Symica

SKILL translator
User Guide

Product version 2.xx
October 2012

Table of Contents

Introduction

Symica's importing and exporting tool provides you with the ability to transfer libraries from Cadence® platforms to Symica DE and the converse.

Symica DE

SKILL Import

In the Symica DE select menu File->Import->SKILL... (fig.1.1)

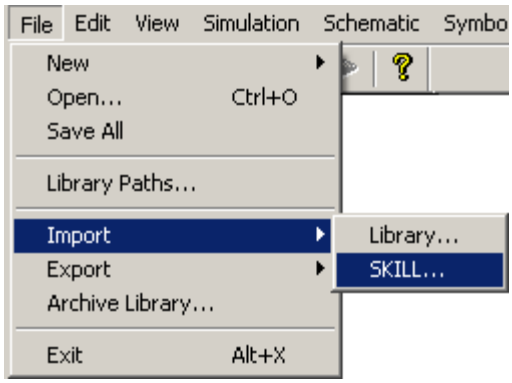

Fig.1.1.

The *Import SKILL* dialog box will appear (fig.1.2).

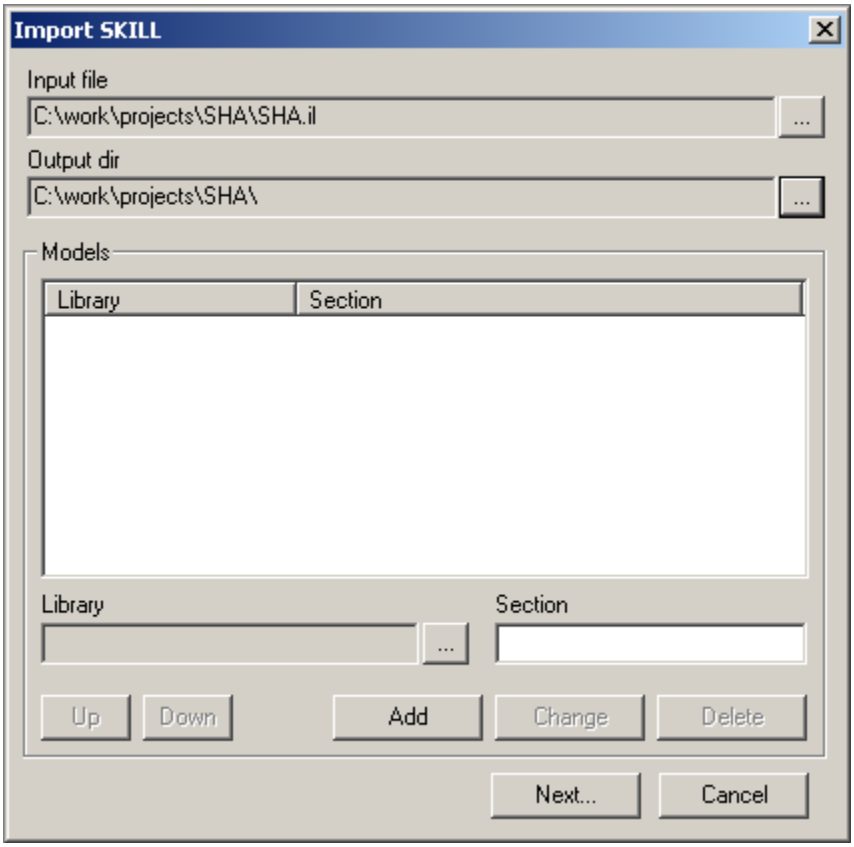

Fig.1.2.

Input File — full path to the input SKILL script file.

Output dir — full path to the directory where the new library will be placed. If the directory does not exist it will be created automatically. If the library already exists in library manager, a new library won't be created and all cells from this library will be added to the existing.

In the **Models** field you can specify SPICE model files. This field is used to simplify the translation process in the next step, when the user must indicate the type of each cell.

After clicking **Next...** the translation program starts parsing the SKILL script and creates a temporary database. A new dialog window appears (fig.1.3)

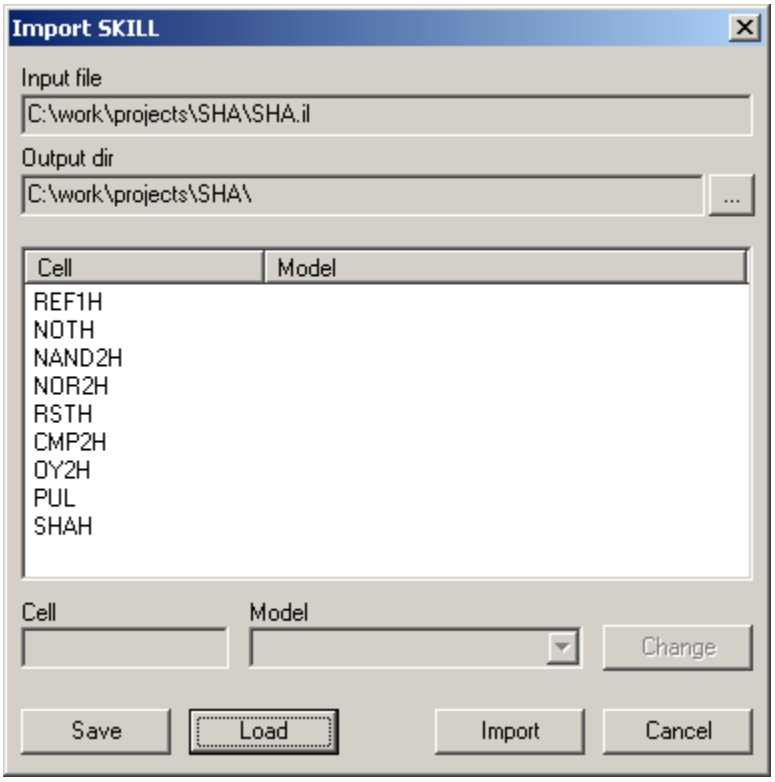

Fig.1.3.

In this window you have to specify the Cell type (Model) for every cell in the SKILL script file. Select cell and from the dropdown list below select model. For most custom cells (represented as SPICE subcircuits) the model has to be assigned to 'Common Cell'. For basic cells (transistors, resistors, capacitors etc.) the model is assigned from the supported model list (fig.1.4).

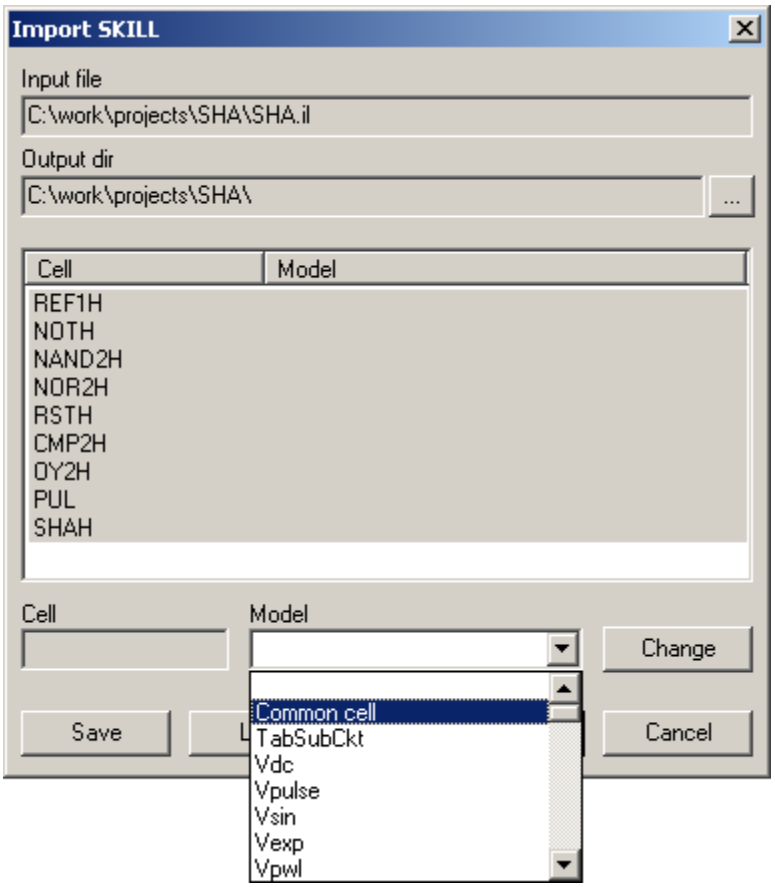

Fig.1.4.

