

Configuration of Model Libraries

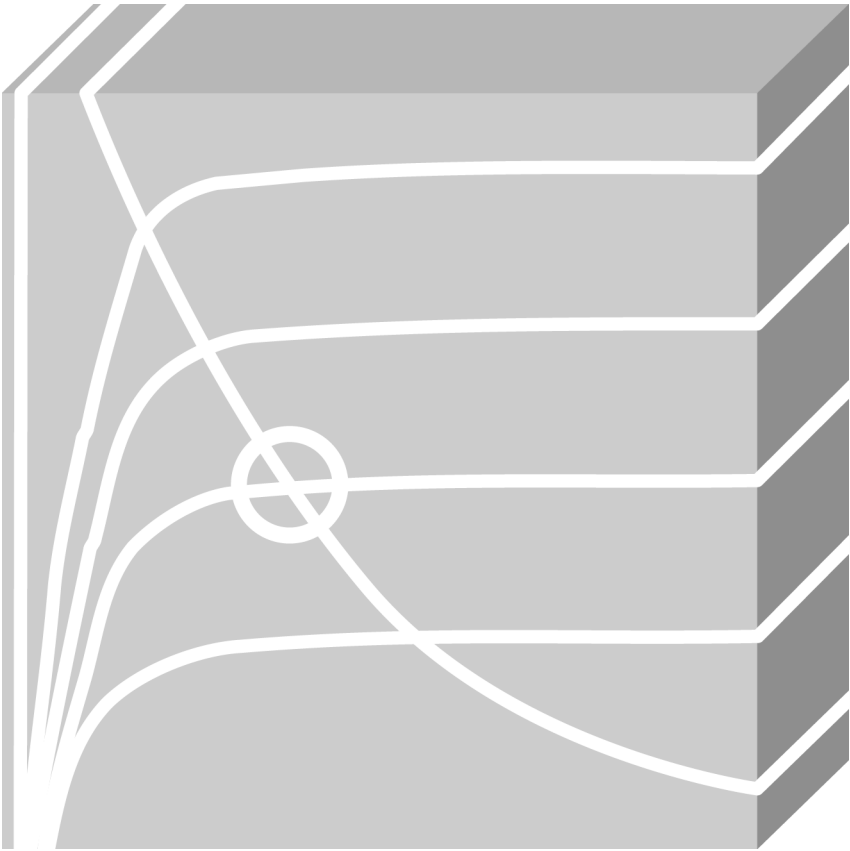


Table of Contents

- 1 Introduction 3**
- 2 Find PSpice Models 3**
- 3 Associate a Model to a Part 3**
- 4 Create a Part Symbol Using PSpice Model Editor 4**
 - 4.1 With the Wizard 5
 - 4.1.1 Rename the File and Open it in Model Editor 5
 - 4.1.2 Create a Part Symbol for Capture 5
 - 4.1.3 Open the Wizard 6
 - 4.2 Manually (Without Wizard)..... 7
 - 4.3 Configure the Libraries 7
 - 4.3.1 Configure the Capture Part Symbol Library MY_DIODE.OLB 7
 - 4.3.2 Configure the PSpice Model Library 9
- 5 Bibliography 11**

1 Introduction

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.

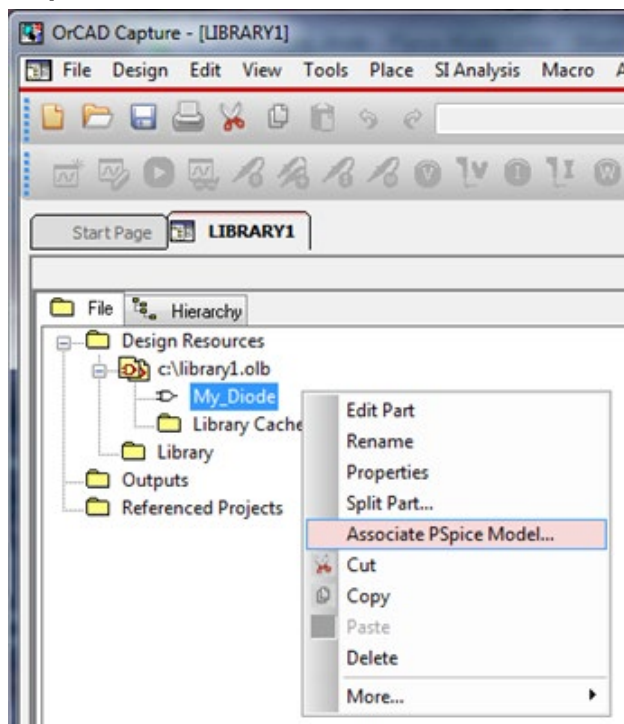
2 Find PSpice Models

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 Cadence website www.PSpice.com
- Search PSpice models in the internet, e.g. www.google.com
- Ask the vendor for the possible PSpice models directly

