

TAIYO YUDEN Component Library for ANSYS Electronics Desktop Circuit Simulator

- Installation manual -

Contents

- * **How to install Component Library (P3-P4)**
- * **How to use Component Library (P5-P8)**
- * **Additional installation procedure for legacy version (P9-P11)**
- * **In case of failing in analyzing models in legacy version (P12)**

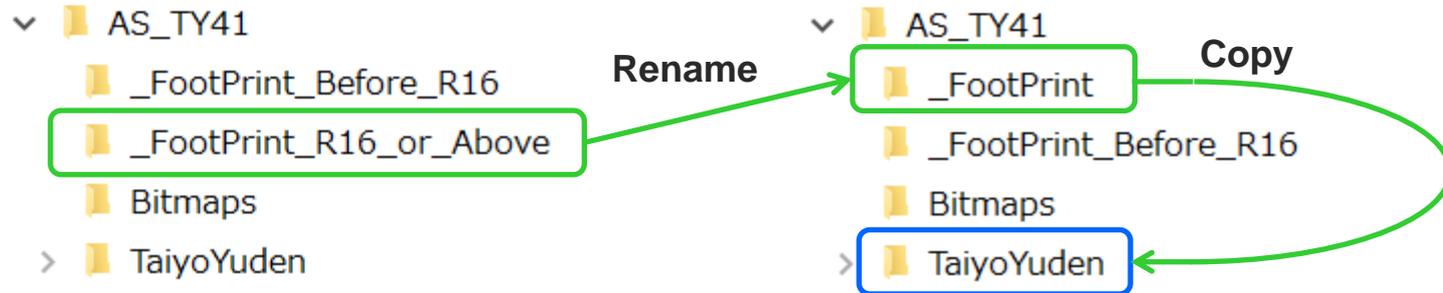
How to install Component Library

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`.

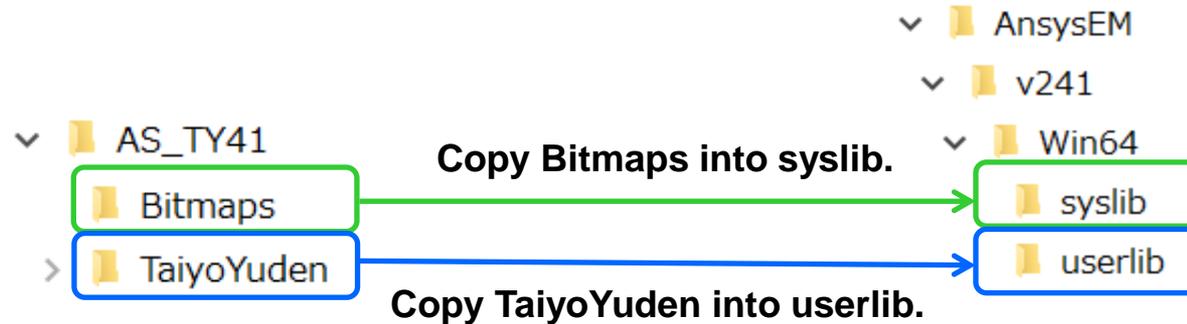
Step 3. Copy `_FootPrint` folder into TaiyoYuden folder.

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).

