

Title: Configuration of PSpice Model Libraries

Product: OrCAD PSpice A/D, OrCAD PSpice AA
and Allegro AMS Simulator

Summary: This application note describes how you can find PSpice models, how you can use PSpice Model Editor to create a part symbol for Capture based on a PSpice model and how you can configure libraries.

Author/Date: Wei Ling / 24.08.2010

Table of Contents

1	Find PSpice Models	2
2	Create a Part Symbol using PSpice Model Editor.....	2
3	Configure the Libraries	4
3.1	Configure the Capture Part Symbol Library MY_DIODE.OLB	4
3.2	Configure the PSpice Model Library.....	6
4	Bibliography	7

1 Find PSpice Models

The most commonly used models are available in the PSpice model libraries shipped with your software. The model library names have a .lib extension. But some models you need can't be found in these standard libraries. You may try with the following ways:

- Go to FlowCAD website <http://www.flowcad.de/Download.htm>, then click the link *PSpice Simulationsmodelle*
- Search PSpice models in the internet e.g. www.google.com
- Ask the vendor for the possible PSpice models directly

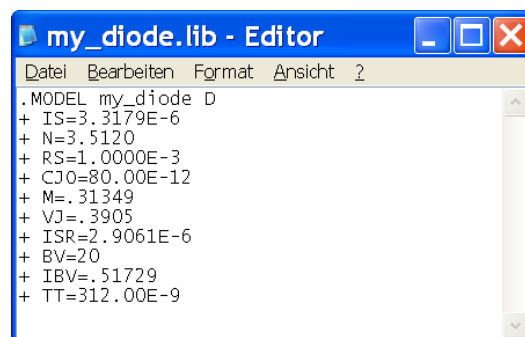
2 Create a Part Symbol using PSpice Model Editor

If you have a PSpice model but don't have the associated part symbol for Capture, you can create it anyhow by using the PSpice Model Editor.

As an example, let's create a part symbol for a PSpice diode model which you have downloaded from the internet and is named my_diode.txt.

First, rename the file and open it with PSpice Model Editor

Rename the file from my_diode.txt to my_diode.lib, then save it.

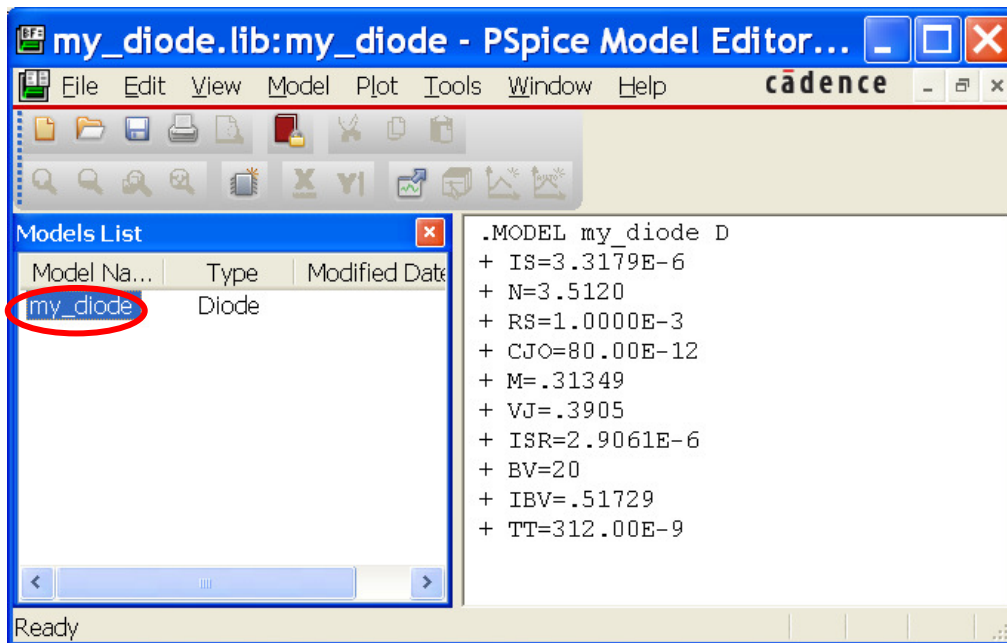


Go into Windows Start Menu: **Start**→ **All Programs**→ **OrCAD 16.2**→ **PSpice Accessories**→ **Model Editor** (the procedure which depends on the software installation may be different at your end)→