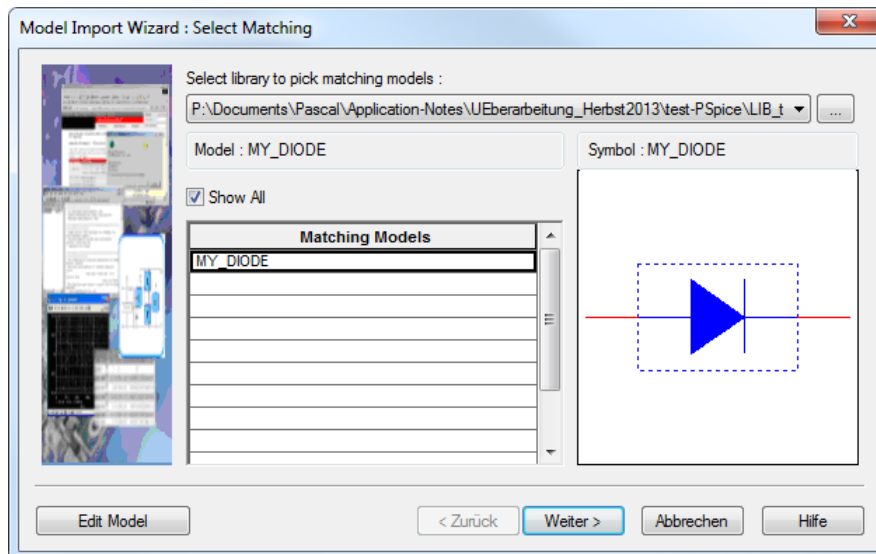
3 Associate a Model to a Part

- Open the .olb in Capture
- Right-click on the Part you want to add a PSpice Model and then click on **Associate PSpice Model...**

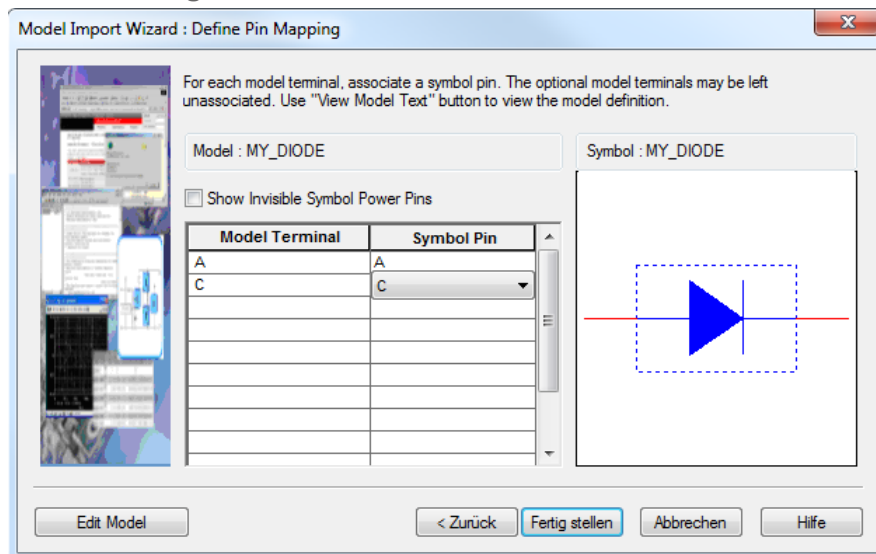


In the Window, which pops up, you can choose the .lib in which your model is.

- Click on **Weiter**.



- Choose the according pin for the Symbol
- Click on **Fertig stellen**.



Note

This procedure does not create a copy of your .olb. Your existing .olb will be modified.

