

Translating Designs from PSpice Schematics to Capture

**Product Version 23.1
September 2023**

© 2023 Cadence Design Systems, Inc.
Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

1	5
Translating Designs from PSpice Schematics to Capture	5
Overview	5
What is not translated	5
Workflow	6
Translating from Within Capture	6
Overview	6
Workflow	6
Setting hierarchical or primitive view in Capture	8
Translating a schematic with hierarchical blocks	8
Translating a single or multi page schematic	9
Translating a symbol library	10
Translating a symbol library with hierarchy	10
Related Topics	11
Translating in Stand-Alone Mode	11
Related Topics	13
Cleaning Up After Translation	13
Overview	13
What does not translate	13
Workflow	13
AKO Parts	14
Attributes	15
Bus labels and bundles	15
Connectivity via wire labels	15
Footprint mapping file	16
Graphics	16
Multiple views	16
Pin names and numbers	16
Symbols and parts	17
Text	17
Variable width pin symbol	17
Xilinx and PLSyn	17
Related Topics	17

Translating Designs from PSpice Schematics to Capture
Table of Contents

Translating Designs from PSpice Schematics to Capture

Overview

The Schematics-to-Capture translator converts MicroSim schematics, symbol libraries, and package libraries to Capture designs and libraries. You can translate designs generated in MicroSim Schematics from versions 5.4 to 8.0.

Capture combines package information with symbol definitions. For this reason, symbol libraries and package libraries are translated together; they cannot be translated separately. If a symbol library does not have a corresponding package library, then the translator generates default packages using the symbols' pin numbers.

Whenever possible, the connectivity of the original schematic is maintained, and the look of the translated schematics and symbols is the same as the look of the original symbols. Where this is not possible, warning messages are displayed and the resulting design requires some editing.

There are two ways to run the translator: either by starting it from within Capture or by running it in a stand-alone mode.

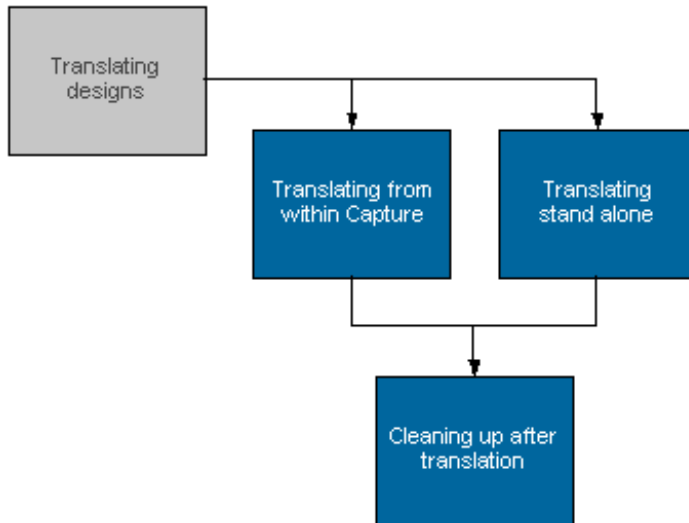
The symbols used in MicroSim Schematics are translated along with the design file; therefore, your MicroSim symbol libraries must be installed. Also, the appropriate configuration (.INI) file must be installed. For PSpice schematics, use `PSPICE.INI`. For older versions of PSpice, use `MSIM.INI`.

What is not translated

The following design objects do not translate from Schematics to Capture:

- custom borders
- no connect symbols
- viewpoints

Workflow



This user guide covers the following topics:

- [Translating from Within Capture](#)
- [Translating in Stand-Alone Mode](#)
- [Cleaning Up After Translation](#)

