Library for Cadence OrCAD Capture

Users Manual

July 2018

Murata Manufacturing Co., Ltd.

Ver.1.0

Index

1. Regarding this Manual

- This manual describes the registration method of the part model library used in OrCAD Capture, and usage examples.

  - The required operations can be implemented according to the procedures indicated in this manual. However, note that some of actual procedures may differ from them depending on your operating environment.

  - The description in this manual is based on the operation method in OrCAD Capture version 16.6 or later.

2. (Preparation) Decompression and Saving of Library

OrCAD Capture uses two types of files including "*****.LIB" and "*****.OLB." These two types of files are generally referred to as a library in this manual.
You can download a zip file which includes these two types of files from the Murata Website.
The ***** indicates the series part number of various parts manufactured and sold by Murata Manufacturing Co., Ltd. (hereinafter Murata).

Working folder: "C:¥orcaduser," and library folder:" C:¥orcaduser¥lib" are used in the following description.

(1) Decompress the downloaded zip file.
(2) Save "*****.LIB" and "*****.OLB" in C:¥orcaduser¥lib.
(3) The no longer required zip file can be deleted.

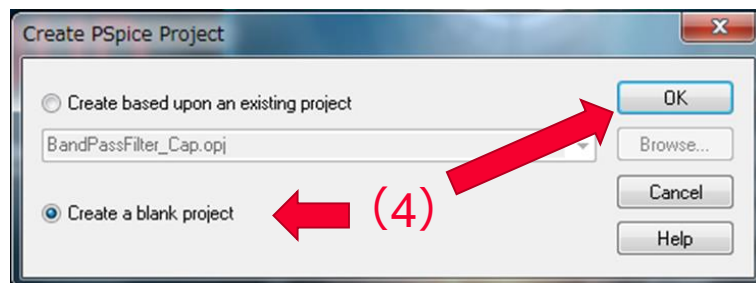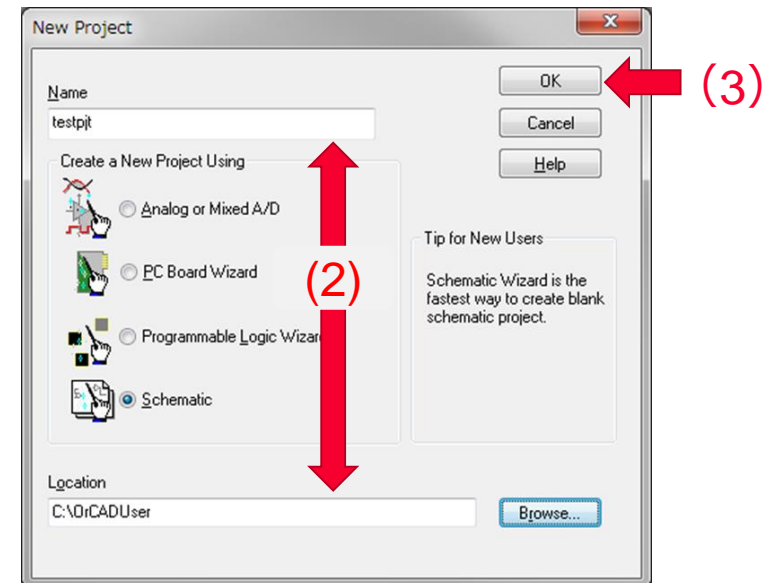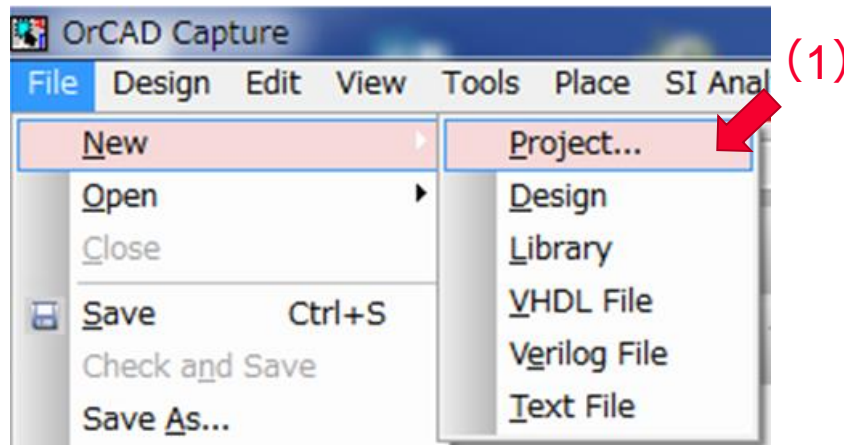The preparation is now completed.

*****.LIB: SPICE Netlist of PSpice format
*****.OLB: A file containing information, such as the symbol figures of a circuit, and list of indicated part numbers, etc.

