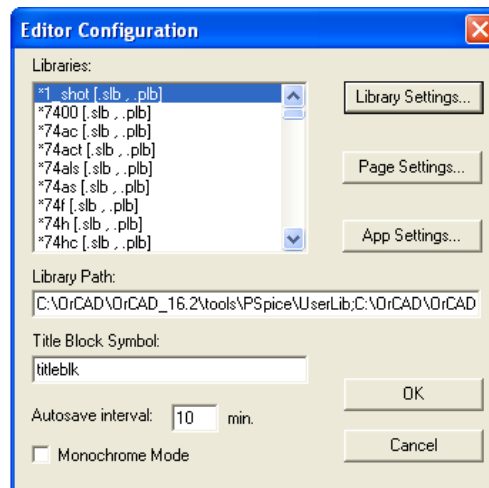
Schematics icon. Schematics

5. From the **Options** menu, select **Editor Configuration**.

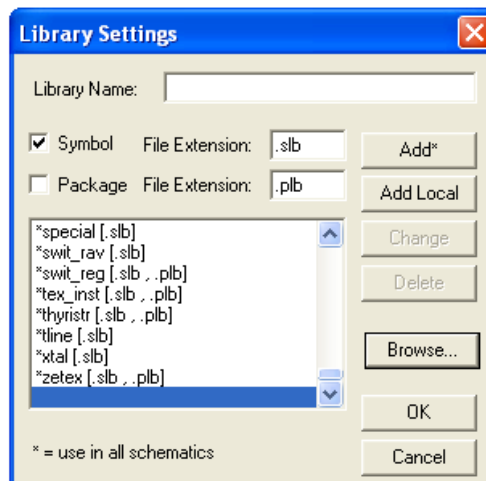


² In Windows 10, there is a common installation problem. If the installation does not seem work, open the Task manager and check if the program appears as a “Background process”. In that case, display its properties and “Go to details”. Next, display its properties, select “Analyze wait chain” and end the nested process

6. Select **Library Settings**.



7. Select **Browse**, choose pvspice.slb from the unzip folder, and click Open (check that the path appears in the **Library Name** text box).



8. Select **Add*** and **pvspice** will be added to the Libraries list box for using it in all schematics.

3 Modeling of PV components

3-1 Introduction

The use of standard electric circuit simulators such PSpice to analyze the operation of PV systems has been proposed in the nineties [5]. This approach requires to develop equivalent electric circuit of individual PV components, which can be used to create electric designs of PV systems for simulation.

PV components have been modelled here from the viewpoint of the external behaviour (I-V characteristics), using Analog Behavioural Modelling (ABM). This PSpice feature allows to describe the electric characteristics of PV components in terms of lookup tables, mathematical relationships or transfer functions using dependent voltage, or current, sources. The use of ABM has several advantages, such as the extension of PSpice capabilities without code modification or the reduction of computational time and convergence problems.

3-2 Models

PV components have been created using a hierarchical structure. At the bottom level, the equivalent electric model of each component is developed. After that, the circuit is represented by a hierarchical symbol, with input and output connections, which can be connected with other components to create a system configuration. This modular design offers the possibility of navigating through the hierarchical structure, providing flexibility for the modification of electric models.

Each symbol has a set of model parameters that can be edited and modified by the user to create custom-made components. Implemented models require only parameters that can be obtained from manufacturer datasheets.

The library contains the irradiance and temperature sources and the basic components of SHS: a PV generator, a lead-acid battery, a charge regulator and DC loads. The description of theoretical models for these components, the electric equivalent circuits and their hierarchical symbols are described below.

3-2-1 PV generator

The characteristic I-V curve of a PV generator can be expressed with a good approximation by [6] :

$$I = I_{SC_gen} - I_{SC_gen} \exp \left[\frac{V - V_{OC_gen} + I \cdot R_{S_gen}}{N_{S_gen} V_t} \right]$$

Where I_{SC_gen} , V_{OC_gen} , R_{S_gen} and $N_{S_gen} = N_{SM} \cdot N_{SC}$ 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, being N_{SM} the number of modules connected in series and N_{sc} the number of cells in series per module. The parameter $V_t = nkT/q$ is the thermal voltage, where n , k , T and q are, respectively, the ideality factor, the Boltzmann's constant, the cell temperature, and the electron charge.

The I-V characteristic under arbitrary operating conditions can be estimated using the following parameters and the equations shown in Table 1 [6]

1. Electric characteristics under Standard Test Conditions (STC) of a single PV module (I_{SCM}^* [A], V_{OCM}^* [V], and maximum power P_{MM}^* [W])
2. Nominal Operating Cell Temperature NOCT [$^{\circ}$ C].
3. Solar irradiance G [W/m^2] and ambient temperature T_A [$^{\circ}$ C].

Parameter	Description	Equation
T_c [$^{\circ}$ C]	Cell temperature	$T_A + \frac{(NOCT - 20)}{800} \cdot G$
I_{SC_gen} [A]	Short-circuit current	$\frac{I_{SC_gen}^* \cdot N_{PM} \cdot (1 + \alpha \cdot (T_c - 25))}{1000} \cdot G$
V_{OC_gen} [V]	Open-circuit voltage	$N_{SM} V_{OCM}^* (1 + \beta \cdot (T_c - 25))$
FF	Fill factor	$FF = \frac{P_{MM}^*}{V_{OCM}^* \cdot I_{SCM}^*}$
v_{oc}	Normalised open circuit voltage, per cell	$V_{OCM}^* / (N_{SC} V_t)$ (at 300K)
FF ₀	Ideal fill factor	$\frac{v_{oc} + \ln(v_{oc} + 0.72)}{1 + v_{oc}}$
R_{S_gen} [Ω]	Series resistance	$\left(1 - \frac{FF}{FF_0}\right) \frac{N_{SM} \cdot V_{OCM}^*}{N_{PM} \cdot I_{SCM}^*}$

Table 1. Electric PV module parameters under arbitrary operating conditions.

The parameters α , β and N_{PM} are, respectively, the coefficients of variation of I_{SCM}^* [A] and V_{OCM}^* [V] with the temperature, and the number of PV modules connected in parallel in the PV generator.

The PV generator model can be represented by the equivalent circuit displayed in Figure 1-b, which is composed of two voltage-controlled current sources and a series resistance.

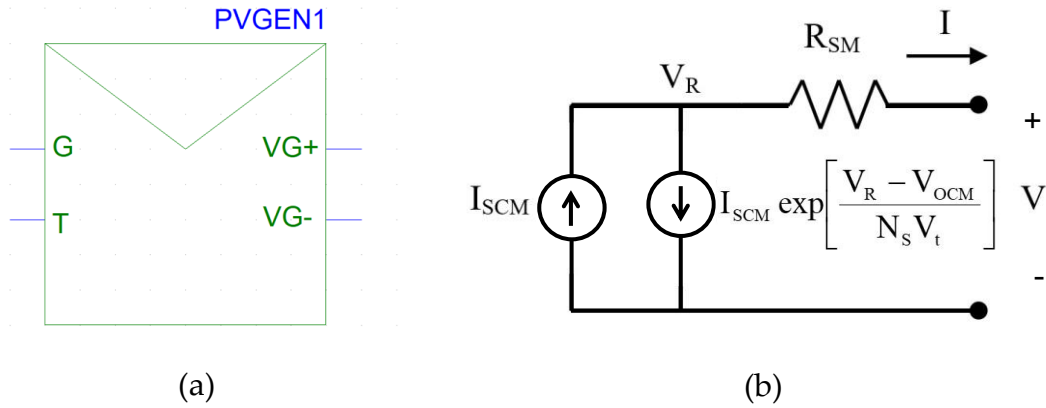


Figure 1. PV generator in PSpice: (a) Hierarchical symbol. (b) Equivalent electric circuit.

