Design Models

How to Create Symbols for PSpice Models

ROHM provides the SPICE models for simulating circuits. The model files include the library and symbol files. However, if a model is not provided with a symbol file, symbols need to be created. This application note describes how to create the PSpice symbols. Since the symbols vary with simulators, refer to the manual of the simulator to be used when creating the symbols for the simulators other than PSpice.

Configuration of provided SPICE models

The SPICE models can be downloaded from the product pages on the ROHM website. An example of the model files is shown below. In this example, the library file and the symbol file for PSpice are shown on the left and right, respectively. If two files are provided as in this example, the model can be registered and used with the simulator. However, the symbol files may not be provided for some products. In such cases, the symbol files need to be created.

Item	Library file	Symbol file for PSpice
File name	BD9E300.lib	BD9E300.olb
Extension	.lib	.olb
File format	Text	Binary

Table 1. Example of the model files. Product name: BD9E300EFJ-LB

How to create symbols

This section introduces two methods using the OrCAD PSpice Designer (Ver. 17.2). Select method A to perform a simulation immediately. Select method B normally.

A. Create the symbols using the PSpice Model Editor

→ To perform a simulation immediately. When the symbol shape is irrelevant.

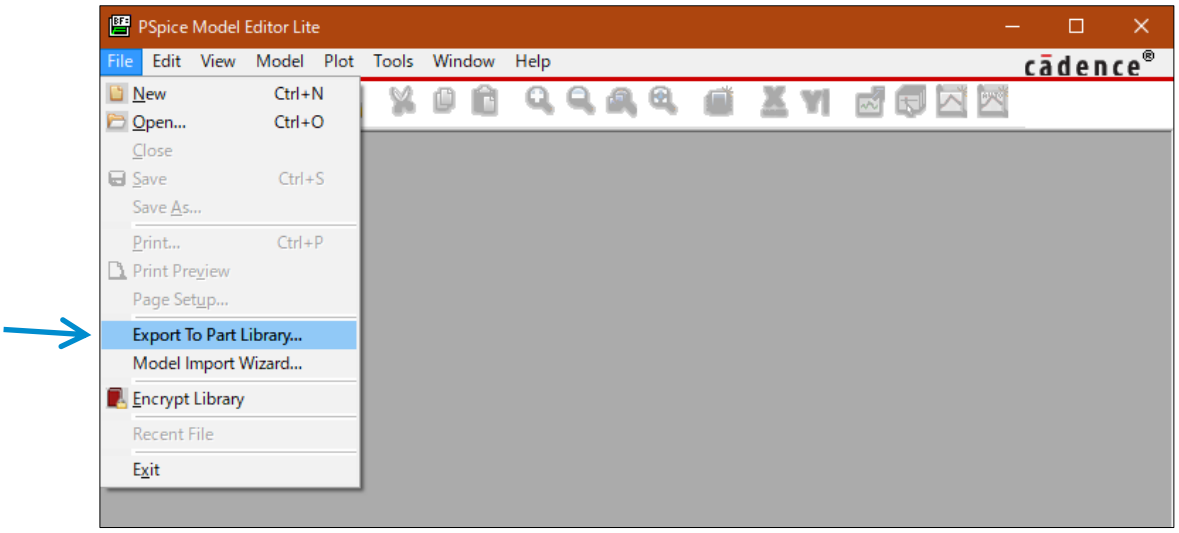
B. Create the symbol library of the OrCAD Capture

→ To create symbol files compatible with the element functions and use them for other projects as well.

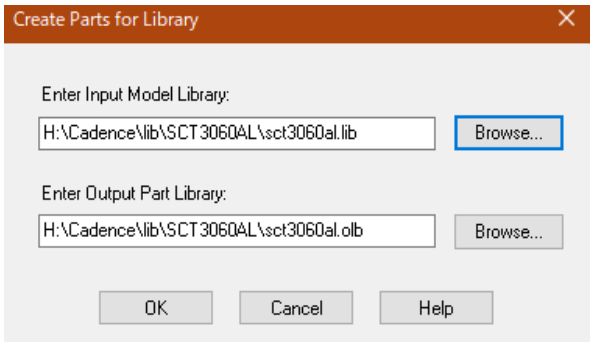
A. Create the symbols using the PSpice Model Editor

1. Start the PSpice Model Editor.

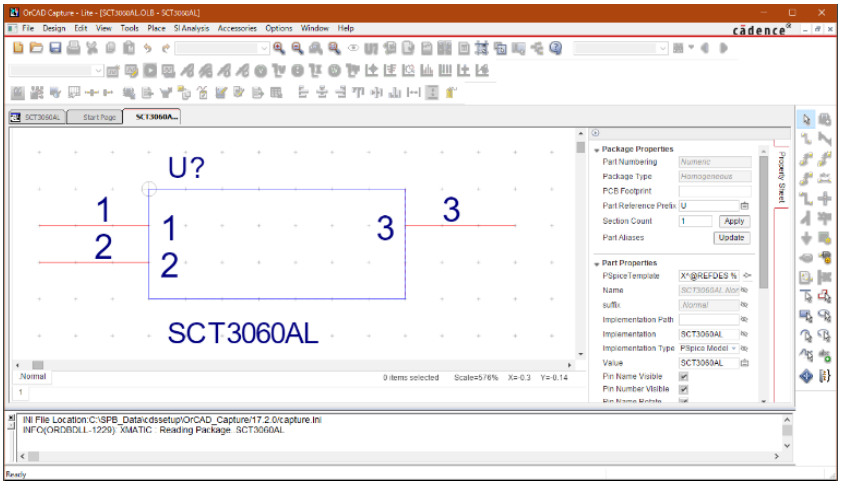
2. In the [File] menu, select [Export To Part Library…].