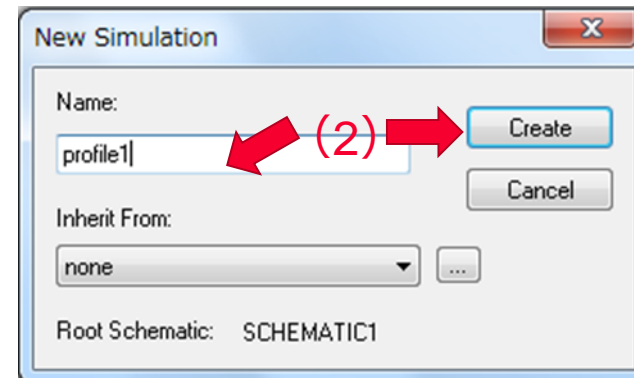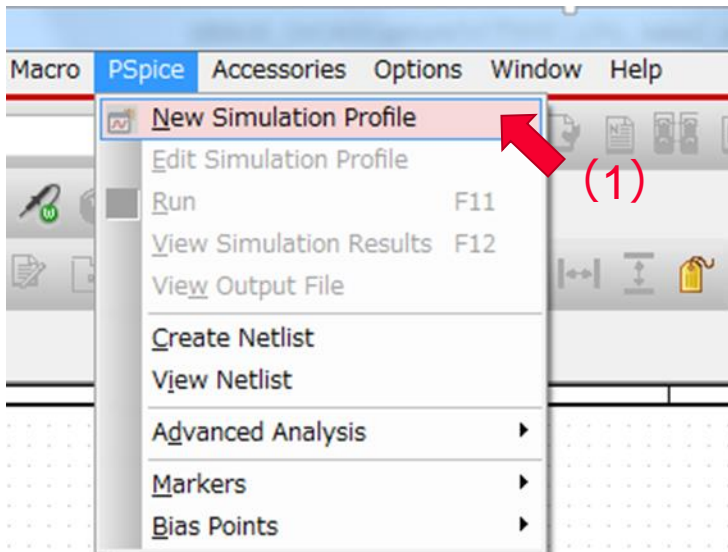# 3. Creation of a Project

Start OrCAD Capture.

(1) From the main menu, select File -> New -> Project.
(2) Set the File Name to "testpjt" and the Save Location to C:¥orcadser.
(3) Click the **OK** button.
(4) Select "New Project" and click the **OK** button.
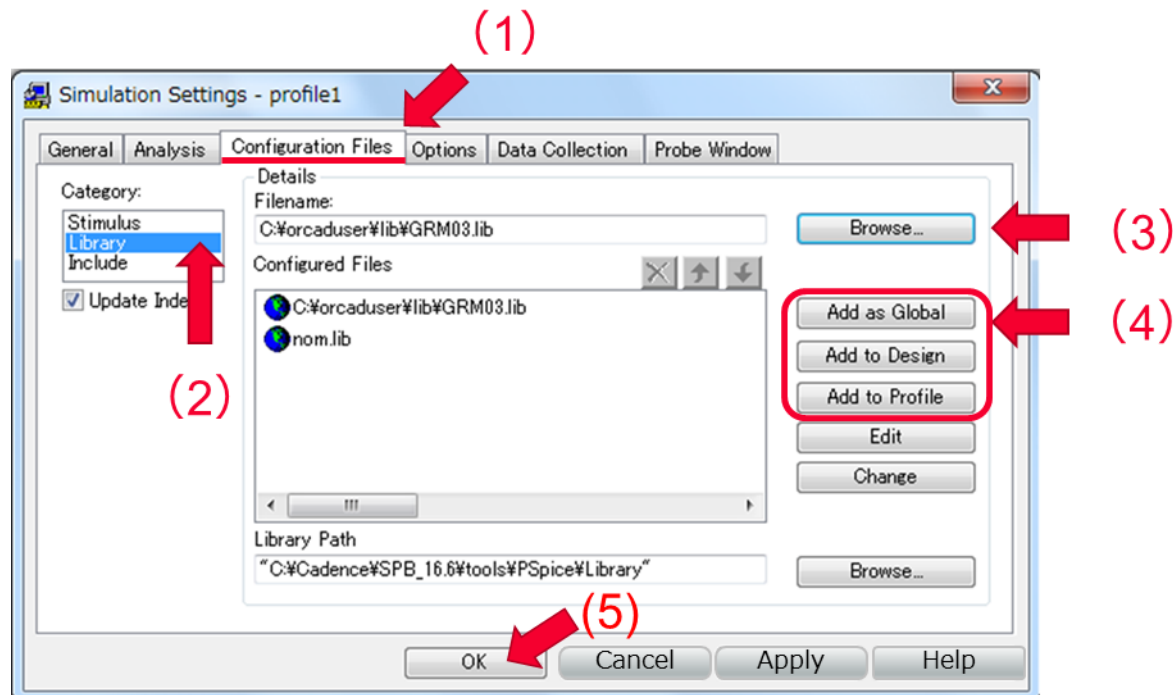
4. Creation of a Simulation Profile

Register the LIB file in Simulation Profile.