Irradiance and ambient temperature are functions of time, which have been modelled in PSpice using constant or piecewise-linear voltage sources. In the second case, the voltage source has a text file attribute that defines the time-evolution of the variable. The current sources of the PV generator circuit are controlled by these irradiance and ambient temperature “voltage sources”.

3-2-2 Lead-acid battery

The battery modelling is based on the model proposed by Copetti [7]. In this model, the battery capacity C_B depends on the discharge current rate and battery temperature, and can be expressed as a function of a reference current as:

$$C_B = \frac{a \cdot C_{20}}{1 + (a - 1) \cdot \left(\frac{I_B}{I_{20}}\right)^b} (1 + c \cdot (T_B - 25))$$

Where C_{20} is the battery capacity in 20 hours, I_{20} is the reference current (constant discharge current in 20 hours), I_B is the battery current, and T_B the temperature of the battery, in [°C]. The coefficients a , b and c must be fitted for each particular battery.

The open-circuit battery voltage, V_{OCB} , can be estimated as a linear function of SOC, disregarding temperature effects, by:

$$V_{OCB} = k_1 + k_2 \cdot SOC$$

Where SOC is the state of charge and coefficients k_1 and k_2 are fitting parameters.

The battery voltage V_B in the external terminals of the battery is the sum of open-circuit voltage V_{OCB} and the voltage drops across its internal impedance.

Under charge, V_B is higher than V_{OCB}

$$V_B = V_{OCB} + I_B \cdot R_C$$

And lower than V_{OCB} under discharge:

$$V_B = V_{OCB} - I_B \cdot R_D$$

Where R_D and R_C are, respectively, non-linear resistances for discharge and charge that depends on the battery current and the SOC, which are calculated using the following equations:

$$R_D = \left(\frac{d_1}{1 + I_B^{d_2}} + \frac{d_3}{SOC^{c_4}} + d_5 \right) \cdot (1 - d_6(T_B - 25))$$

$$R_C = \left(\frac{c_1}{1 + I_B^{c_2}} + \frac{c_3}{(1 - SOC)^{c_4}} + c_5 \right) \cdot (1 - c_6(T_B - 25))$$

Where the coefficients d_i and c_i are fitting parameters. The battery model for discharge is represented by the equivalent electric circuit shown in Figure 2. During the charge, I_B has opposite sense and R_D is replaced by R_C .

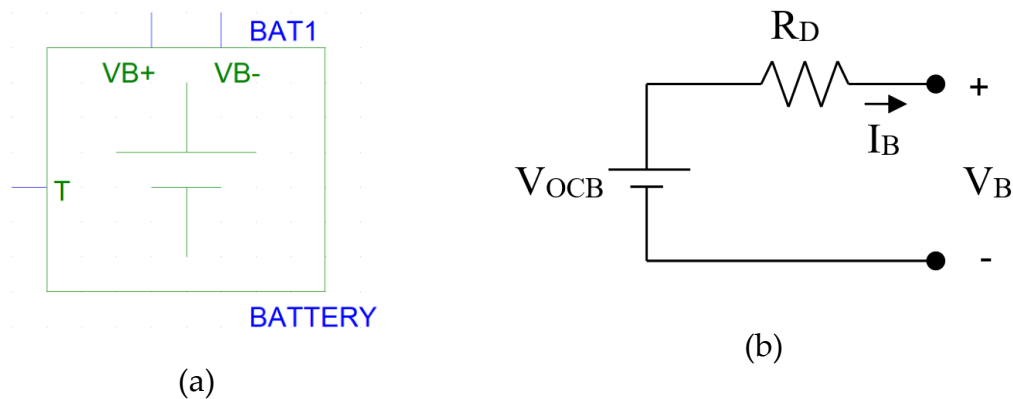


Figure 2. Battery in PSpice: (a) Hierarchical symbol. (b) Equivalent electric circuit.

3-2-3 Charge regulator

The charge regulator has been modelled as an ON-OFF series type. The equivalent electric circuit is displayed in Figure 3.

The model is composed of two voltage-controlled resistors that behave as nearly ideal ON-OFF switches. In the ON state the resistance is very low and the resistors essentially are a short-circuit and, in the OFF state, their resistance is very high and operate as open-circuits.

The switch R_{OC} protects the battery against overcharge disconnecting the PV generator when the battery exceeds a given high voltage disconnection voltage HVD. The PV generator is reconnected when the battery voltage decreases below a high voltage reconnection voltage HVR.

The switch R_{OD} protects the battery against overdischarge disconnecting the load when battery voltage decreases below the low voltage disconnection voltage LVD. The load is reconnected when the battery voltage exceeds a given low voltage reconnection voltage LVR.

Both switches are controlled by the hysteresis cycles shown in Figure 4. The voltages of the overcharge hysteresis cycle are corrected with the temperature using the following equations:

$$HVD = HVD(25^{\circ}\text{C}) + \lambda(T_B - 25)$$

$$HVR = HVR(25^{\circ}\text{C}) + \lambda(T_B - 25)$$

Where λ is the coefficient of correction with the temperature, which is typically in the range from -4 to -5 mV/ $^{\circ}\text{C}/\text{cell}$. For example, for 12 V battery, with six 2V-cells connected in series, this correction ranges from -24 to -30 mV/ $^{\circ}\text{C}$.

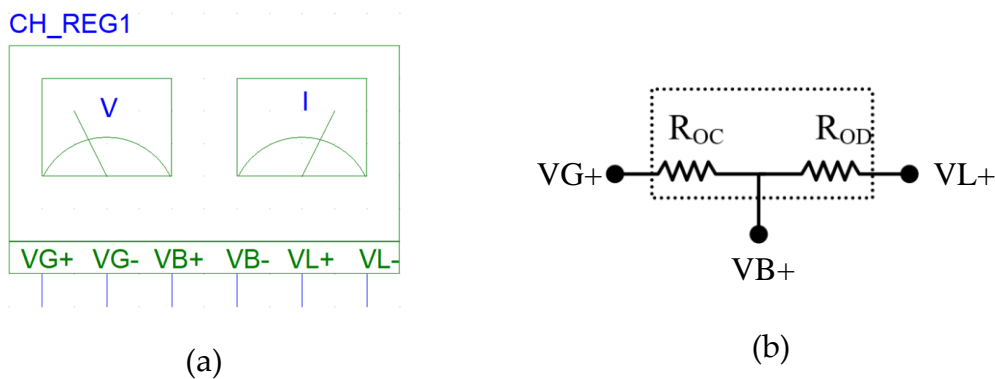


Figure 3. Charge regulator in PSpice: (a) Hierarchical symbol. (b) Equivalent electric circuit between the positive terminals of the generator (G), battery (B) and load (L), while the negative terminals are connected to the same point.