3. When the [Create Parts for Library] window is displayed, select a library file (.lib) in [Enter Input Model Library:] where the symbol is created. Next, in [Enter Output Part Library:], specify the location and symbol name (.olb) to save the symbol. Finally, clicking [OK] automatically creates the symbol.



4. To change the symbol shape, use the [Draw] menu on the right.



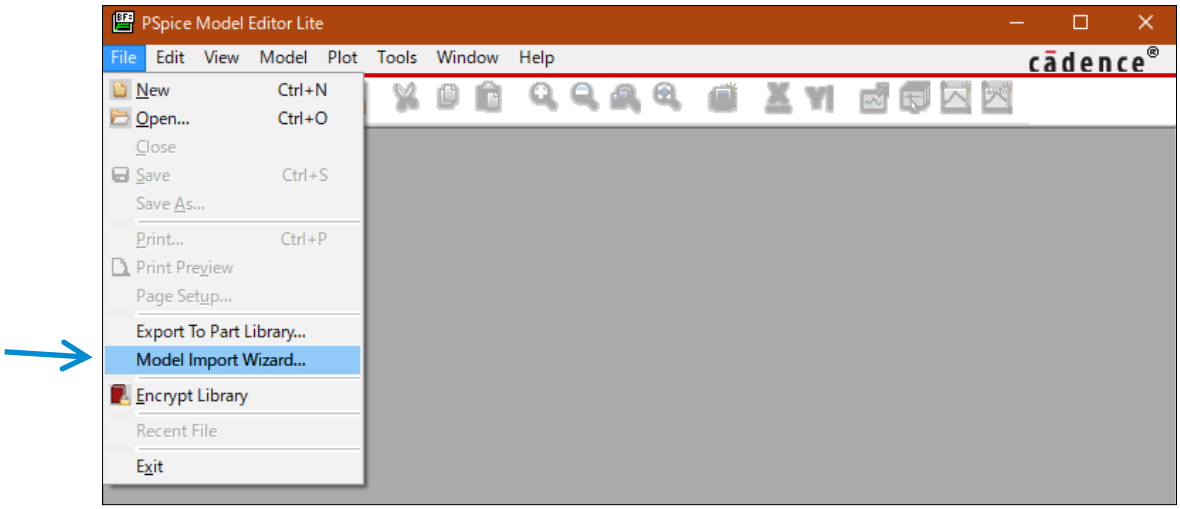
B. Create the symbol library of the OrCAD Capture

For explanation, this section takes Nch Power MOSFET RD3G400GN as an example.