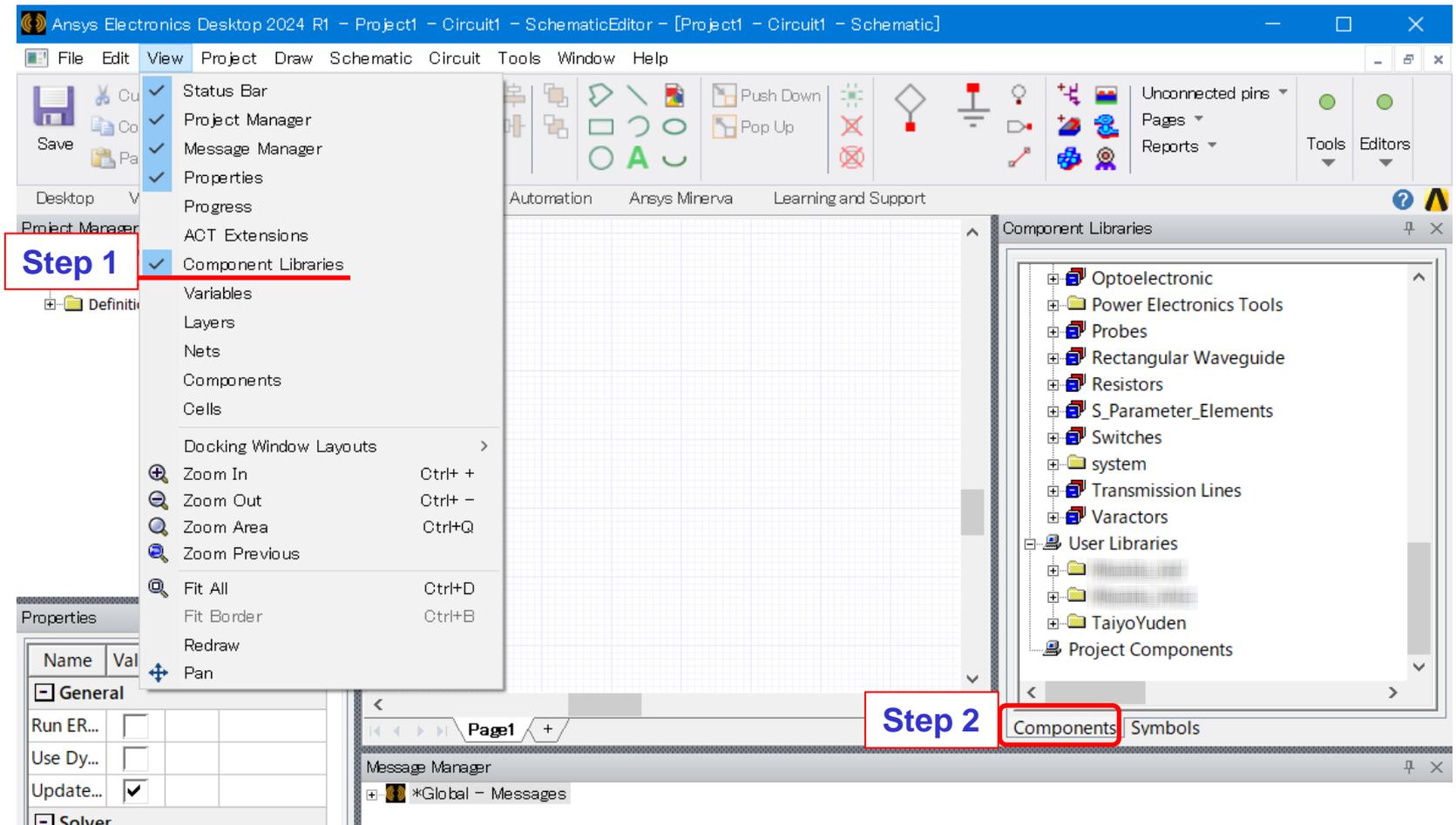


- *1 In case that the older version of the library is already installed, delete Taiyo_Yuden folder from userlib folder before installing the new library.
- *2 If you use the legacy version of Electronics Desktop or ANSYS Designer, Bitmaps folder may exist in the folder where Electronics Desktop is installed. 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.