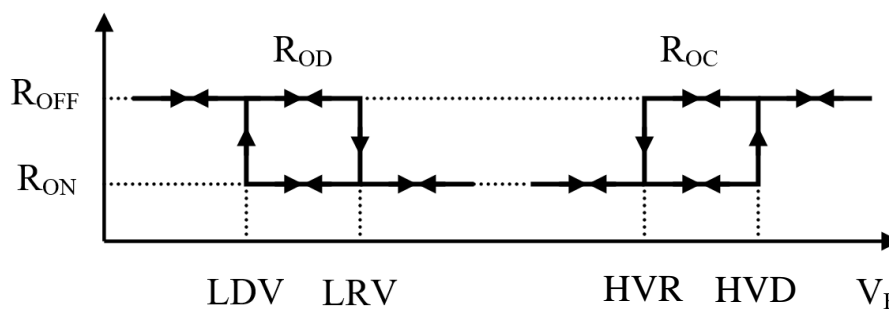


Figure 4. Hysteresis voltage control of charge regulator switches.

3-2-4 Load

Power consumption has been modelled as a constant power load. A first approximation to the modelling is using a voltage-controlled current source:

$$I_L = \frac{P_L}{V_L}$$

Where P_L is the demanded load power and V_L and I_L are, respectively, the load voltage and load current.

However, the behavior of this approximation is not good for low voltages because PSpice assumes electric parts are turned OFF when power supplies are close to zero volts. This unusual operating condition happens when the charge regulator activates the overdischarging protection and disconnects the load from the battery. An improved model consists in the series connection of two resistors [9] :

$$R_L = R_{MIN} + R_{VAR} = R_{MIN} + \frac{V_L^2}{P_L}$$

Where R_L is the load resistance, R_{MIN} is a constant resistance and R_{VAR} is a voltage-controlled resistor. Hence:

$$I_L = \frac{V_L}{R_L} = \frac{V_L}{R_{MIN} + \frac{V_L^2}{P_L}} = \frac{1}{\frac{R_{MIN}}{V_L} + \frac{V_L}{P_L}}$$

For low voltages, $I_L \approx V_L/R_{MIN}$ (the load behaves like a resistor of value R_{MIN}), and for high voltages, $I_L \approx P_L/V_L$ (the load behaves like a constant power consumption).

Figure 5 shows this equivalent electric load model, where the constant parameter P_L has been substituted by a time-dependent generic function, $p_L(t)$, which allows to specify the load power consumption in each instant.

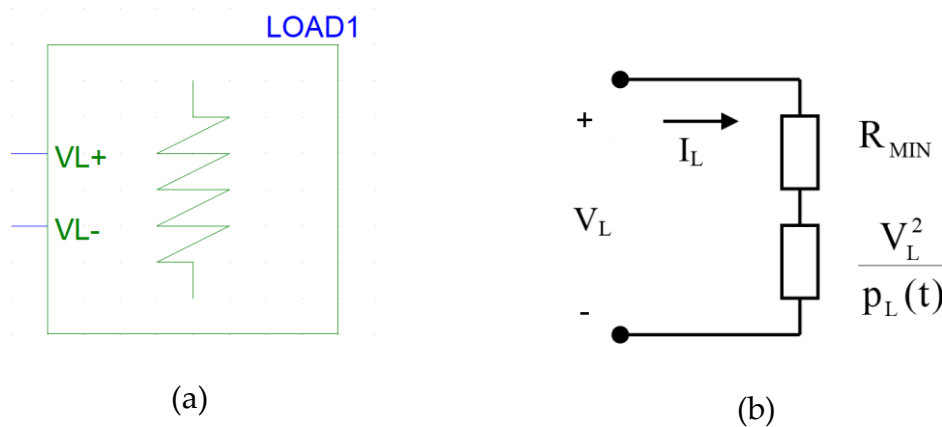


Figure 5. Load in PSpice: (a) Hierarchical symbol. (b) Equivalent electric circuit.

For example, daily load profiles have been implemented defining this function $p_L(t)$ as:

$$p_L(t) = L \cdot h(t)$$

Where L is the daily energy consumption [Wh] and $h(t)$ is a normalised daily profile, which verifies that:

$$\int_{day} h(t)dt = 1$$

Both parameters, L and $h(t)$, are specified as load attributes, which can be edited and modified by the user to define custom-designed consumption profiles.

3-2-5 System simulations

Once the electric models have been created, their hierarchical symbols are included into a symbol library, called `pvspice.slb`, which should be added to the list of libraries available in PSpice as described in the previous chapter. After that, the program is ready to run SHS simulations.

The simulation procedure is similar to the analog one. First, the SHS design is prepared for simulation using Microsim Schematics, a graphical editor which allows to select, placing and connecting the symbols to create the desired system configuration. For example, figure 7 shows a typical SHS configuration, just as it is displayed in the Schematics editor.

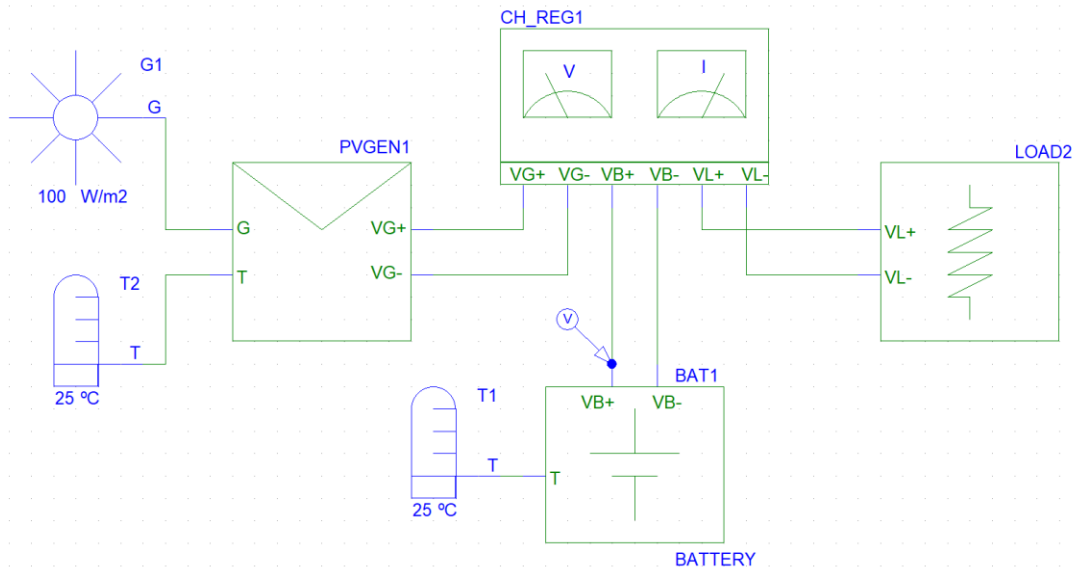


Figure 6. Example of SHS configuration.

Once the SHS design is prepared and the analysis type is configured, the simulation can start. As result, PSpice generates a set of data containing the voltage and current values in all circuit nodes.

The data produced by the simulation can be analyzed and interactively manipulated using Microsim Probe, a graphical analyzer available for PSpice. Among other functions, Probe can display complex arithmetic functions of analog simulation output variables. This feature can be used to evaluate the simulation results, computing several figures of merit, such as: array and final yield, components losses, performance ratios, loss of load probability, battery efficiency, etc.

The complete simulation procedure and the Microsim program used in each step is shown in Figure 7.