4 Create a Part Symbol Using PSpice Model Editor

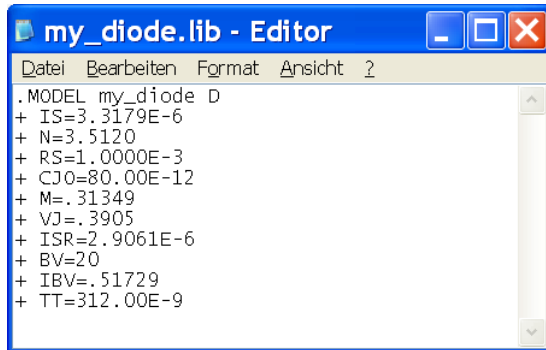
If you have a PSpice model but don't have the associated part symbol for Capture, you can create it anyhow by using the PSpice Model Editor.

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**.

4.1 With the Wizard

4.1.1 Rename the File and Open it in Model Editor

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



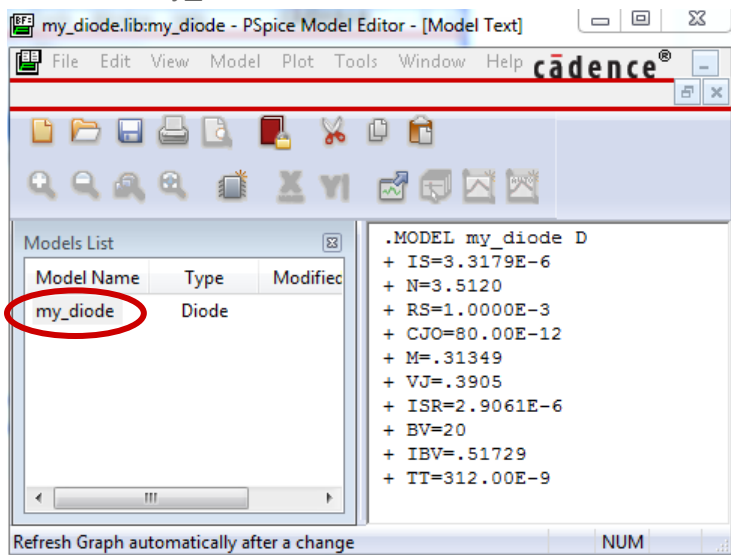
```

.MODEL my_diode D
+ IS=3.3179E-6
+ N=3.5120
+ RS=1.0000E-3
+ CJO=80.00E-12
+ M=.31349
+ VJ=.3905
+ ISR=2.9061E-6
+ BV=20
+ IBV=.51729
+ TT=312.00E-9
  
```

- Go into Windows Start Menu: **Start > All Programs > Cadence > Release17.2 > 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...** and select the `my_diode.lib` file you just saved and renamed. You can see the model definition below.

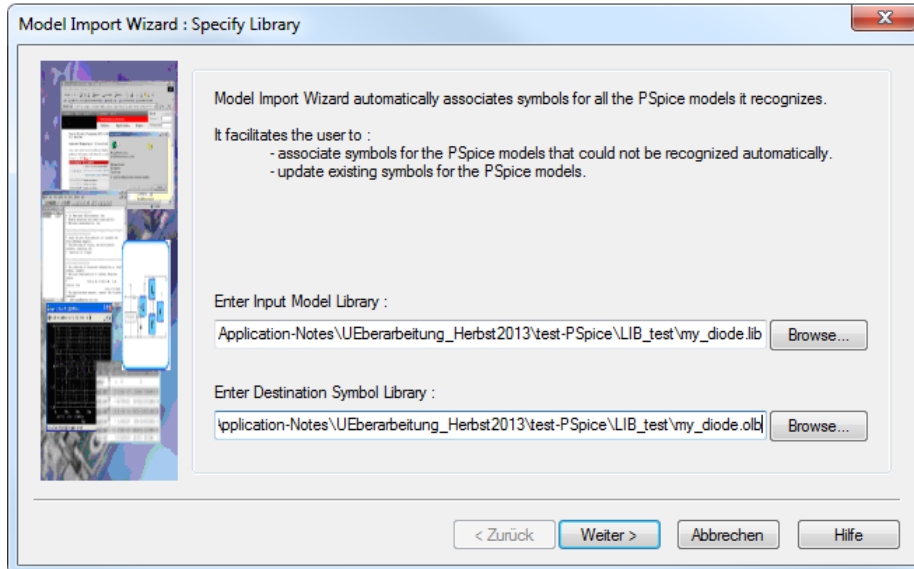
4.1.2 Create a Part Symbol for Capture

- Select the `my_diode` in the **Models List** window.