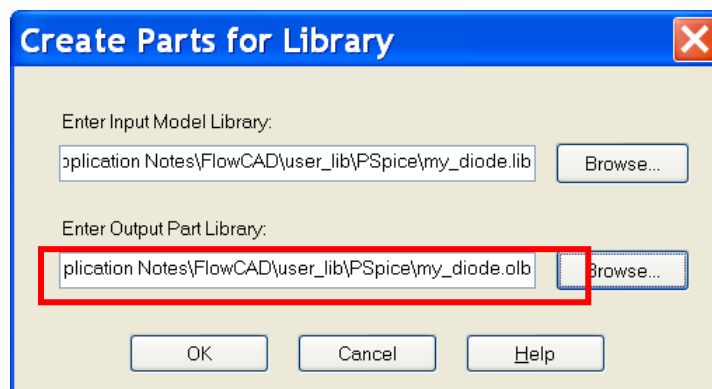
If the **Select Design Entry Tool** window comes up, check **Capture** → **Done**. In PSpice Model Editor window click **File** → **Open...** → Open the my_diode.lib file you just saved and you can see the model definition here.

Create a Part Symbol for Capture

Select the my_diode in the **Models List** window, then click **File**→ **Export to Capture Part Library...**→

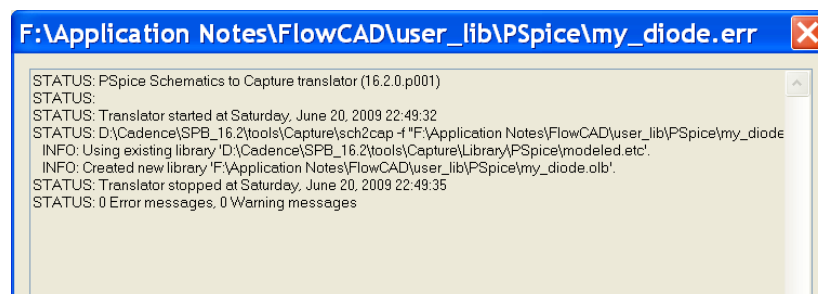


The **Create Parts for Library** window comes up. Here you can assign the path for the output part library.



Click **OK**→

A window with the status information will appear as follows:



Click **OK** to close the information window and make sure that a Capture part symbol library MY_DIODE.OLB is created and saved correctly.

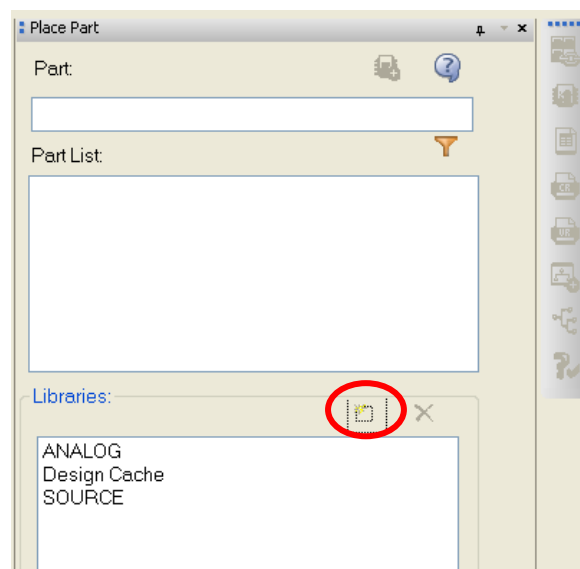
3 Configure the Libraries

Now you have prepared the PSpice model library and the part symbol library for you design. You need to configure them to simulate your circuit.

3.1 Configure the Capture Part Symbol Library MY_DIODE.OLB

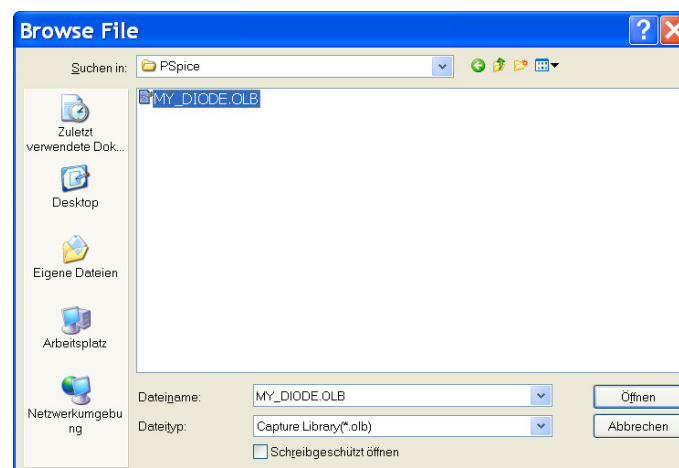
Assuming you have a PSpice project opened e.g. PSpice_Model_Config.opj and the schematic page is active.