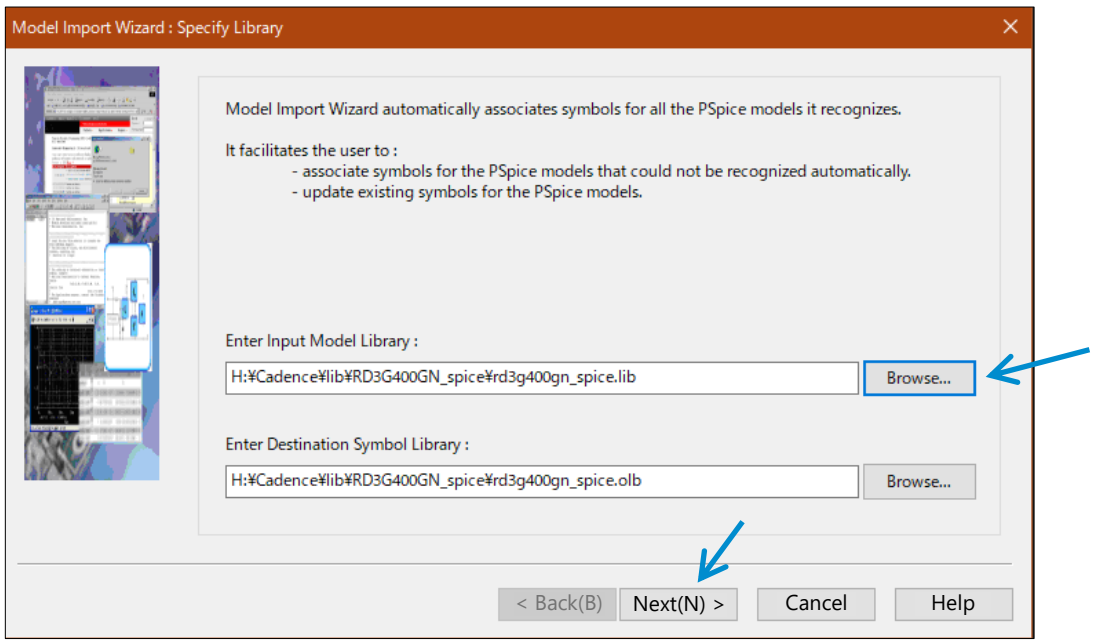
Library file name: rd3g400gn_spice.lib

1. Start the PSpice Model Editor.

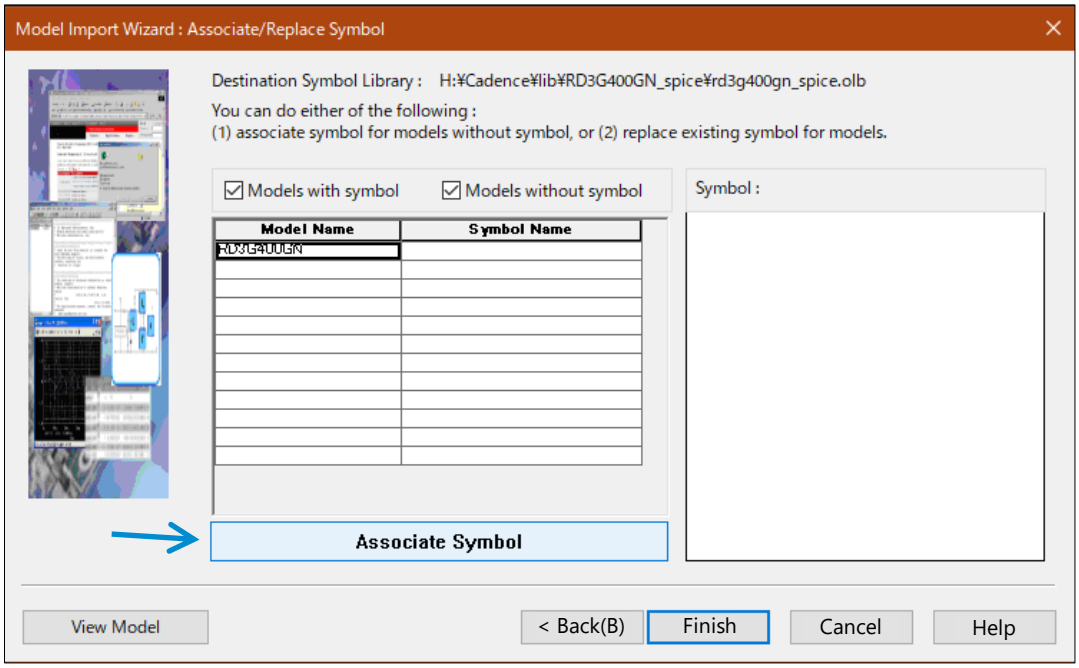
2. In the [File] menu, select [Model Import Wizard…].



3. When the [Model Import Wizard] window is displayed, specify a library file (.lib) in [Enter Input Model Library:] where the symbol is created. When a library file is selected with [Browse…], the location and symbol name (.olb) to save the symbol file are automatically entered in [Enter Destination Symbol Library:]. If no issue is found, click [Next].

