



# Using P-SPICE Models for Vishay Siliconix Power MOSFETs

By Kandarp Pandya

## Introduction

Vishay provides P-SPICE models for each of its power MOSFETs on the Vishay Web site, allowing design engineers to evaluate device performance at any stage of the design process. This application note explains how to download the P-SPICE models from the Vishay web site, prepare them for use on OrCAD electrical simulation software, build an electrical schematic circuit, and run electrical simulations.

## Downloading the P-SPICE Model

For this application note, we will use the 30 V n-channel SiR158DP power MOSFET as an example. So our first step will be to create a part folder called SiR158DP\_PS in the user OrCAD data library. An example of the user library path for a part folder can look like: C:\OrCAD\_Data\SiR158DP\_PS. This means the folder named SiR158DP\_PS is stored in a folder name OrCAD\_Data, on "C" drive

To find the part you're interested in, browse by specification range or package type under MOSFETs from the Vishay home page, or do a search on the part number (figure 1). Either type of search will bring you to a page with all the documentation for a given device (figure 2).

On the power MOSFET product information page, scroll down to select the P-SPICE Model Library (\*.LIB) file. Right click on "SiR158DP-P LIB." Save the target in the local user data part library folder "SiR158DP\_PS" after changing the extension to lib -- "SiR158DP.LIB." Click the back arrow at the top left to return to the product information page.

Now scroll down to P-SPICE Model Library (\*.OLB). Right click on "SiR158DP-P OLB." Save the target in the local user data part library after changing the file extension to olb - "SiR158DP.OLB."

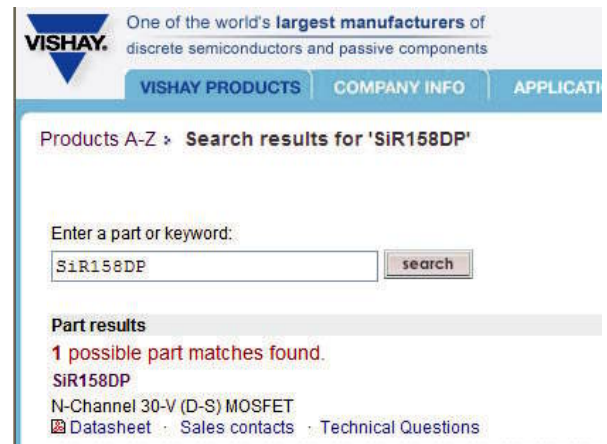


Fig. 1 - Search Results

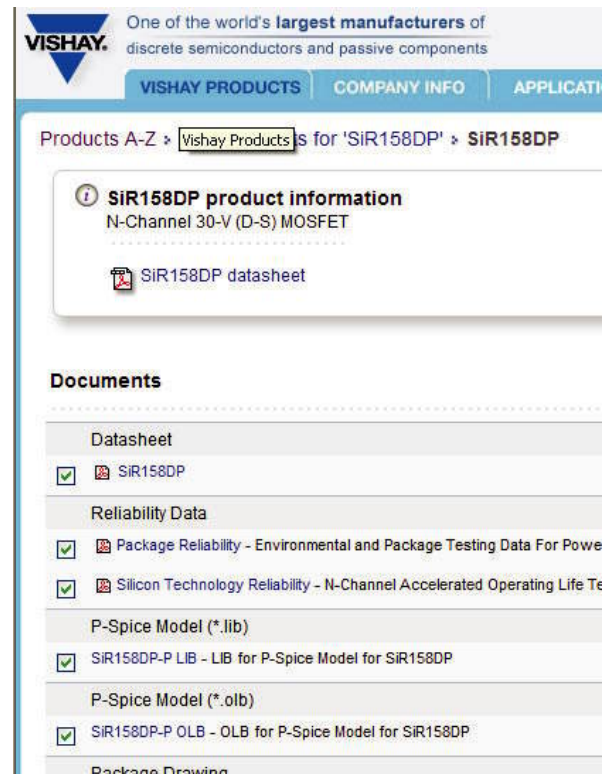


Fig. 2 - Product Information

### Using P-SPICE Models for Vishay Siliconix Power MOSFETs

#### Adding the Part Library to an OrCAD Capture Project

Use the mouse to click on the capture icon on your desktop, or to use the program file path see figure 3 to launch OrCAD capture.

