TAIYO YUDEN Component Library for Cadence PSpice (Standard Model)

- Installation manual -

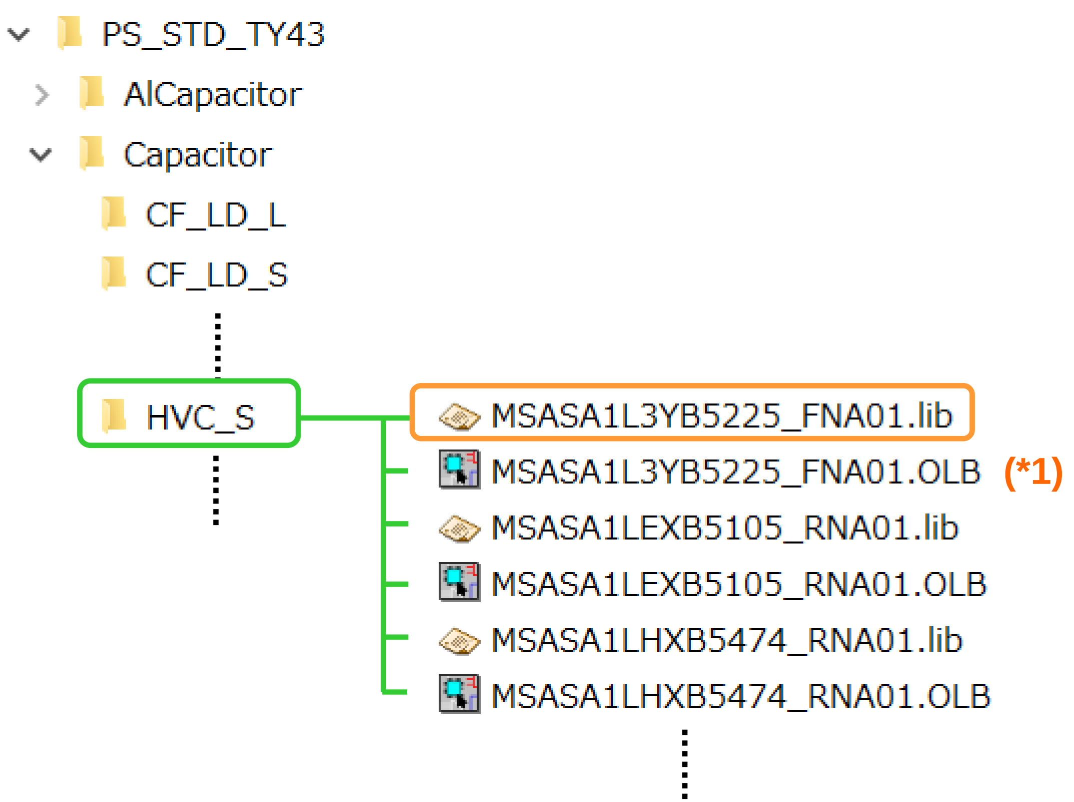
Contents

How to install Component Library

Step 1. **Unzip "PS_STD_TY**.zip".**

Step 2. **Copy the netlist file(.lib) you would like to use to any folder you like.**

***1 Symbol files(.OLB) are for simulation in OrCAD.**
Please refer to P6-P11 to use symbol files(.OLB) in OrCAD.



How to use Component Library

Step 1. Describe the library in the netlist.

netlist example

Sample.cir - メモ帳

ファイル(F) 編集(E) 書式(O) 表示(V) ヘルプ(H)