From menu bar click **Place**→ **Part**→

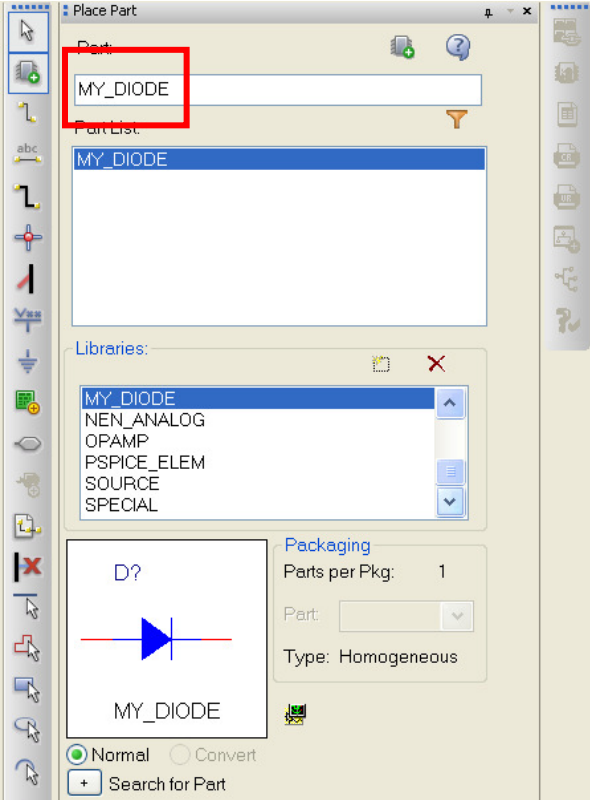


Click the Add Library button and find the library MY_DIODE.OLB in the output directory which you have specified under 2b).

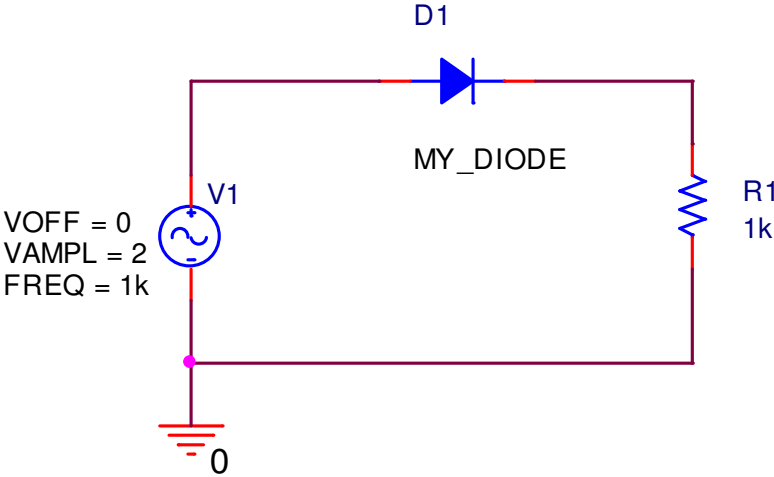
Select MY_DIODE.OLB and open it.



Place the part MY_DIODE into the Schematic Page.



The example circuit we would like to build up and simulate later on is a half wave rectifier. Build this half-wave rectifier test circuit as follows:



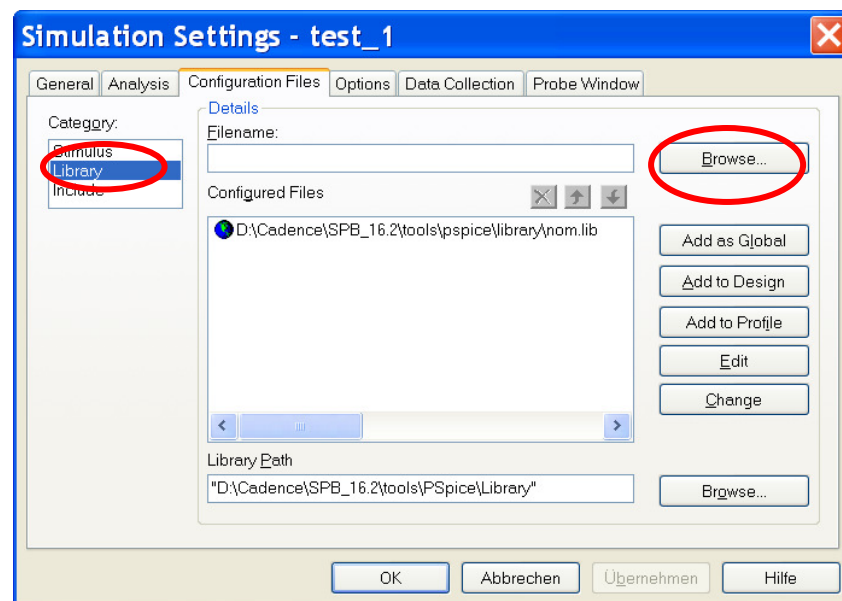
3.2 Configure the PSpice Model Library

PSpice searches model libraries for the model names specified by the MODEL implementation for parts in your design. These are the model definitions that PSpice uses to simulate your circuit.