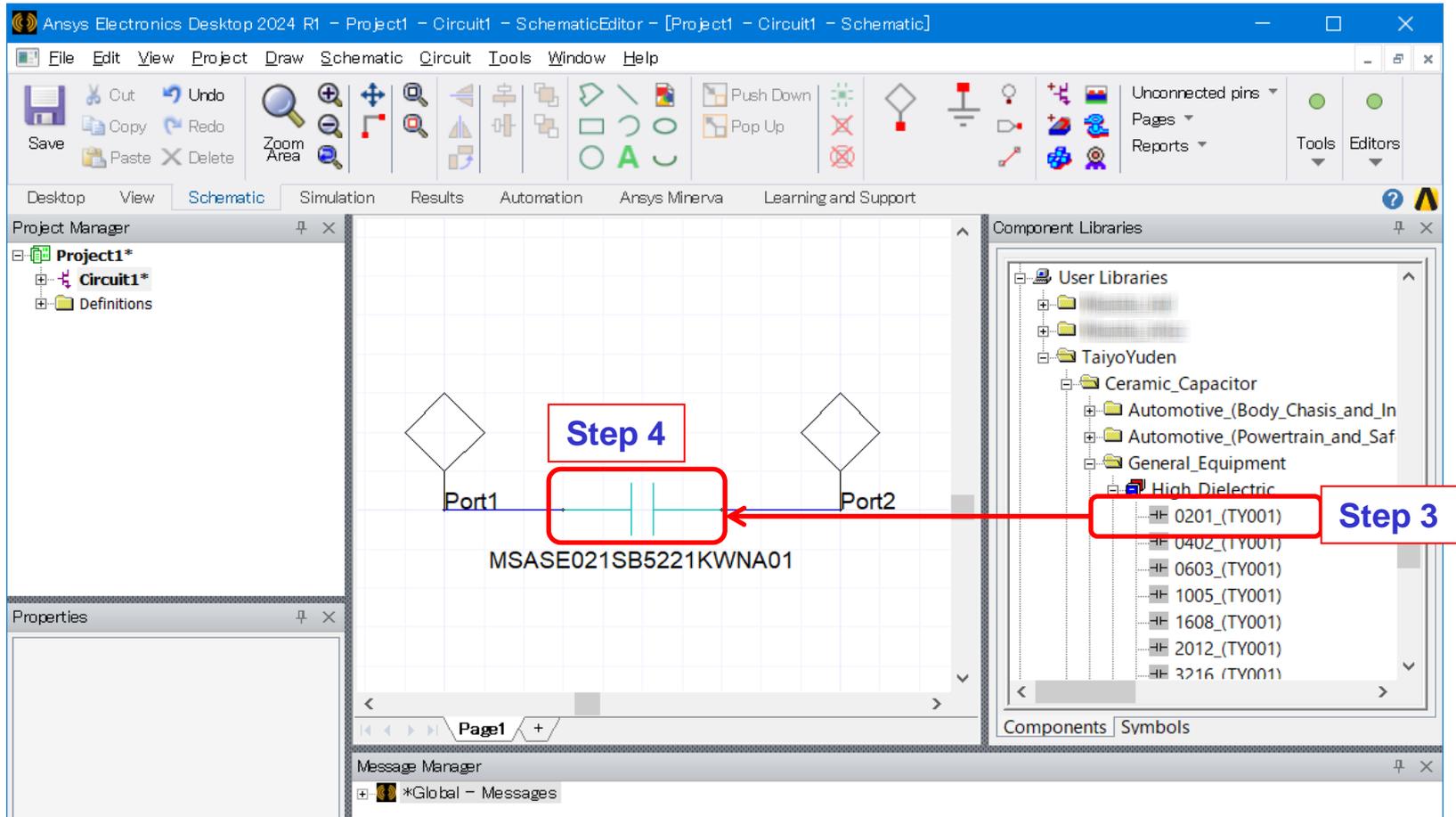
Step 2. Select Components tab on Component Libraries window.



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.