Step 1

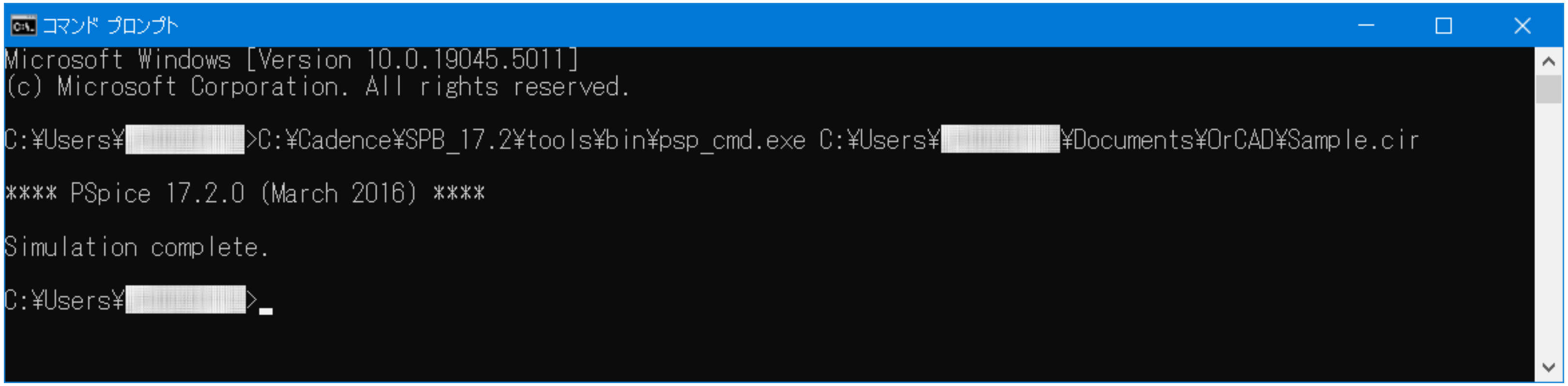
```
V1 1 0 AC 0.01
X1 1 0 MSASA1L3YB5225_FNA01
.ac oct 100 100Hz 3GHz
.lib MSASA1L3YB5225_FNA01.lib
.option numdgt=9
.print AC IR(X1:1) II(X1:1)
.end
```