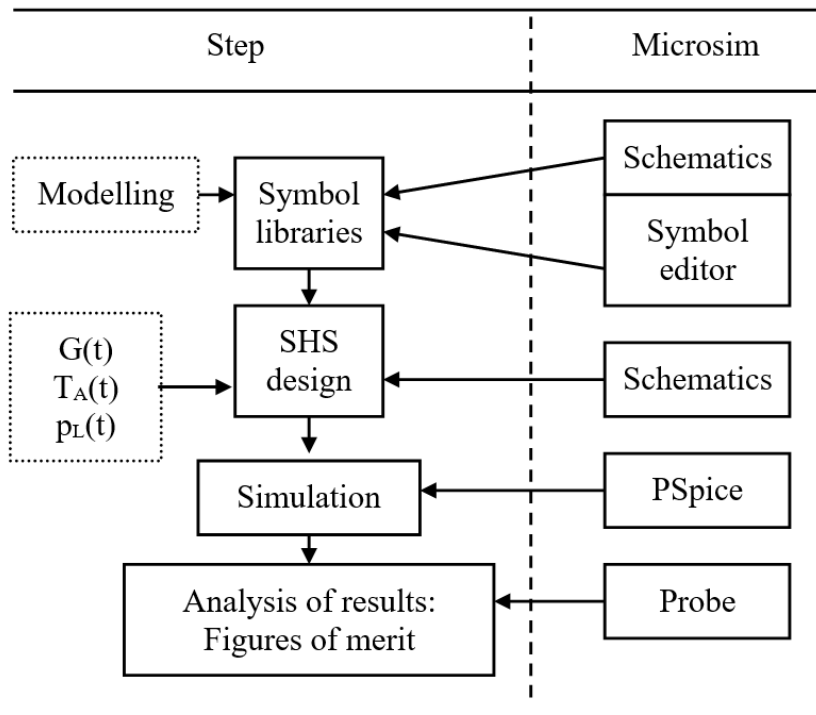


Figure 7. Simulation procedure.

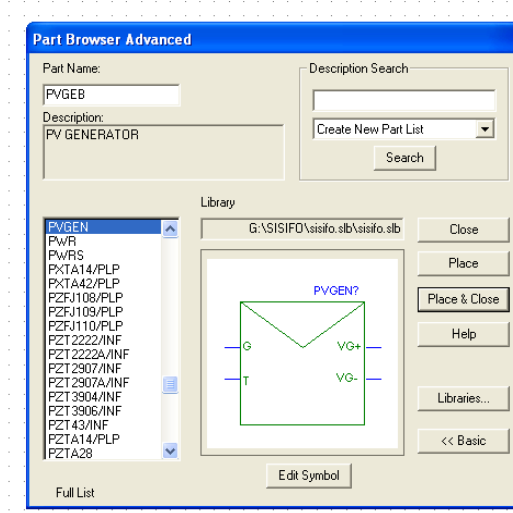
4 Simulation examples

4-1 My first simulation

This section describes, step-by-step, the simulation of the current-voltage curve of a PV generator: circuit drawing, edition of parts attributes, configuration of the analysis type and analysis of simulation results. The final file for simulation is the first example provided with PVspice (“Example 1 - PV generator curves”).

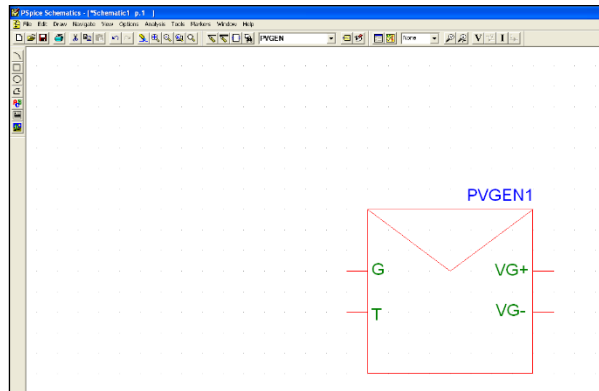
4-1-1 Drawing the circuit



1. From the Draw menu, select Get New Part to display the Part Browser dialog box.

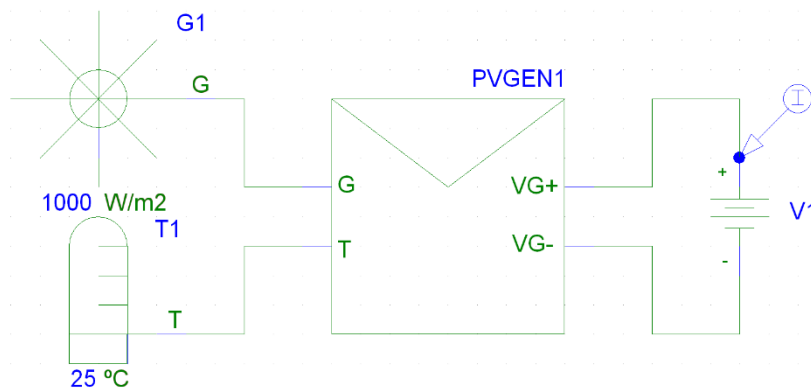


2. Select the PV generator in the Part Browser dialog box typing PVGEN in the Part Name text box. Other way is clicking in the Libraries to view the Library Browser dialog box, select pvspice.slb, and select PVGEN from the pvspice list of components.


- Click Place to place the PV generator in the schematic.



- Repeat steps 1 to 3, place the following components: IRRADIANCE, TEMPERATURE and VDC, arrange the following circuit interconnecting the components with wires , and place a current marker  at the positive terminal of V1.



4-1-2 Editing symbol attributes

- Select the PV generator symbol (PVGEN1), click the Edit Attributes button , and set the following attributes:

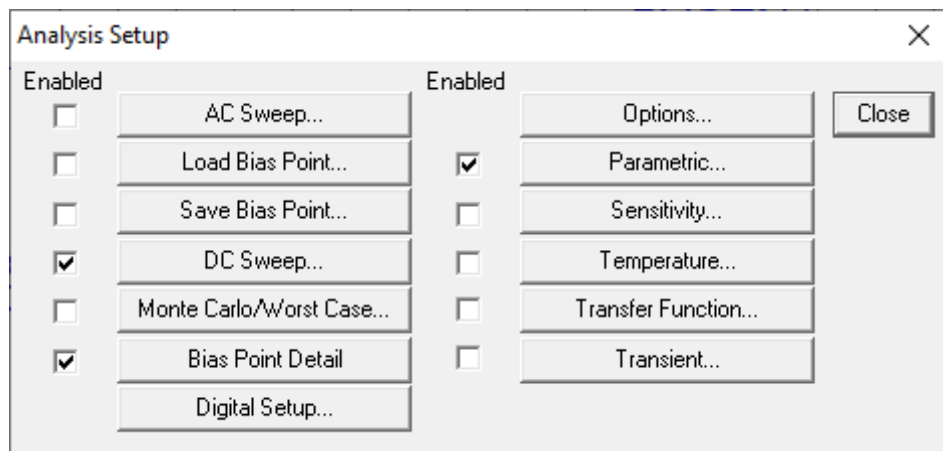
Attribute	Value	Units
ALFA	0	$^{\circ}\text{C}^{-1}$
BETA	-0.004	$^{\circ}\text{C}^{-1}$
ISCM	3.2	A
NOCT	47	$^{\circ}\text{C}$
NPC	1	
NPM	1	
NSC	36	
NSM	1	
PMM	50	W
VOCM	21.5	V

- Repeat the step 1 with the other symbols setting the following attributes:

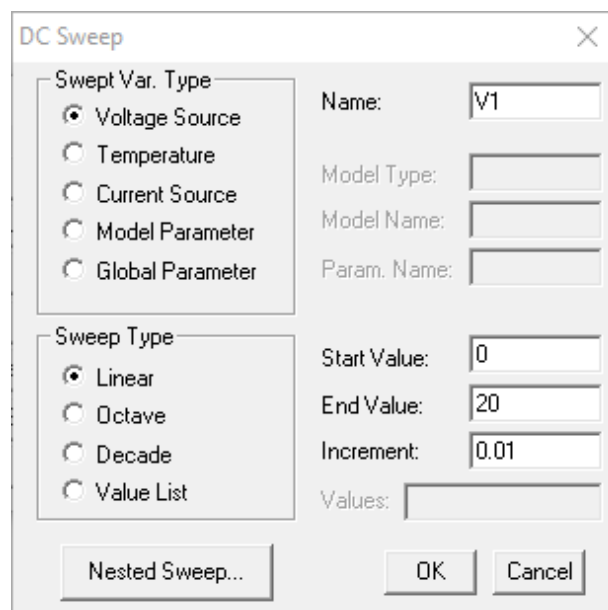
G1 (IRRADIANCE):	DC=1000
T1 (TEMPERATURA):	DC=25
V1 (VDC):	DC=12

4-1-3 Configuring the analysis and running the simulation

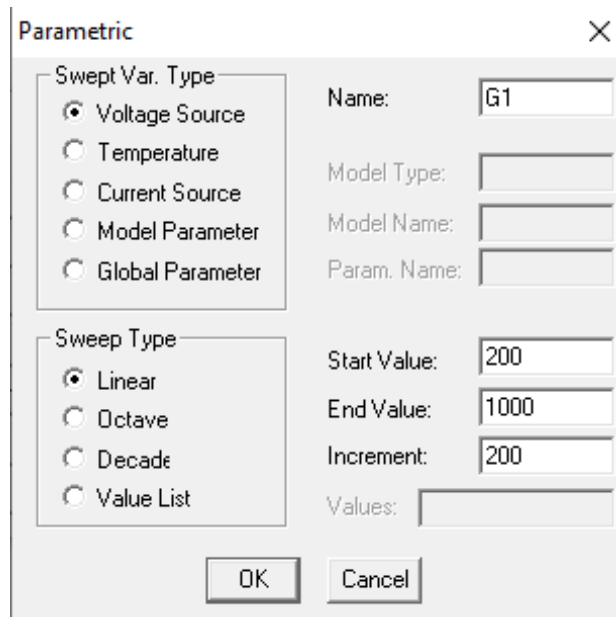
- From the Analysis menu, select Setup



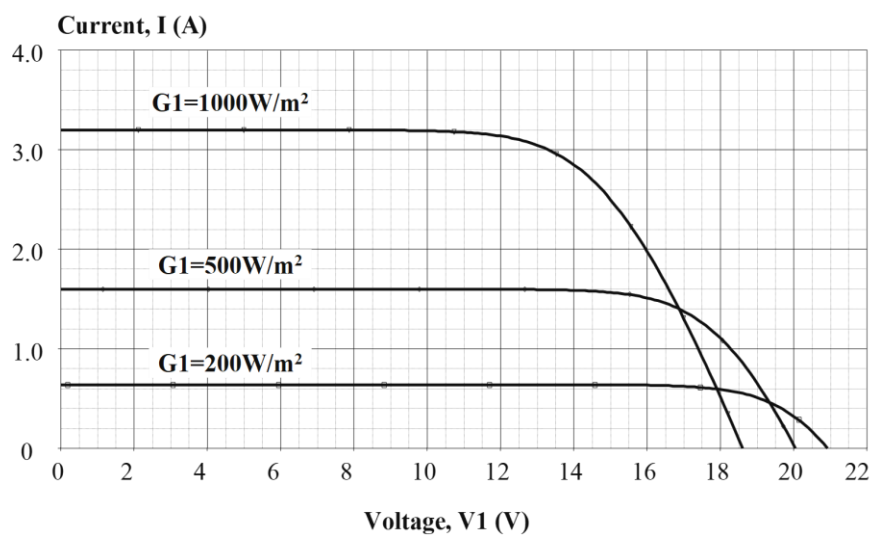
- Select DC sweep ... and set a linear sweep for V1 from 0 to 20V.



- Select Parametric ... and set a linear sweep for the irradiance (voltage source G1) from 200 to 1000 W/m² with an increment of 200 W/m².



- Finally, in the Analysis menu, click on Simulate (F11). The tool Probe will show you the current-voltage curves of the PV generator similar to the figure displayed below.



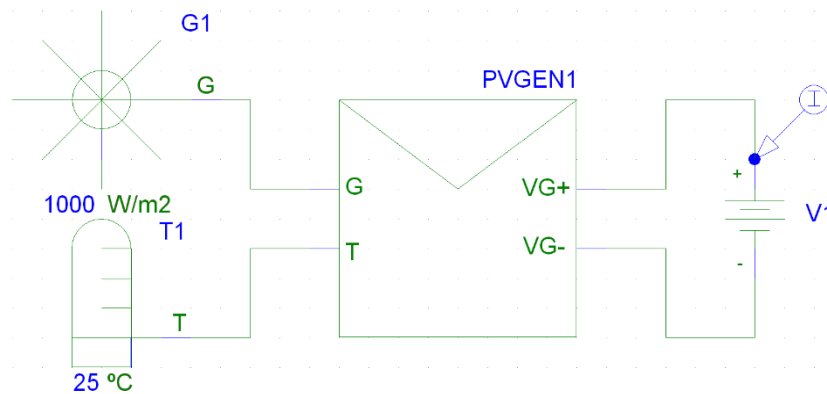
4-2 Examples

Next sections describe several simulation examples. The first one is the circuit of the simulation described in the previous section. With PSpice student 9.1 only examples 1 to 3 can be simulated owing to the limitation of components of this demo version.

4-2-1 Example 1

File: Example 1 - PV generator curves.sch

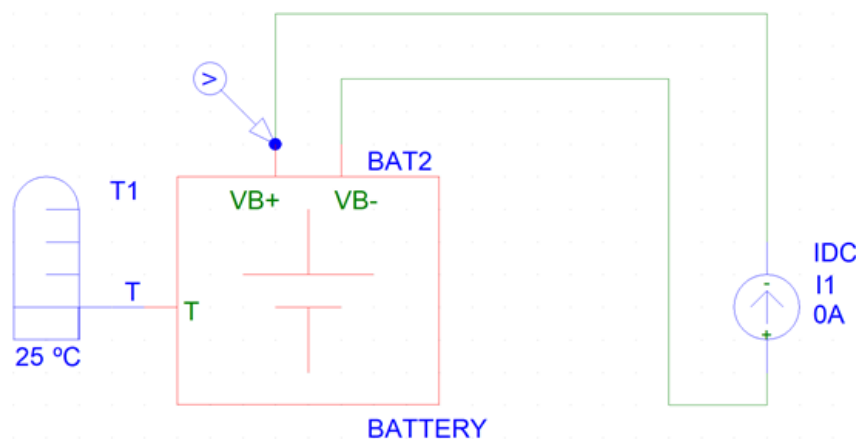
Displays current-voltage curves of a PV generator as a function of the incident irradiance.



