PSpice Simulation of PhotoMOS Relays

SPICE is a powerful general purpose analog and mixed-signal circuit simulator that is used to verify circuit designs and to predict the circuit's behaviour. The original Spice program has given rise to a variety of commercial implementations. Among them is PSpice, a version for use on a PC which provides a comfortable graphic user interface. For this reason, PSpice has become very common and is widely used by engineers. An essential requirement for achieving quick and accurate simulation results is a well assembled library containing the circuit's parts. A lot of libraries come with the standard installation. However not every part is included in these libraries. Therefore Panasonic Electric Works provides PSpice libraries for its PhotoMOS relays.

The basic operation of a PhotoMOS relay is described below: on the input side of the semiconductor relay a light emitting diode is located. If a current flows through the LED it starts emitting light. This light is converted to a photo voltage by solar cells located at least 0.4 mm away. With the help of a control MOSFET, this photoelectric voltage drives the gates of two source-coupled DMOSFET. Hence the output of the relay is controlled.

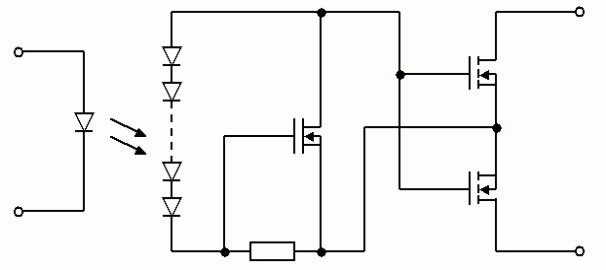

Figure 1: PhotoMOS equivalent circuit

In order to build a model for the PhotoMOS relay, existing PSpice parts like diodes, resistors, MOSFETs, and controlled sources are used. The input characteristic is described by a diode model. The LED's light emission on the input side is proportional to the current through the LED. This current is sensed with the help of a voltage source, and a controlled current source reflects the intensity of detected light by the solar cells. Since solar cells have a p-n-junction they are represented by diodes, connected in parallel to the current source reflecting the detected light intensity. Capacitance and serial resistance of the solar cells are included in the diode model. The current flowing through the solar cells results in a voltage drop which is used to switch the output of the relay. The MOSFET for controlling the photo voltage and the resistance between gate and source of the control transistor can be easily modelled by using existing PSpice models. Together these parts comprise the DIC, which has a nonlinear I-V characteristic due to the behaviour of the solar cells.

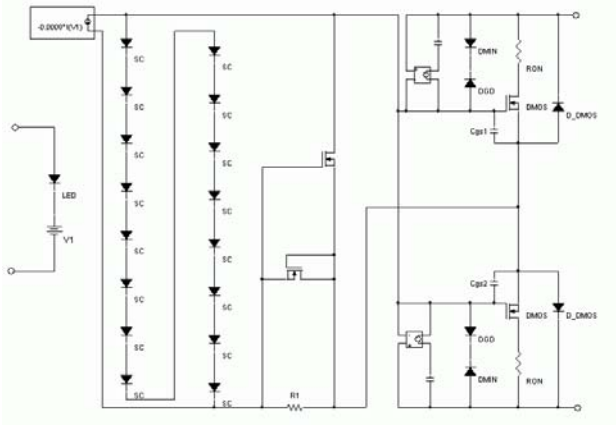

Figure 2: PSpice Model Subcircuit

The nonlinear source is used for driving the gates of two source-coupled DMOSFET providing bi-directional switching capability. The model of the DMOSFET contains a simple MOSFET, capacitors, resistors, and diodes. Since the PSpice models for MOSFETs were developed for lateral structured field effect transistors, the vertical structure of the DMOSFET transistor needs some additional components to simulate its behaviour.
The primary device for gate controlled switching of a DMOSFET is a level 1 MOSFET model that includes threshold voltage. The drain resistance is modelled by a separate resistor, making it possible to include 1st and 2nd order temperature effects. The parasitic diode of a DMOSFET is included by implementing a diode across drain and source. Avalanche breakdown, leakage current and output capacity of the device are modelled by this diode. Gate to source capacity can be easily included while modelling the capacitance between drain and gate is more complex. In order to achieve a square root dependency with drain to source voltage, a capacitor and a diode in parallel are used. Voltage between drain and gate, and a second diode prevent steady-state currents and provide curve-fitting possibilities.

The model subcircuit accurately simulates the PhotoMOS relay's behaviour. It includes temperature effects as well as voltage dependencies and nonlinear effects. If you have questions or requests regarding PSpice simulation of our PhotoMOS relays please contact: info-eu@eu.pewg.panasonic.com