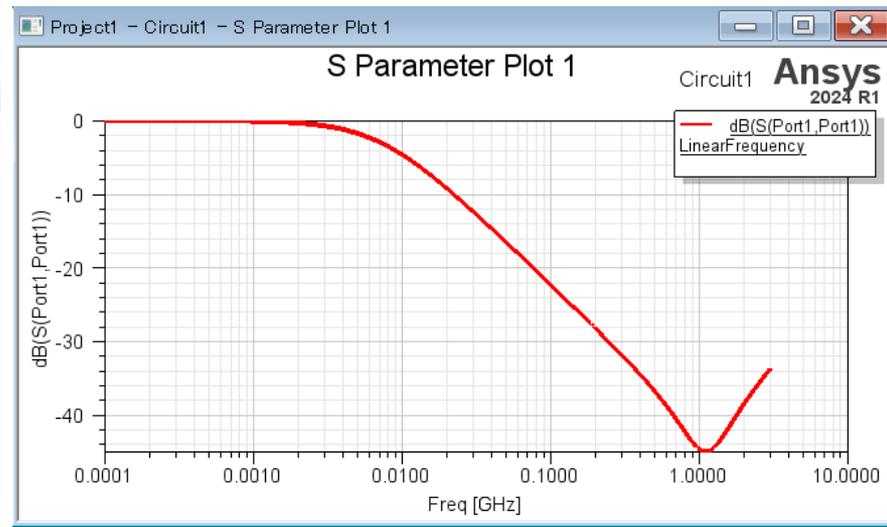
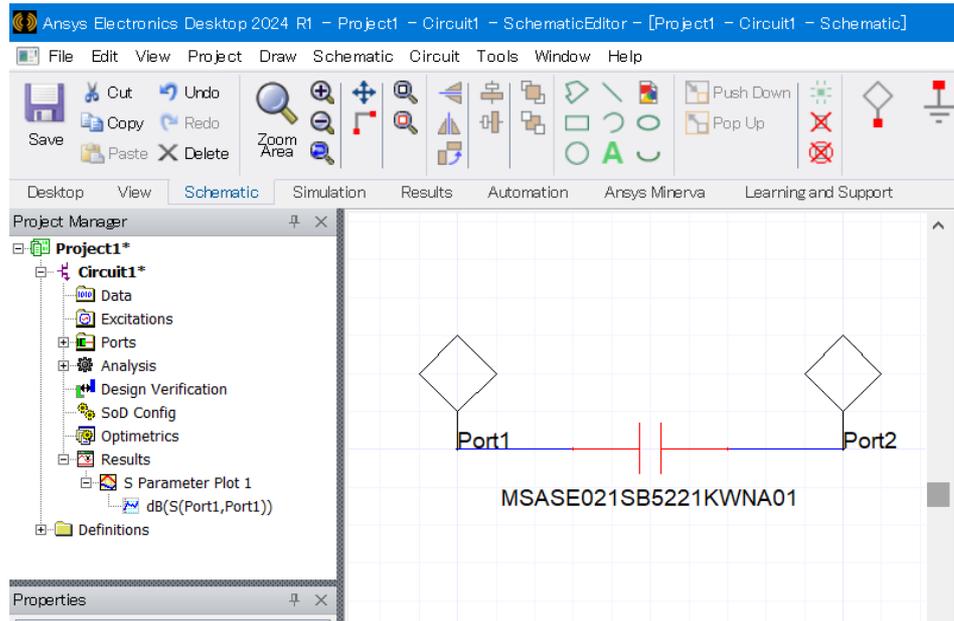
Step 6. Select the desired model from Model List window and click OK.

The screenshot shows the 'Properties: Project1 - Circuit1' window with the 'Value' tab selected. The 'VComp' field is set to 'MSASE021SB5221KWNA01' and has a 'Choose Model' button next to it. The 'Model List' dialog box is open, showing a table of component models. The first row is highlighted in blue, and the 'OK' button is circled in red.