4-2-2 Example 2

File: Example 2 – Battery charging curves.sch

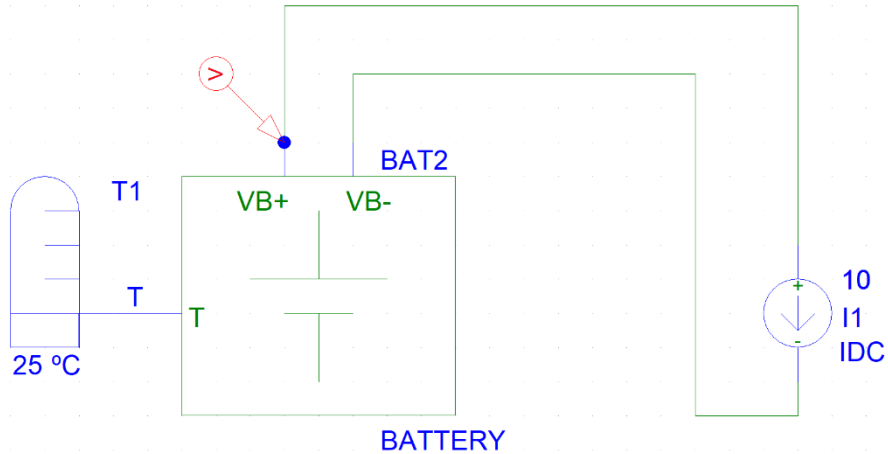
Displays charging curves, battery voltage as a function of time, of a 100 Ah lead acid battery at different currents with an initial SOC=50%. In simulations, each second represents one hour.



4-2-3 Example 3

File: Example 3 – Battery discharging curves.sch

Displays discharging curves, battery voltage as a function of time, of a 100 Ah lead acid battery at different currents with an initial SOC=100%.

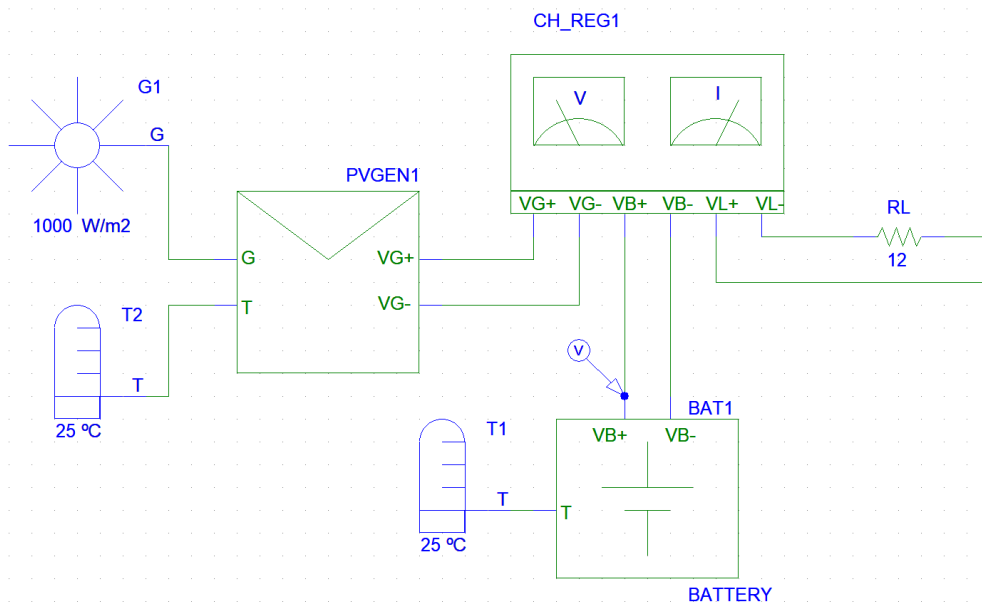


4-2-4 Examples 4 y 5

Files: Example 4 – Overcharge.sch

Example 5 – Overdischarge.sch

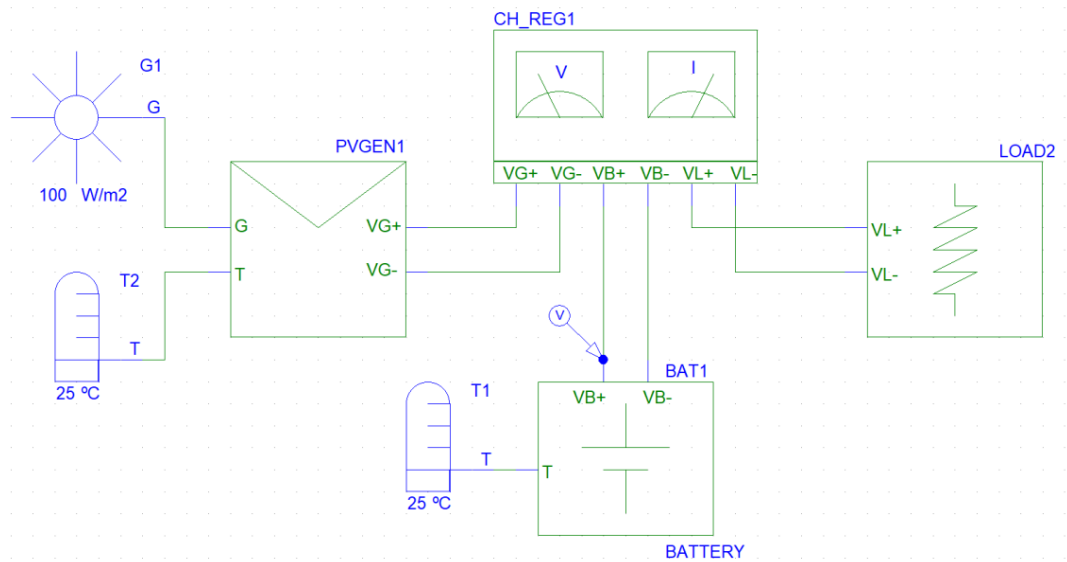
Displays the battery voltage and the operation of the charge regulator during the operation of a stand-alone PV system.



4-2-5 Example 6

File: Example 6 – Load.sch

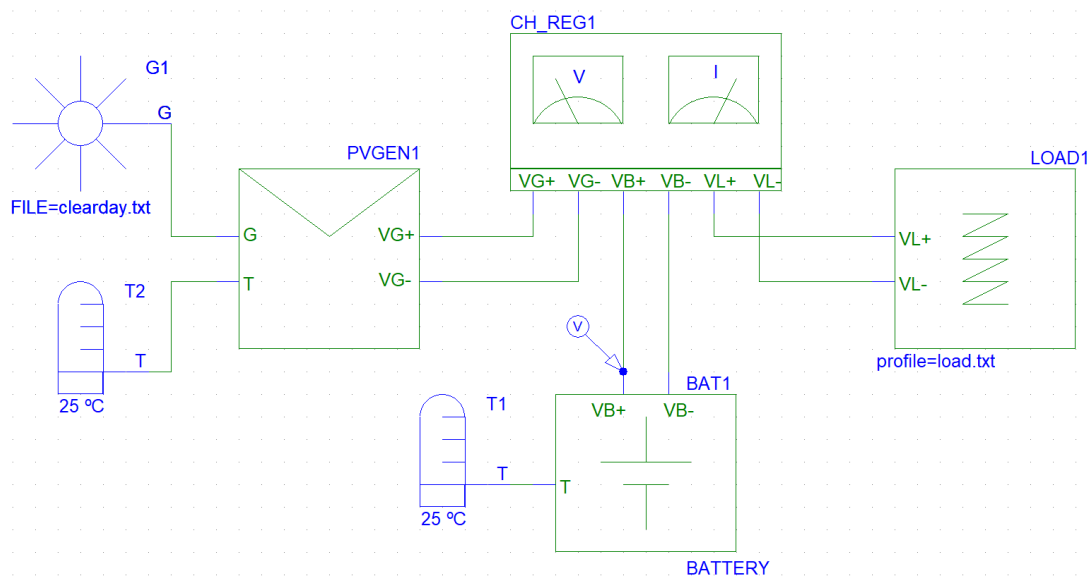
Illustrates the operation of the constant load model.