Note: nowadays, many basic cells from foundries are represented as subcircuits. If you have specified SPICE model files in the previous dialog box (fig. 1.5), select the appropriate “subckt” name (fig.1.6) and all subckt parameters will be configured automatically. SPICE model names or subckt names are usually the same as the cell names in the library and are selected automatically (fig.1.7). Otherwise, select “Common cell” as type and adjust subckt parameters manually afterwards.

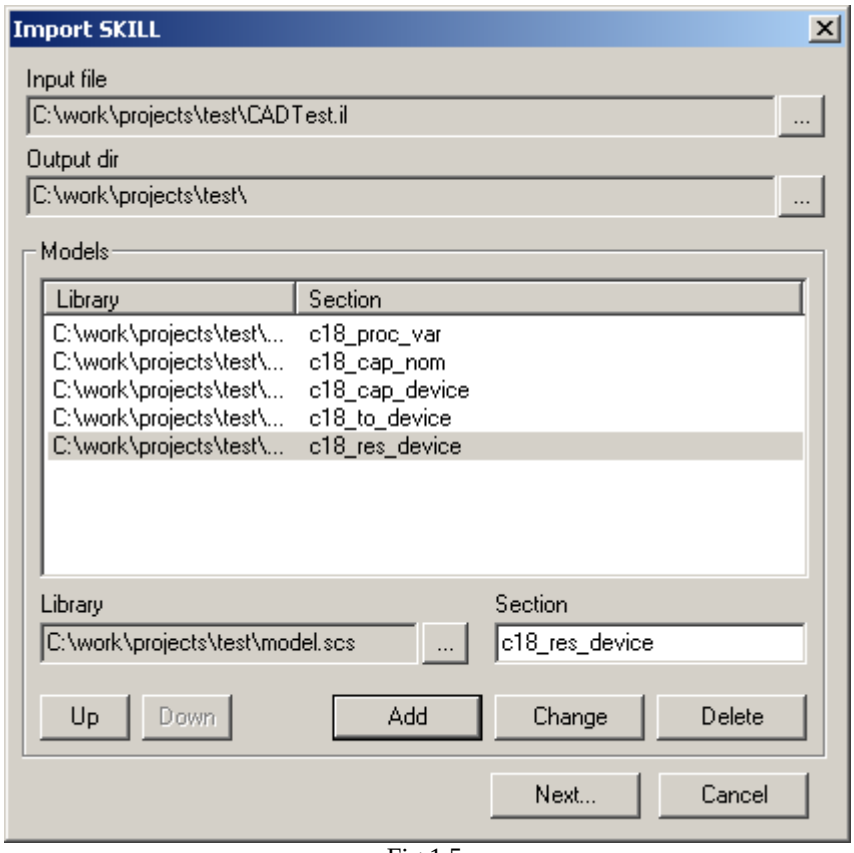

Fig.1.5.

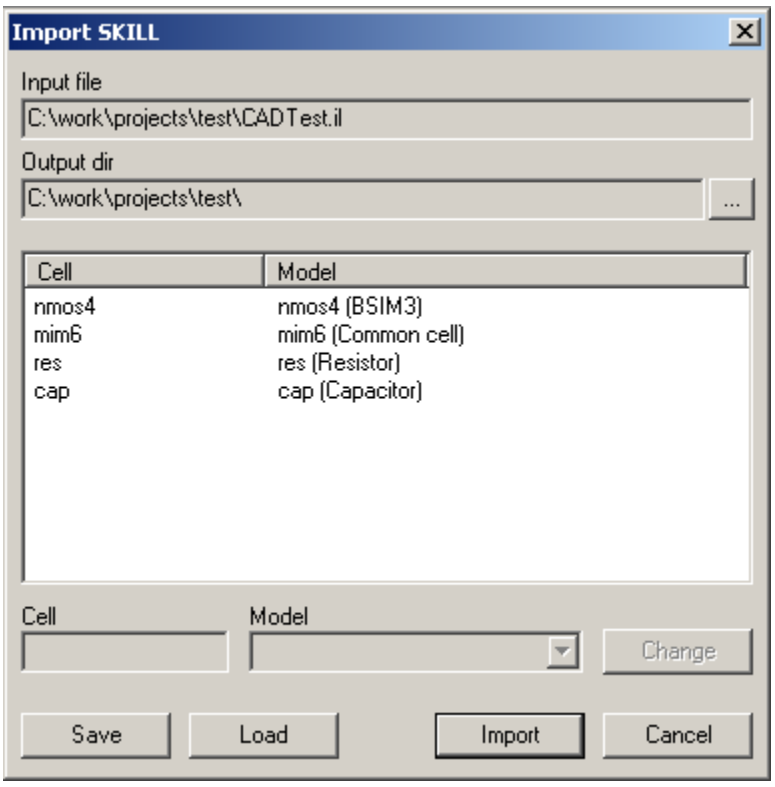

Fig. 1.6.

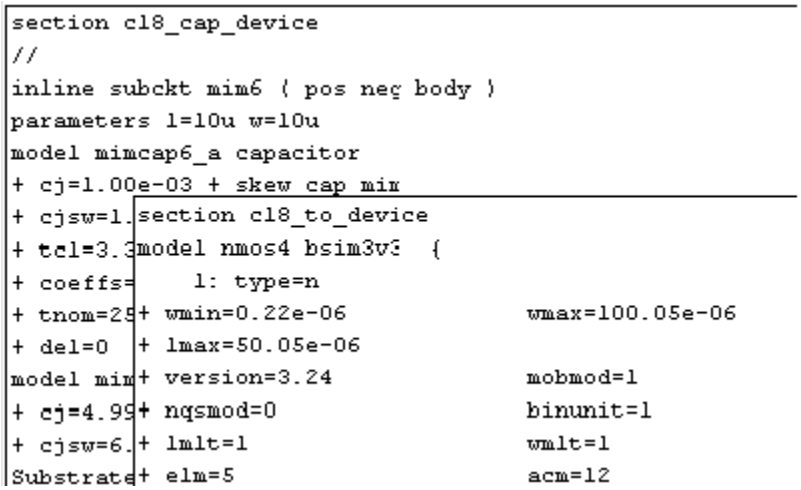

Fig. 1.7.

For a large library it may be convenient to save the current configuration, so that it can be reused for future library translations. After clicking the *Import* button, all cells for which models are specified will be translated.

Note:

- all voltage/current sources are translated into *isource* or *vsource* from the *analogLib* library;
- all content in schematics is scaled and positioned to the upper left corner of screen, all symbols are snapped to the defined grid;
- Symica currently does not support callbacks (future release).

SKILL Export

To start creating SKILL script in the Symica DE click on menu File->Export->SKILL... (fig.1.8.)

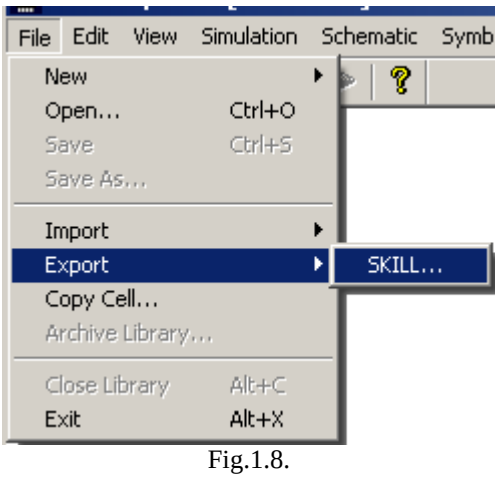

Fig.1.8.

An *Export SKILL* menu (fig.1.9):

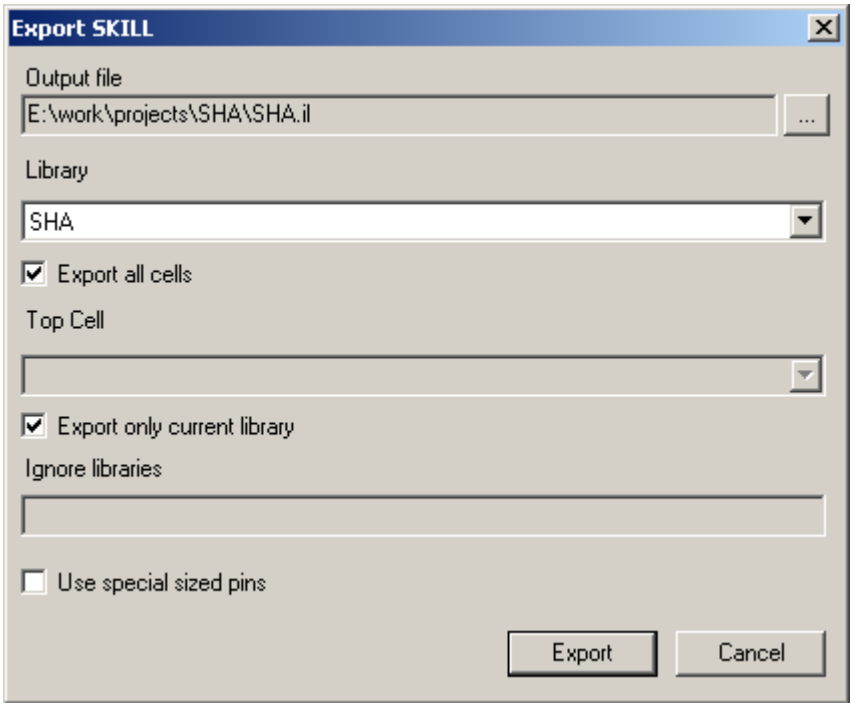

Fig.1.9.