Model	Capacitance	Tolerance	Case Size	Thickness	Rated Voltage	Temperature Characteristics	Frequency Range
MSASE021SB5221KWNA01	220[pF]	+/-10[%]	0.25x0.125[mm]	0.125[mm]	16[V]	X5R	100kHz - 3GHz
MSASE021SB5221MWNA01	220[pF]	+/-20[%]	0.25x0.125[mm]	0.125[mm]	16[V]	X5R	100kHz - 3GHz
MSASE021SB5471KWNA01	470[pF]	+/-10[%]	0.25x0.125[mm]	0.125[mm]	16[V]	X5R	100kHz - 3GHz
MSASE021SB5471MWNA01	470[pF]	+/-20[%]	0.25x0.125[mm]	0.125[mm]	16[V]	X5R	100kHz - 3GHz
MSASE021SB5102KWNA01	1000[pF]	+/-10[%]	0.25x0.125[mm]	0.125[mm]	16[V]	X5R	100kHz - 3GHz
MSASE021SB5102MWNA01	1000[pF]	+/-20[%]	0.25x0.125[mm]	0.125[mm]	16[V]	X5R	100kHz - 3GHz
MSASJ021SB5222KWNA01	2200[pF]	+/-10[%]	0.25x0.125[mm]	0.125[mm]	6.3[V]	X5R	100kHz - 3GHz
MSASJ021SB5222MWNA01	2200[pF]	+/-20[%]	0.25x0.125[mm]	0.125[mm]	6.3[V]	X5R	100kHz - 3GHz
MSASJ021SB5472KWNA01	4700[pF]	+/-10[%]	0.25x0.125[mm]	0.125[mm]	6.3[V]	X5R	100kHz - 3GHz

How to use Component Library

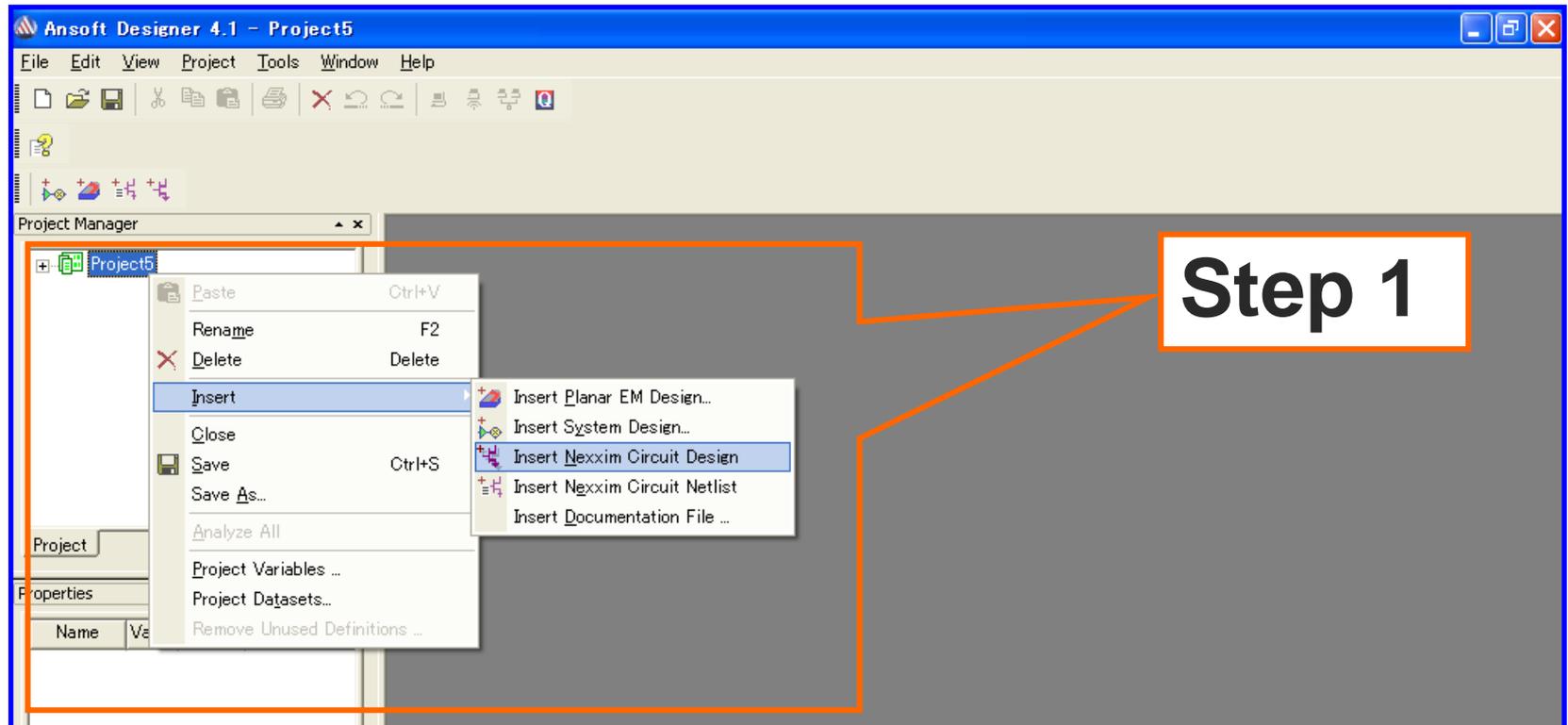
Step 7. Perform the simulation.