4-2-6 Example 7

File: Example 7 – Day simulation.sch

Simulates the system operation during a day using solar irradiance and load profiles from the text files clearday.txt and load.txt, respectively.



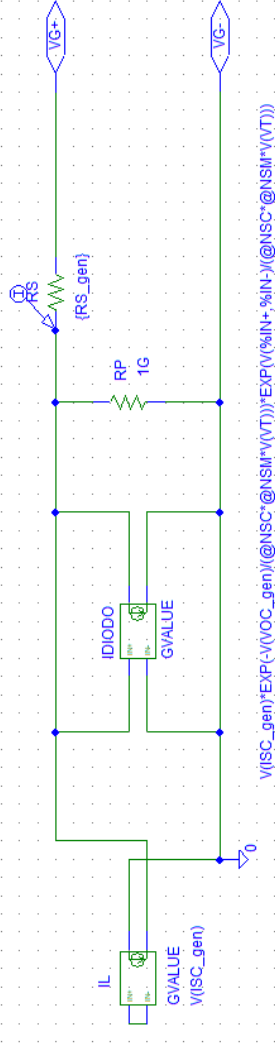
References

- [1] J. Muñoz and P. Díaz. A virtual photovoltaic power systems laboratory. IEEE EDUCON 2010 Conference, Madrid, Spain, 2010, pp. 1737-1740, <https://www.doi.org/10.1109/EDUCON.2010.5492411>. (Preprint available from <https://blogs.upm.es/javiermunoz/>).
- [2] J. Muñoz, L. Marroyo, M. Collares-Pereira, N. Tyutyuyndzhiev, M. Conlon, A. Elmoussaoui, B. Wilkin. An Open-Source Simulation Tool of Grid-Connected PV Systems. 28th European Photovoltaic Solar Energy Conference and Exhibition, pages 3882–3887. <https://www.doi.org/10.4229/28thEUPVSEC2013-5BV.4.18>
- [3] PSpice Schematics. Schematic Capture Software. User's Guide. Cadence Design Systems 1985-2000.
- [4] PSpice® A/D. Schematics version. User's Guide. Cadence Design Systems 1985-2000.
- [5] L. Castañer, R. Aloy and D. Carles. Photovoltaic System Simulation Using a Standard Electric Circuit Simulator. Progress in photovoltaics: Research and applications, vol 3, 239-252 (1995).
- [6] E. Lorenzo. Solar electricity. Engineering of photovoltaic systems. Progensa, 1994.
- [7] J. B. Copetti, E. Lorenzo, F. Chenlo. A General Battery Model for PV System Simulation. Progress in photovoltaics: Research and applications, vol 1, 283-292 (1993).
- [8] J.B. Copetti, F. Chenlo. Lead/acid batteries for photovoltaic applications. Test results and modeling, Journal of Power Sources, Volume 47, Issues 1–2, 1994, Pages 109-118, [https://doi.org/10.1016/0378-7753\(94\)80054-5](https://doi.org/10.1016/0378-7753(94)80054-5).
- [9] MicroSim Application Notes. Version 8.0. June, 1997. MicroSim Corporation.

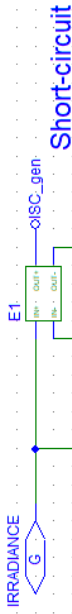
Annex. Circuit models

PV generator

PV generator



$$V(SC_gen) * \exp(-V(VOC_gen) / (@NSM * V(T))) * \exp(V(IN) + \%IN) / (@NSC * @NSM * V(T)))$$



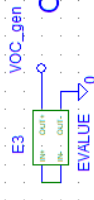
Short-circuit

$$@NPM * @ISC * V(IN) + \%IN * 1000 * (1 + @ALFA * V(TC) - 25)$$



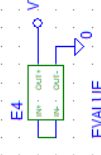
Cell temperature

$$V(IN) + (@NOCT - 20) / 800 * V(IN) +$$



Open-circuit voltage

$$@NSM * VOC * (1 + @BETA * (V(TC) - 25))$$



Thermal voltage, VT

$$1.38E-23 * V(TC) + 273 / 1.6E-19$$

Series resistances calculation

1° Cell parameters

PARAMETERS:
 ISC_cell { @ISC @NPC }
 VOC_cell { @VOC @NSC }
 PIM_cell { @PIM @NSC @NPC }

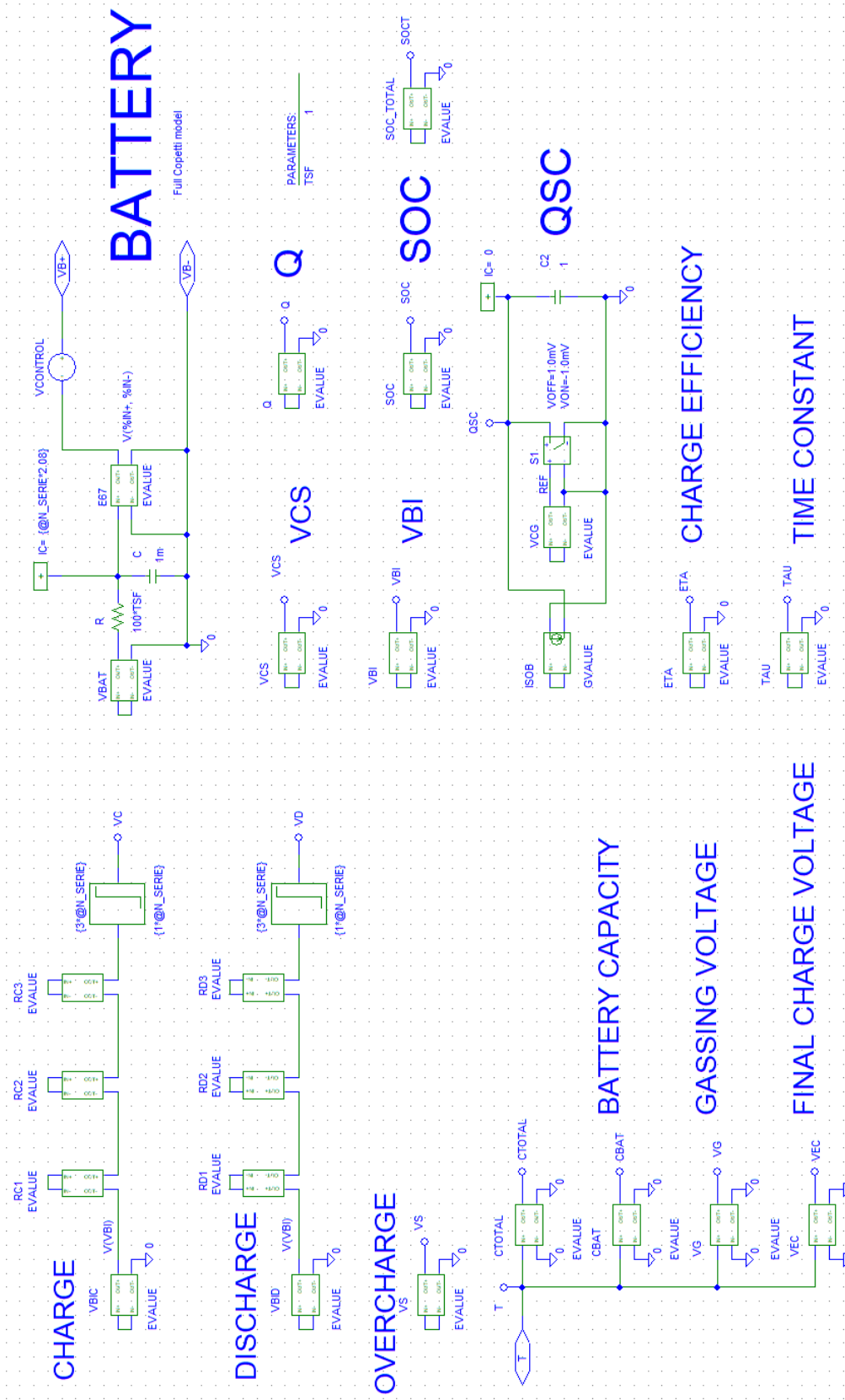
2° Other parameters ($\tau = 300K$)