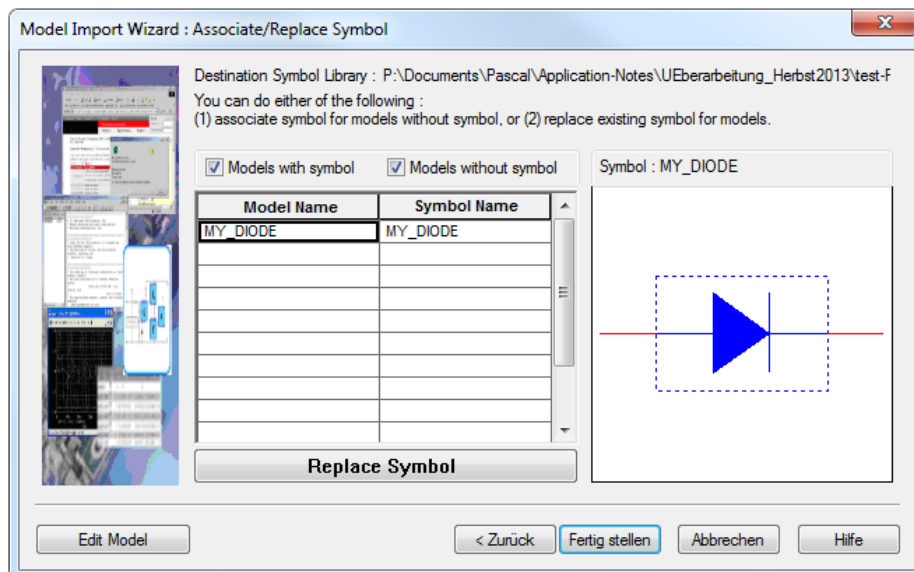


4.1.3 Open the Wizard

- Select **File > Model Import Wizard [Capture]...** The following window comes up.



- You can change the paths if they are not already according to your purpose and click on **Weiter**.



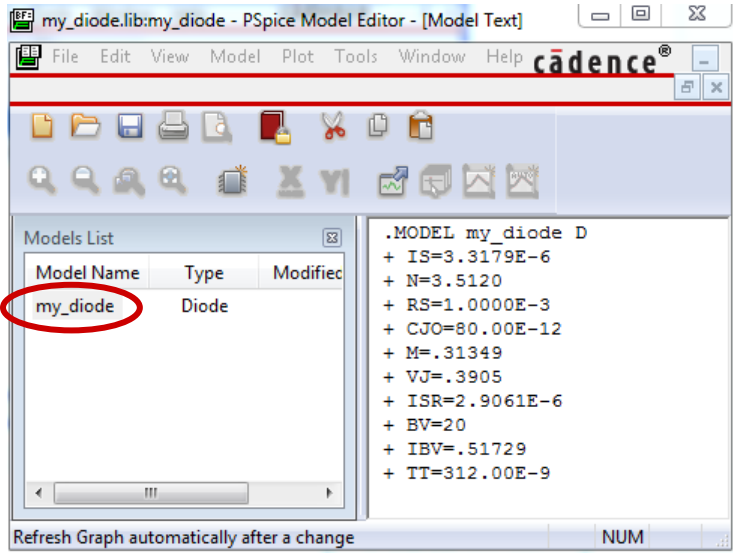
- If you want to use another symbol as suggested, click on **Replace Symbol** and choose a library and the symbol you want.
- Click on **Fertig stellen** to finish the process.
You can check the Log File to see if everything was created how you intended.

Note

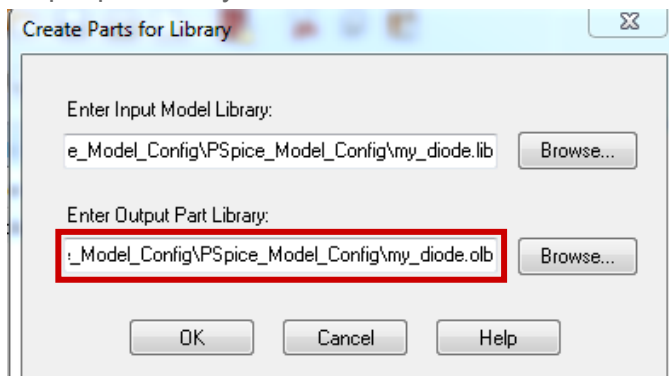
There will be a copy of the selected symbol with the property for the PSpice Model. The Original Symbol won't be altered in any way.