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

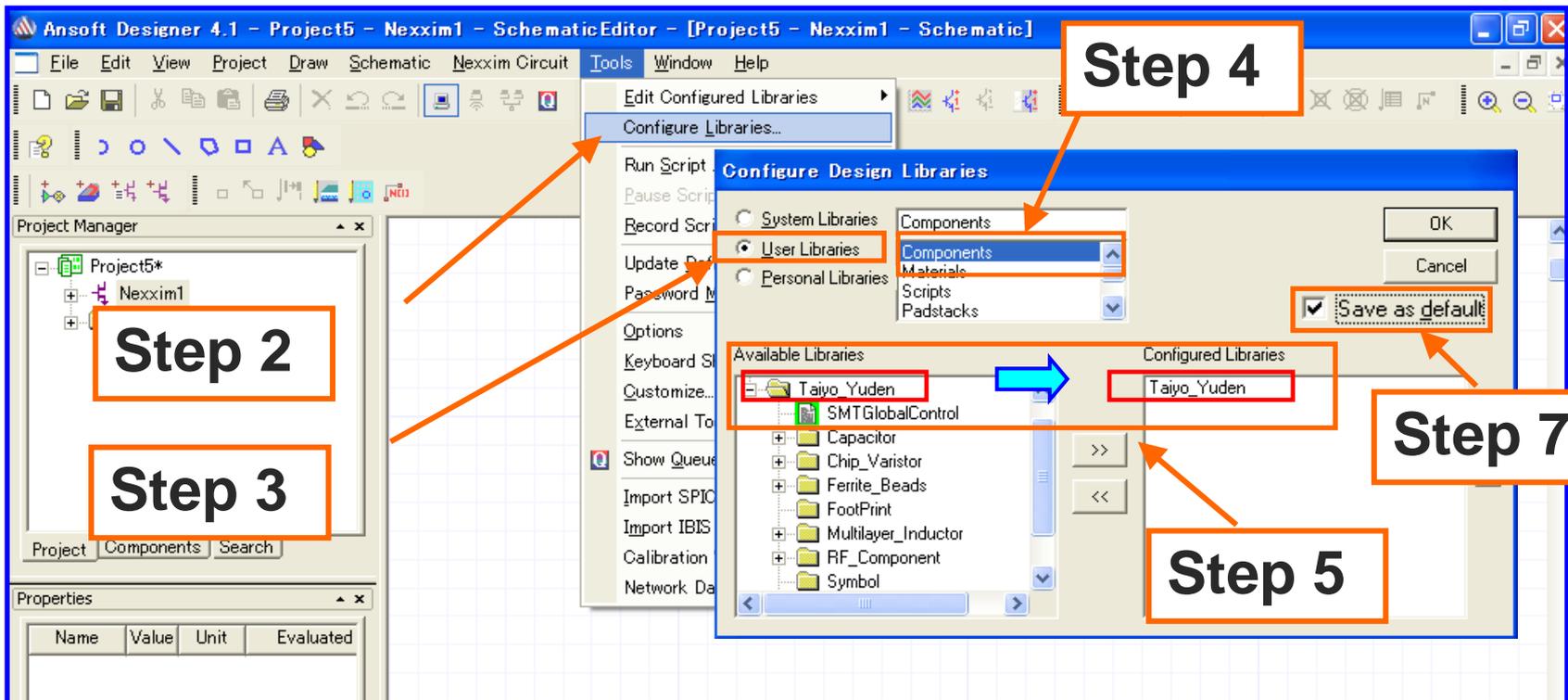
Additional installation procedure for legacy version

