TAIYO YUDEN Component Library for ANSYS Electronics Desktop Circuit Simulator

- Installation manual -

Contents

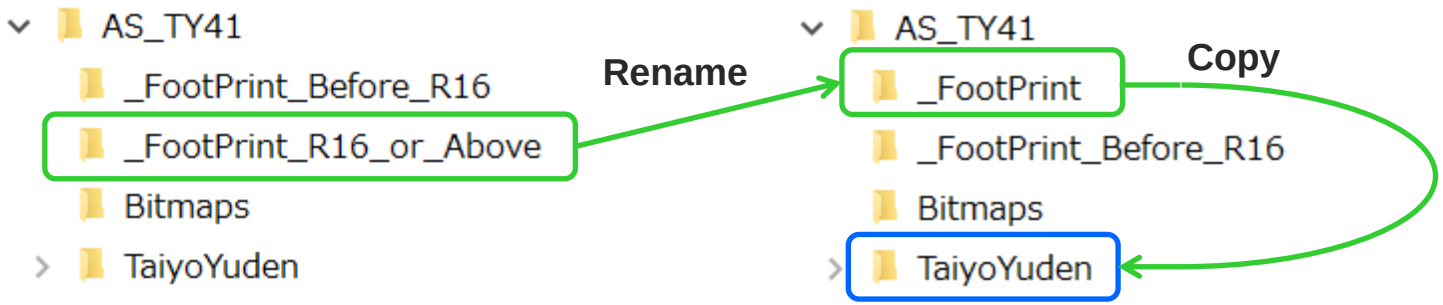
How to install Component Library

Step 1. **Unzip AS_TY**.zip.**

Step 2. **In case of using R16.0 or above versions, rename _FootPrint_R16_or_Above folder to _FootPrint. Otherwise, rename _FootPrint_Before_R16 to _FootPrint.**

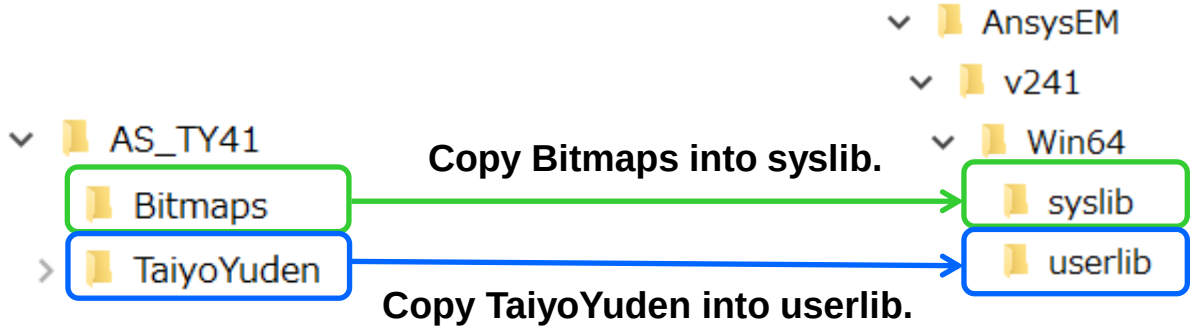
Step 3. **Copy _FootPrint folder into TaiyoYuden folder.**

Example for R16.0 or above versions



How to install Component Library

Step 4. **Copy Bitmaps folder and TaiyoYuden folder into specified folders(*1, *2) below where Electronics Desktop is installed. The installation is then complete(*3).**



***1 In case that the older version of the library is already installed, delete Taiyo_Yuden folder from userlib folder before installing the new library.**