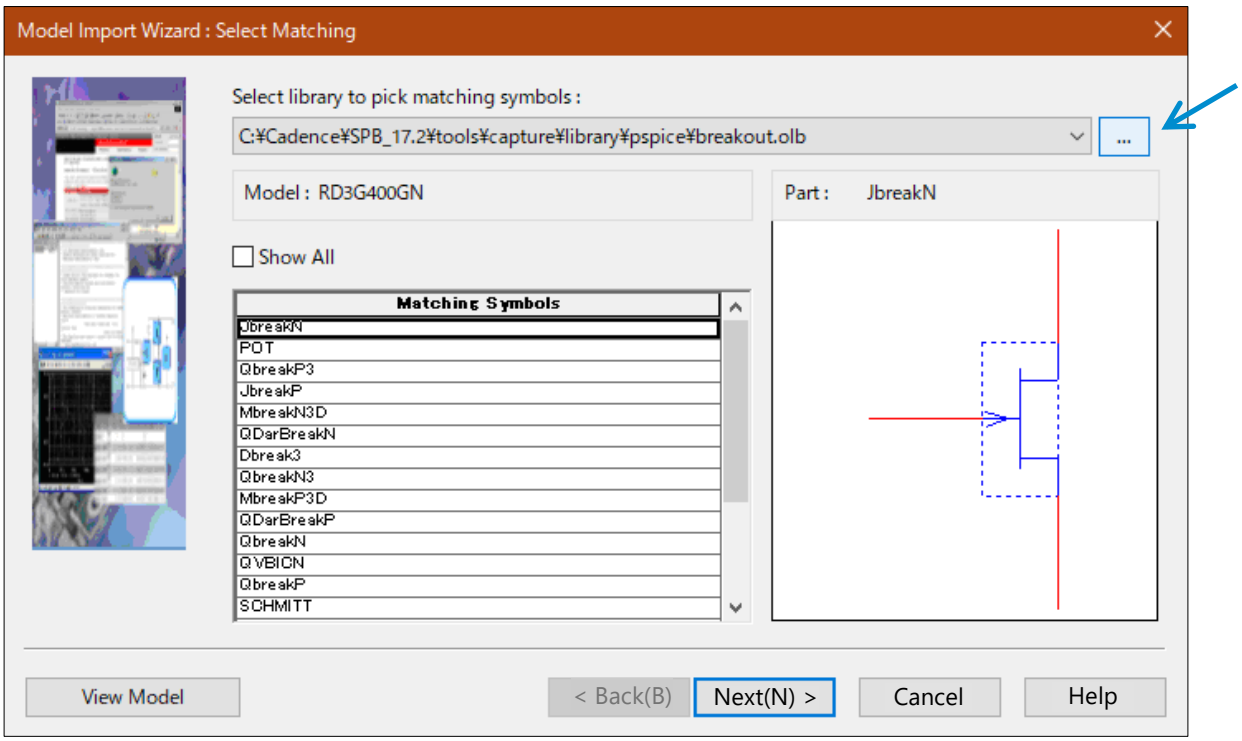


4. When the [Associate/Replace Symbol] window is displayed, click [Associate Symbol].

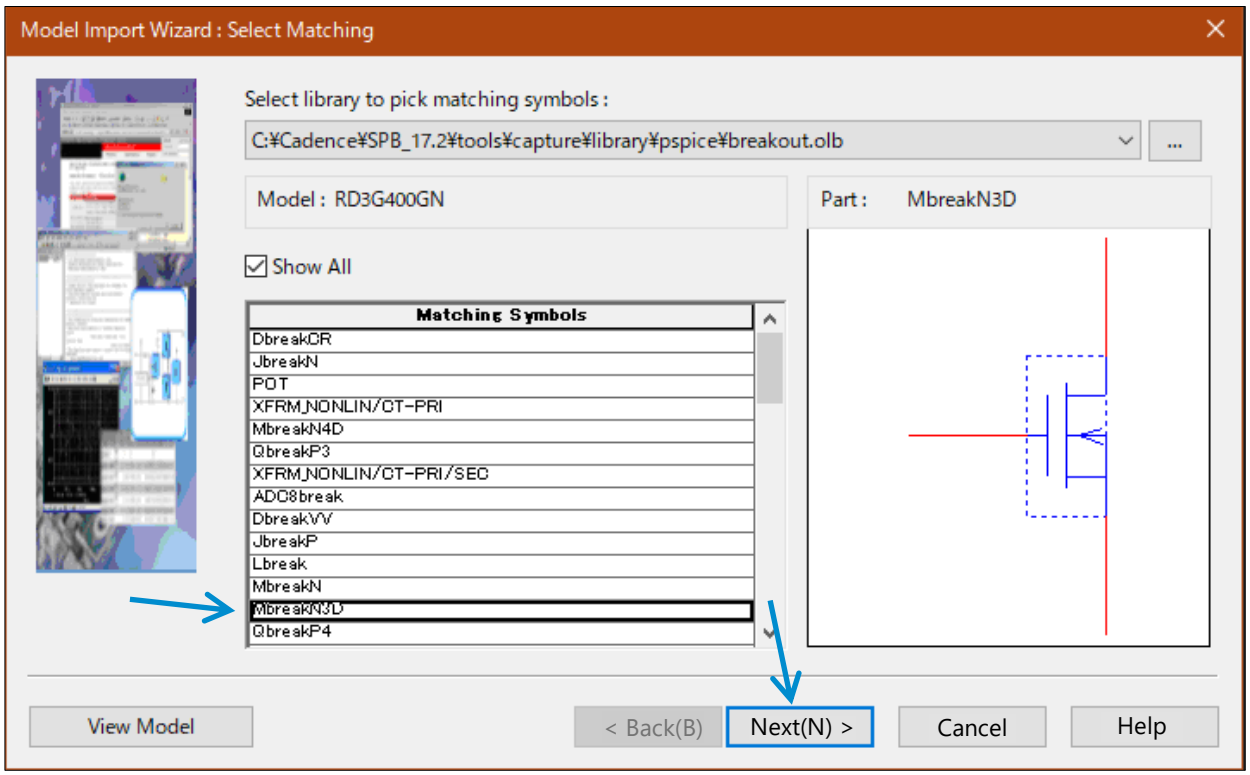


5. When the [Select Matching] window is displayed, click […] in the [Select library to pick matching symbols:] field to specify the [breakout.olb] file. This symbol file is usually located in the directory where the OrCAD PSpice Designer is installed. For example, the file may be found in the following location.

   Location of the [breakout.olb] file: C:¥Cadence¥SPB_17.2¥tools¥capture¥library¥pspice



6. In the list of symbols displayed in [Matching Symbols], select an appropriate symbol and click [Next]. In this example, [MbreakN3D] is selected for the Nch MOSFET model.



Regarding the question of which symbol included in [breakout.olb] should be used, any symbols can be used if the number of terminals matches. The usage examples are shown below.

Device	BREAKOUT symbol
Diode	Dbreak
Diode (2 in 1)	Dbreak3
Nch MOSFET	MbreakN3D

Device	BREAKOUT symbol
Pch MOSFET	MbreakP3D
Bipolar NPN	QbreakN
Bipolar PNP	QbreakP

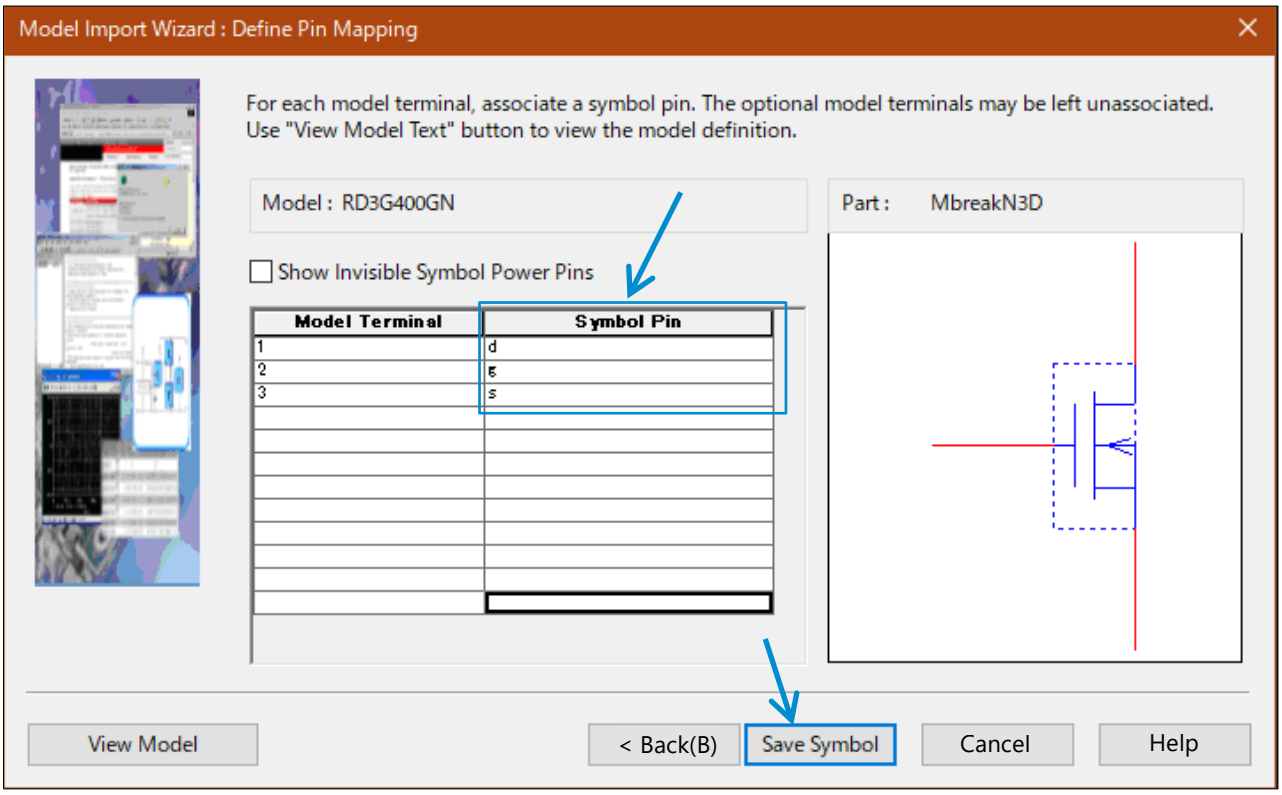
7. When the [Define Pin Mapping] window is displayed, perform mapping of the terminal names of the SPICE model and the pin names of the symbol. Basically, the terminal names and order of the SPICE model are arranged as follows.