1 行、1 100% Windows (CRLF) UTF-8

***1 Refer to the PSpice manual for the description of the netlist.**

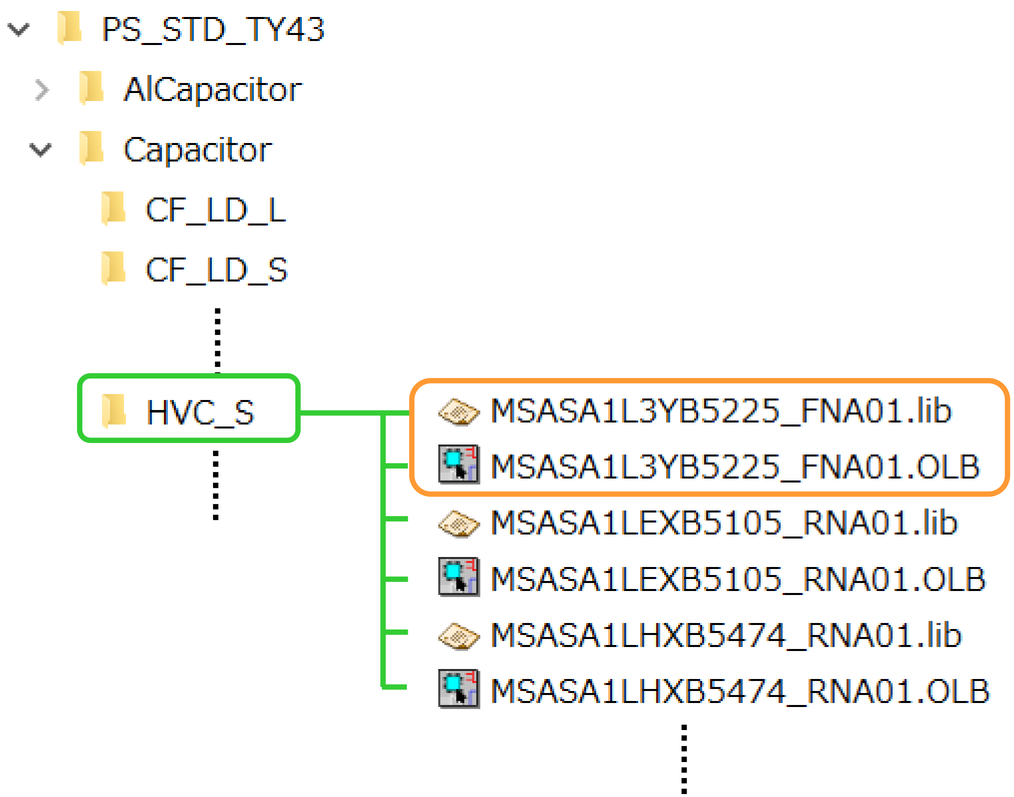
How to use Component Library

Step 2. **Perform the simulation from the command line such as windows command prompt.**



How to use Component Library on OrCAD

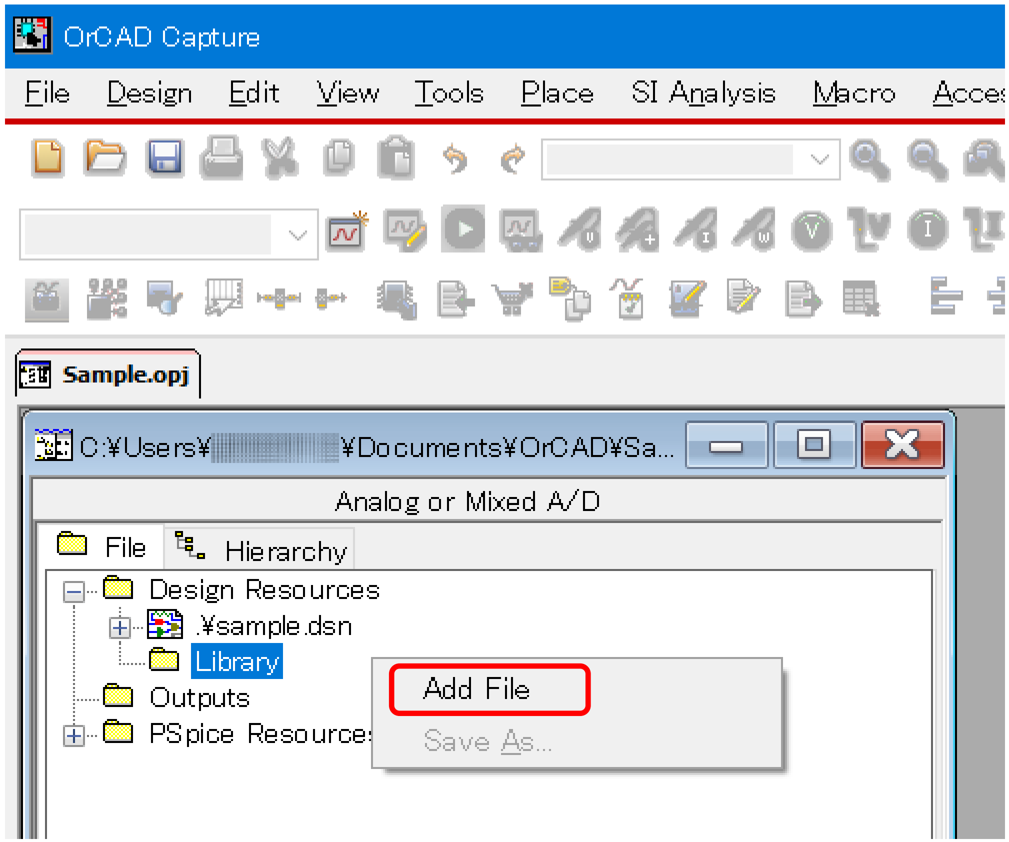
Step 1. **Copy the netlist file(.lib) and the symbol file(.OLB) you would like to use from the folder where you unzipped the component library to any folder you like.**