Output File — full path to the SKILL script to be placed. Directory should be existed or an error message will appear. If the library contains either *verilog, verilogA* or spicenl cellview types, the corresponding source files will also be created in the destination directory with the following names:
<libraryname>_<cellname>_<viewname>_<filename>, where

libraryname — name of the library;
cellname — name of the cell;
viewname — name of the verilog view;
filename — name of the verilog or spicenl file (usually verilog.v, veriloga.va or spicenl.lib).

Library — select library to export from dropdown list (fig.1.10):

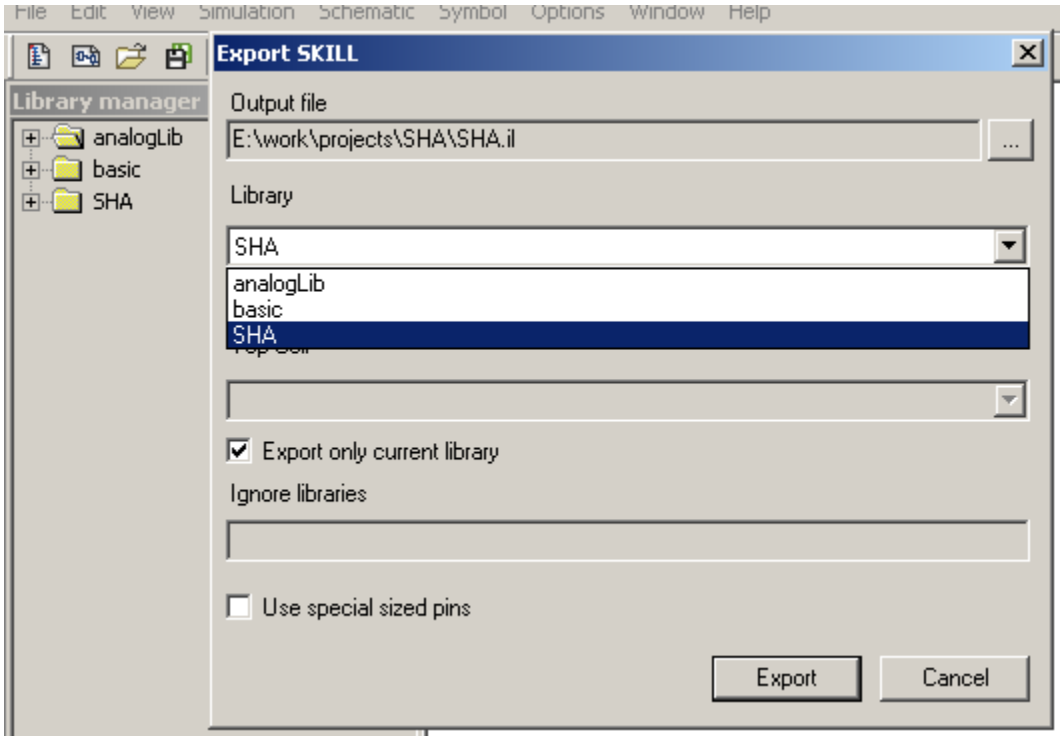

Fig.1.10.

Export All Cells — check this option to translate the entire library. If this option is unchecked a **Top Cell** filed becomes editable.

Top Cell — from dropdown list select top cell that will be translated hierarchically (fig 1.11). This means that all cells which it contains will also be added to a script. Empty string is the same as if **Export All Cells** option is checked.

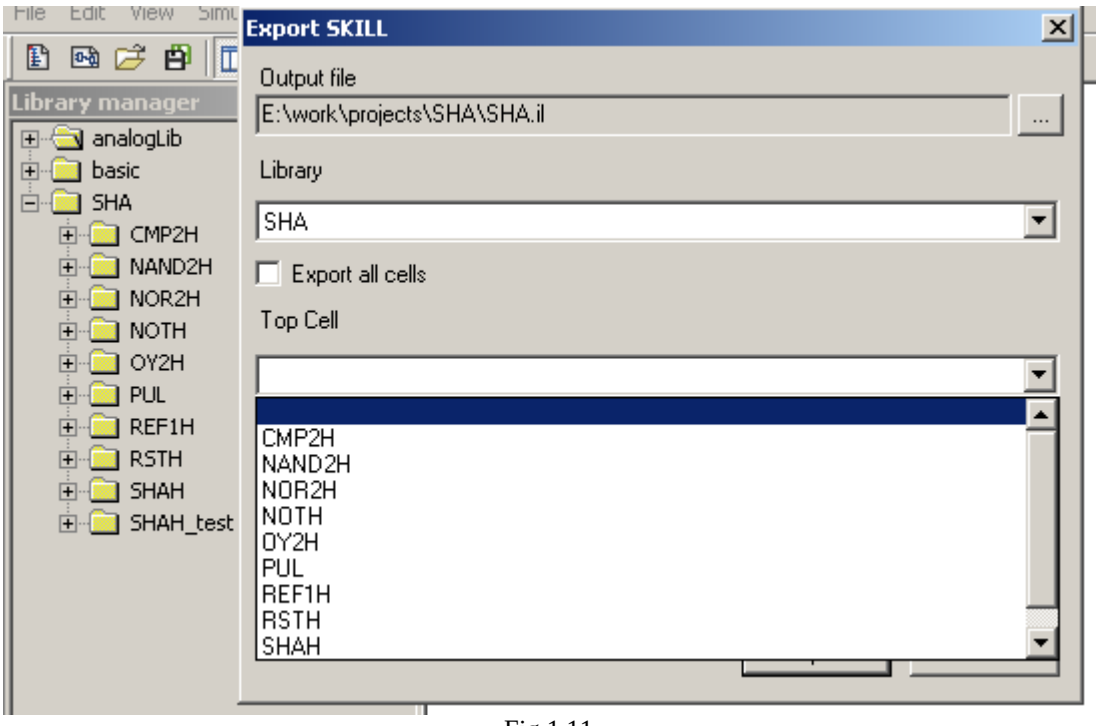

Fig 1.11

A SKILL description will be created for the following view types:

- *schematic* view;
- *symbol* view;
- *verilog* (or *functional*) view;
- *verilogA* view;
- *spicenl* view

The cell properties (part of Cadence CDF options) of each cell will also be translated and added to the script. Because of *spicenl* view has not any analog in Cadence library manager it will be translated as text view. Other view types such as “config” and “layout” won't be translated and added to the script; they should be created with the Cadence Library Manager.

Export Only Current Library — if this option is checked only the library defined in the Input File field will be translated. It usually means that all other necessary libraries already exist in the Cadence data base and don`t need to be converted. If this option is unchecked all cells from other libraries that are used in the current library will be also added to the script.

Ignore Libraries — list of the libraries which cells won't be added to the script (in fig 1.12 library *basic* is ignored and library *analogLib* which cells are used in SHA will be translated).