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 **“Taiyo Yuden”** in **“Available Libraries”** box.
4. Select **“Components”**.
5. Select **“Taiyo Yuden”** 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”**.

Configure libraries for this circuit

Additional installation procedure for legacy version

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



8. Reboot ANSYS Designer.

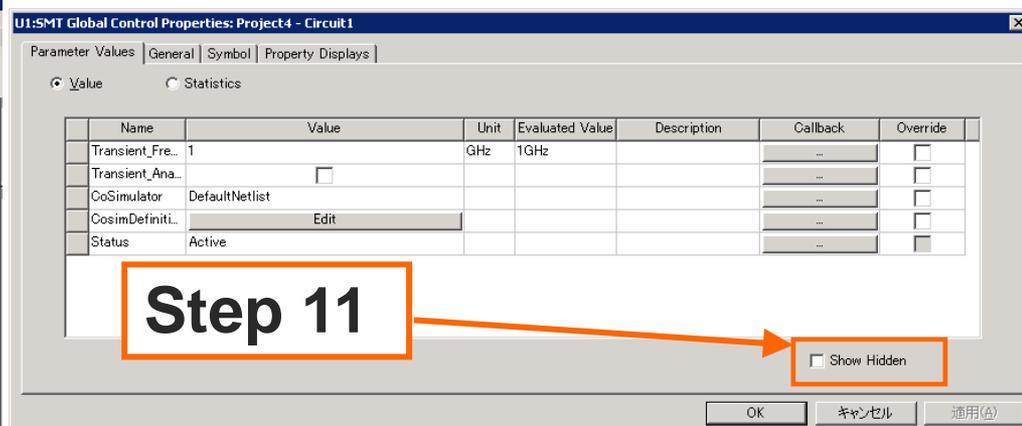
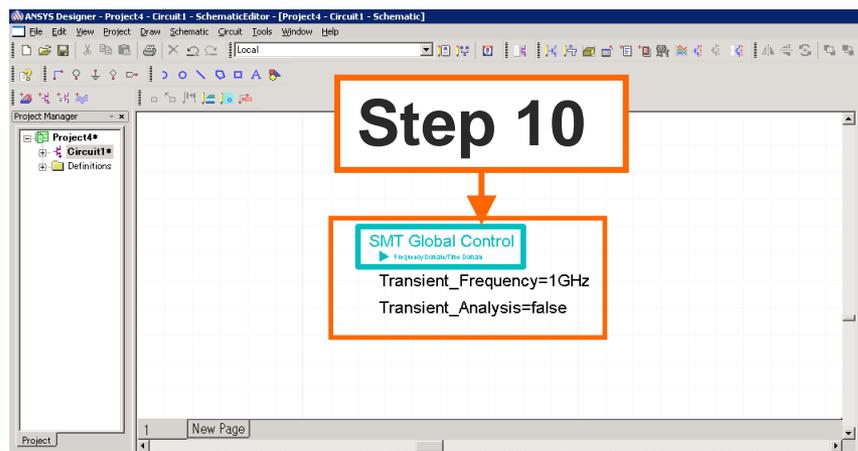
Do the following procedure with schematic window open.