(1) From the main menu, select PSpice -> New Simulation Profile.
(2) Set the Name of the simulation profile to "profile1" and click the **Create** button.
    The Simulation Settings window will open separately from the main window.
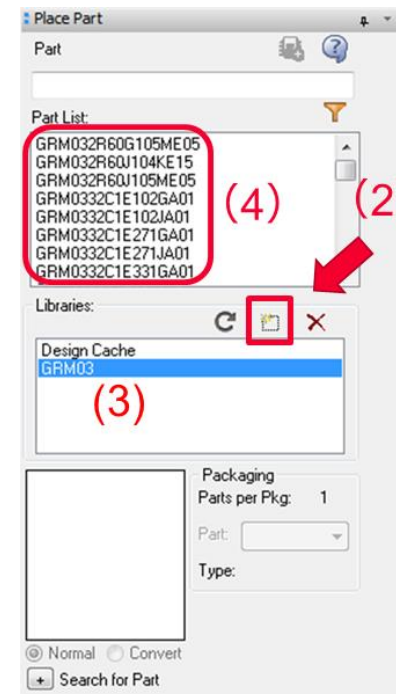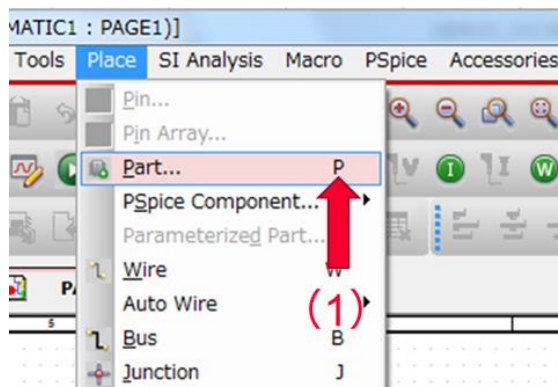
5. Registration of LIB Files

(1) Select the Configuration Files tab.
(2) Set the category to "Library."
(3) Click the **Browse** button of the file name and select GRM03.LIB of "C:¥orcaduser¥lib."
(4) Select "Add as Global" ("Add to Design" and "Add to Profile" can also be selected).
(5) Click the **OK** button to end the registration of the LIB file.

6. Registration of OLB Files

(1) From the main menu, select Place -> Parts.

(2) In "Place Part," click [ ] and select "GRM03.OLB" of "C:¥orcaduser¥lib."

(3) "GRM03" will be displayed in the library window.

(4) A list of part numbers which can be used in GRM03 will be displayed in Parts List.
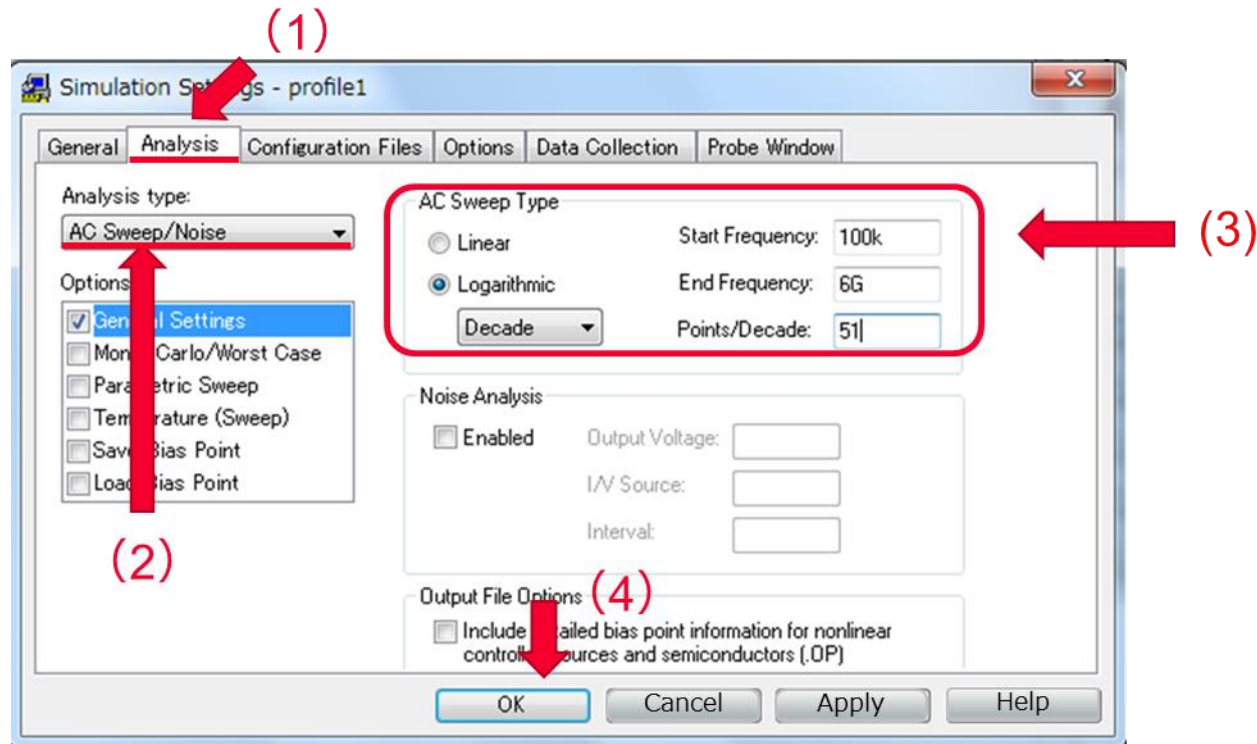   The OLB file is now registered.