Diode	: 1. A (Anode), 2. K (Cathode)
MOSFET	: 1. D (Drain), 2. G (Gate), 3. S (Source)
Bipolar transistor	: 1. C (Collector), B (Base), 3. E (Emitter)

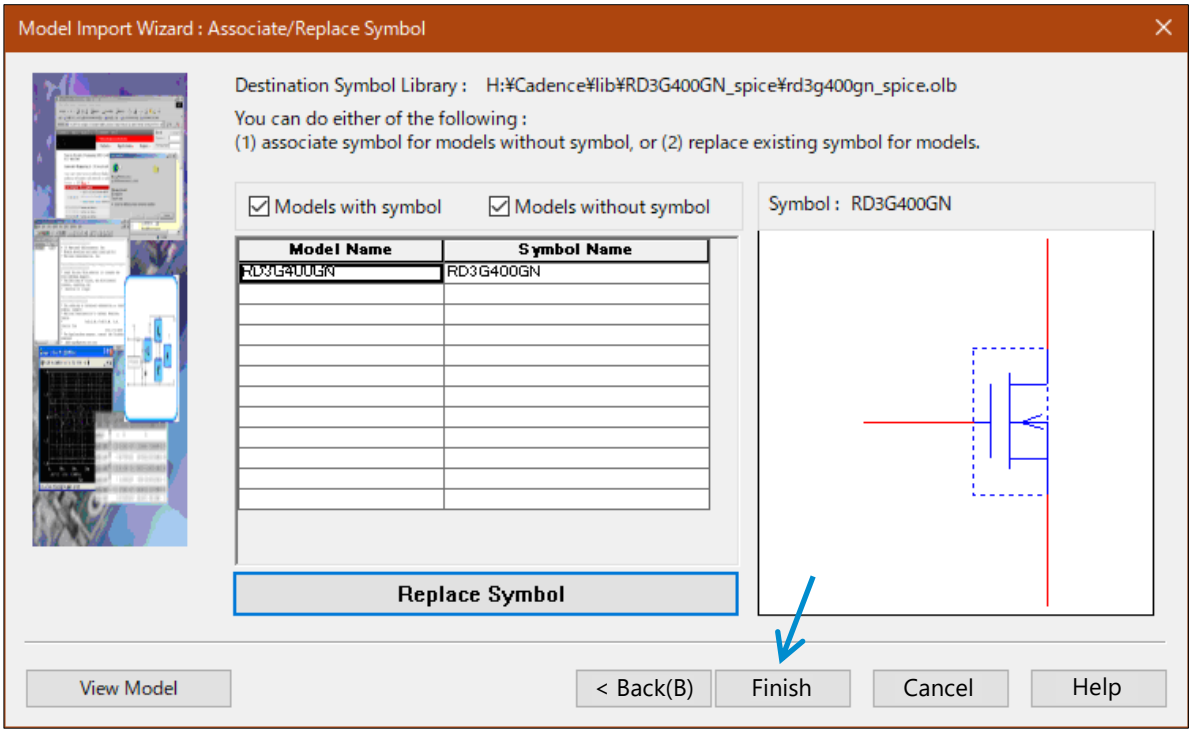
However, in rare cases, the order might be different for equivalent circuit models (subcircuit models). Therefore, it is necessary to check the description in the library file. As an example, the description in the library file for Nch MOSFET RD3G400GN is shown on the right. Since “D”, “G”, and “S” are arranged in the order of 1, 2, and 3, the basic description is confirmed.

```
* RD3G400GN NMOSFET model
* Model Generated by ROHM
* All Rights Reserved
* Commercial Use or
* Resale Restricted
* Date: 2015/08/07
********************D G S
.SUBCKT RD3G400GN 1 2 3
M1    11 22 3 3 MOS_N
D1          3 1 DDS
R1          1 11 RTH 2m
D2         22 11 DDG
R2          2 22 6.5
.MODEL MOS_N NMOS
+ LEVEL=3
+ L=2.0000E-6
+ W=1
```

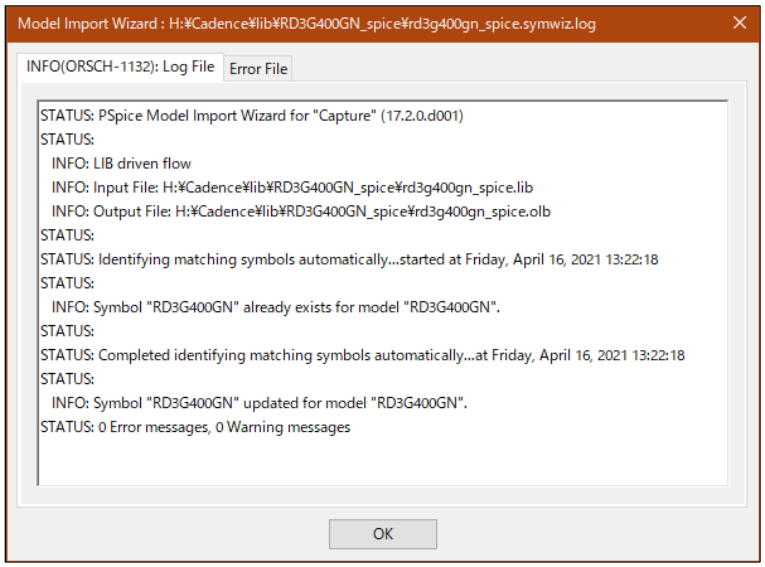
8. In the [Symbol Pin] field of the [Define Pin Mapping] window, assign the pin names in the order confirmed previously. When the assignment is completed, click [Save Symbol].