How to use Component Library on OrCAD

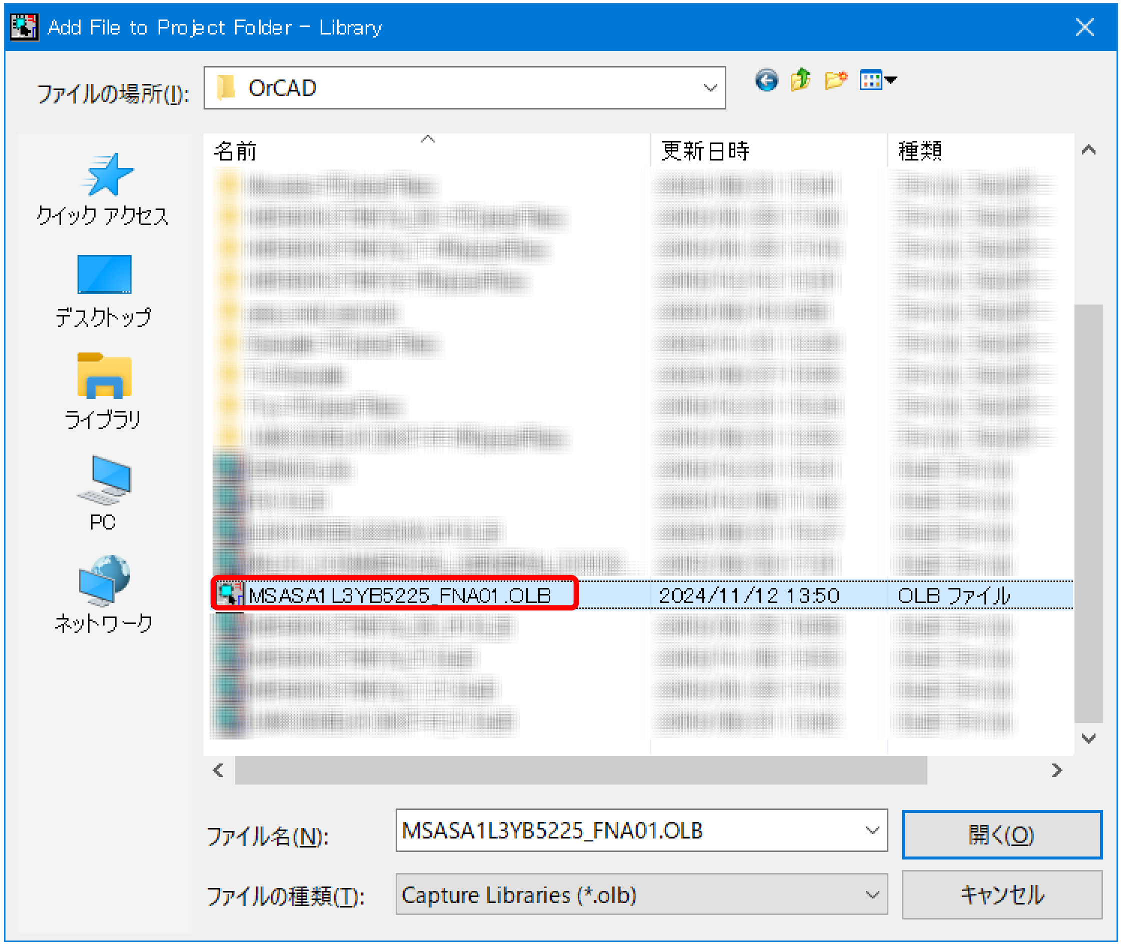
Step 2. **Launch OrCAD Capture and open the project file or make a new project file.**

