UM2167
User manual

OrCAD PSpice model usage instructions

Introduction

This document describes how to use ST's PSpice models available for SMPS devices. The models are useable in the OrCAD system environment of Cadence Design Systems and will not work in other simulation platforms. Furthermore, we recommend using the latest version of OrCAD to avoid convergence problems and speed up the simulation.

PSpice models describe the characteristics of typical devices and don't guarantee the absolute representation of product specifications and operating characteristics; the datasheet is the only document providing product specifications.

Although simulation is a very important tool to evaluate the device's performance, the exact device's behavior in all situations is not predictable, therefore the final laboratory test is necessary.

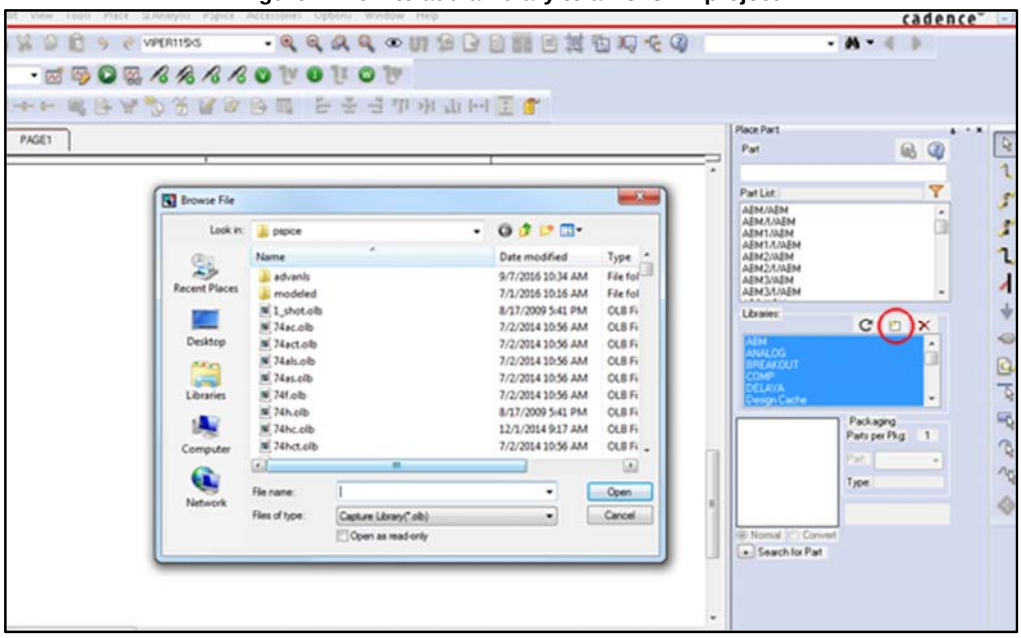
1 PSpice models – instructions to simulate

The model is included in a zip package that contains two files:

- “<DEVICE_NAME>.OLB”, the symbol to place in the schematic
- “<DEVICE_NAME>.LIB”, the library containing the actual implementation of the model

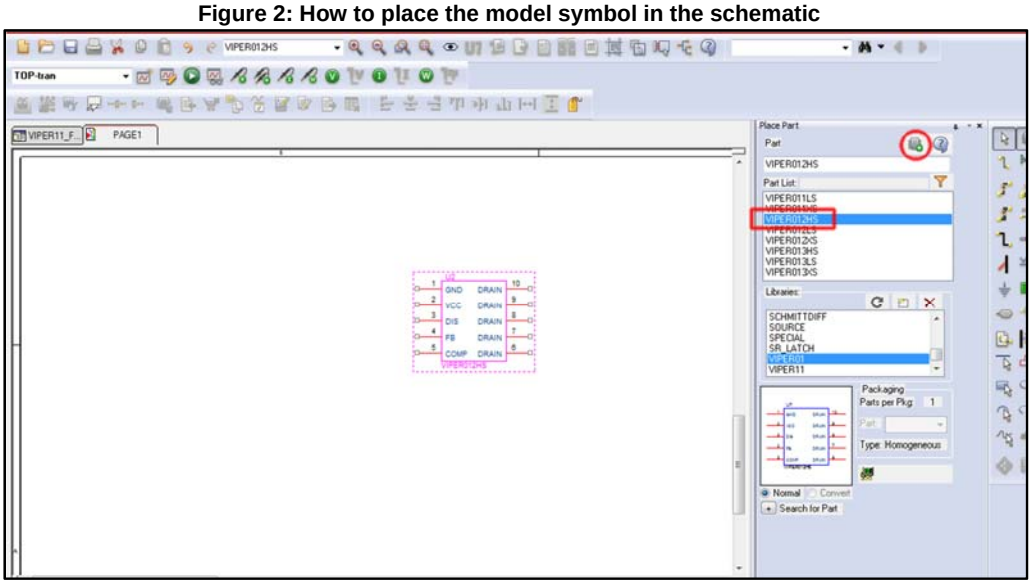
To use the model, unzip the package and add the library “<DEVICE_NAME>.OLB” to your OrCAD project, as illustrated in the following figure: click the button highlighted by the red circle and browse to search the library you want to add.