From the main menu, select PSpice -> Edit Simulation Profile.
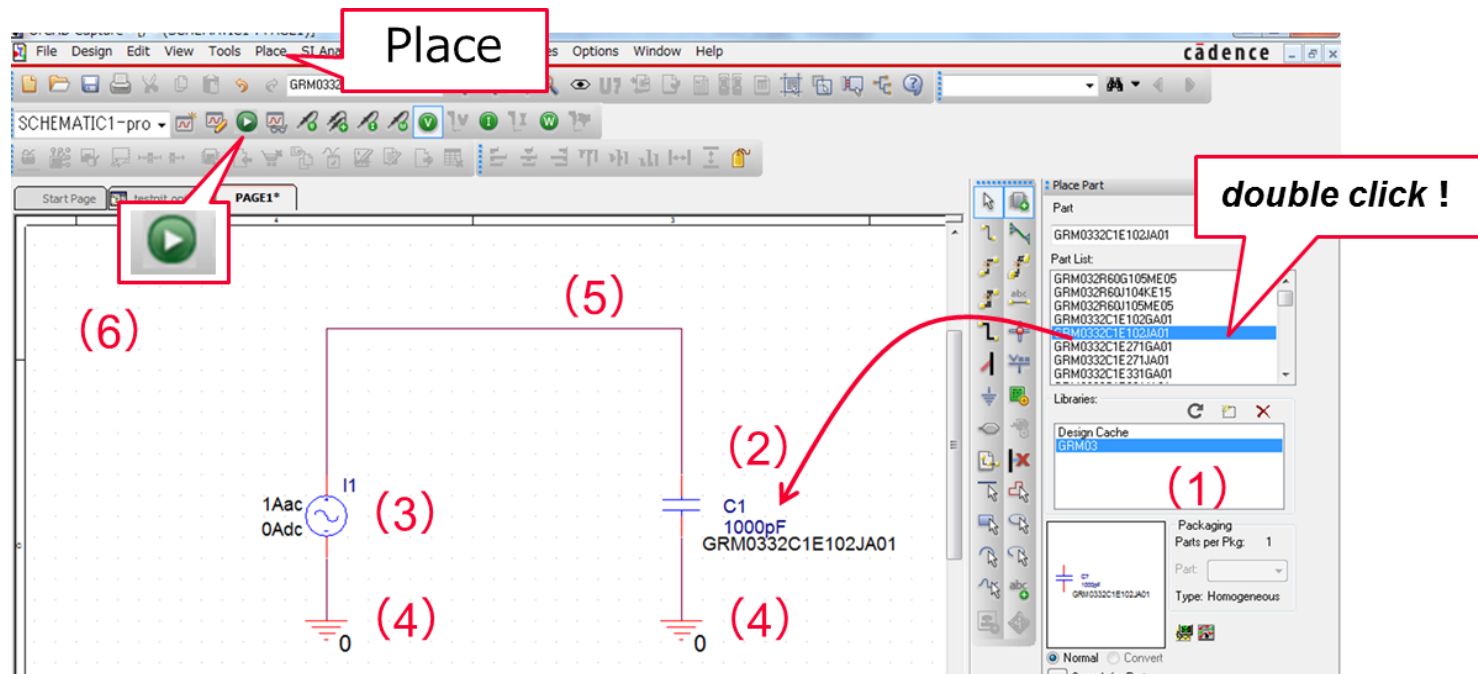(1) Select the Analysis tab.
(2) Set the type of analysis to "AC Sweep/Noise."
(3) In AC Sweep Type, set the Start Frequency to 100 kHz, End Frequency to 6 GHz and Point/Decade to 51.
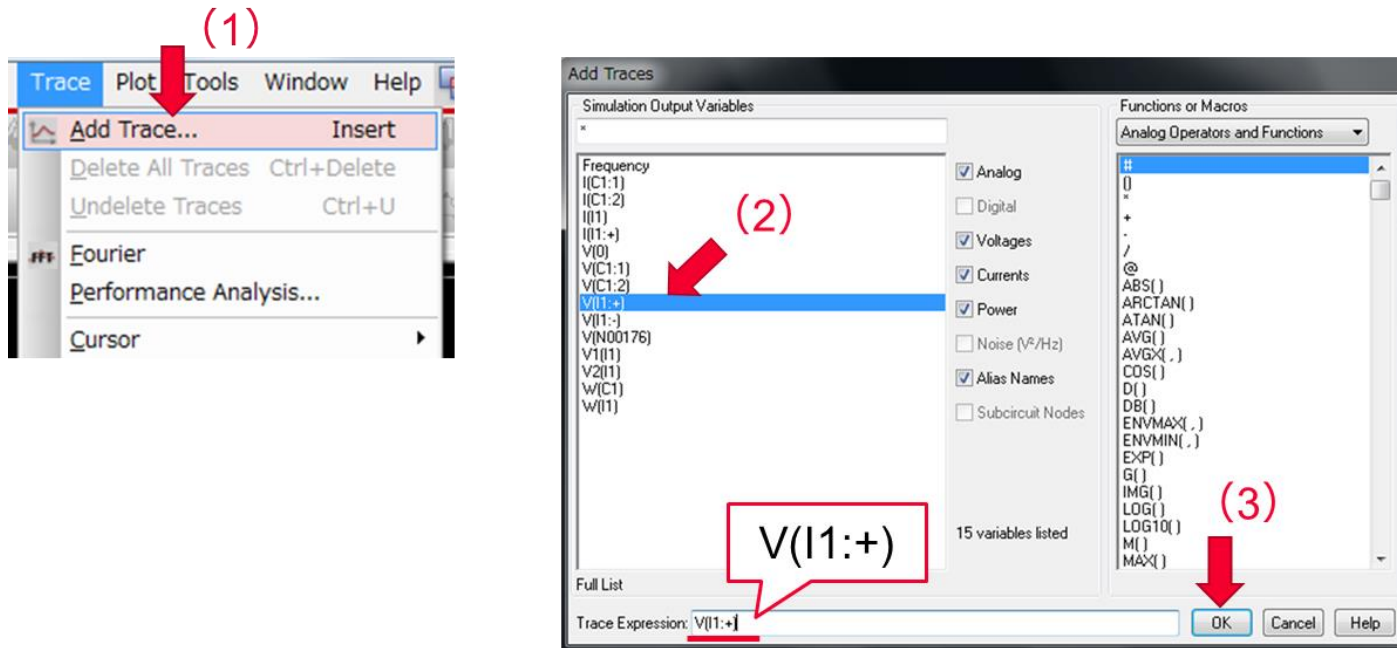(4) Click **OK**.