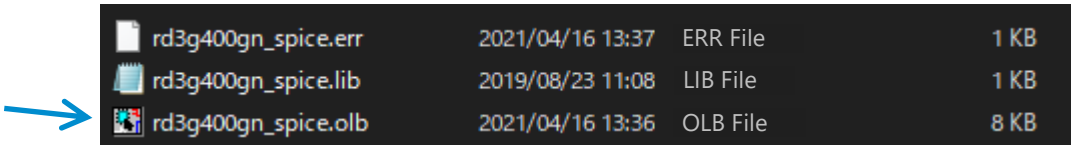
9. In the [Associate/Replace Symbol] window, click [Finish].



10. When the log window is displayed, confirm that no error has occurred.



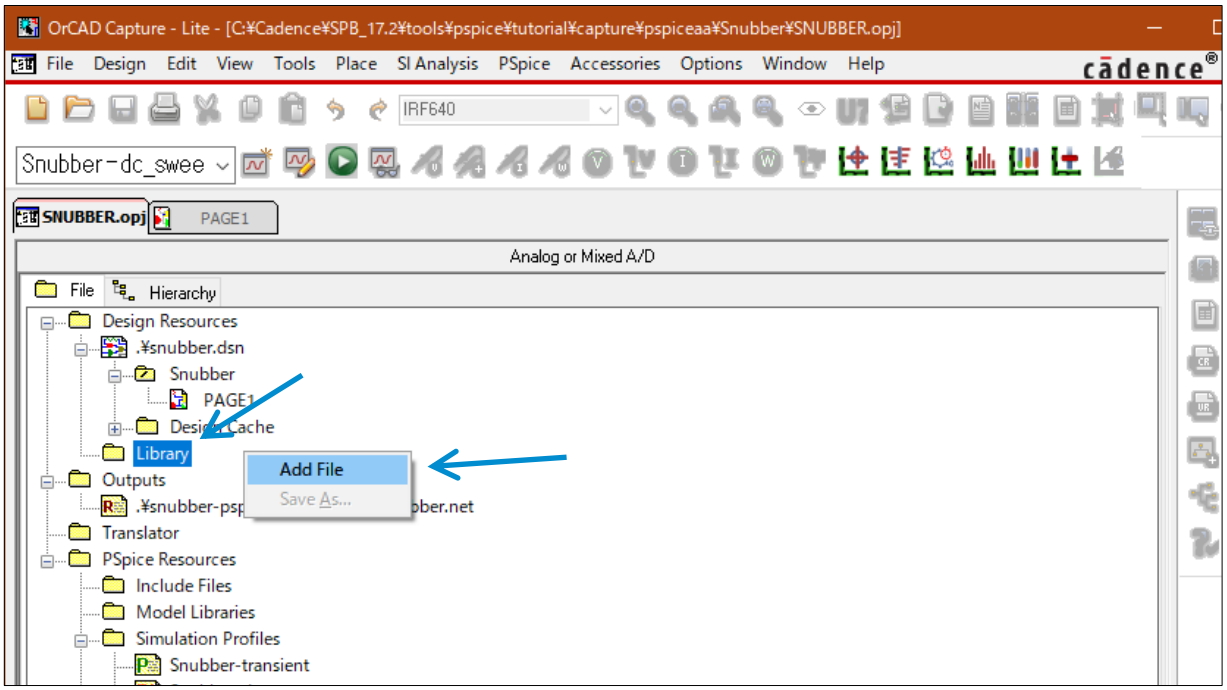
11. Confirm that the symbol file (.olb) is created in the specified folder. This completes the creation of the symbol file (.olb).