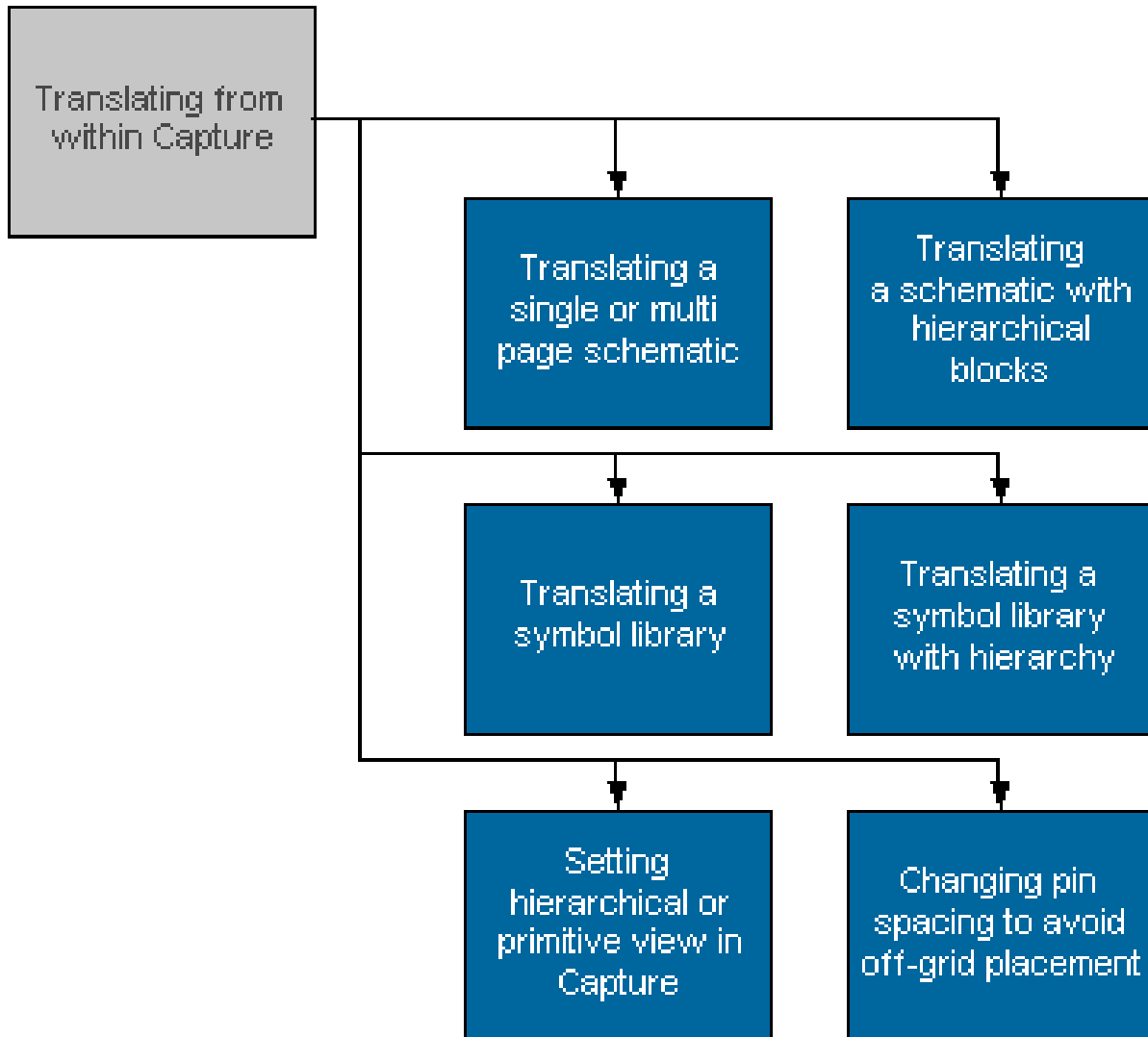
Translating from Within Capture

Overview

Use the translator from within Capture when you want to translate a single schematic and have it immediately ready for editing.

Wherever possible, the electrical connectivity and the “look” of the translated schematic pages (and symbols) are the same as the originals. Where this is not possible, warning messages are displayed in the Capture Session Log that tell you which items need to be edited. For descriptions of items that may need editing, see the *Converting MicroSim Schematics Designs to OrCAD X Capture Designs Quick Start* manual.

Workflow



This section covers the following topics:

- [Setting hierarchical or primitive view in Capture](#)
- [Translating a schematic with hierarchical blocks](#)
- [Translating a single or multi page schematic](#)
- [Translating a symbol library](#)
- [Translating a symbol library with hierarchy](#)

Setting hierarchical or primitive view in Capture

In Schematics, a part that could be both primitive and hierarchical could be simulated either way, depending on the translator view assigned. To perform the same function in Capture, you must edit certain properties, using Capture's property editor.

The table below shows the property values to use for primitive and hierarchical views.

Property editor field	Hierarchical	Primitive
Primitive	NO	YES
Implementation	<Schematic name>	<Model name>
Implementation type	Schematic View	PSpice Model

After translation from Schematics to Capture, the values in these fields reflect the translator view used in Schematics. See Capture's online help for detailed information about part properties, primitive parts, hierarchical parts, and the property editor.