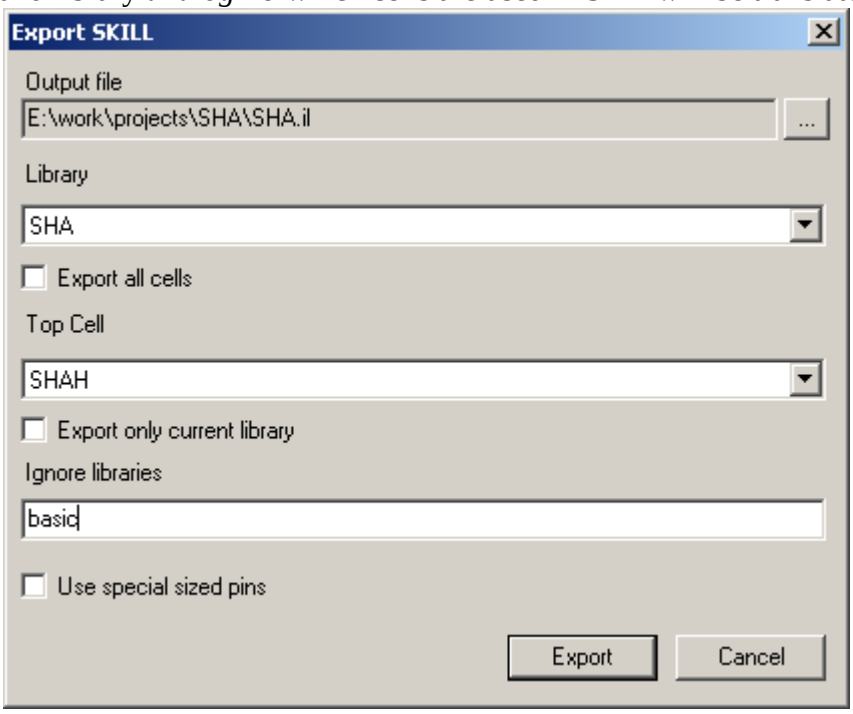

Fig 1.12

Use Special Sized Pins — this option will add a special pin library named «ExportPinLibrary» to a script. It contains three types of pins (input pin, output pin and inout pin) that geometrically are smaller than standard pins in basic library.

Click *Export* button to run the SKILL script creation.
If there are some pins or nets that have same names in different cases a warning window will appear (fig 1.13)

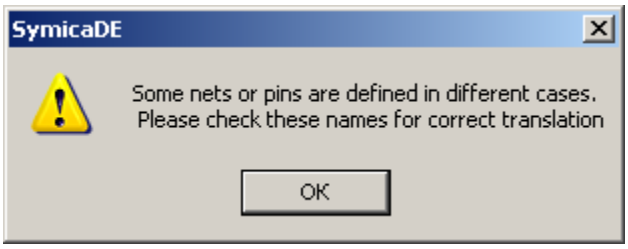

Fig 1.13

After clicking Ok button a window with list of these pins or nets will appear (fig 1.14)

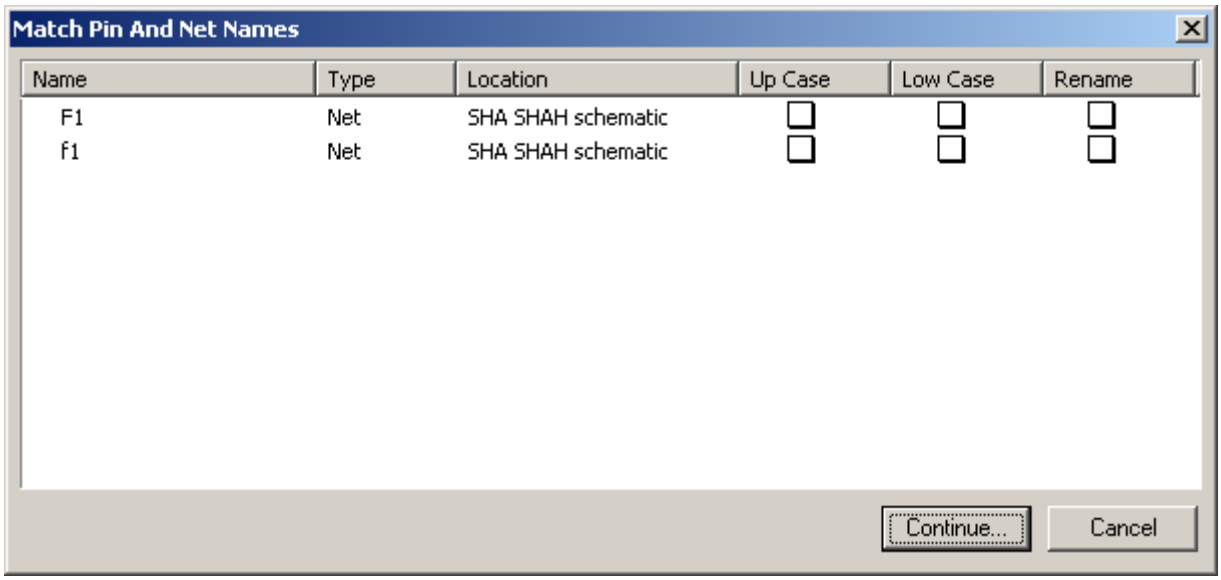

Fig 1.14

In this table you can use checkboxes to rewrite selected name to lower case, upper case or rename it adding «net_» or «pin_» to the beginning of the name (fig 1.15 and fig 1.16).

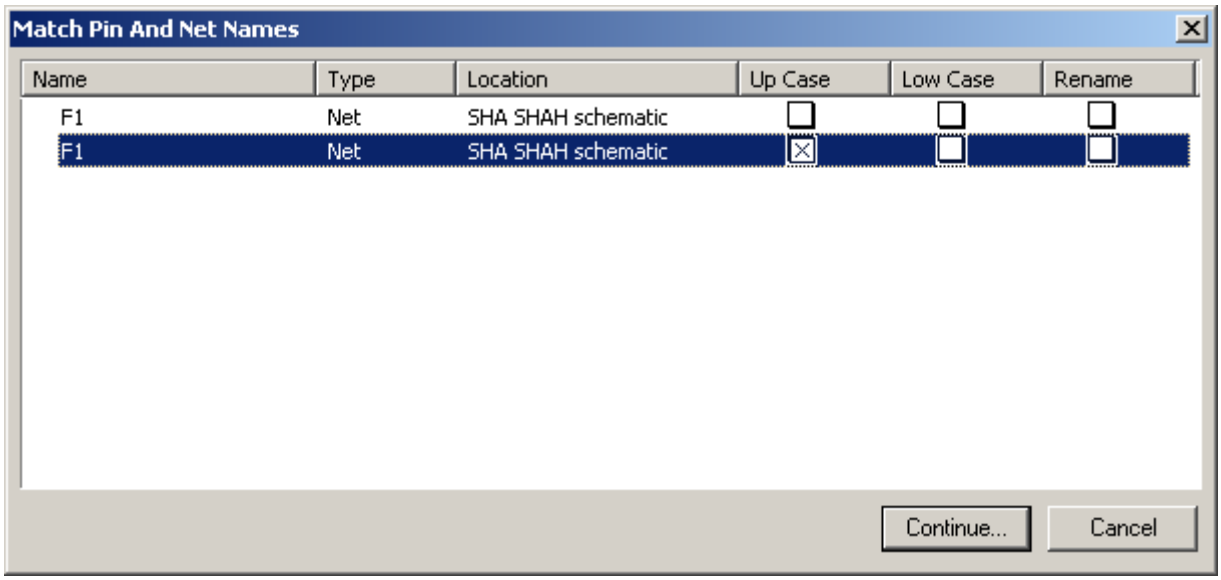

Fig 1.15

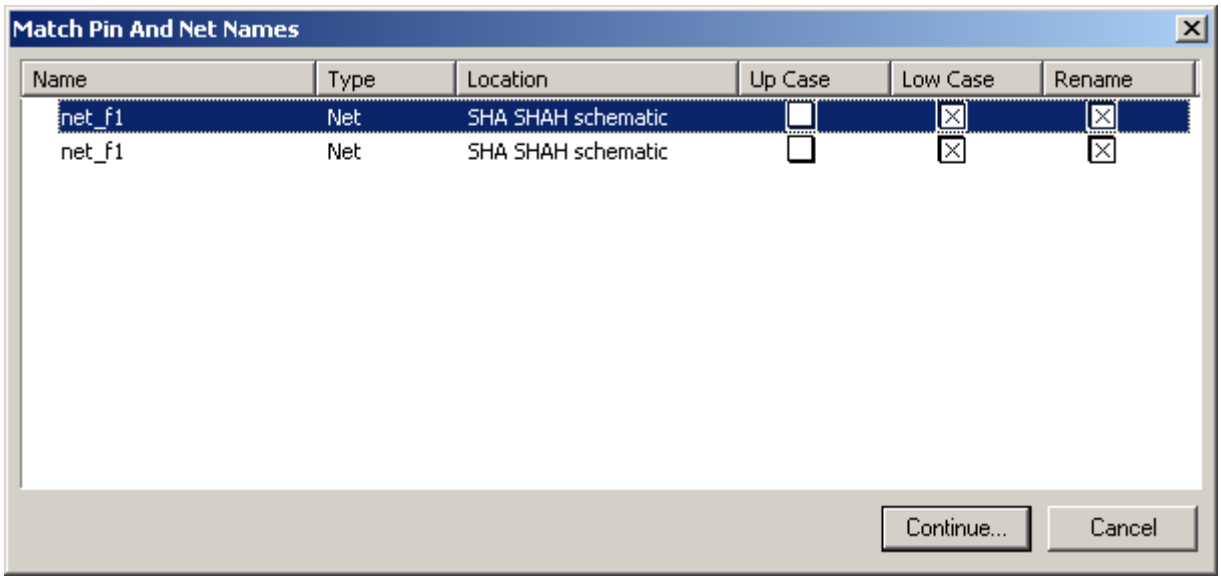

Fig 1.16

The generated SKILL script together with the supplementary generated files of the sample project SHA are shown in fig.1.17.

