TAIYO YUDEN

TAIYO YUDEN Component Library for Cadence PSpice (Temperature/DC Bias Model)

- Installation manual -

Contents

- * How to install Component Library (P3)
- * How to use Component Library (P4-P5)
- * How to use Component Library on OrCAD (P6-P15)

How to install Component Library

Step 1. Unzip "PS_TY**.zip".

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



How to use Component Library

- **Step 1.** Describe the library in the netlist.
- Step 2. Add the ambient temperature after the described library. (In case of writing no parameter, 25C is set as the ambient temperature.)

📕 Sample.cir - メモ帳 — **D** ファイル(E) 編集(E) 書式(O) 表示(V) ヘルプ(H) 0 DC 0.3 AC 0.01 016KKT1ROM_PARAMS: Temperature=85 Step 1 sHz 3GHz Step 2 6KKT1ROM P.Lib options numdgt=9. .print AC VR(1) VI(1) .end 1行、1列 100% Windows (CRLF) UTF-8

netlist example

- *1 Refer to the PSpice manual for the description of the netlist.
- *2 The ambient temperature here only works for the described library, not for the whole circuit.

How to use Component Library

Step 3. Perform the simulation from the command line such as windows command prompt.

| _ | | × |
|--------|-------------|-----|
| | | ^ |
| sample | e.cir | |
| | | |
| | | |
| | | |
| | | |
| | | ~ |
| | — sample | - D |

Step 1. Launch PSpice Model Editor, then select Model Import Wizard.



Step 2. Select the library on the Model Import Wizard window, then go to next.



Step 3. Click finish button without associating the existing symbols and also click yes on the next dialog.

| Model Import Wizard : As | ssociate/Replace Symbol | × |
|--------------------------|---|-----|
| | Destination Symbol Library : C:¥Orcad_Users¥PSpice¥LSEPC2016KKT1R0M_P.olb You can do either of the following : (1) associate symbol for models without symbol, or (2) replace existing symbol for models. | |
| | Models with symbol Models without symbol Symbol : Model Name SEPCZUTERRITRUM | |
| | QUESTION(ORSCH-1014): Do you want to attach default QUESTION(ORSCH-1014): Do you want to attach default rectangular symbols to models without corresponding symbols in the destination symbol library? | |
| | | |
| View Model | | ヘルプ |

Step 4. Click OK on the final log window.

| Model Import Wizard : C:¥Orcad_Users¥PSpice¥LSEPC2016KKT1R0M_P.symwiz.log | × |
|--|---|
| INFO(ORSCH-1132): Log File Error File | |
| | |
| STATUS: PSpice Model Import Wizard for "Capture" (17.2.0.d001) | |
| STATUS: | |
| INFO: LIB driven flow | |
| INFO: Input File: C:¥Orcad_Users¥PSpice¥LSEPC2016KKT1R0M_P.lib | |
| INFO: Output File: C: ¥Orcad_Users ¥PSpice ¥LSEPC2016KKT1R0M_P.olb | |
| STATUS: | |
| STATUS: Identifying matching symbols automaticallystarted at Monday, November 01, 2021 12:07:42 | |
| STATUS: | |
| STATUS: | |
| STATUS: Completed identifying matching symbols automaticallyat Monday, November 01, 2021 12:07:4 | |
| STATUS: | |
| STATUS: 0 Error messages, 0 Warning messages | |
| | |
| | |
| | |
| < > | |
| | |
| ОК | |

Step 5. Verify that the symbol file(.olb) is generated to use on the OrCAD schematic.



Step 6. Launch OrCAD Capture, then right-click Library on the project tree and select Add File.

| 🚼 OrCA | D Capture | e – [C# | ¥Orcad_L | lsers¥F | Spice | ¥LSE | PC201 | 16KKT | 1R0M | l.opj] | | | | | | |
|----------|---|---|----------------------------------|-----------------------------------|--------------|----------|--------|-------|----------|-------------|------|---------|-------|--------|-------|------|
| 💽 File | Design | Edit | View | Tools | Plac | e S | 6I Ana | lysis | PSp | oice | Acce | ssories | Op | otions | Wir | ndow |
| | | - 1 | 4 0 | Ĉ | 9 | ¢ [| | | | | ~ | 0 | ٩ | 9 | ۹, | ۲ |
| SCHEN | MATIC1 | -bia | < ~ 📈 | • 🍫 | | V | R | A | ß | R | Ø | Ţv | Ô | Jī | Ŵ | Ъ |
| * | 5 | - | | 4 | 8- | - | • | ě | 1 | > | | | | | 1 | |
| | 2016 | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | А | nalog | or Mis | ed A/ | ′D |
| E Filler | e ^E , Design Ise Output: Output: PSpice | Hierarch Resou ppc2011 prary s Resou | ny irces 6kkt1r0m arc S | ndsn dd File ave <u>A</u> s | 5 | | | | | | | | | | | |