See figure 5 below. Name the project same as the part number followed by underscore and PS. Select radio button Analog or Mixed A/D. Browse to point to location C:\OrCAD\_Data\SiR158DP\_PS. Click OK.

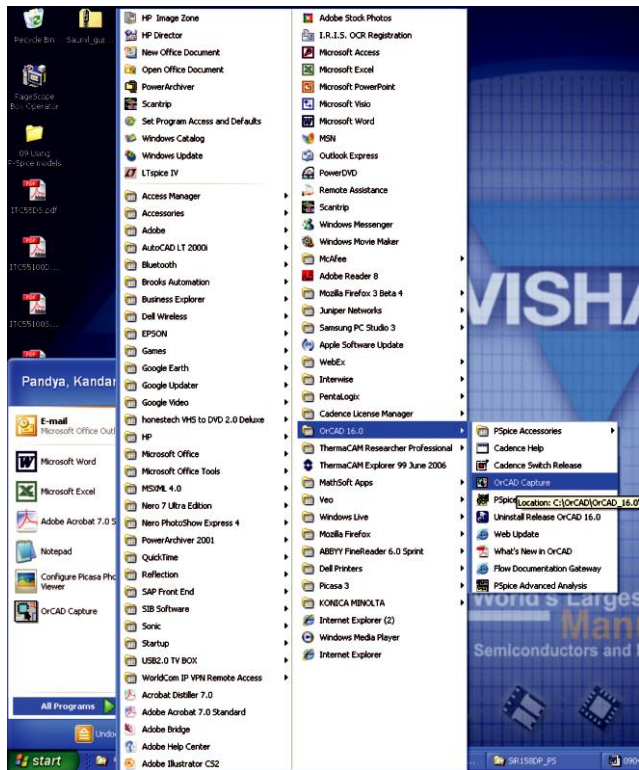


Fig. 3 - Launch OrCAD Capture



Fig. 5 - New Project Name, type, and location

Select the settings as shown in figure 6 and click OK open a blank schematic page.

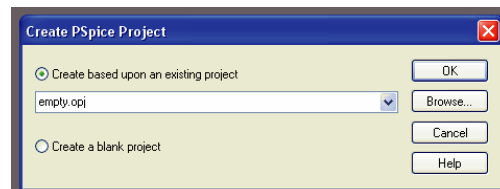


Fig. 6 - Project bases

Start a new project by selecting File, New, Project as shown in figure 4.

Open the OrCAD capture project. Click "Place Part" icon on the pull-down menu on right side. Select "Add Library." Go to the location of the part library folder "SiR158DP\_PS." The "SiR158dp.olb" file is listed. See figure 7. Select this file and click open to add the library to the design. The part window displays the SiR158DP in the library listing. Select the library "SiR158DP" that is listed in the part window. To add the parts to the schematic page, select the same and click "OK." See figure 8. The MOSFET symbol attached to the cursor is ready for placement on the schematic page. Every mouse click places one part symbol at a desired location. See figure 9.