Figure 1: How to add a library to an OrCAD project



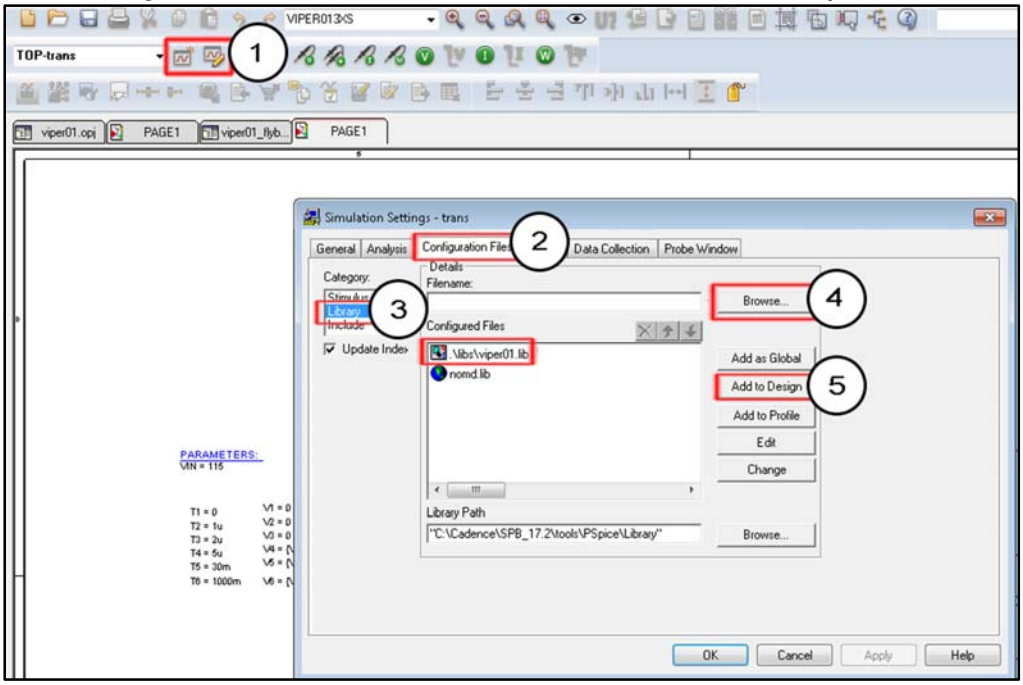
Then, select the part of interest from the “Part List” and place it on the schematic as shown in the following figure.

Figure 2: How to place the model symbol in the schematic



Then, create or open your simulation profile (1), go to “Configuration Files” (2), select the category “Library” (3), browse and search the file <DEVICE_NAME>.lib (4) and click to “Add to Design” (5), as shown in the following figure.

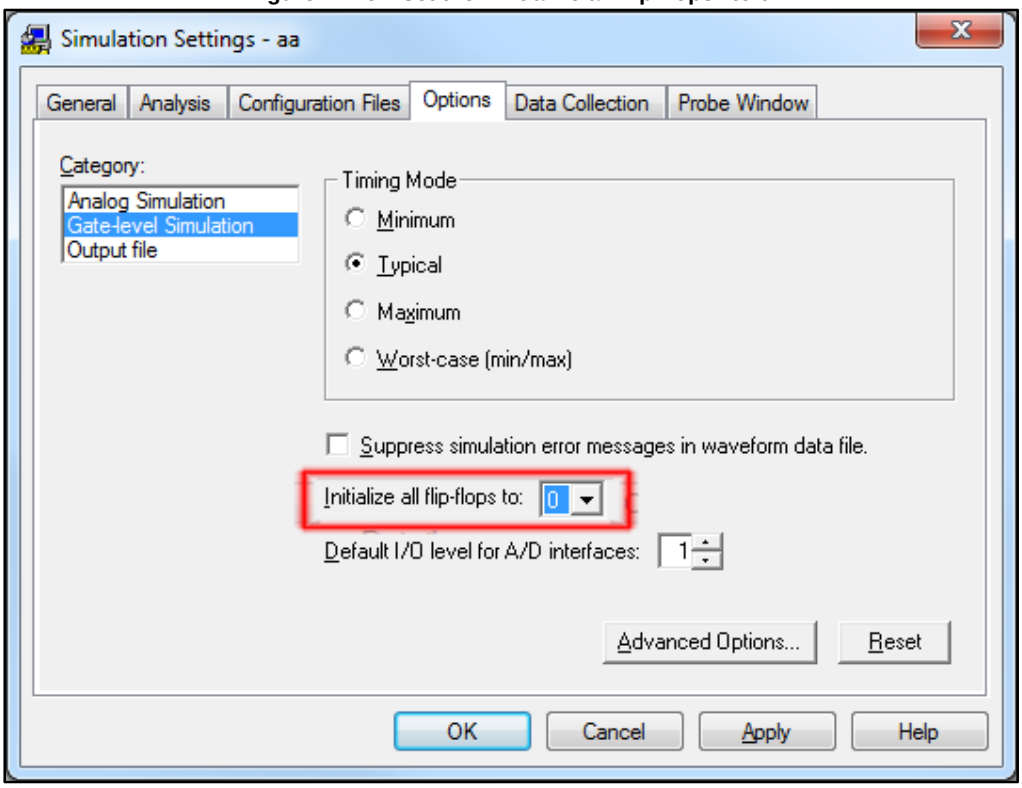
Figure 3: How to add the reference to the model to the simulation profile