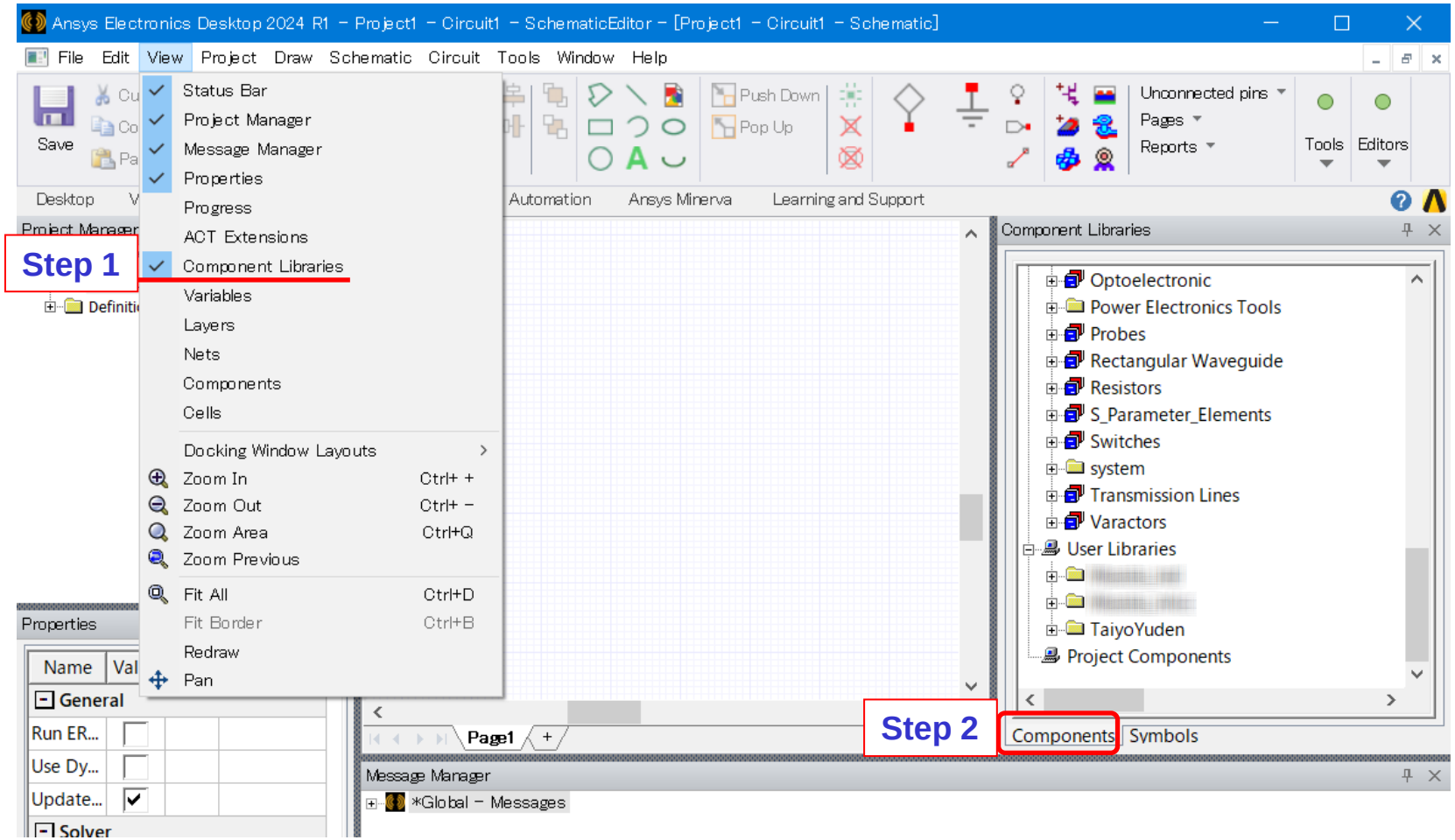
***2 If you use the legacy version of Electronics Desktop or ANSYS Designer, Bitmaps folder may exists in the folder where Electronics Desktop is installed. In that case, copy Bitmaps folder of the library onto the Bitmaps folder of Electronics Desktop.**

***3 If you use the legacy version of Electronics Desktop or ANSYS Designer, additional installation procedures may be required. If the library is not registered to Electronics Desktop after step 4, refer to P9-P11.**

How to use Component Library

Step 1. **Select View-> Component Libraries from menu bar after entering Circuit Design.**