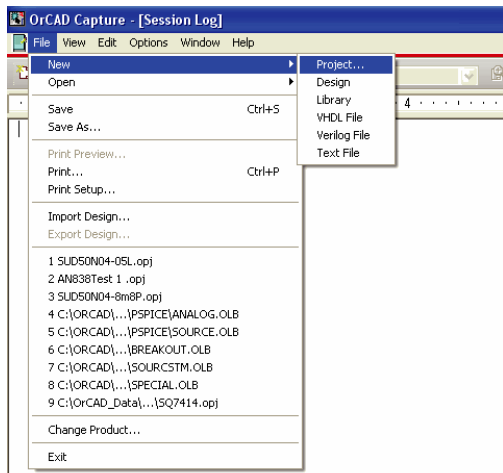


Fig. 4 - New Project in OrCAD Capture

APPLICATION NOTE

## Using P-SPICE Models for Vishay Siliconix Power MOSFETs

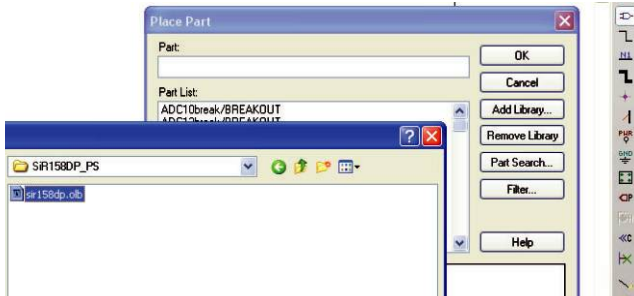


Fig. 7 - Add Part Library

To add the part library file to the design, click "Library" under Category (red arrow). Use "Browse" (blue arrow) for the library file location. See figure 11. Select the "sir158DP.lib" file and click open. The file name with path appears in the browse bar. Click "Add to Design" (red arrow). The library file appears in the configuration files listing (blue arrow). See figure 12. Click "Apply" and "OK" to close the window.

The OrCAD capture is now ready for simulating the SiR158DP power MOSFET.