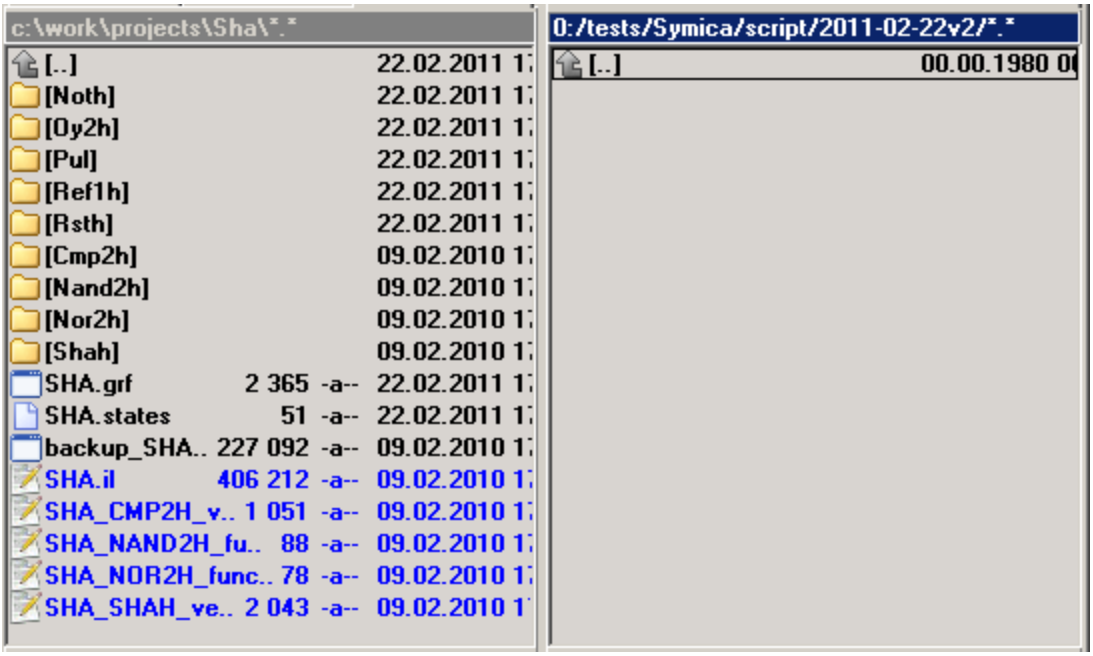

Fig.1.17.

Notes for script creation procedure:

- all voltage/current sources are translated into *isource* or *vsource* from the *analogLib* library;
- the grid length for all symbols and schematics is 0.065 points per inch (Virtuoso® Schematic Editor default); in case of using another grid setting, please change the grid step in Symica DE manually.

After creating the script, the script and all supplementary files can be moved to the Cadence design platform. It is recommended to copy them to a new directory.

Cadence platform

Verified with versions:
- Cadence v.5.0.33
- Cadence v.6.10
- SKILL v.06.30

Symica Integration Tools

You can load 'Import SKILL script' or 'Export SKILL script' using the Symica Integration Tools dialog box. Script files are located in <Symica_install_dir>\share\tools directory under OS Windows and <Symica_install_dir>\share\symica\tools directory under OS Linux. File «SymTools_cds5.il» is used for Cadence version 5.x and «SymTools_cds6.il» file is used for Cadence version 6.x. These scripts can be loaded into Cadence environment in two ways:

1. By loading from Cadence CIW (fig.2.1).
2. By loading from *.cdsinit* file (fig.2.2).

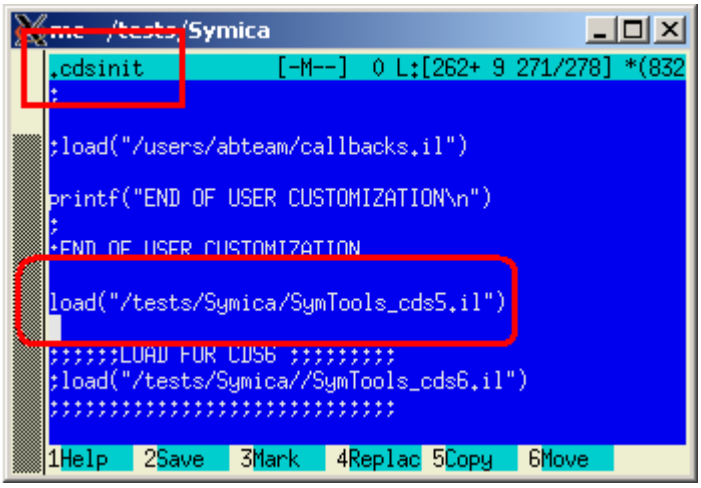

Fig.2.1.

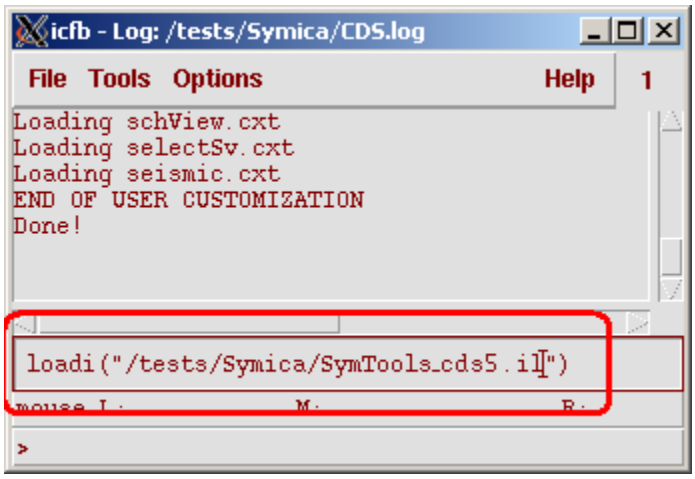

Fig.2.2.

SKILL Import

After CADTools.il is loaded, run the command '**SymicaTools**' in Cadence CIW command prompt (fig.2.3) and in the pop-up dialog box click the *"Import Script"* button (fig.2.4)

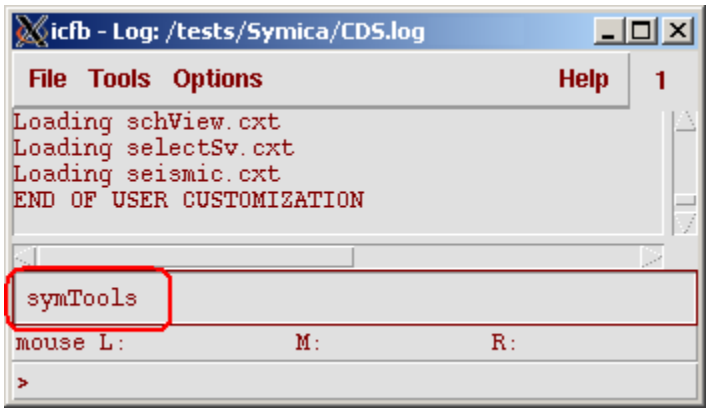

Fig.2.3.

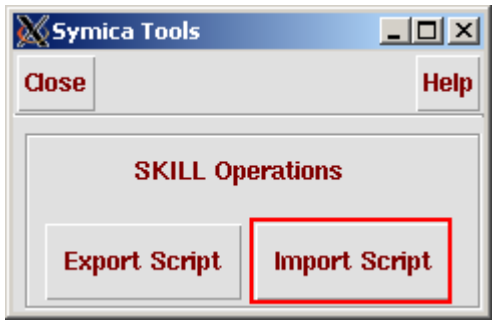

Fig.2.4.