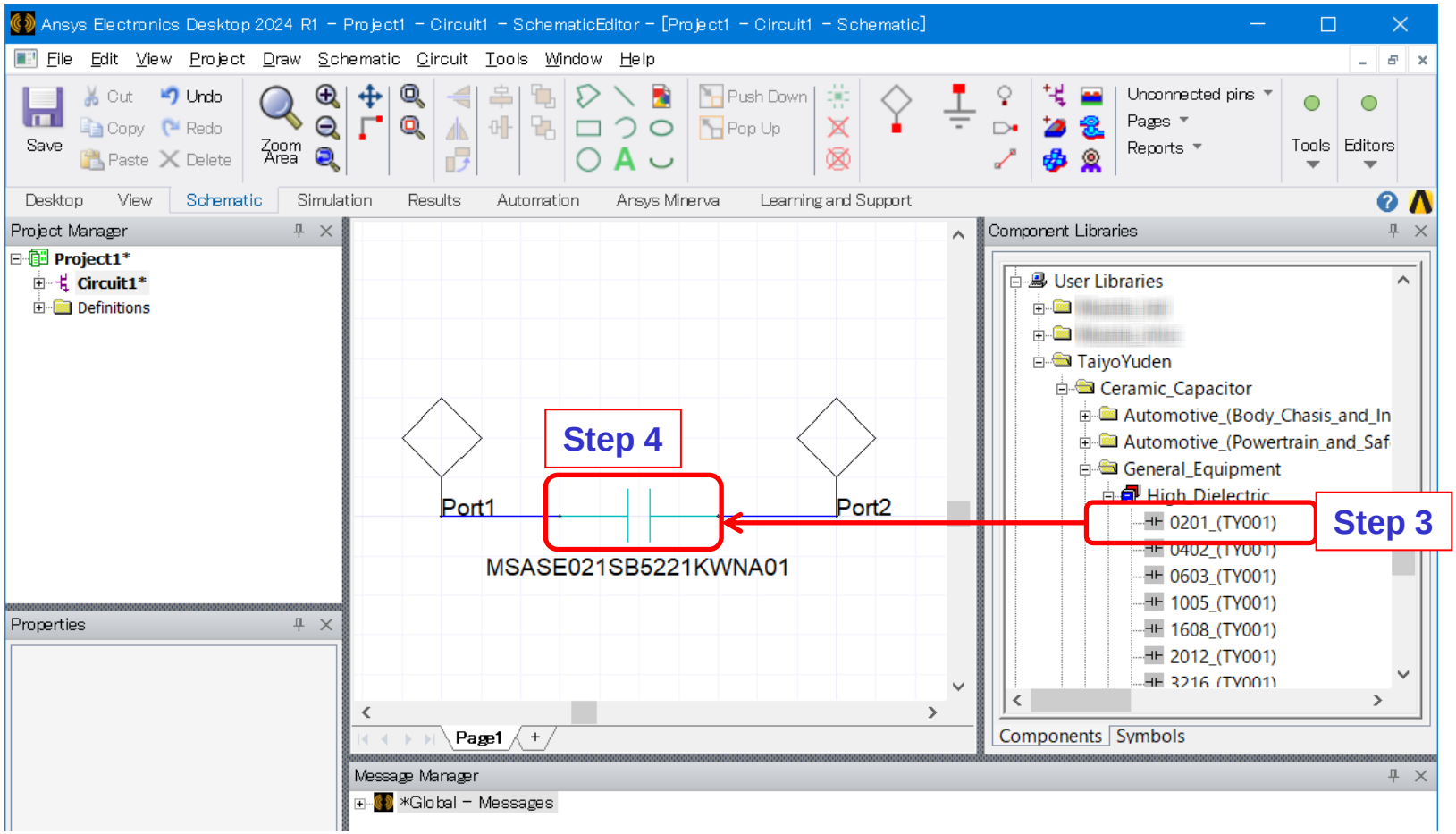
Step 2. **Select Components tab on Component Libraries window.**



How to use Component Library

Step 3. **Double-click the desired component from TaiyoYuden folder at User Libraries, and put it onto the schematic.**