For PSpice to locate these model definitions, you must configure the libraries. Model libraries and the models they contain have either profile, design or global application to your designs.

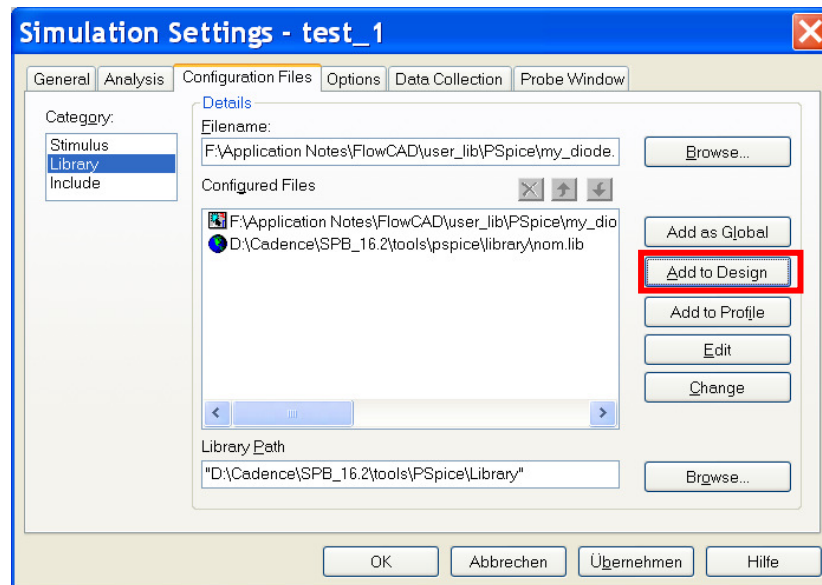
- Profile models: Profile models apply to one profile. Example usage: To set up device and lot tolerances on the model parameters for a particular part instance when running a Monte Carlo or Sensitivity/Worst-Case analysis using a specific profile.
- Design models: Design models apply to one design. The model library is then available for the entire current design (recommended).
- Global models: Global models are available to all designs you create. For example the nom.lib library is configured automatically as global after creating a new PSpice project.

Normally if there is no Simulation Profile, you can easily create a new one. If the Simulation Profile exists already, just open it and the **Simulation Settings** window comes up. Click the tab **Configuration Files** → in field **Category** select **Library** → click **Browse...**

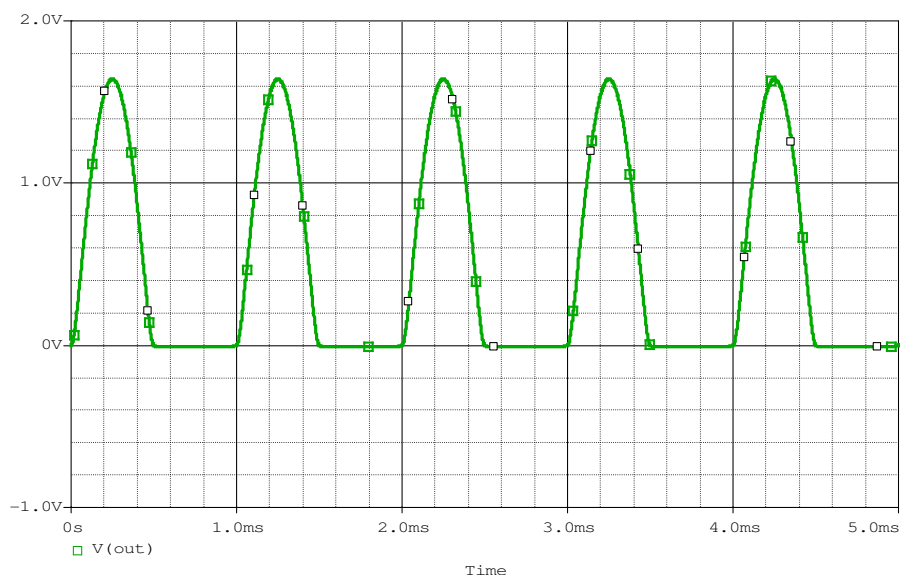


Find and select the PSpice model library my_diode.lib and open it →

Add to Design → The pspice library my_diode.lib is added to the current design and ready for use. It should look like the following when you finish the library configuration.



Now you can simulate your half-wave rectifier test circuit with the newly created diode model. Run the simulation for e.g. 5ms. The voltage V(out) is displayed as follows:



The Simulation output shows the rectified signal of the AC voltage source. Only the positive half wave is transmitted by the newly implemented Diode.

4 Bibliography

- [1] PSpice User's Guide, Product Version 16.2, Cadence
- [2] OrCAD Capture User's Guide, Product Version 16.2, Cadence