Symica

Symbol Wizard
User Guide

Product version 1.xx
March 2011

Table of Contents

Introduction

The Symica PDK generator allows you to convert model descriptions from spice model files into a library.

Conversion is accomplished using the Symica converting wizard providing fast processing and complete results.

Conversion achieved in two steps:

- Parsing models from spice model files
- Matching symbols for parsed models

Symbol wizard

1. Select *File->Import->Library…* to open Import Library menu window (fig. 1):

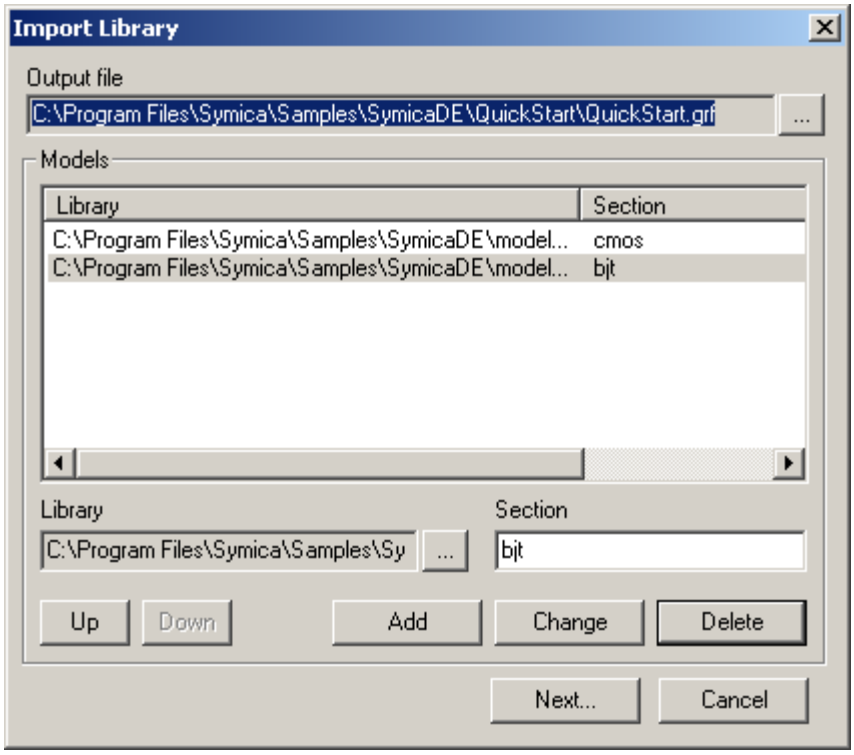

Fig. 1.

Output File – specify full path to the library where elements from model files will be added. (If library file doesn't exist you must create it)
Models field shows path to model files and selected section:
Library – shows full path to the spice model file.
Section – selected section name in a spice model file which contains one or more models (only models from selected sections will be converted).

- If there are several sections in file then each section should be added separately.

- If there are no sections in model file you must create it manually (see attachment "how to create section").

Click ... button and select model file. Then in Section field enter section name and click *Add*. File path and section appear in the *Models* field.
Add - button adds current library and section to the list.
Change or **Delete** - edit selected model file or section.
Up or **Down** - change a parse priority.

2. Click **Next…** a new window appears with extracted cells (fig. 2).

Fig. 2.

Cell column – list of parsed model/subcircuit entities (cells).
Cell Type – if model is described using keyword *model* – its cell type is *basic cell*. If model is described using key word *subckt* or *inline subckt* – its cell type is *subsurcuit*.
Symbol Type – symbol type associated with the cell. If no type is associated — the cell's symbol defaults to a Blackbox.
Create Cell – if checkbox is checked this cell will be created with symbol view according to symbol type. When you change value from "unchecked" to "checked" the *Symbol Properties* window appears (fig. 4)).

The black field to the right displays a symbol figure of the selected cell (fig. 3). When you clear "Create Cell" checkbox, figure refreshes (to the empty field) after switching between selected cells.