7. Example of Impedance Calculation of a Capacitor (2/4)

(1) Select Place -> Parts and select "GRM03" of Place Part.

(2) Double-click on "GRM0332C1E102JA01" of the parts list to place it in the circuit diagram.

(3) Select Place -> PSpice Component -> Source -> Current Source -> AC to place it in the circuit diagram.

(4) Select Place -> Ground to select and place a ground.

(5) Select Place -> Wire to connect each component.

(6) Select PSpice -> Run, or click the ⏵ button on the Toolbar.
   A display window of the calculation results will open separately from the main window.
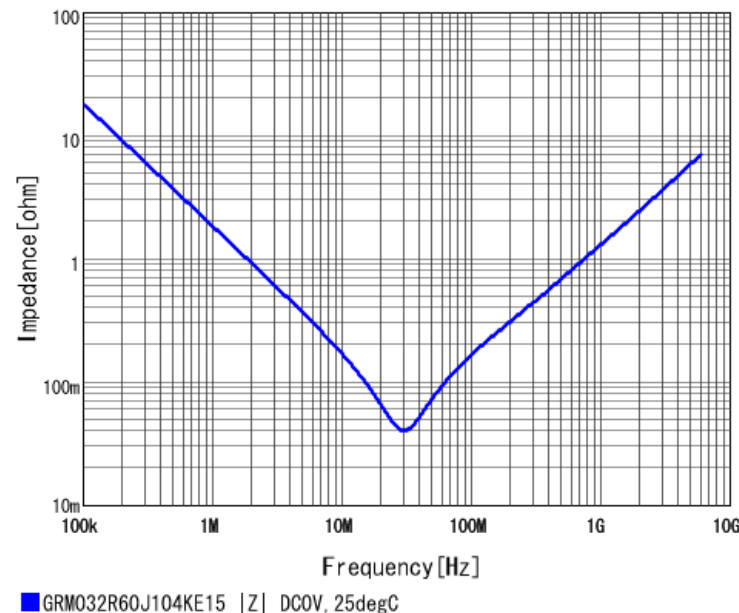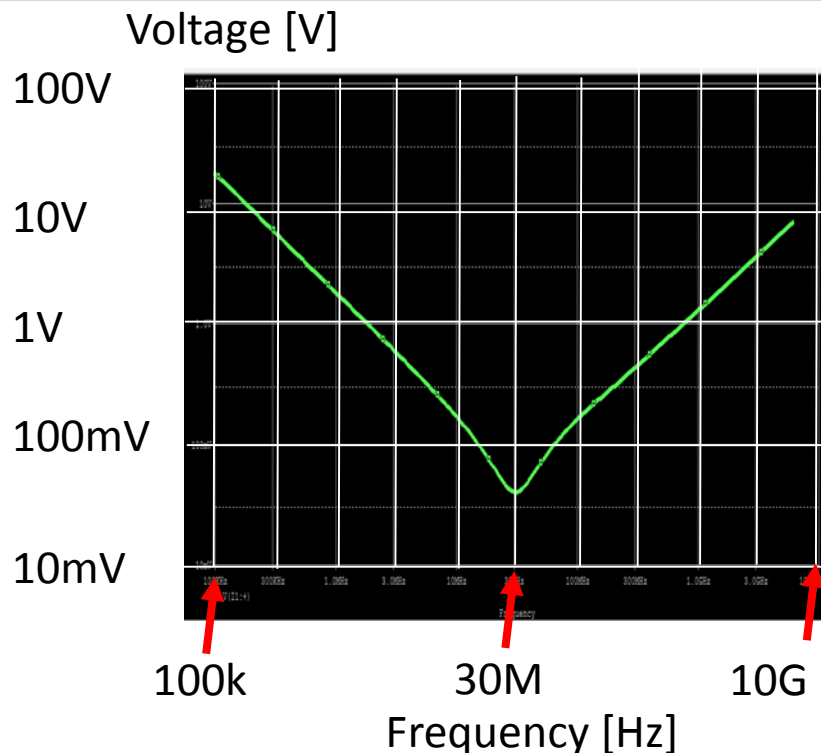
7. Example of Impedance Calculation of a Capacitor (3/4)