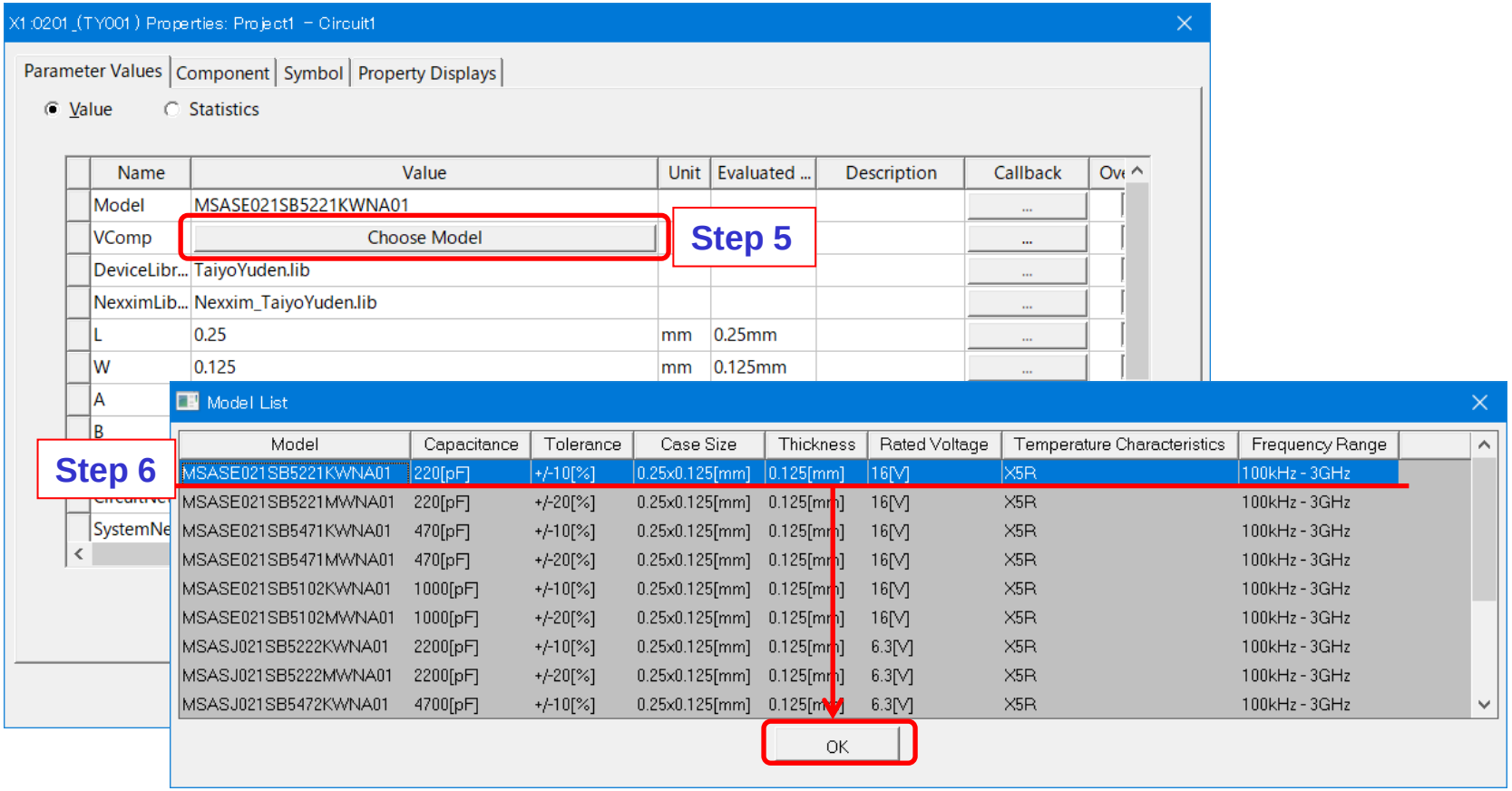
Step 4. **Double-click the component on the schematic.**