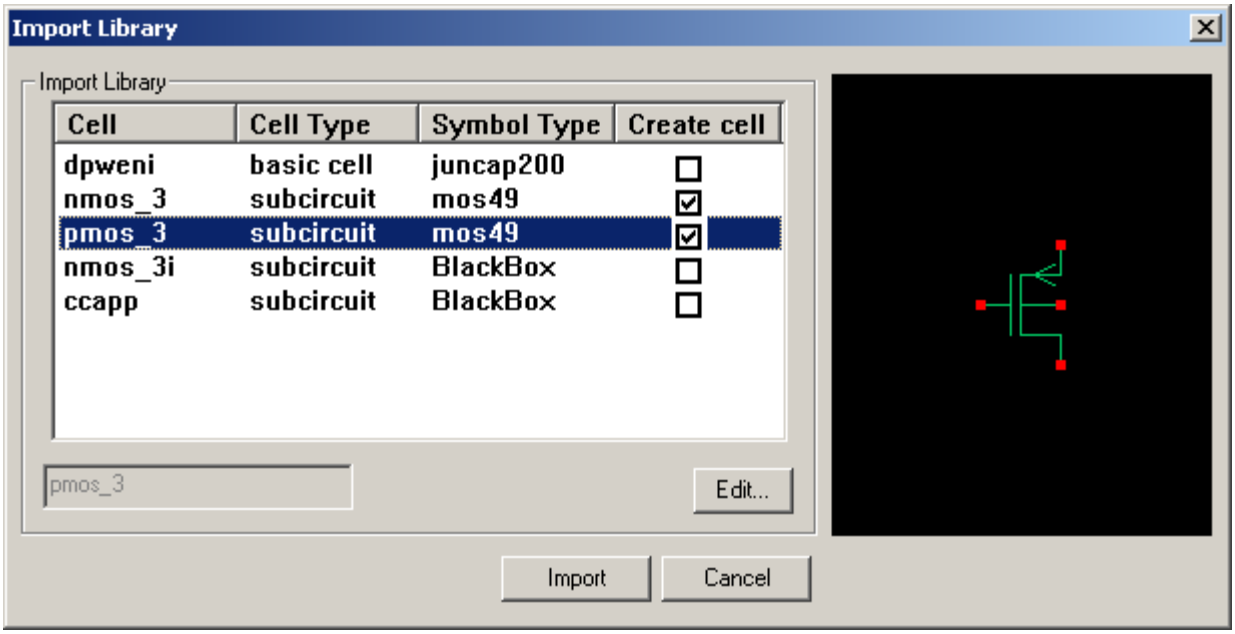

Fig. 3.

Double click on any row or select row and click **Edit…** button to open the *Symbol Properties menu*.

Symbol View Properties (fig. 4)

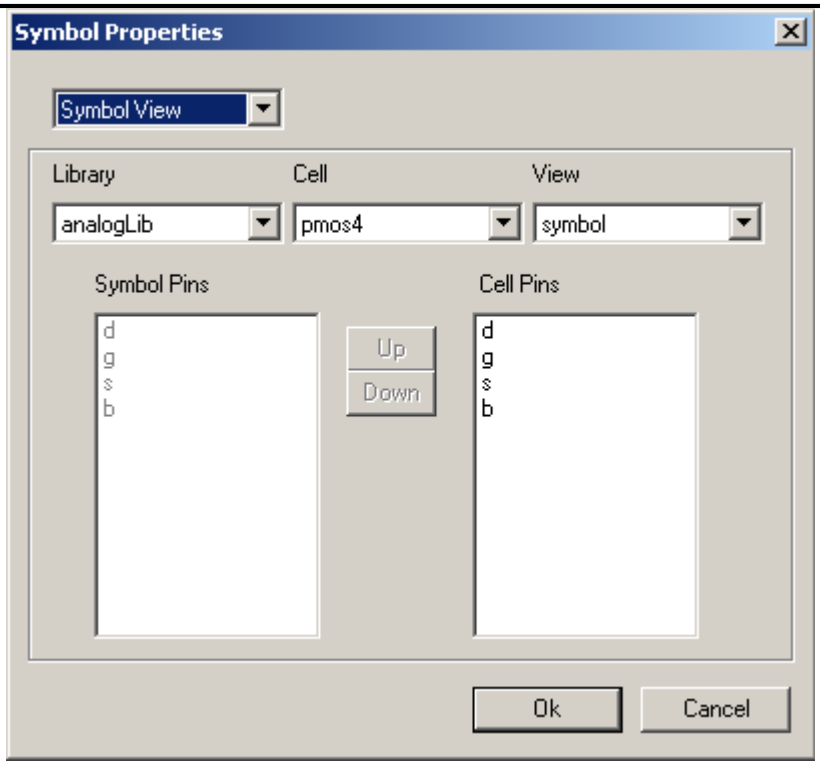

Fig. 4.

In boxes **Library**, **Cell** and **View** you may choose a symbol. While symbol is selected, in the *Symbol Pins* field, pin names are shown according to their order in symbol view. In *Cell Pins* field, pin names are shown according to theirs order in model file. You may match symbol pins and model pins in any order by selecting pin names and clicking buttons **Up** and **Down**. Click **Ok** button and *Create cell* value for the current cell changes to "*Yes*".