In SKILL Import dialog box specify two fields (fig. 2.5):
SKILL Script Full Path — full path to the SKILL script containing information about the library being converted. It can be typed manually or selected by clicking the "*Browse*" button (fig.2.6)
Verilog Dir Path — path to the verilog files to be converted. This field is taken from path to the SKILL script (fig.2.7) by default.

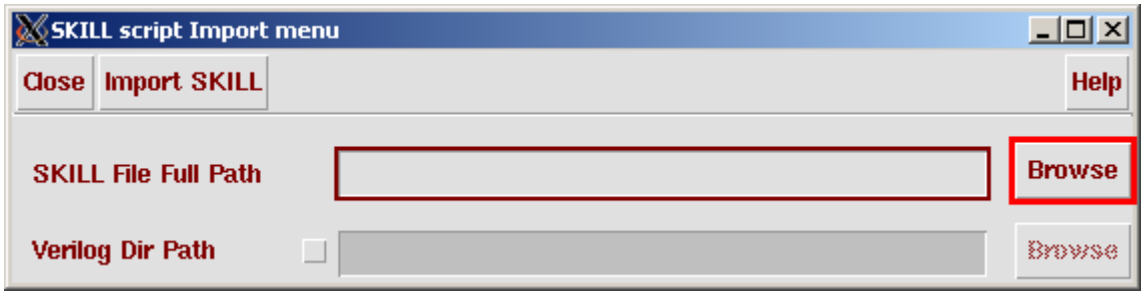

Fig.2.5.

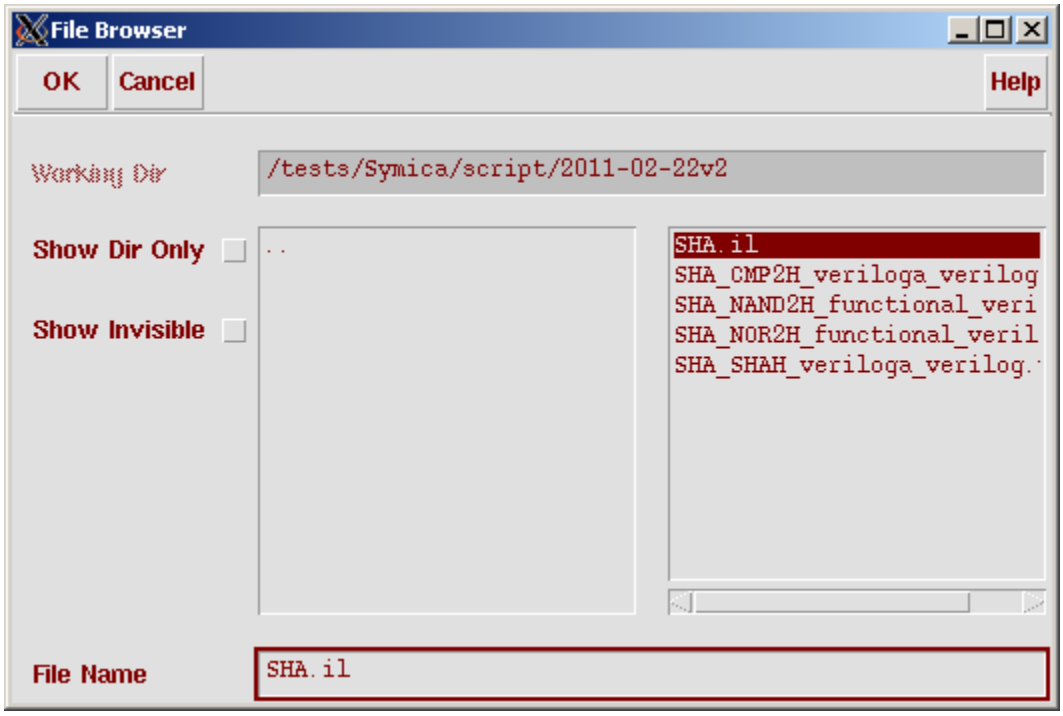

Fig.2.6.

When all fields are specified (fig.2.7) click the "*Import SKILL*" button.

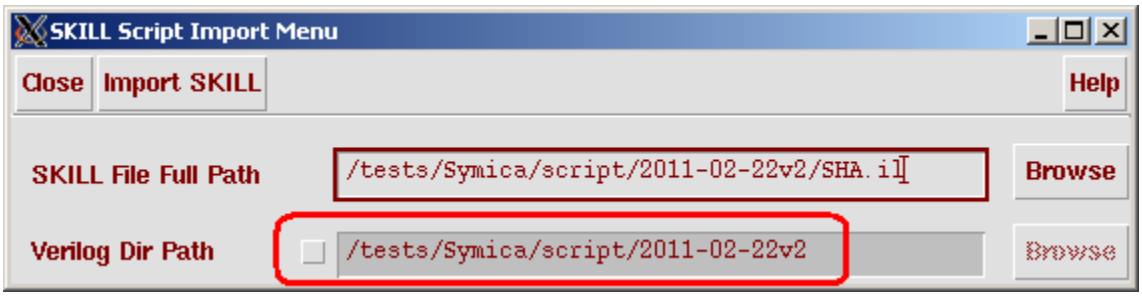

Fig.2.7.

Warnings and errors that arise during import procedure will be shown in CIW. The new library will appear in the Cadence Library Manager (fig.2.8).

Fig.2.8.

Usually there is no necessity to convert the FAB's PDKs as they were already initially created in Cadence.

For most custom libraries all cellviews will be created exactly as they are represented in Symica. There are some specifics regarding the conversion of cell properties to the Cadence CDF specification:

– only three fields of cell properties will be transfered: *name*, *prompt*, *defvalue;*
– all *namePreffix* values for all cells are set to “X” and all *componentName* values are set to *model name;*
– Symica does not support callbacks (future release).

SKILL Export

Run '**SymicaTools**' from the CIW command prompt (fig.2.9) and in the pop-up dialog click the *Export Script* button (fig.2.10)

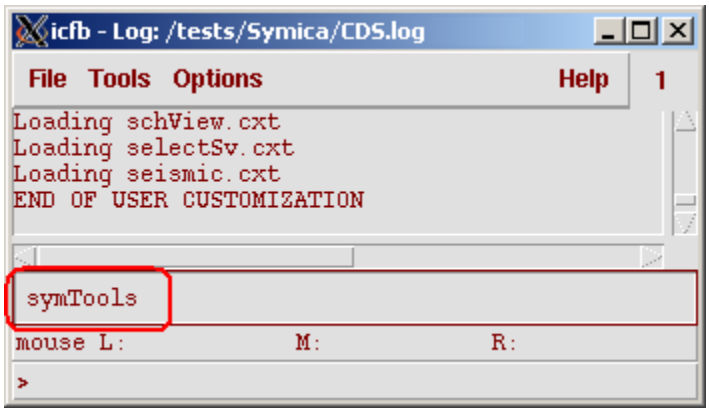

Fig.2.9.

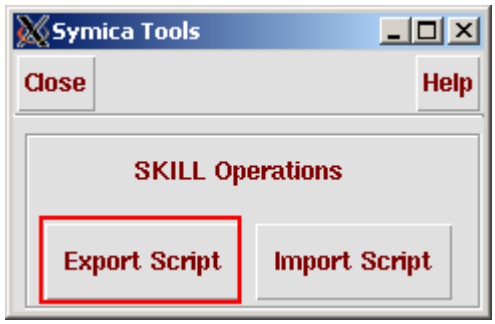

Fig.2.10.

The following dialog appears (fig.2.11).