Since the current value of the current source is set to AC1 [A], the voltage applied to the capacitor is equivalent to the impedance.

(1) From the menu of the results display window, select Trace -> Add Trace.
(2) Selecting voltage V (I1: +) of current source I1 adds V (I1: +) in the Trace Expression.
(3) Clicking the **OK** button will draw a graph of the horizontal axis frequency and vertical axis voltage.

7. Example of Impedance Calculation of a Capacitor (4/4)

The figure on the left shows the characteristics of V (I1: +) set on the previous page. Since the current value is set to AC1 [A], this voltage value can be considered as the impedance.

In this step, the axis setting was changed to display a double-logarithmic graph. (Click the **Axis Label -> Setting Menu**)

The figure on the right shows the impedance characteristics of a capacitor of the same part number displayed in the design support tool SimSurfing of Murata indicated as a reference.

8. Inquiries

- Please use the Contact Form on Murata's Website for any inquiries regarding this library. (https://www.murata.com/ja-jp/contactform)

Entry example

Specify the subject

Enter the contents

For inquiries regarding the usage method of OrCAD, please contact the distributor where the product was purchased.

9. Others

- Microsoft, Windows are registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries.

- Cadence, OrCAD, PSpice registered trademarks or trademarks of Cadence Design Systems, Inc. in the United States and/or other countries.

The following describes how to create OLB files from the Spice netlist for each part number downloaded from Murata's design support tool SimSurfing.

For details, refer to the "Model Import Wizard" manual provided by Cadence Design Systems, Inc.

(Appendix 1) Changing File Name of Netlist

The following describes the netlist of "GRM31CR60J476ME19" as the subject.

(1) From the "multilayer ceramic capacitors" of the design support software SimSurfing, download the netlist of "GRM31CR60J476ME19."