Step 3. **Right-click Library on the project tree and select Add File.**



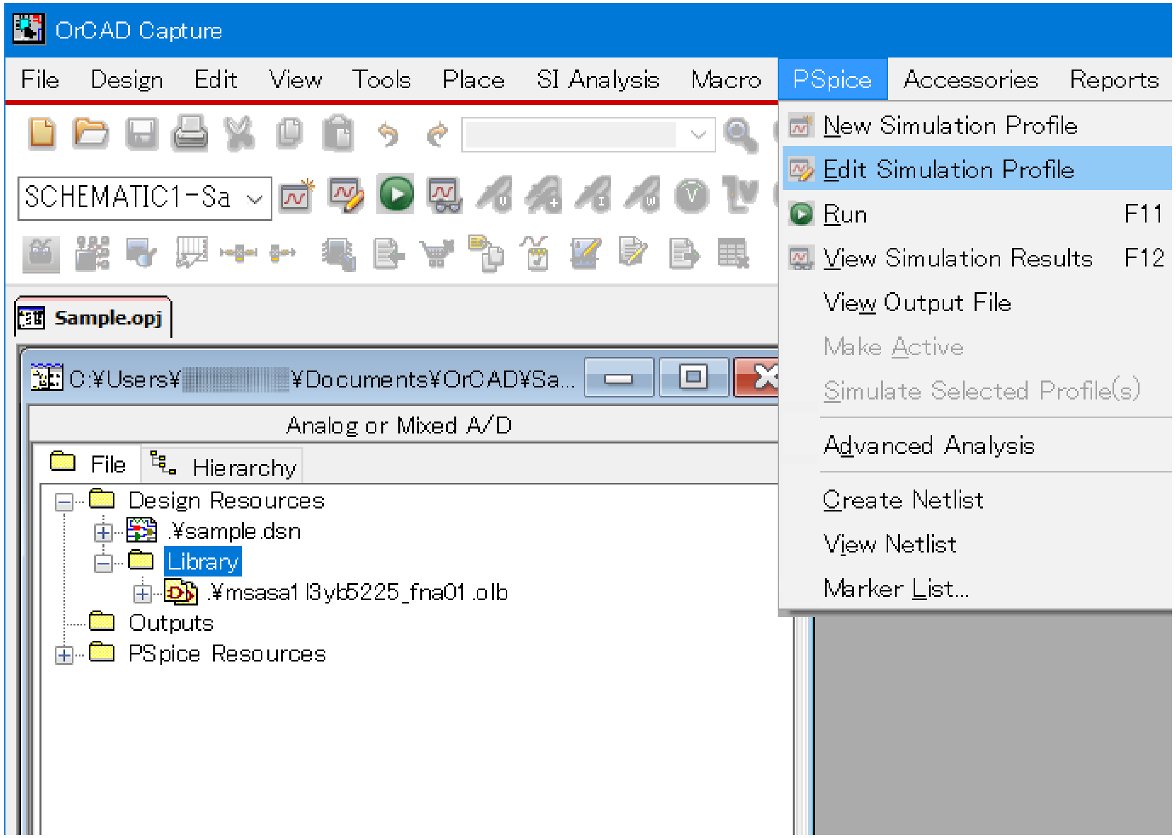
How to use Component Library on OrCAD

Step 4. **Select the symbol file(.OLB) prepared at step 1 to register symbol.**



How to use Component Library on OrCAD

Step 5. Select PSpice > Edit Simulation Profile from the menu bar.