Translating a schematic with hierarchical blocks

To translate a schematic with hierarchical blocks:

1. From the File menu, choose *Import – PSpice*.
The Import Design dialog box opens.
2. In the the *PSpice* tab, in the *Open* text box, specify the path for the original Schematics design file that contains a hierarchy, or use the Browse button to find the file.
3. In the *Save As* text box, specify the directory where you want to save the translated design, or use the Browse button to specify a path. The default location is the directory of the original Schematics file. The default name is the same as the original Schematics file name with an .OPJ extension.
4. In the *Schematic Configuration File* text box, specify the location of the appropriate configuration file. For PSpice Schematics Release 8 and prior, use MSIM.INI. For all more recent versions use PSPICE.INI. This file is normally in the Windows directory.
5. Select the *Translate Hierarchy* check box.
6. Select the *Consolidate all Schematic files into one Design file* check box.
7. Click *OK*.

The Project Manager appears.

All the schematics referenced in that design are listed.

If the design uses multiple views, the default implementation given to hierarchical blocks is the PSpice translator view.

All subschematics for hierarchical blocks are translated as well and they are consolidated into the single design file.

Only symbols referenced from the schematic are included in the design cache of the translated project. All custom symbol libraries should be translated individually or in a batch by [Translating in Stand-Alone Mode](#).

Translating a single or multi page schematic

To translate a single- or multi-page schematic:

1. From the File menu, choose *Import – PSpice*.
The Import Design dialog box opens.
2. In the *Open* text box, specify the path for the original Schematics design file, or use the Browse button to find the file.
3. In the *Save As* text box, specify the directory where you want to save the translated design, or use the Browse button to specify a path. The default location is the directory of the original Schematics file. The default name is the same as the original Schematics file name with a .OPJ extension.
4. In the *Schematic Configuration File* text box, specify the location of the appropriate configuration file. For PSpice Schematics Release 8 and prior, use MSIM.INI. For all more recent versions use PSPICE.INI. This file is normally in the Windows directory.
5. Click *OK*.

The translated schematic appears in the Capture schematic page editor and is ready for editing.

Only symbols referenced from the schematic are included in the design cache of the translated project. All custom symbol libraries should be translated individually or in a batch by [Translating in Stand-Alone Mode](#).

Translating a symbol library

For each symbol, if there is a package library, the pin numbers come from the package. If there is no package library, the pin numbers come from the symbol. When there are multiple gates, you can choose View Package and see multiple packages.

If the symbol library you are translating has the same name as a previously existing OrCAD X part library in the same directory, the translator overwrites the old library with the new one. To keep the old library, specify a different directory for the new library in the Save As dialog box.

To translate a symbol library:

1. From the File menu, choose *Import – PSpice*.
The Import Design dialog box opens.
2. In the *Open* text box, specify the path for the MicroSim symbol library (*.SLB), or use the Browse button to find the file.
3. In the *Save As* text box, specify the directory where you want to save the translated library, or use the Browse button to specify a path. The default location is the directory of the original library. The default name is the same as the original Schematics file name with an .OPJ extension.
4. In the *Schematic Configuration File* text box, specify the location of the appropriate configuration file. For PSpice Schematics Release 8 and prior, use MSIM.INI. For all more recent versions use PSPICE.INI. This file is in the Windows directory.
5. Click *OK*.

An OrCAD X part library (*.OLB) is created, and the Capture Project Manager appears. When the Design Manager window is maximized, the Capture title bar reads:

```
OrCAD X Capture for Windows - \[<library name>\]
```

Translating a symbol library with hierarchy

If the symbol library you are translating has the same name as a previously existing OrCAD X part library in the same directory, the translator overwrites the old library with the new one. To keep the old library, specify a different directory for the new library in the Save As dialog box.

To translate a symbol library with hierarchy:

1. From the File menu, choose *Import – PSpice*.
The Import Design dialog box opens.
2. In the *Open* text box, specify the path for the MicroSim symbol library (*.SLB) that contains

hierarchical symbols, or use the Browse button to find the file.

3. In the *Save As* text box, specify the directory where you want to save the translated library and design, or use the Browse button to specify a path. The default location is the directory of the original schematic. The default name is the same as the original Schematics file name with an .OPJ extension.
4. In the *Schematic Configuration File* text box, specify the location of the appropriate configuration file. For PSpice Schematics Release 8 and prior, use MSIM.INI. For all more recent versions use PSPICE.INI. This file is normally in the Windows directory.
5. Select the *Translate Hierarchy* check box.
6. Select the *Consolidate all Schematic files into one Design file* check box.
7. Click *OK*.