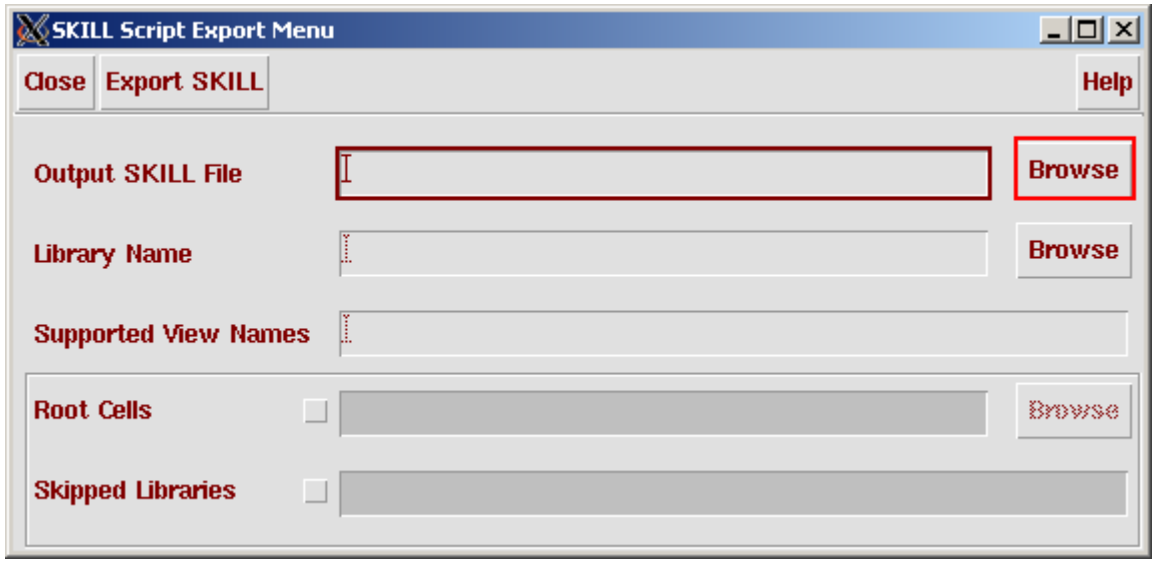

Fig.1.11.

Output SKILL File – full path to the SKILL script file. In the file browser dialog select the file name of target script (fig.2.12).

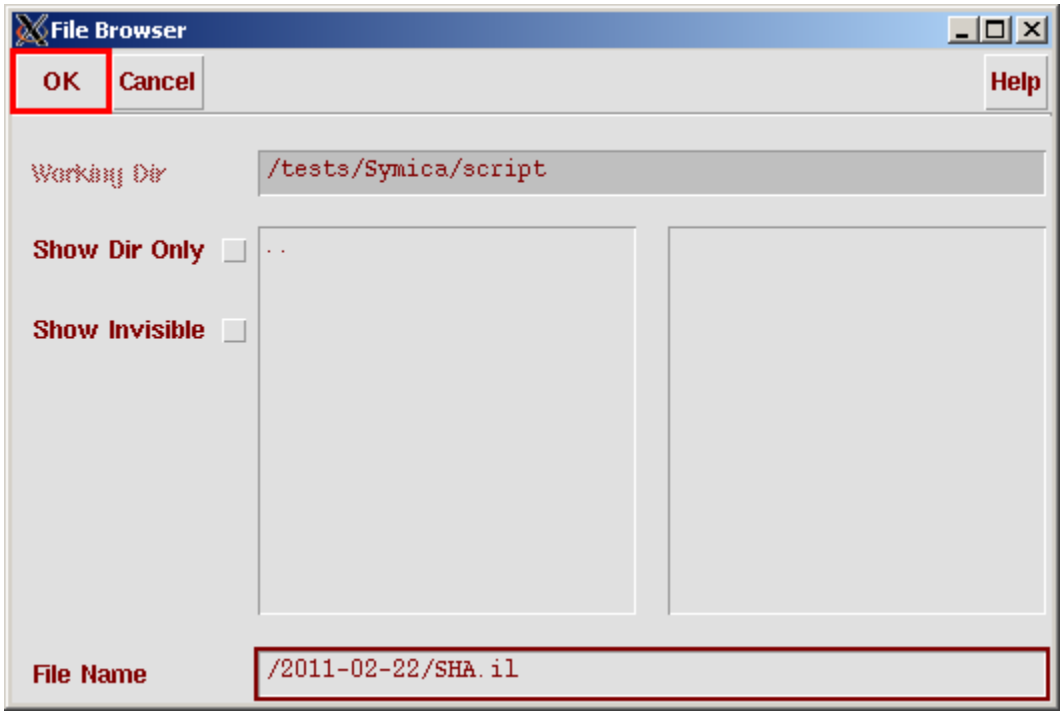

Fig.2.12.

Library Name – name of the library to translate in the Cadence data base. It may be typed in manually or selected from the Cadence Library manager by clicking the *Browse* button (fig.2.13).

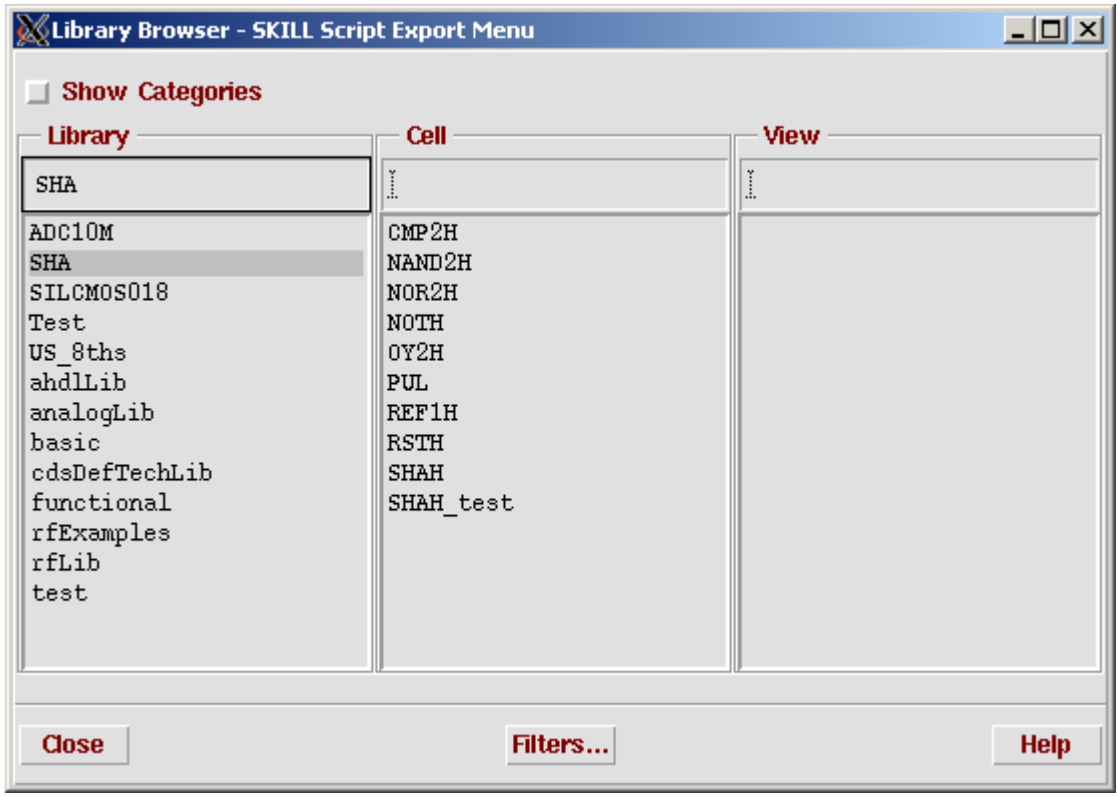

Fig.2.13.

Supported View Names – list of supported view names separated by spaces. The field is filled automatically after the library or root cell has been selected (fig.2.14).

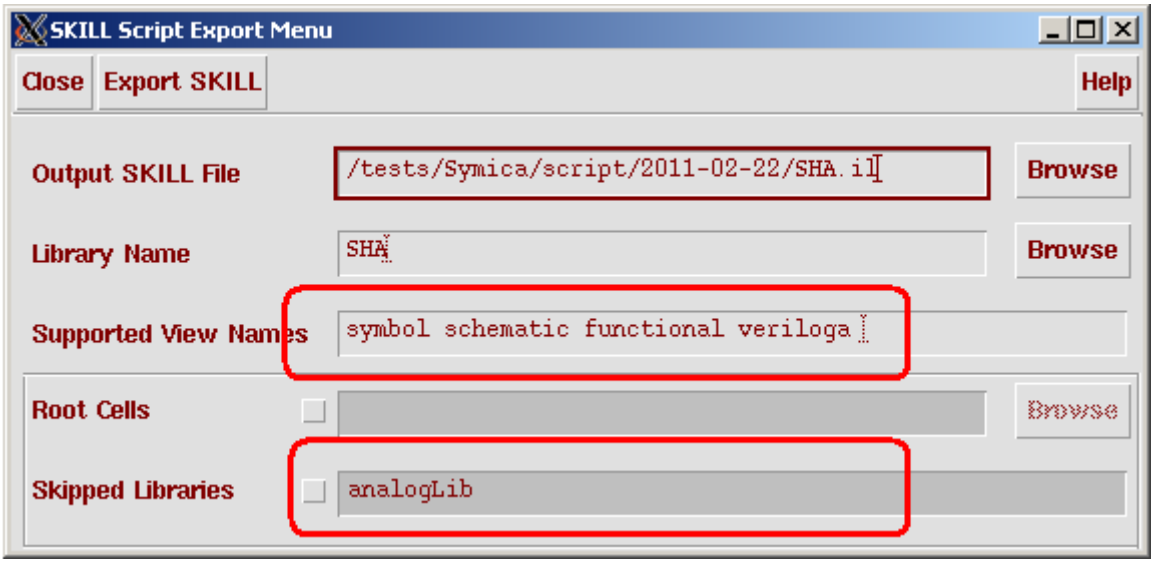

Fig.2.14.