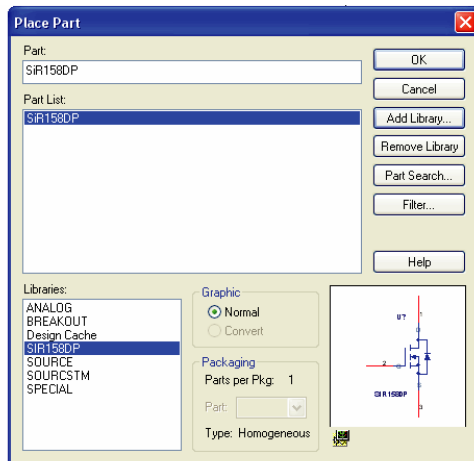


Fig. 8 - Place Part on Schematic Page

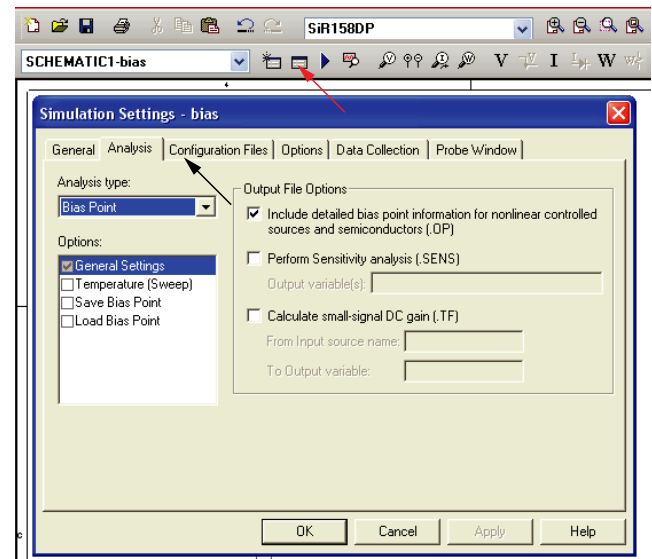


Fig. 10 - Simulation Settings

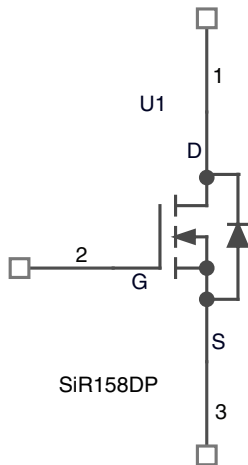


Fig. 9 - Part Symbol on Schematic Page

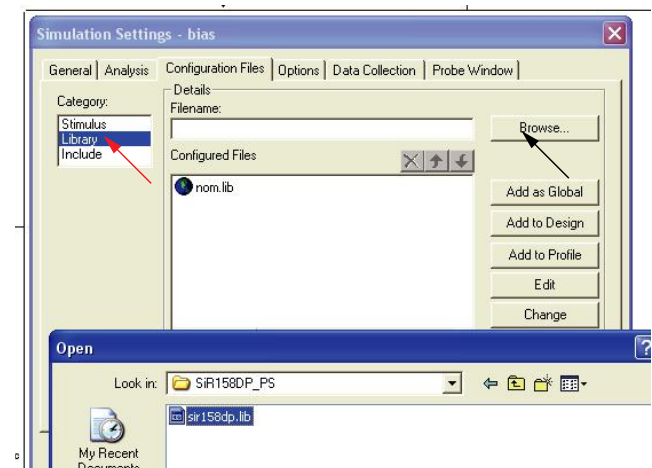


Fig. 11 - Brows and Add Library File

### Configuring the Part Library in the OrCAD capture Project

On the schematic page, click the tab to open "Simulation Settings" as shown by the red arrow in figure 10. Click the tab "Configuration Files" as shown by the blue arrow.

### Using P-SPICE Models for Vishay Siliconix Power MOSFETs

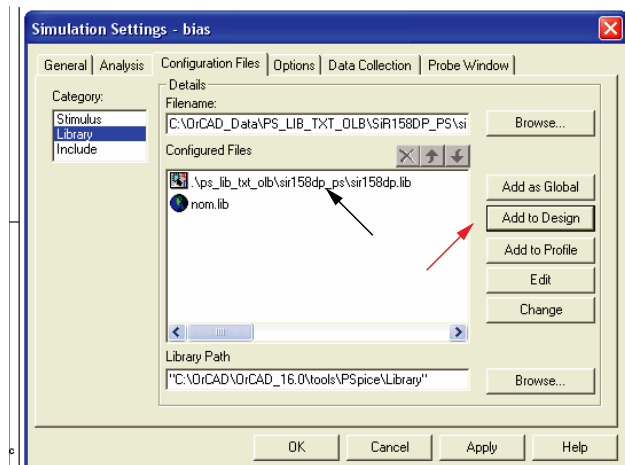


Fig. 12 - Add to Design

#### Summary

Using Vishay Siliconix power MOSFET P-SPICE models on the OrCAD capture-P-SPICE platform involves four steps:

- Download \*.OLB - part symbol file
- Download \*.LIB - P-SPICE library file
- Add \*.OLB file to part library of design
- Add \*.LIB file to configuration files -- category library

The SiR158DP example is repeated in more detail in Appendix A.

## Using P-SPIICE Models for Vishay Siliconix Power MOSFETs

### Appendix A

#### Working Example of Using P-SPIICE Model for Electrical Simulation

This exercise uses the OrCAD platform to validate the basic functionality of the electrical schematic part symbol file name SiR158DP.OLB and the P-SPIICE library file SiR158DP.LIB used for electrical simulation.

##### Step 1: Download:

Create a new folder name: SiR158DP\_PS in OrCAD Data Folder as described in the application note proceed with down loads. Link: [www.vishay.com/product?docid=64730&query=SiR158DP](http://www.vishay.com/product?docid=64730&query=SiR158DP)

Right click and save target SiR158DP\_P\_OBL as SiR158DP.OLB in the folder SiR158DP\_PS.

Right click and save target as SiR158DP\_P\_LIB as SiR158DP.LIB in the folder SiR158DP\_PS.

##### Step 2: OrCAD Project: SiR158DP\_PS:

Create a new project name: SiR158DP in OrCAD Capture and save in the folder SiR158DP\_PS. Add the part schematic symbol SiR158DB.OLB to the part library as discussed in the application note. Build a schematic as shown in figure 1.