A Capture library (*.OLB) is created, and the Capture Project Manager appears.

All of the symbol and package information is stored in the part library (*.olb) and the schematics for any hierarchical symbols will be stored in the same library. Each hierarchical part will have a schematic folder with the same name in the library.

Related Topics

For information about...	Click this topic...
Translating designs without using Capture	Translating in Stand-Alone Mode
Cleaning up your design after translation	Cleaning Up After Translation

Translating in Stand-Alone Mode

Use the stand-alone translator when you want to translate a set of designs without opening all the converted files immediately in Capture.

To translate in stand-alone mode:

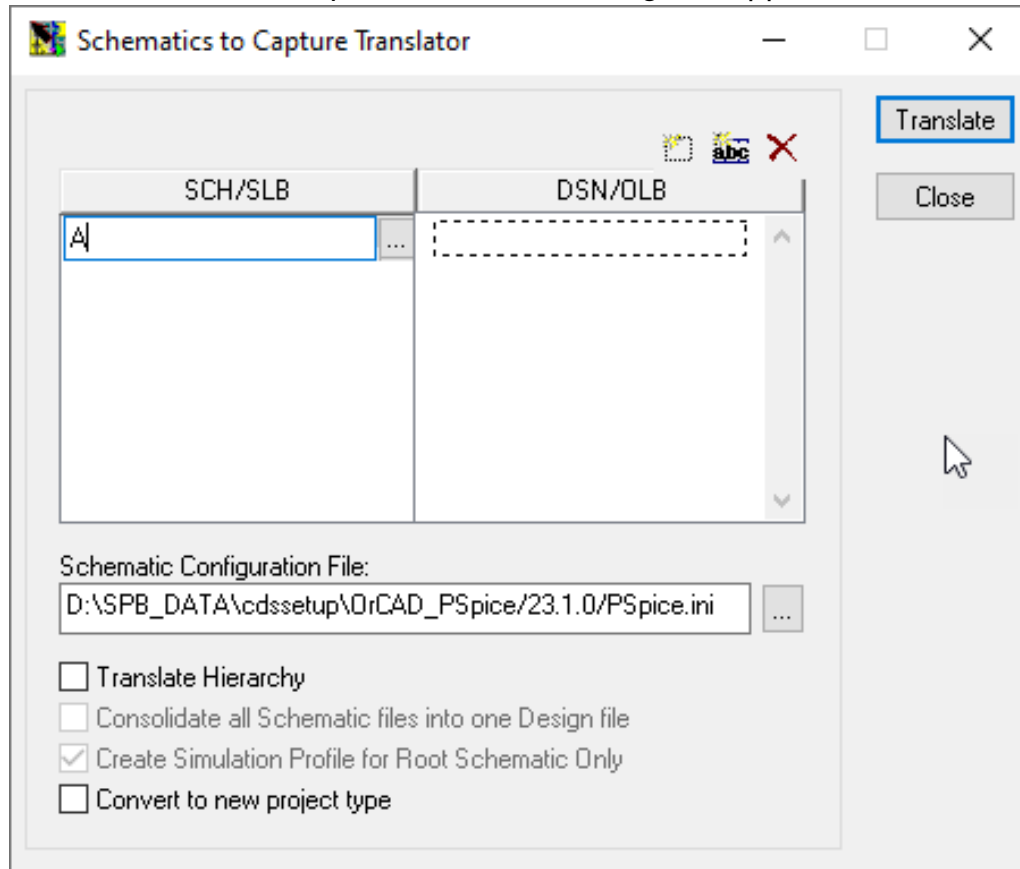
1. From the Start menu, choose Run. Specify the path and filename for `sch2cap.exe` and click *OK*. The default location for this file is: <OrCAD

X_installation_directory>\tools\bin\sch2cap.exe



Or

Double-click `sch2cap.exe` in Windows File Browser.

The *Schematics to Capture Translator* dialog box appears.



The frame on the left lists the original Schematics files that you want to convert while the frame on the right displays the destination paths and filenames of the translated files.