How to use Component Library

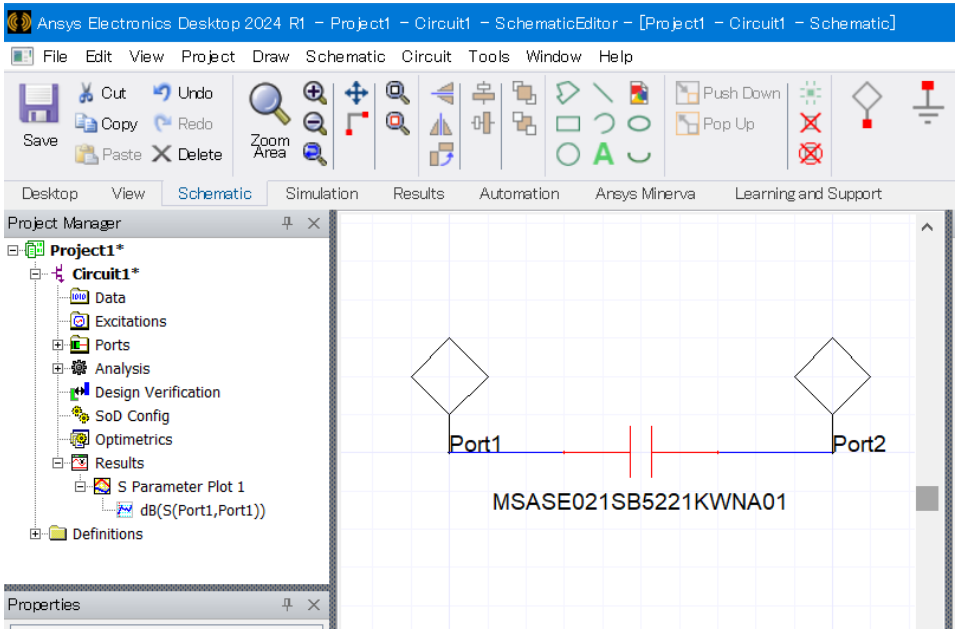
Step 5. **Click Choose Model.**

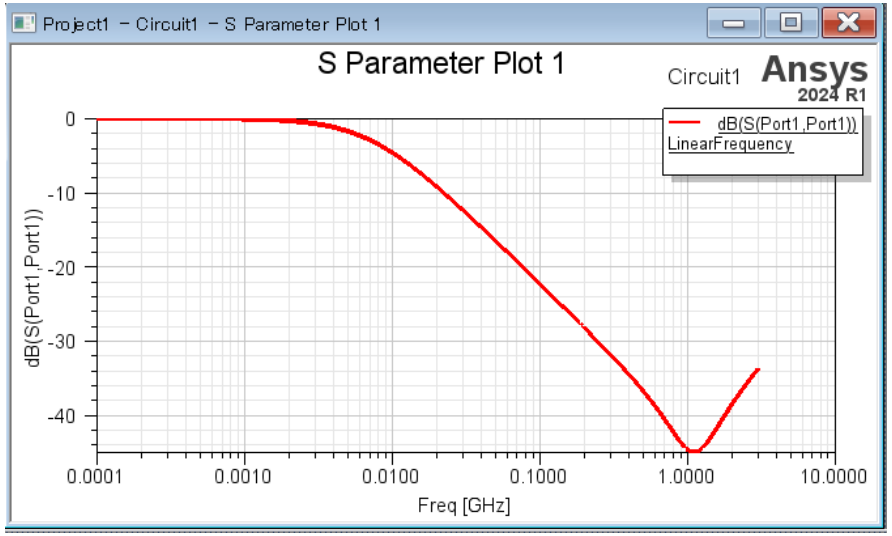
Step 6. **Select the desired model from Model List window and click OK.**



How to use Component Library

Step 7. Perform the simulation.

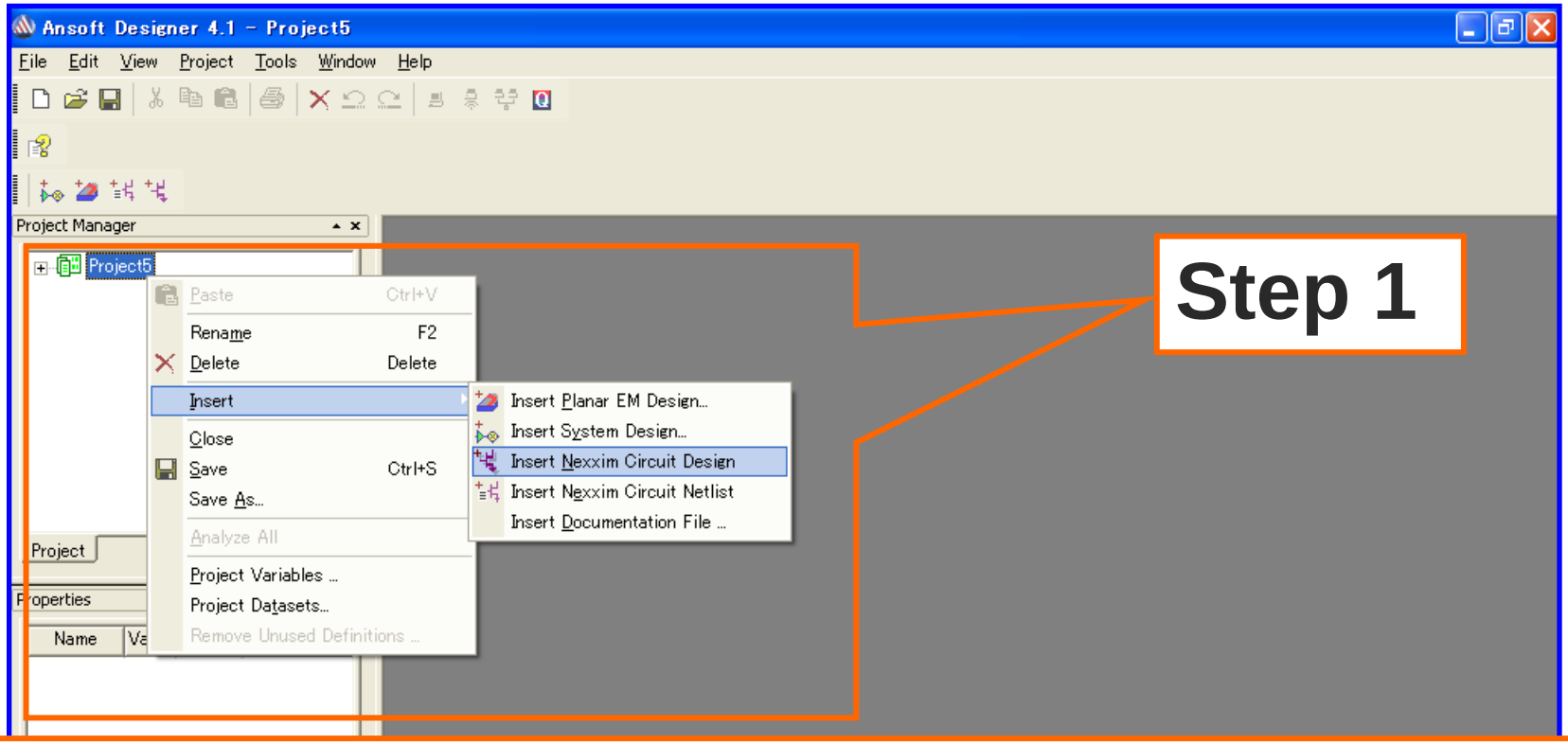




* In case that failing in analyzing models in legacy version using nexxim engine, refer to P12.