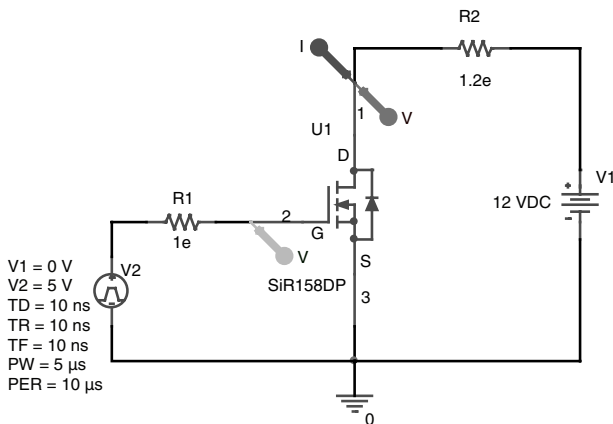


Fig. 1 - Electrical Schematic Diagram

##### Step 3: Simulation settings:

Open simulation settings window:

- (1) Click on tab "Analysis." Select analysis type "Time Domain (Transient)." Set Run Time to 12 μs.
- (2) Click on tab "Configuration Files." Select Category "Library". Under Details, Browse and Add to Design the P-SPIICE model file SiR158DP.LIB.

##### Step 4: Run Simulations and Results

- (1) Figure 2 shows turn-on excursion of MOSFET

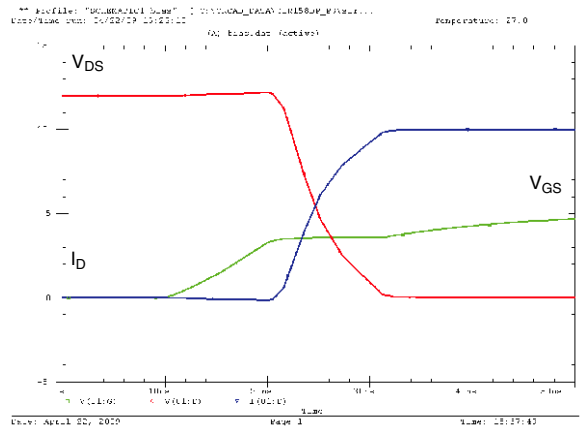


Fig. 2 - Turn-On Excursion of SiR158DP

We can observe the clean turn-on of MOSFET, validating its functionality in the turn-on area.

- (2) Figure 3 shows turn-off excursion of MOSFET. The clean turn-off indicates trouble-free model behavior.
- (3) Figure 4 shows one complete cycle excursion of the MOSFET and validation of the model.

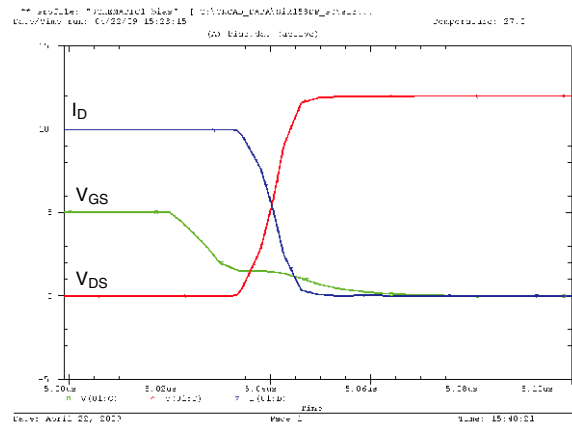


Fig. 3 - Turn-Off Excursion of SiR158DP

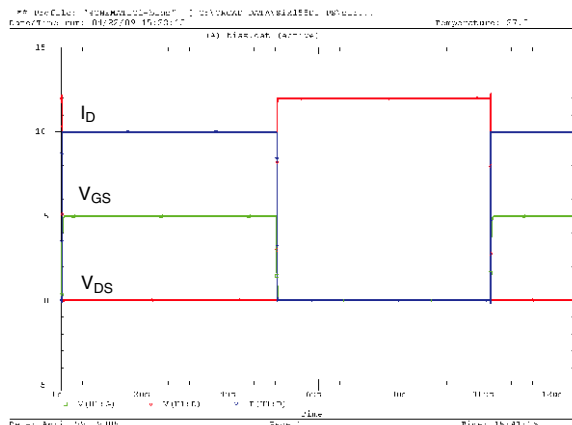


Fig. 4 - One Complete Cycle Excursion of SiR158DP