2. Click the new filename button ().
3. In the text box that appears in the left frame, specify the path and filename of the Schematics file you want to translate, or click the Browse button next to the text box and select the file.
4. The destination path and filename for the selected file appears in the right frame. You can edit this by double-clicking it or clicking the edit filename button () while it is selected.
5. Repeat steps 2 to 4 to add all of the files you want to translate.
6. In the *Schematic Configuration File* text box, specify the location of the appropriate configuration file. For PSpice Schematics Release 8 and prior, use `MSIM.INI`. For all more

recent versions use `PSPICE.INI`. This file is in the Windows directory.

7. Select *Translate Hierarchy* to translate all the subschematics in addition to the top-level schematics of a hierarchical design.
8. Select *Consolidate all Schematic files into one Design file* to merge the schematic and library files together into one design.
9. Click *Translate*.

Related Topics

For information about...	Click this topic...
How to translate a design from within Capture	Translating from Within Capture
Cleaning up your design after translation	Cleaning Up After Translation

Cleaning Up After Translation

Overview

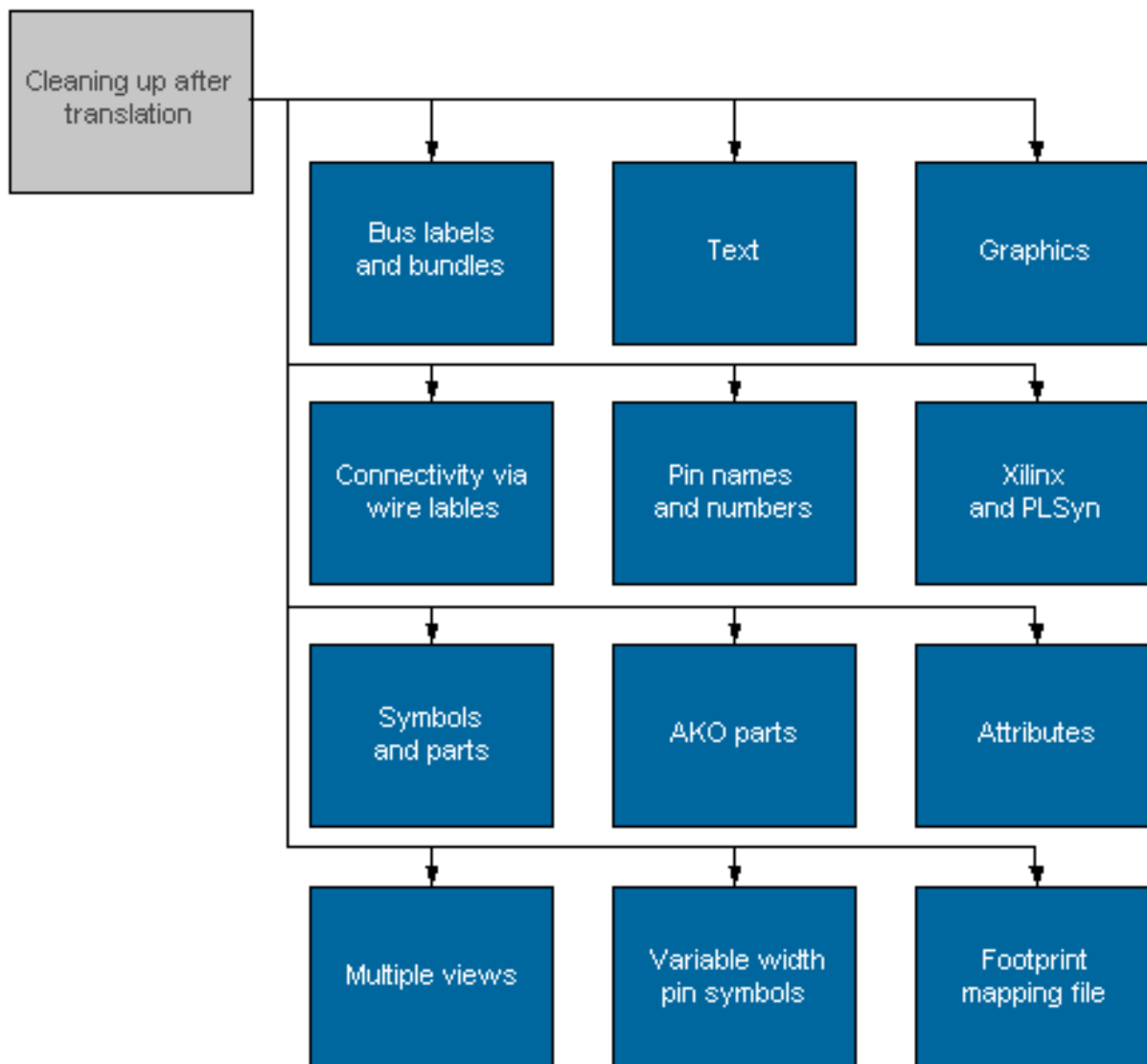
After the process is complete, you may see new items or changes in your translated design. Some issues listed below require you to take some action, while the translator automatically takes the action for others. In either case, a warning appears, keeping you informed of the situation.