2 Important notes

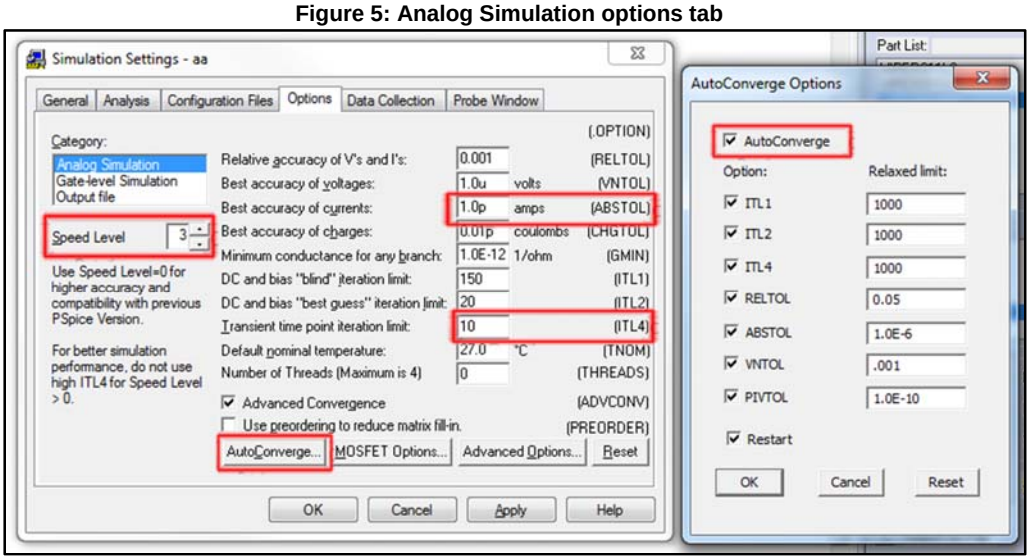
- Before starting the simulation, make sure that the option “Initialize all flip-flops” is set to “0” as shown in the following figure: in your simulation profile go to “Options” tab, select the Category “Gate-level Simulation” and select “0” in the dropdown list “Initialize all flip-flops to:”.

Figure 4: How set the "Initialize all flip-flops" to 0



- Almost all our converter models need to detect a rising edge of the input voltage to start the power-up sequence. For this reason, do not use a VDC component as VIN source but a VPULSE or a VPWL.
- In case you should experience convergence problems, try to play around with the “Analog Simulation” options of your simulation profile. We suggest to apply the following changes (sorted by priority):
  - Change the parameter ABSTOL from 1.0p to 1.0n.
  - Increase the value of the parameter ITL4 from 10 to 50 or 100.
  - Increase the Speed Level from 3 to 4 or 5.
  - Set the option “Auto Convergence”.

Figure 5: Analog Simulation options tab



3 Revision history

Table 1: Document revision history

Date	Version	Changes
14-Mar-2017	1	Initial release.
11-Apr-2017	2	Updated *Figure 3: "How to add the reference to the model to the simulation profile"* (replaced by new figure) and its description.

IMPORTANT NOTICE – PLEASE READ CAREFULLY

STMicroelectronics NV and its subsidiaries (“ST”) reserve the right to make changes, corrections, enhancements, modifications, and improvements to ST products and/or to this document at any time without notice. Purchasers should obtain the latest relevant information on ST products before placing orders. ST products are sold pursuant to ST’s terms and conditions of sale in place at the time of order acknowledgement.

Purchasers are solely responsible for the choice, selection, and use of ST products and ST assumes no liability for application assistance or the design of Purchasers’ products.

No license, express or implied, to any intellectual property right is granted by ST herein.

Resale of ST products with provisions different from the information set forth herein shall void any warranty granted by ST for such product.

ST and the ST logo are trademarks of ST. All other product or service names are the property of their respective owners.

Information in this document supersedes and replaces information previously supplied in any prior versions of this document.

© 2017 STMicroelectronics – All rights reserved