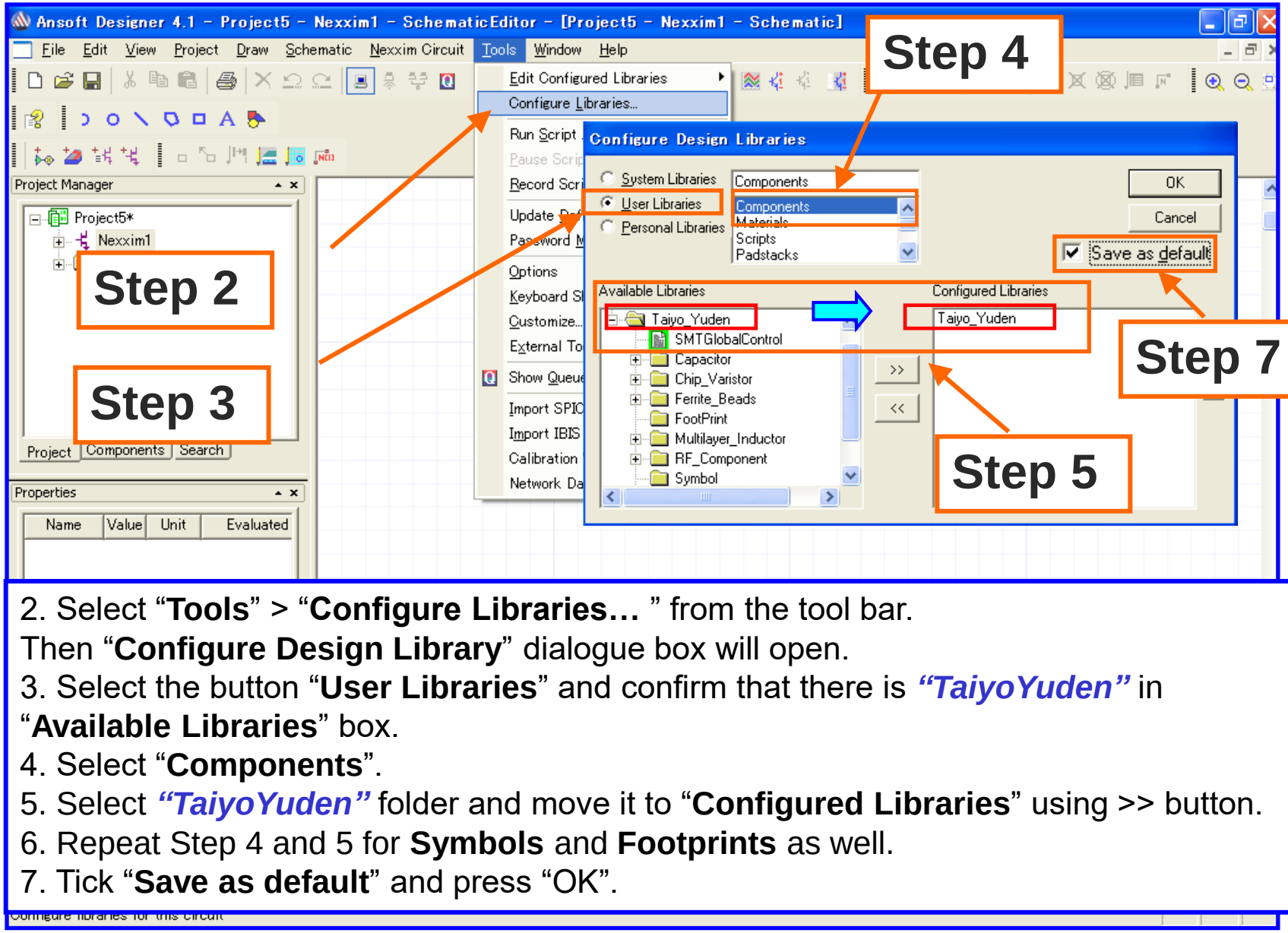
Additional installation procedure for legacy version

The following explanation is for ANSYS Designer 4.1, however, the whole procedure is almost all the same for other versions.