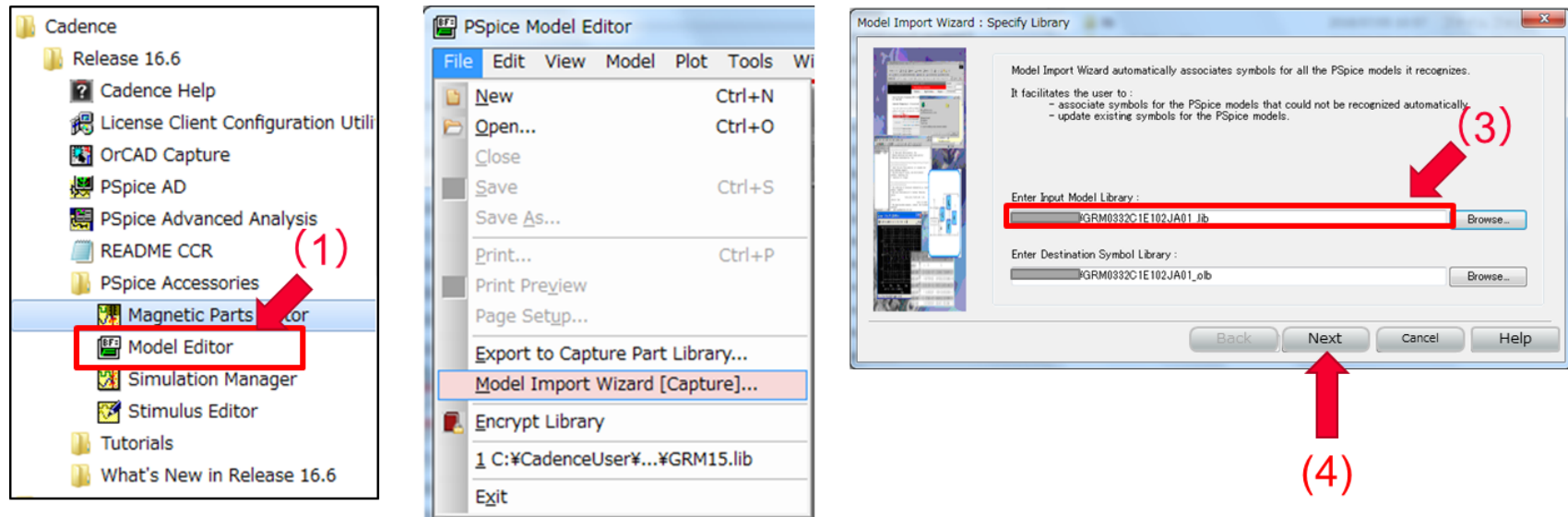GRM31CR60J476ME19_25degC.mod

Reference   https://ds.murata.co.jp/simsurfing/mlcc.html?lcid=ja

(2) Change the file extension from ".mod" to ".lib."
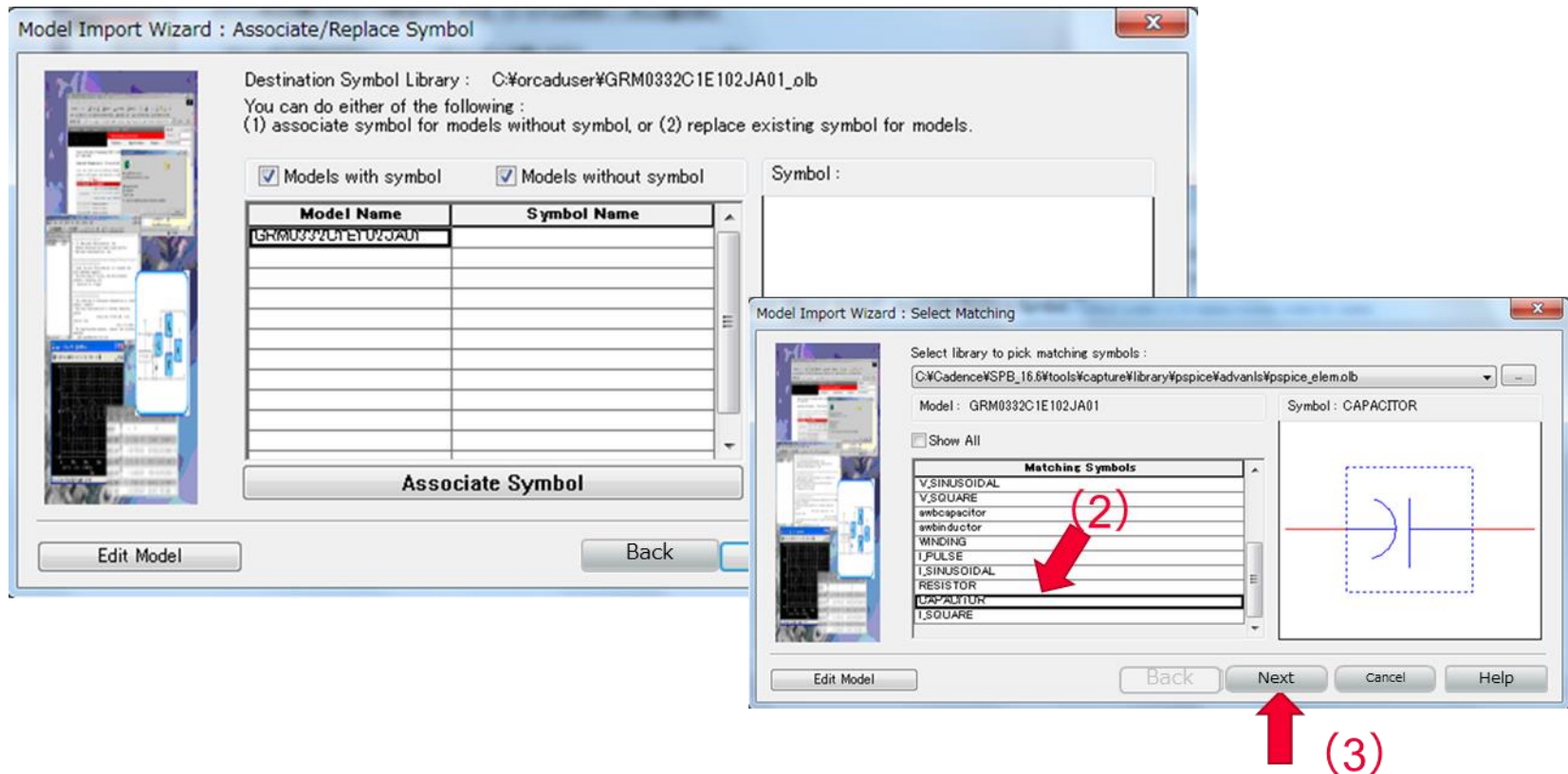
GRM31CR60J476ME19_25degC.mod

GRM31CR60J476ME19_25degC.lib

(Appendix 2) Selection of a LIB File

(1) From the Start menu of Windows, start Model Editor.