4.2 Manually (Without Wizard)

- Do the same steps as described in 3.1.1 and 3.1.2.



- Select **my_diode** in the **Models List** window, and 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.
- Click **OK** to close the information window and make sure that a Capture part symbol library **MY_DIODE.OLB** is created and saved correctly.

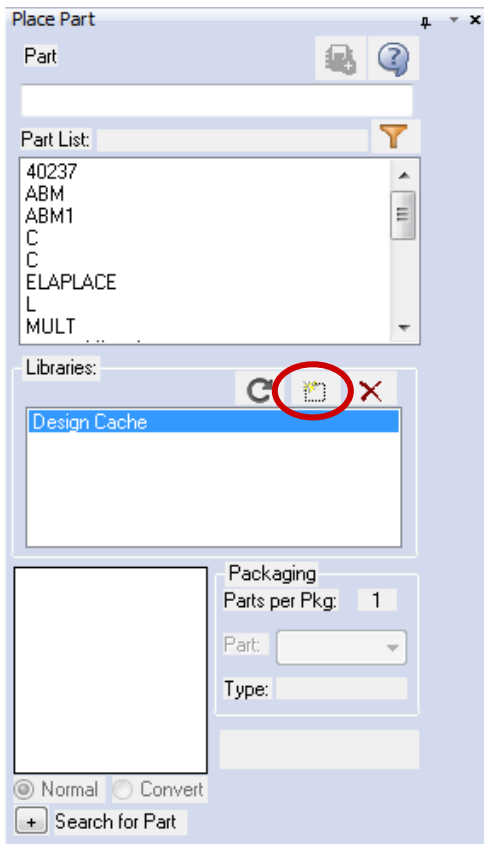
4.3 Configure the Libraries

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.

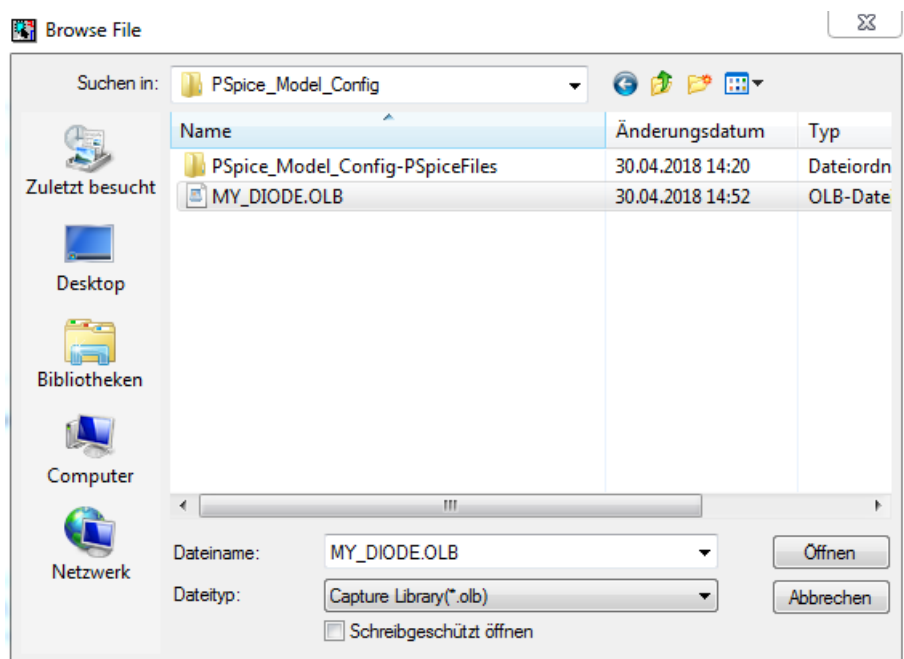
4.3.1 Configure the Capture Part Symbol Library MY_DIODE.OLB