PARAMETERS:
 FF { PIM_cell / (VOC_cell * ISC_cell) }
 FFO { (VN - LOG(VN - 0.72)) / (VN + 1) }
 VN { VOC_cell / (1.38E-23 * 300 / 1.6E-19) }

3° Series resistances

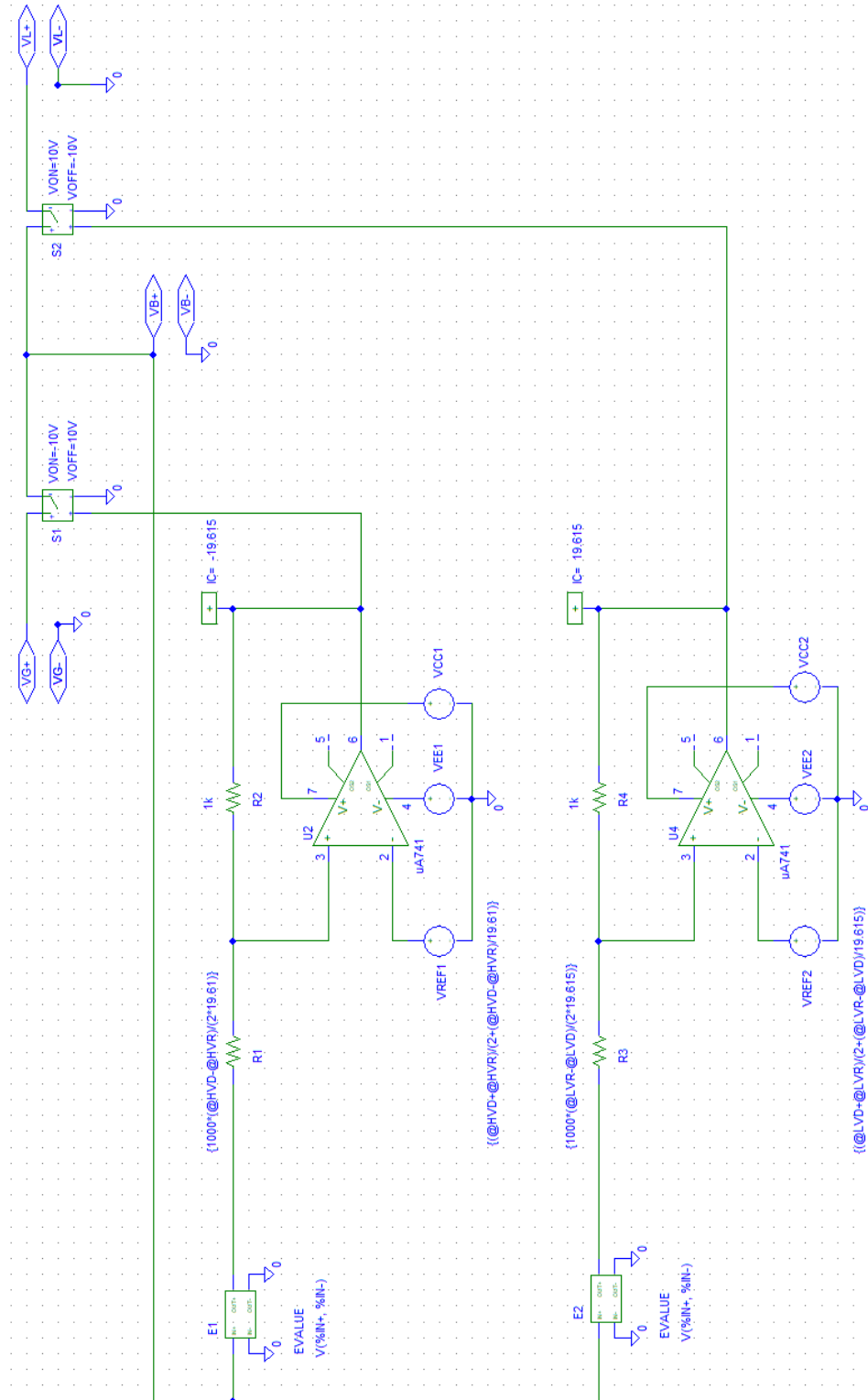
PARAMETERS:
 RS_cell { (1 - FFO) * VOC_cell / ISC_cell }
 RS_gen { (@NSC * @NSM) / (@NPC * @NPM) * RS_cell }

Battery



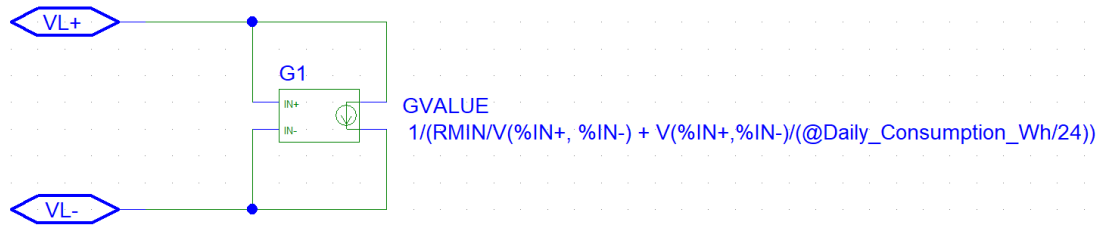
Charge regulator

ON/OFF CHARGE REGULATOR



Loads

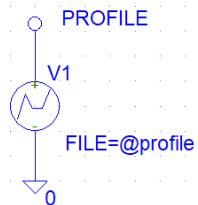
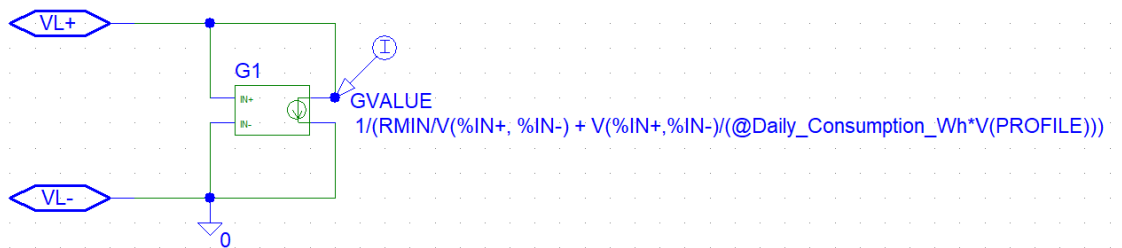
CONSTANT LOAD



PARAMETERS:

RMIN 1

VARIABLE LOAD



PARAMETERS:

RMIN 1