9. Click the component tab, then select User Libraries -> Taiyo Yuden -> SMT Global Control and put it on the schematic.

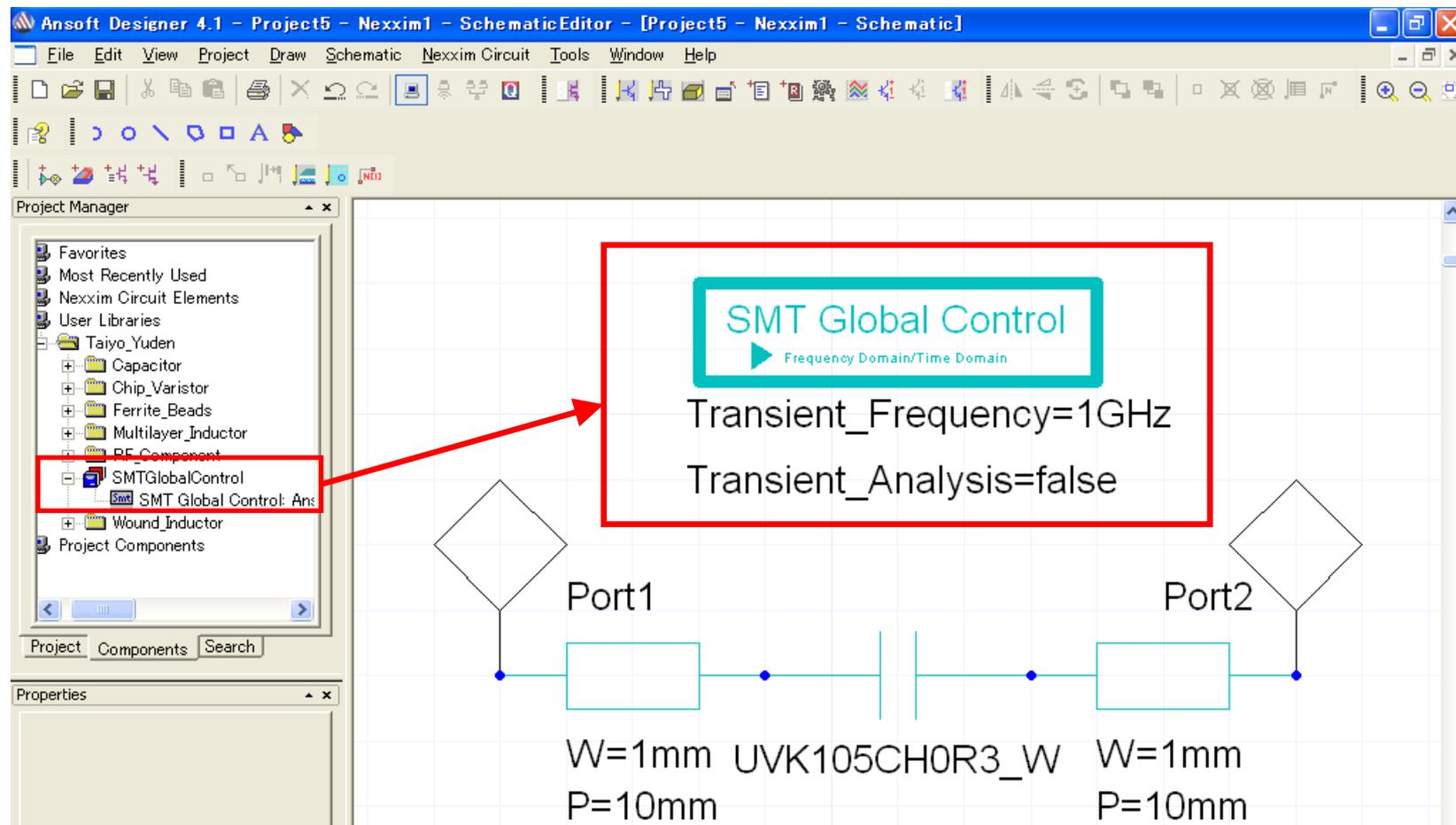
10. Double click the SMT Global Control to open the property window.

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