Step 7. Select the symbol file(.olb) generated at step 5 to register symbol.

| 🔀 Add File to Proje | ect Folder – Library | | | | × |
|----------------------|---------------------------------------|--------------------------------|--|------------------------------|---------------|
| ファイルの場所(<u>l</u>): | PSpice | | ~ G | ø 🖻 📴 | |
| クイック アクセス | 名前 LSEPC2016KKT 【【】LSEPC2016KKT | 1R0M-PSpiceFiles 1R0M_P.OLB | 更新日時 2021/11/01 12:20 2021/11/01 12:07 | 種類 ファイル フォルダー OLB ファイル | サイズ 7 KB |
| テスクトック | | | | | |
| マクトワーク | | | | | |
| | ファイル名(<u>N</u>): | 1 | | ~ | 開<(<u>O)</u> |
| | ファイルの種類(工): | Capture Libraries (*.olb |) | × : | キャンセル |

Step 8. Select PSpice > Edit Simulation Profile from the menu bar.

| OrCAD Capture - [C:¥Orcad_Users¥PSpice¥LSEPC2016KKT] | 1R0M.opj] | | | | |
|---|---------------------------------------|-----|--|--|--|
| 🔁 File Design Edit View Tools Place SIAnalysis | PSpice Accessories Options Win | dow | | | |
| | 📷 New Simulation Profile | 6 | | | |
| | 🖾 <u>E</u> dit Simulation Profile | Г | | | |
| SCHEMATIC1-bias 🗸 ൽ 👰 💽 礘 🔏 🚑 | Run F11 | b. | | | |
| | ₩ <u>V</u> iew Simulation Results F12 | Γ. | | | |
| 🗎 🕷 🗤 🚧 🖛 🖬 📲 📑 👘 🕲 | Vie <u>w</u> Output File | 17 | | | |
| | Make <u>A</u> ctive | | | | |
| [38] LSEPC2016 | Simulate Selected Profile(s) | | | | |
| | Advanced Analysis | | | | |
| 💼 File 💐 Hierarchy | <u>O</u> reate Netlist | | | | |
| 🖃 💼 Design Resources | View Netlist | | | | |
| 😥 🎬 Isepc2016kkt1r0m.dsn | Marker List | | | | |
| | | | | | |
| G:¥Cadence¥SPB_17.2¥tools¥capture¥library | y¥pspice¥analogolb | | | | |
| Ci#Cadence#SPB_17.2#tools#capture#library Ci#Cadence#SPB_17.2#tools#capture#library Ci#Cadence#SPB_12.2#tools#capture#library | y¥pspice¥breakoutolb | | | | |
| C+Cadence+SPB_17.2+tools+capture+library | y#pspice#source.old | | | | |
| C¥Cadence¥SPB 17.2¥tools¥capture¥library | v¥pspice¥specialolb | | | | |
| | , | | | | |
| Outputs | | | | | |
| PSpice Resources | | | | | |
| | | | | | |
| | | | | | |
| , , | | | | | |

- **Step 9.** Select Configuration Files tab.
- **Step 10.** Select "Library" on the Category pane.
- **Step 11.** Select the library(.lib) at the Filename section.
- Step 12. Click Add to Design, then OK to register library(.lib).



Step 13. Open the schematic and select the Part icon.

Step 14. Select the library on the Libraries pane on the Place Part window.

Step 15. Double-click the component on the Part List pane to put on the schematic.

Step 16. Perform the simulation after completing the schematic.

| 👪 OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)] | – 🗆 🗙 |
|---|-----------------------|
| 👔 File Design Edit View Tools Place SIAnalysis PSpice Accessories Options Window Help 🛛 cāden (| ce [®] - = × |
| 📔 🗁 🔚 🚔 🐒 🗊 💼 🥱 🥐 LSEPC2016KKT1R0M 🗸 🍳 🛛 🗸 🗸 | M - 4 🕨 |
| Q Q Q ∞ UI \$ D D D D D D D Q 4 Q | |
| SCHEMATIC1-bias -> ळ 🖓 💽 礘 🔏 🔏 🔏 🔗 🖤 🕐 🗊 📜 🎯 | |
| · · · · · · · · · · · · · · · · · · · | 🍧 Step 13 |
| Place Part + * × Part Part | k 🚯 L |
| | N 5 5 |
| Part List: | 🍠 🚢 ጌ |
| 1 U1 2 2 Step 15 | 🕂 🖌 🎬 |
| LSEPC2016KKT1R0M | ≑ 喝 👄 |
| 0.1Asc | |
| | To 🖧 🚅 |
| | S & |
| | · 많 신동 💩 |
| © SOURCE SOURCSTM | ●] } |

* Refer to page 5 of this manual for setting the ambient temperature of the component.