After launching ANSYS Designer,
1. Right-click on the project name (e.g. Project1) in the Project Manager.
Select “Insert” > “Insert Nexxim Circuit Design”
Then a schematic window will open.

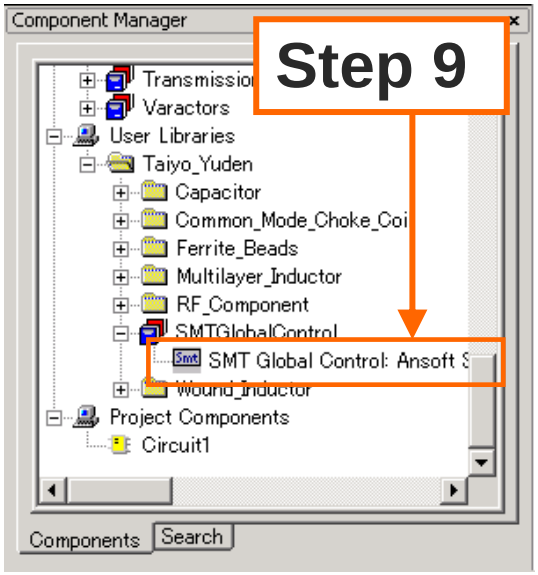
Additional installation procedure for legacy version



2. Select "**Tools**" > "**Configure Libraries…** " from the tool bar.
Then "**Configure Design Library**" dialogue box will open.
3. Select the button "**User Libraries**" and confirm that there is ***"TaiyoYuden"*** in "**Available Libraries**" box.
4. Select "**Components**".
5. Select ***"TaiyoYuden"*** folder and move it to "**Configured Libraries**" using >> button.
6. Repeat Step 4 and 5 for **Symbols** and **Footprints** as well.
7. Tick "**Save as default**" and press "OK".

Additional installation procedure for legacy version

The following procedures are not required for versions not using nexxim engine.

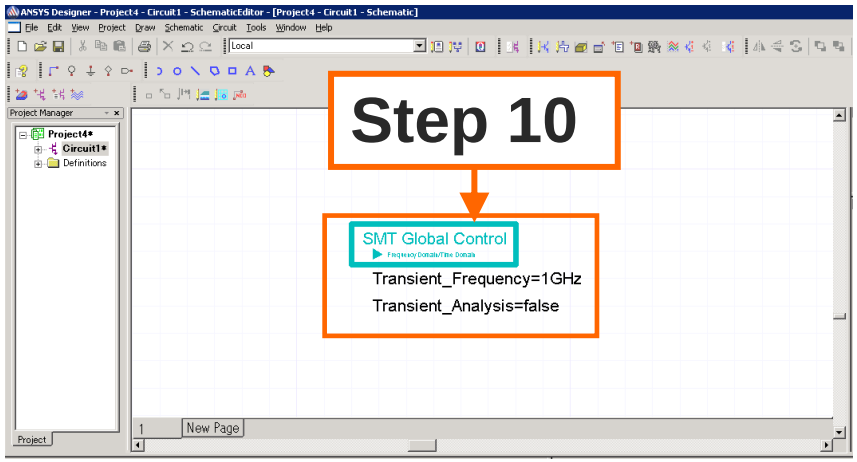


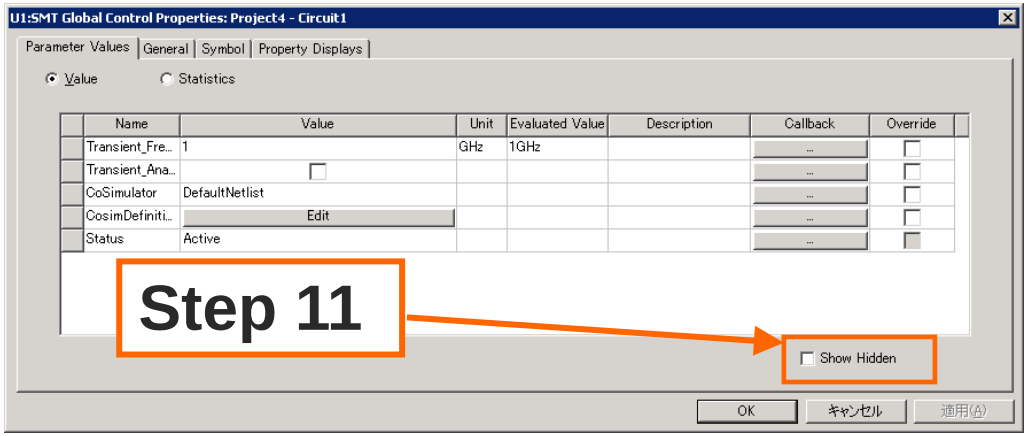
8. Reboot ANSYS Designer.