Only specified cellviews will be exported to the script. You can add more cellview names manually.

SKILL descriptions will be created for the following view types:

- schematic;
- symbol;
- verilog or functional;
- veriloga.

Symbol and schematic views can have any names. Verilog view names must start with a “verilog”, “veriloga” or “functional” substring. For these verilog views corresponding files are copied to the 'Output SKILL file” directory with dedicated names:
<libraryname>_<cellname>_<viewname>_<filename>

libraryname — name of the library;
cellname — name of the cell;
viewname — name of the verilog view;
filename — name of the verilog file (usually verilog.v or veriloga.va).

Cell properties (part of the Cadence CDF options) for each cell will also be translated and added to the script. Other view types such as *config* and *layout* won't be translated to the script, they can be created manually using Cadence Library Manager.

Root Cells – list of root cells that will be exported to the script together with all hierarchically linked cells. If this field is inactive all cells in the specified library will be translated. You may enter cell names manually or select them using the Cadence Library Manager (fig.2.15).

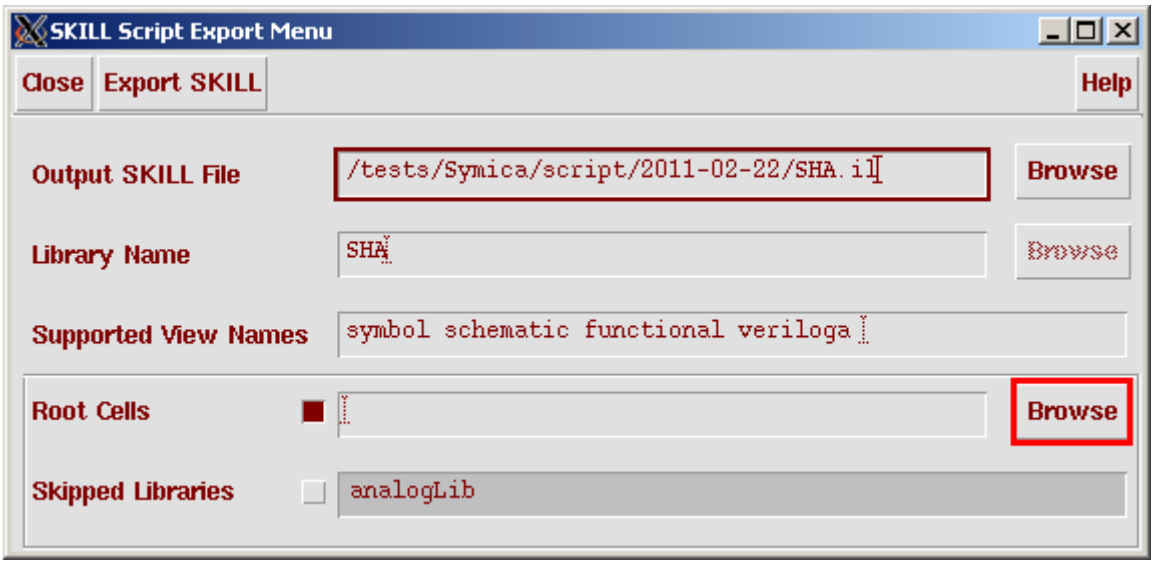

Fig.2.15.

Skipped Libraries – list of libraries that won't be exported to the script. The list is filled automatically with all cross-reference libraries as defined in 'Library Name' and 'Root Cells'. You also may specify the libraries to be skipped manually.

Click the Export SKILL button to start the execute the translation to the script (fig.2.16).

Fig.2.16.

The translation process is logged into Cadence CIW (fig.2.17).

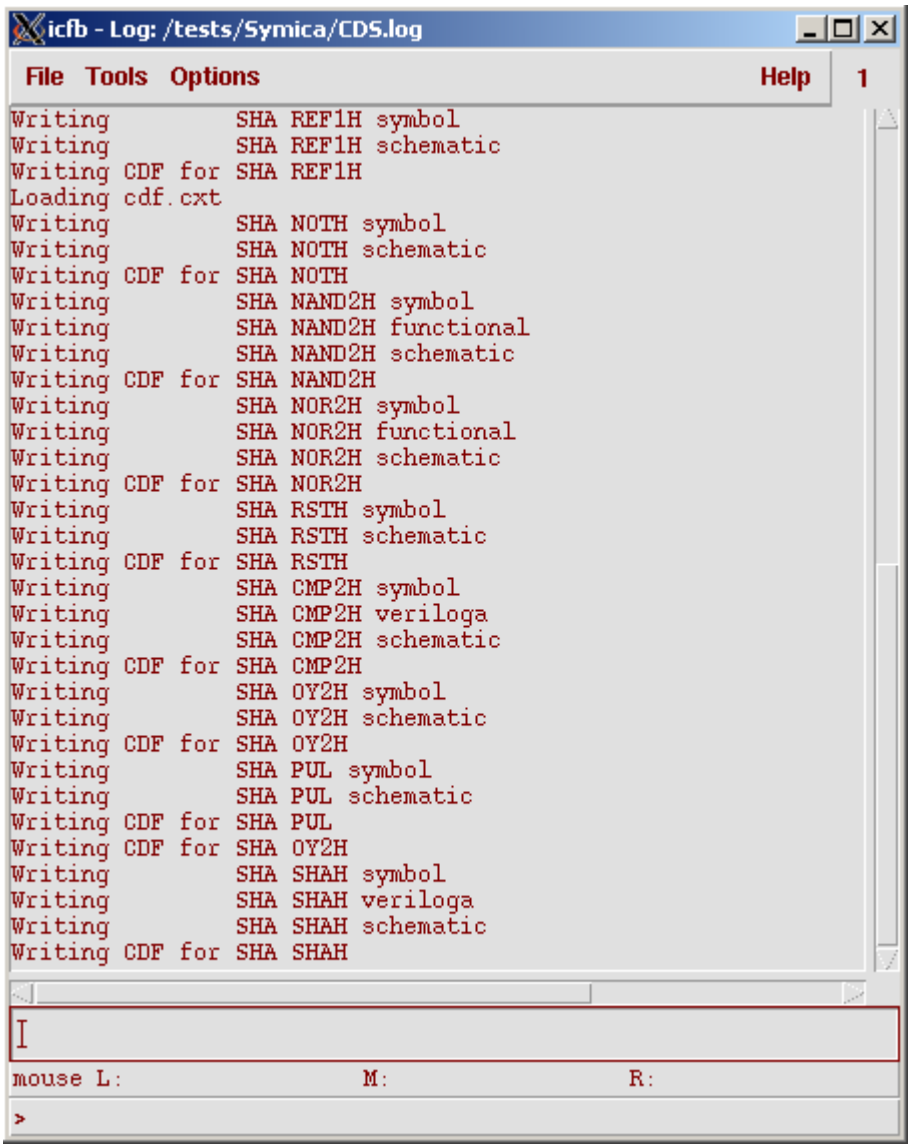

Fig.2.17.

If the translation has completed successfully the script with the added files will be created (fig.2.18).

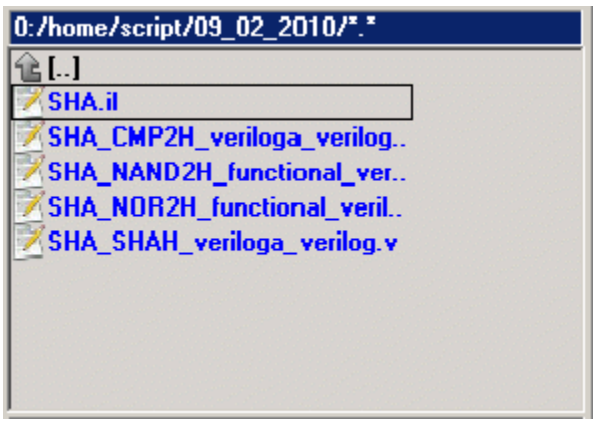

Fig.2.18.

Move generated files from the Cadence installed platform to the Symica installed platform.