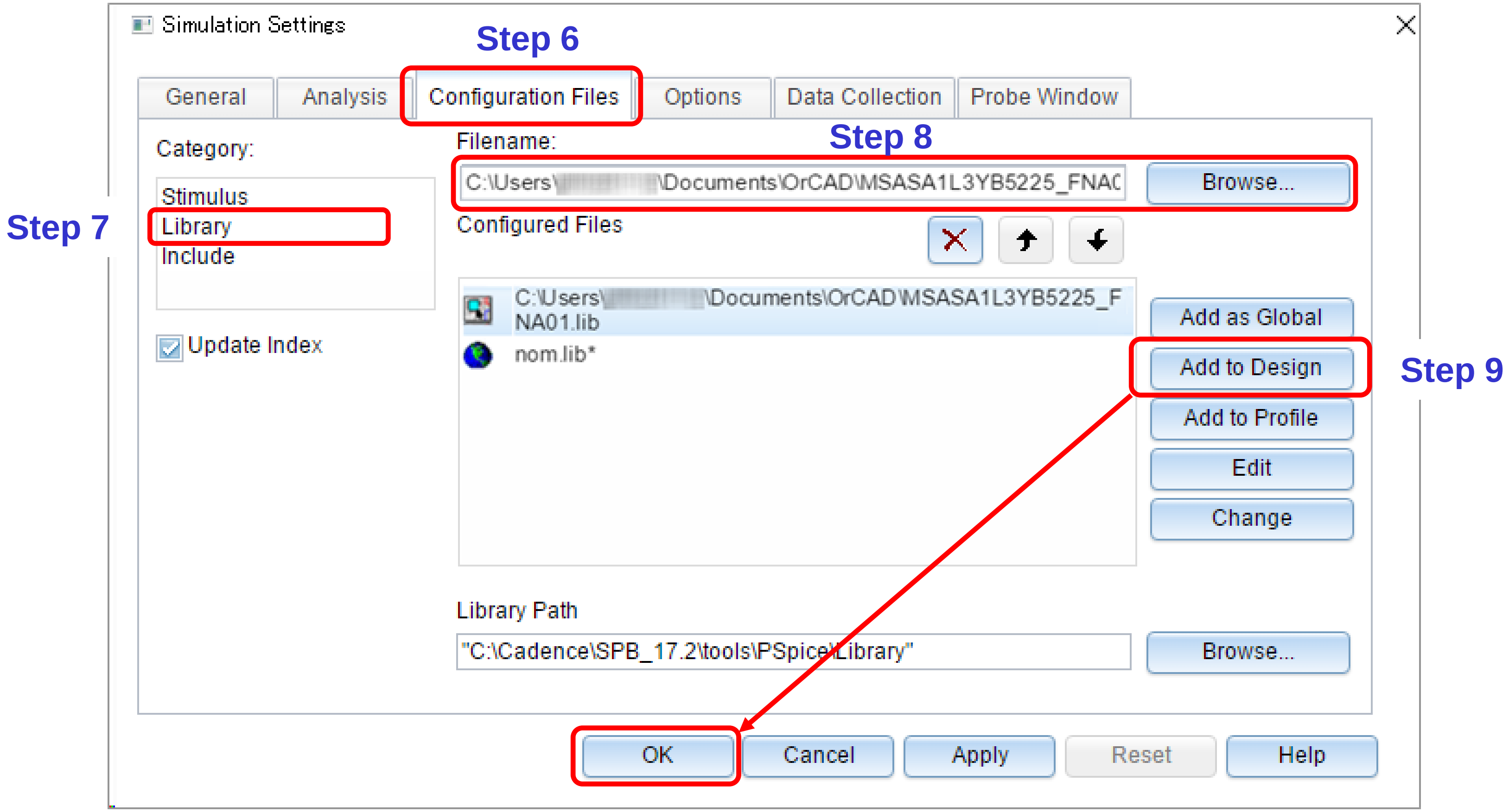
How to use Component Library on OrCAD

Step 6. Select Configuration Files tab.

Step 7. Select “Library” on the Category pane.