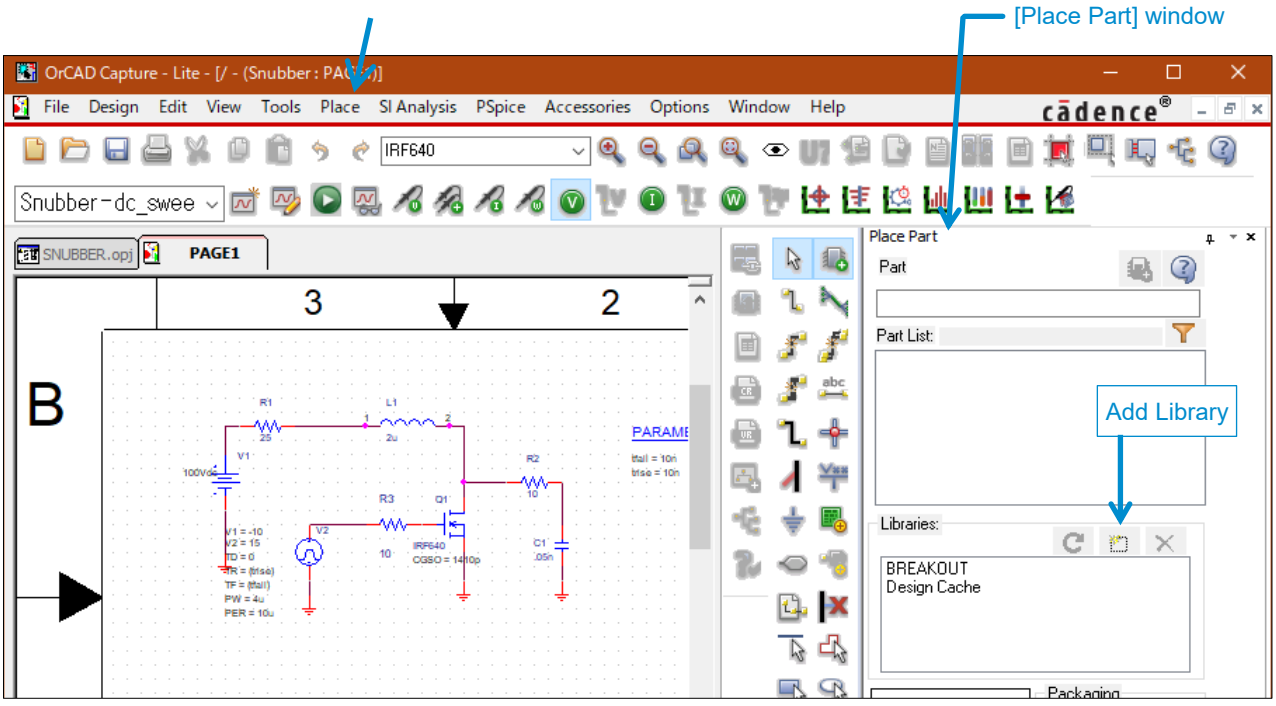
From here, the usage of the symbol file (.olb) is explained.

12. Open the project file in the OrCAD Capture.

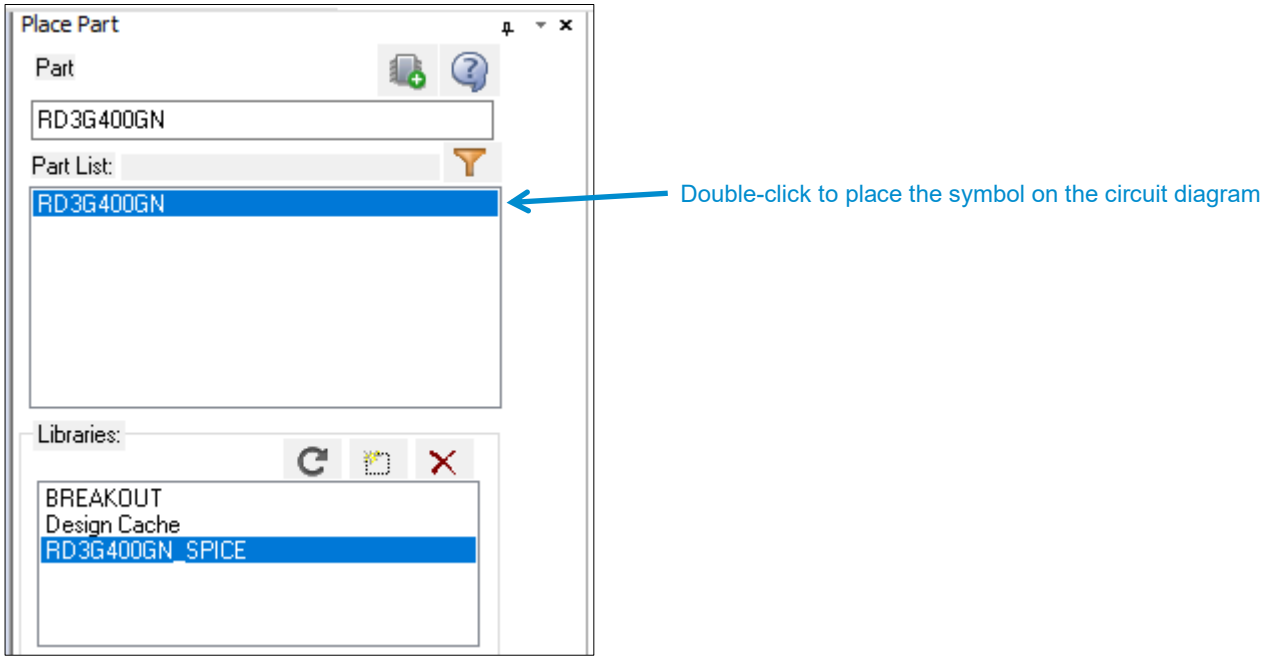
13. First, add the symbol to the library. Right-click on the [Library] folder in [Design Resources] and select [Add File] from the pop-up menu. When the file selection window is displayed, select the created [.olb] file. Confirm that the symbol is added to the [Library] folder.



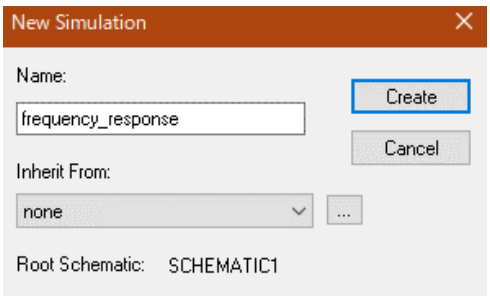
14. Alternatively, while the circuit diagram is open, select [Place] and then [Parts] in the menu bar. The [Place Part] window is displayed on the right edge. In the [Libraries:] field, clicking the [Add Library] icon displays the [Browse File] window. Select the created [.olb] file.



15. The added symbol file (.olb) is displayed in the Part List. Double-click the part to place the symbol on the circuit diagram.