BlackBox View (fig. 5)

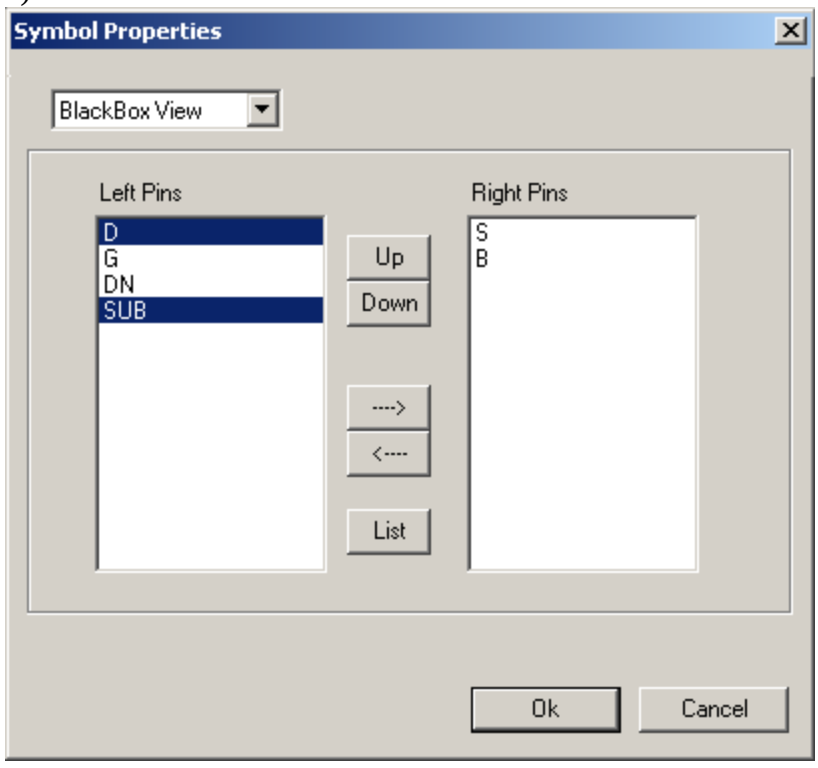

Fig. 5.

Generated symbol for subcircuit will be created as a black box (rectangle with pins located at the left and right sides). Pin location can be changed with buttons → or ← and **Up** or **Down**. By clicking **List** button the *Pin Attributes* dialog box appears (fig. 6).

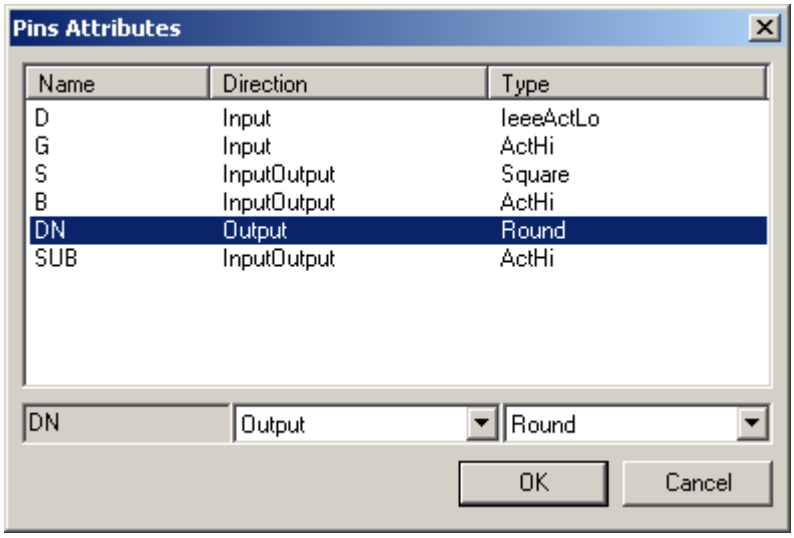

Fig. 6.

Press **Ok**. Now in Import Library window, the *Create cell* checkbox for the current cell is checked and its symbol figure changes are shown.

Click **Import** button on the **Import Library** window and all cells with checked *Create cell* value will be generated and added to the selected library.

How to create section

Examples of section declaration (180nm_bulk.txt):

```
simulator lang=local

section bjt

model bjt1 bjt type=pnp

endsection bjt

simulator lang=spice

.lib cmos
.model N NMOS
+Level = 49 Lint = 4.e-08 Tox = 4.e-09
+Vth0 = 0.3999 Rdsw = 250 Tref=27.0 version =3.1
............
............

.endl
```