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

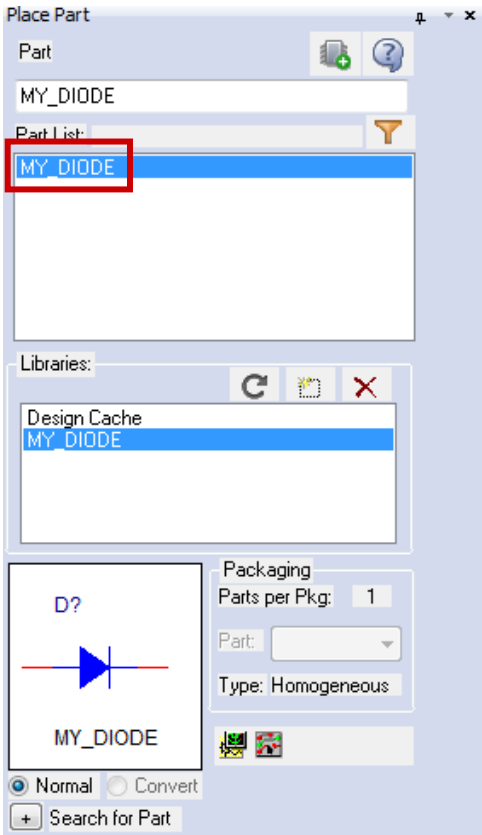
- From menu bar click **Place > Part**



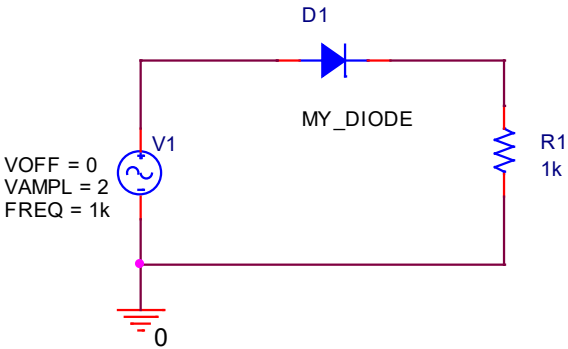
- Click the **Add Library** button and find the library **MY_DIODE.OLB** in the output directory which you have specified in 4.2
- Select **MY_DIODE.OLB** and open it.



- 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:

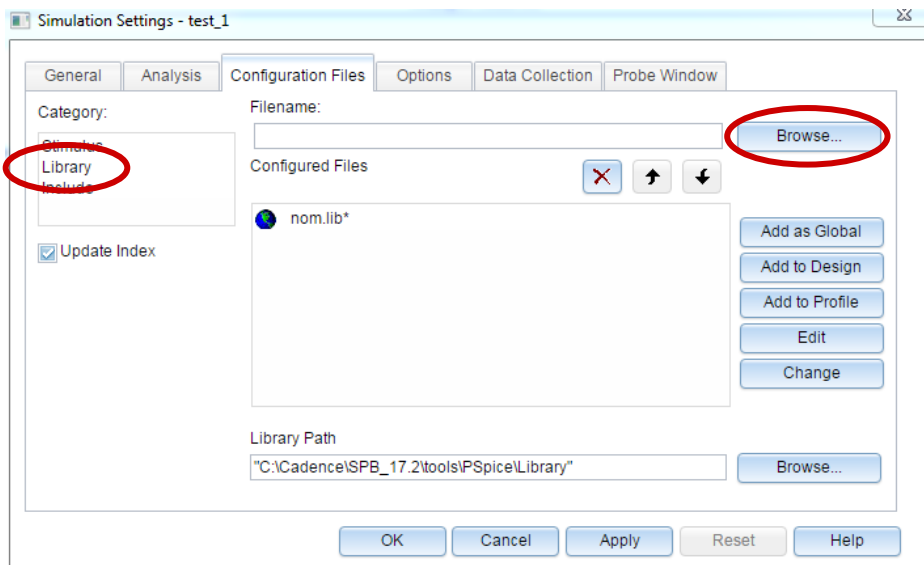


4.3.2 Configure the PSpice Model Library

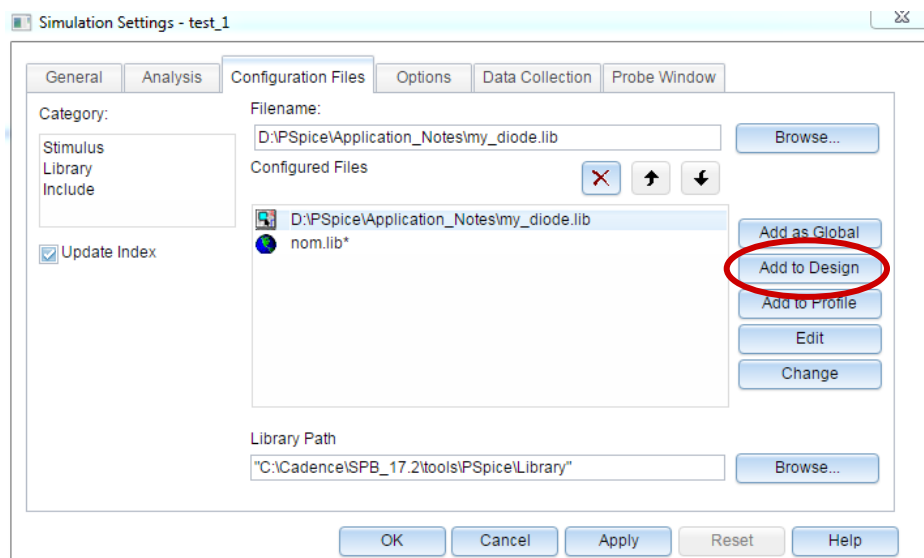
PSpice searches model libraries for the model names specified by the MODEL implementation for parts in your design. These are the model definitions that PSpice uses to simulate your circuit.

For PSpice to locate these model definitions, you must configure the libraries. Model libraries and the models they contain have one of the three following levels:

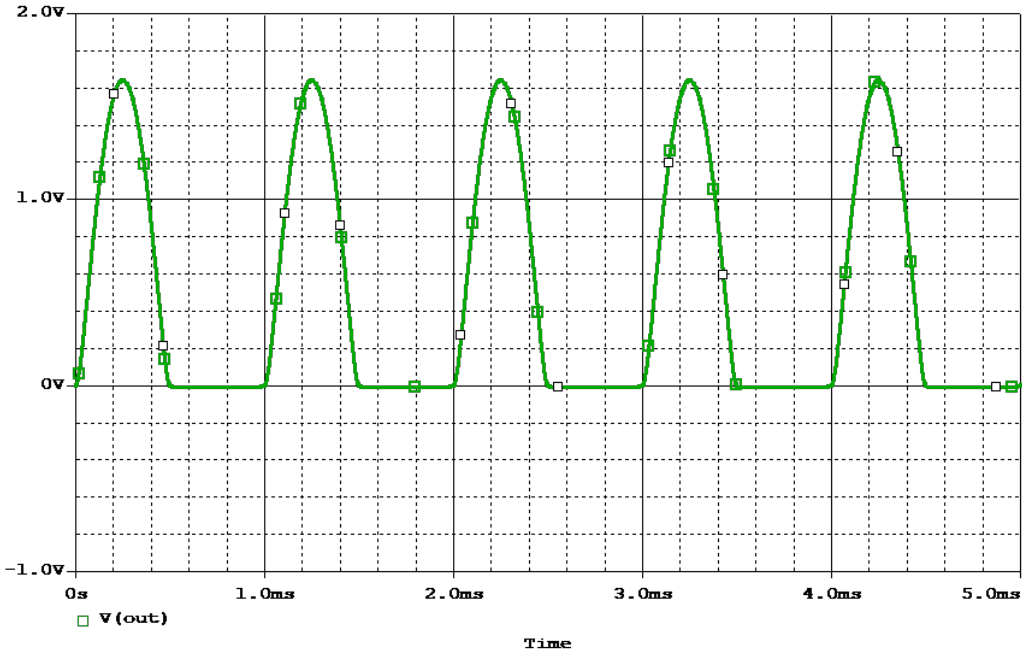
1. Profile models: Profile models apply to one simulation profile. Example usage: To set up device and lot tolerances on the model parameters for a particular part instance when running a Monte Carlo or Sensitivity/Worst-Case analysis using a specific profile.
 2. Design models: Design models apply to one design. The model library is then available for the entire current design (recommended).
 3. 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 already exists then open it. Click on the tab **Configuration Files** and under the tab Category, select **Library** and click on **Browse...**



- Find and select the PSpice model library **my_diode.lib**, open it and click on **Add to Design**.



- 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.

5 Bibliography

- [1] PSpice User's Guide, Product Version 17.2, Cadence
- [2] OrCAD Capture User's Guide, Product Version 17.2, Cadence