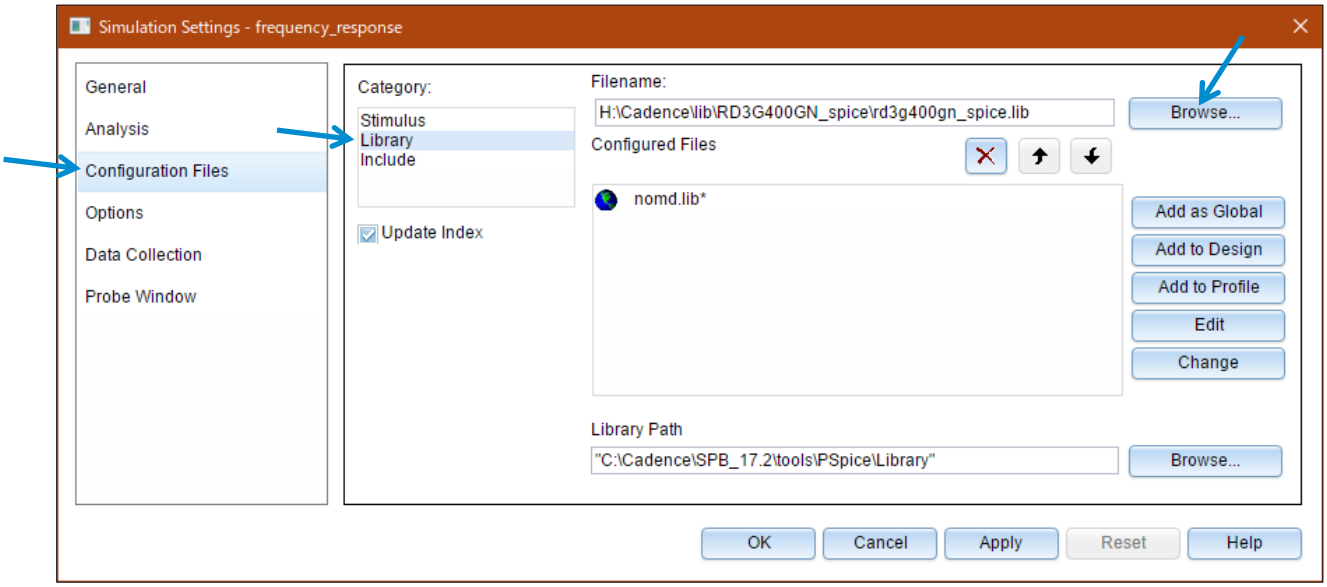


16. Next, link the library file (.lib) to the project. In the menu bar of the circuit diagram, select [PSpice] > [New Simulation Profile] to open a new simulation profile. When the [New Simulation] window is opened, enter an appropriate simulation name in [Name:] and click [Create].



To open an existing simulation profile, select [PSpice] > [Edit Simulation Profile] in the menu bar.

17. When the [Simulation Settings] window is opened, select [Configuration Files] in the left column and select [Library] in the [Category:] field. Next, click [Browse…] to select the library file (.lib).



18. According to your purpose, select one of the [Add…] buttons in the right column to add the library file.
    a. [Add as Global]: The library file is linked to all project files of OrCAD.
    b. [Add to Design]: The library file is linked only to this project file.
    c. [Add to Profile]: The library file is linked only to this simulation profile.

19. Click the [Apply] button and then the [OK] button to complete the setting.

Notes

1) The information contained herein is subject to change without notice.

2) Before you use our Products, please contact our sales representative and verify the latest specifications :

3) Although ROHM is continuously working to improve product reliability and quality, semiconductors can break down and malfunction due to various factors.
Therefore, in order to prevent personal injury or fire arising from failure, please take safety measures such as complying with the derating characteristics, implementing redundant and fire prevention designs, and utilizing backups and fail-safe procedures. ROHM shall have no responsibility for any damages arising out of the use of our Poducts beyond the rating specified by ROHM.

4) Examples of application circuits, circuit constants and any other information contained herein are provided only to illustrate the standard usage and operations of the Products. The peripheral conditions must be taken into account when designing circuits for mass production.

5) The technical information specified herein is intended only to show the typical functions of and examples of application circuits for the Products. ROHM does not grant you, explicitly or implicitly, any license to use or exercise intellectual property or other rights held by ROHM or any other parties. ROHM shall have no responsibility whatsoever for any dispute arising out of the use of such technical information.

6) The Products specified in this document are not designed to be radiation tolerant.

7) For use of our Products in applications requiring a high degree of reliability (as exemplified below), please contact and consult with a ROHM representative : transportation equipment (i.e. cars, ships, trains), primary communication equipment, traffic lights, fire/crime prevention, safety equipment, medical systems, servers, solar cells, and power transmission systems.

8) Do not use our Products in applications requiring extremely high reliability, such as aerospace equipment, nuclear power control systems, and submarine repeaters.

9) ROHM shall have no responsibility for any damages or injury arising from non-compliance with the recommended usage conditions and specifications contained herein.

10) ROHM has used reasonable care to ensure the accuracy of the information contained in this document. However, ROHM does not warrants that such information is error-free, and ROHM shall have no responsibility for any damages arising from any inaccuracy or misprint of such information.

11) Please use the Products in accordance with any applicable environmental laws and regulations, such as the RoHS Directive. For more details, including RoHS compatibility, please contact a ROHM sales office. ROHM shall have no responsibility for any damages or losses resulting non-compliance with any applicable laws or regulations.

12) When providing our Products and technologies contained in this document to other countries, you must abide by the procedures and provisions stipulated in all applicable export laws and regulations, including without limitation the US Export Administration Regulations and the Foreign Exchange and Foreign Trade Act.

13) This document, in part or in whole, may not be reprinted or reproduced without prior consent of ROHM.

ROHM SEMICONDUCTOR

Thank you for your accessing to ROHM product informations.
More detail product informations and catalogs are available, please contact us.

ROHM Customer Support System

http://www.rohm.com/contact/