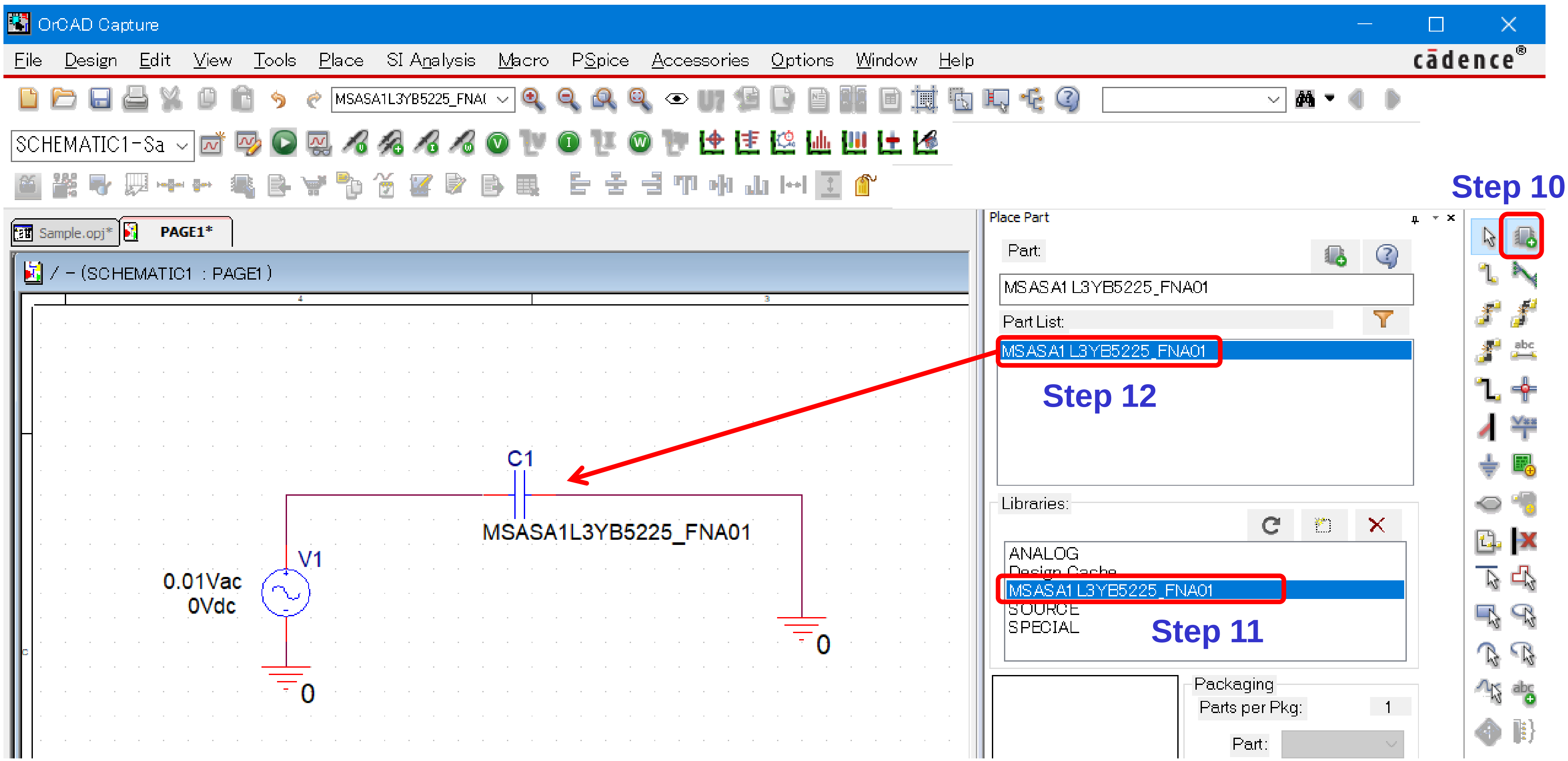
Step 8. Select the library(.lib) at the Filename section.

Step 9. Click Add to Design, then OK to register library(.lib).



How to use Component Library on OrCAD

Step 10. Open the schematic and select the Part icon.
Step 11. Select the library on the Libraries pane on the Place Part window.
Step 12. Double-click the component on the Part List pane to put on the schematic.
Step 13. Perform the simulation after completing the schematic.



TAIYO YUDEN