Panasonic
INDUSTRY

PSpice simulation of PhotoMOS® relays

PSpice simulators allow users to conveniently verify circuit designs and predict behaviour on their PCs. Panasonic Industry offers the complete component library necessary to run simulations for PhotoMOS® relays.

Application Note

Your Committed Enabler

IN Your Future

Circuit simulation

Product

PSpice simulation libraries for PhotoMOS® relays

Purpose

Design PhotoMOS® circuits more accurately and efficiently with a well-assembled component library.

Features

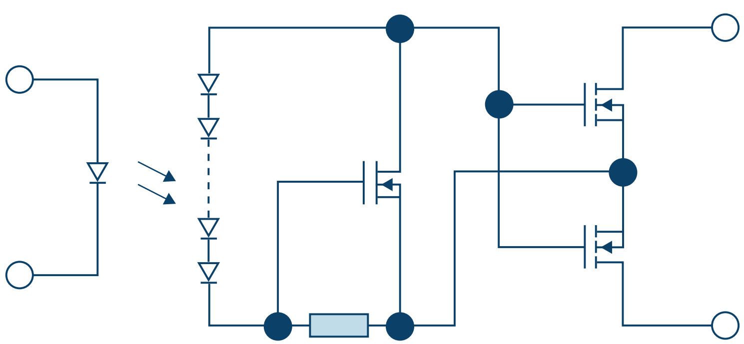
Includes all parts of a PhotoMOS® relay
Realistic and detailed simulation
Takes into account temperature effects, voltage dependencies and nonlinear effects

Circuit Simulation

Facts & Figures

SPICE is a powerful general-purpose analog and mixed-signal circuit simulator used to verify circuit designs and predict circuit behaviour. One successful commercial implementation of this tool is PSpice, a PC version with a convenient graphic user interface. PSpice has become a popular simulator among engineers, but in order to achieve quick and accurate simulation results, a well-assembled library containing all the circuit's parts is necessary. While many libraries are already included in the standard programme, this is not the case for more specialised circuit configurations, such as PhotoMOS® relays.

This type of relay typically consists of several elements: An LED diode on the input side emits light to a photodiode array (PDA) with solar cells, which converts the incoming light into electrical current and voltage. These in turn drive two power DMOSFETs on the output side. An intermediate control circuit is responsible for the safe and reliable turn-on and turn-off of the output DMOSFETs once a certain trigger current is reached.
In order to build a model for a PhotoMOS® relay with PSpice, engineers need to include all these elements in their simulation. 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. A voltage source senses this current, and a controlled current source reflects the intensity of light detected by the PDA. Since the solar cells making up this PDA have a p-n-junction, they are themselves represented by diodes connected in parallel to the current source reflecting the detected light intensity. The diode model also takes into account the capacitance and serial resistance of the solar cells. The current flowing through these cells results in a voltage drop, which is used to switch the output. The MOSFET serving as the control circuit can be easily modelled by using existing PSpice libraries.

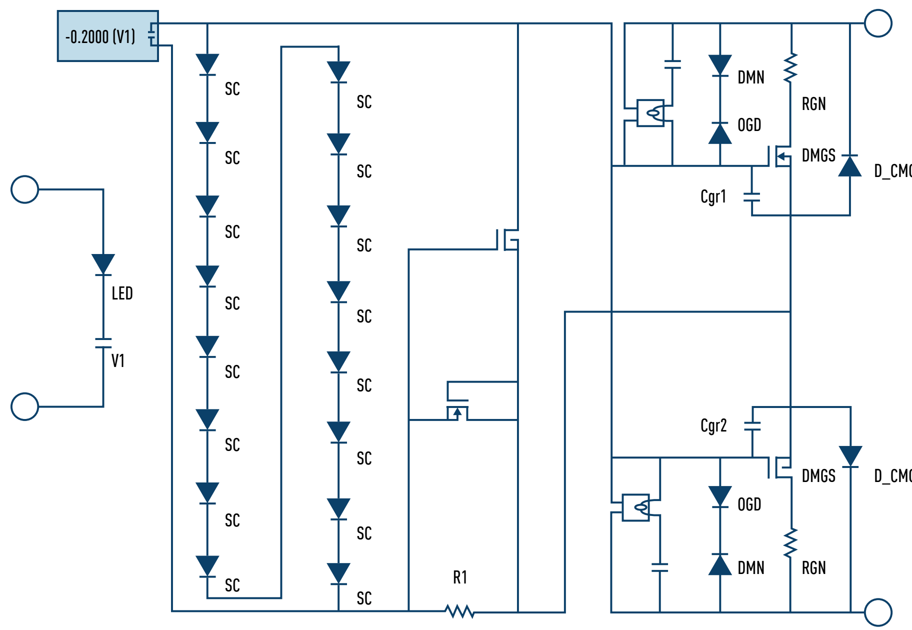


IN Your Future

Circuit Simulation

Facts & Figures

Together, these parts comprise the DIC, which has a nonlinear I-V characteristic due to the behaviour of the solar cells. The nonlinear source is used for driving the gates of two source-coupled DMOSFETs, providing bidirectional switching capability. The model for these DMOSFETS contains a simple MOSFET, capacitors, resistors, and diodes. It is here where the standard libraries included in PSpice usually reach their limits: Since the PSpice models for MOSFETs were developed for lateral structured field effect transistors, additional components are needed to simulate the behaviour of the vertical structure of the DMOSFET transistor. The primary device for gate-controlled switching 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 also modelled by this diode. But while gate-to-source capacity can be easily included, 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 are used in parallel. Voltage between drain and gate, together with a second diode, prevent steady-state currents and provide curve-fitting possibilities. By including this complex structure in their circuit models, engineers can more accurately simulate the behaviour of a PhotoMOS® relay.



IN Your Future

Learn more about PhotoMOS® technology

Get here the PhotoMOS® relay App!

Application Note - How to solve various tasks with PSpice simulation of PhotoMOS® relays
Date: August 2024
Contact: Panasonic Industry Europe GmbH, photomos@eu.panasonic.com

Panasonic INDUSTRY

Your Committed Enabler