(2) From the menu of the Model Editor, select File -> Model Import Wizard.

(3) As the model library to be input, select the previous LIB file.

   The save destination of the symbol library will be displayed automatically.

(4) Click **Next**.

(Appendix 3) Association of Symbols and LIB File
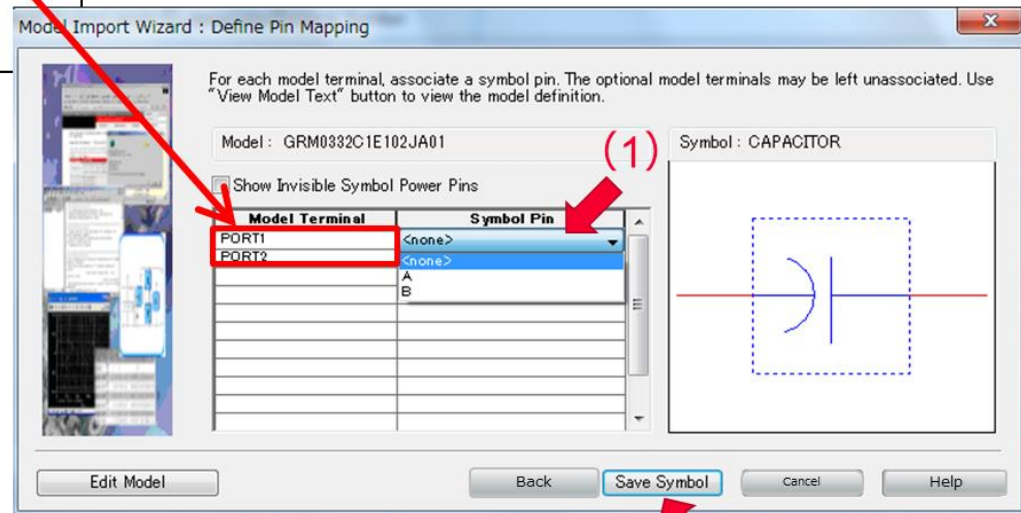

(1) Click **Associate Symbol**.

(2) In the Select Matching window, select CAPACITOR.

(3) Click **Next**.

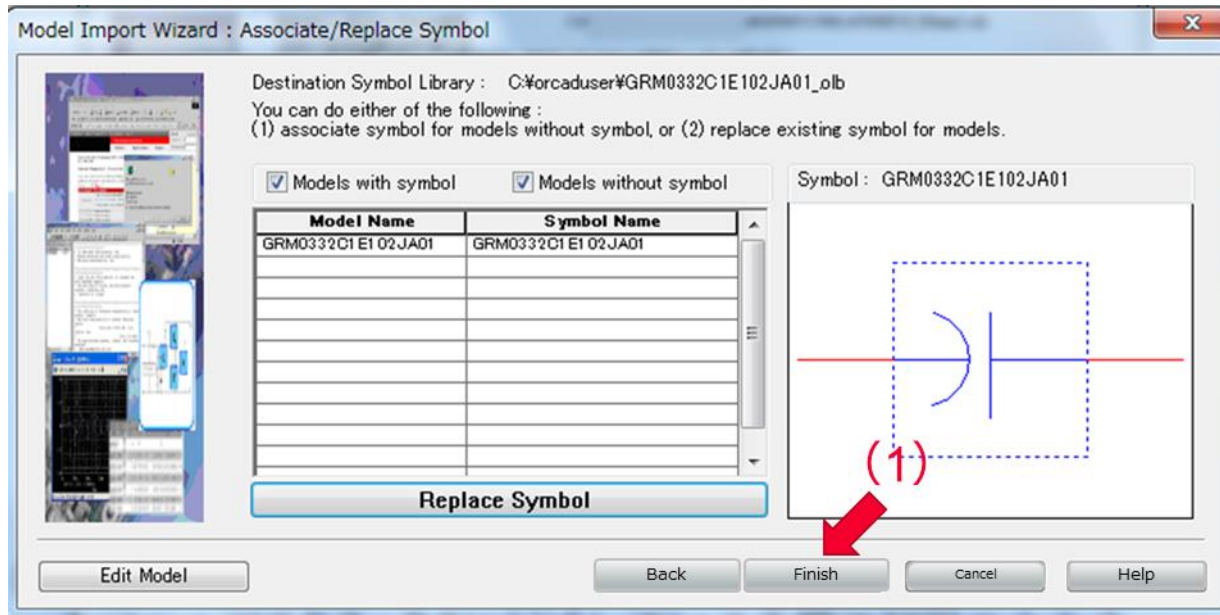# (Appendix 4) Association of Terminal and PIN

Contents of Netlist

(1) From the menu of Symbol Pin, associate PORT1 with "A" and PORT2 with "B."
(2) Click **Save Symbol**.

(Appendix 5) Creation of an OLB File

(1) Clicking **Finish** will create an OLB file.
　　It can be used by the procedure described in Pages 7 and 8 of this manual.