Do the following procedure with schematic window open.

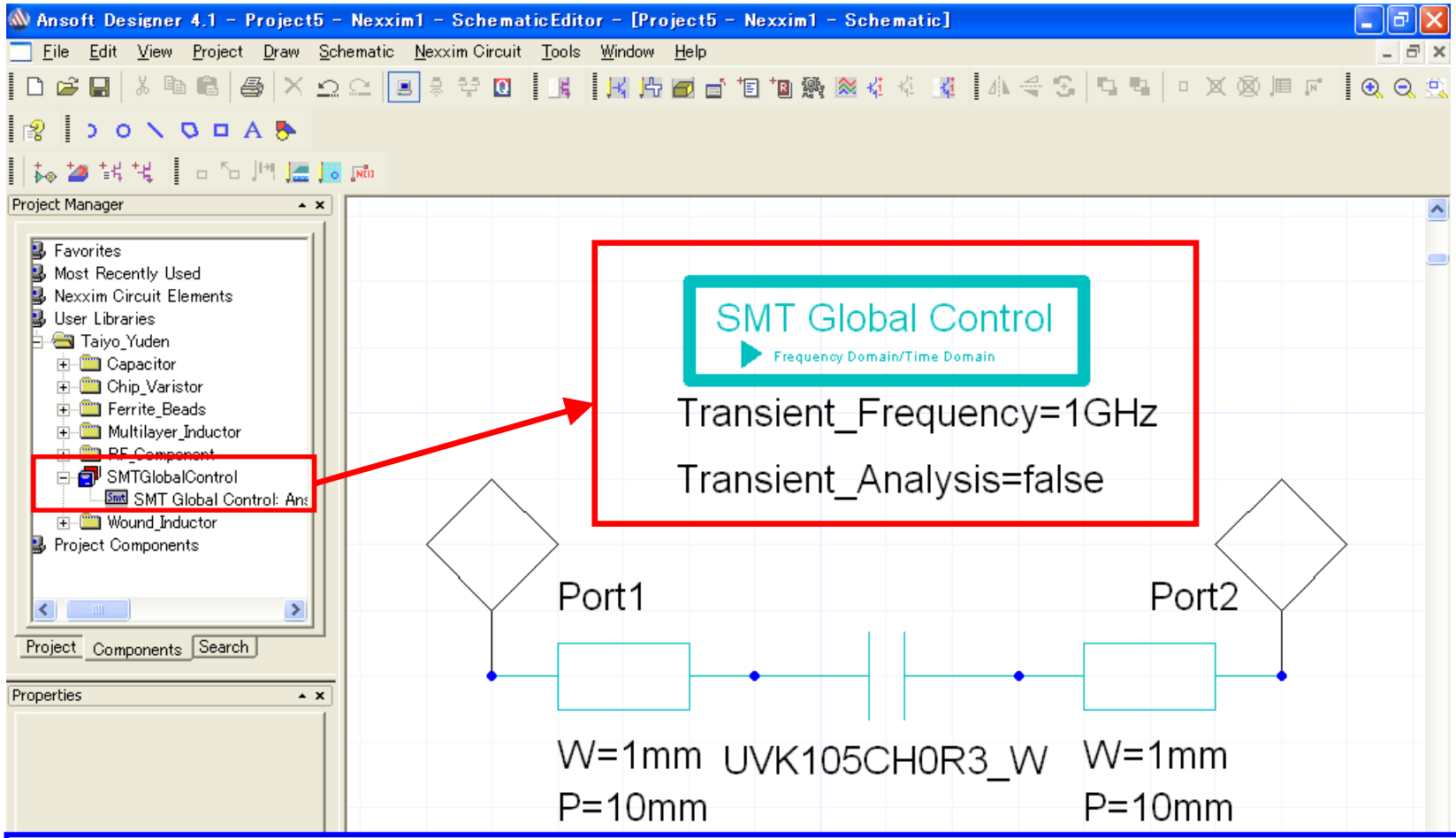
9. Click the component tab, then select User Libraries -> Taiyo Yuden -> SMT Global Control and put it on the schematic.
10. Double click the SMT Global Control to open the property window.
11. Uncheck “Show Hidden” checkbox and click OK button.

That’s all for the library configuration.





In case of failing in analyzing models in legacy version



1. Put a [SMT Global Control] in Taiyo_Yuden onto the schematic.

2. For a transient analysis, tick [Transient_Analysis] in the property of [SMT Global Control]. Then set [Transient_Frequency] value to the fundamental frequency of the signal to be analyzed.

TAIYO YUDEN