What does not translate

The following design objects do not translate from Schematics to Capture:

- custom borders
- no connect symbols
- viewpoints

Workflow



AKO Parts

The translator duplicates attribute graphics, pins and other aspects of symbols for each instance of an AKO part.

Attributes

In Capture, attributes are called properties. Pin and property names are case-sensitive. In other words, the property "VALUE" is distinct from the property "Value". Therefore, the following property names change as shown during translation:

Attribute	Property
before&ldots;	after&ldots;
TEMPLATE	PSpiceTemplate
VALUE	Value

The IPIN attribute, when translated, makes hidden pins visible in Capture. A small global port is added to these.

Bus labels and bundles

Busses labeled with only one signal (for example, Bus1) are translated as wires.

Busses can only carry a contiguous sequence of signals with the same name, such as A[0-4]. Because of this, bus bundles and bundled pin names, such as A[0-1],B[0-1] or A,B,C, do not translate. You need to rename these busses in Capture.

Busses labeled with comma separators change to hyphenated sequences (for example, A[0,1,2] becomes A[0-2] and A0,A1,A2 becomes A[0-2]).

Bus labels that end in numbers (for example, Bus13[0-7]) will not be translated because these are not supported by Capture. You need to rename these busses to end in a non-numeric value.

Connectivity via wire labels

Connectivity via wire labels works within a schematic page, but not across schematic pages; for example, a wire labeled CLK on schematic page one is not connected to CLK on schematic page two.

To maintain connectivity where the original schematic used wire labels across schematic pages, the translator adds small off- page connectors to wires.

Footprint mapping file

The mapping file (S2CFPMAP.TXT) is a text file you can use to control how the translator maps footprint names from the Schematics naming convention to the convention used in the layout editor. You can add your custom footprint names and set how they are renamed for use in your translated design.

Graphics

Symbols use only one color. If the symbols in your original schematic used multiple colors, they are one color in the translated design.

Custom colors defined in the original schematic are mapped to the closest colors available.

Multiple views

If a hierarchical block or symbol contained multiple views in Schematics, all subschematics referenced are translated. However, the hierarchical block or symbol is translated to refer to the subschematic referenced by the PSpice view. You can change the hierarchical block's or symbol's subschematic by editing the block or part in Capture and changing its implementation.

Pin names and numbers

In Capture, you can either display all the pin names or display none. Pin names and numbers are displayed as text next to the respective pins. All pin name text is made the same point size in the translated design.

Port pins do not have names or numbers, so any names and numbers assigned to port pins in the original design are not present in the translated design.

Pin names and number positions are automatically set by Capture, relative to the pin. If pins are placed on the corners of a symbol, then the pin number orientation may be different in the translated design from the original schematic.

Translated symbol libraries may have overlapping pin names. You can correct this in the translated design by setting the font size in your schematic to be different from that of the originating symbol library.

Symbols and parts

Stimulus, current probe, initial condition, node set and PSpice command symbols all translate into parts in Capture. Annotation symbols translate into title block symbols.

Heterogeneous parts (those made up of different type gates within one package) translate into one part in Capture that has multiple gates.

Text

If the text in your original schematic used fonts that were not TrueType, that text will appear in the translated design using the default font (Arial). If you want to add more text, you need to use a TrueType font.

Text justification may be altered in the translated design because Capture does not support justification.

Annotation text boxes in the resulting schematic may have lines that wrap. MicroSim Schematics truncated words outside the text box, while OrCAD X Capture wraps the word to the next line.

Variable width pin symbol

If you translate a symbol library with symbols that contain variable-width pins, you will need to create parts with fixed-width pins. For example, instead of the FSTIM symbol, create FSTIM16 or FSTIM32.

For schematics that contain variable-width pin symbols, the translator creates fixed-width versions and places them in the design's design cache.

Xilinx and PLSyn

Designs containing symbols for Xilinx parts or symbols created using MicroSim PLSyn will translate to Capture but they will not simulate. You can delete the Xilinx or PLSyn parts and reuse the remainder of the circuit for other designs in Capture.

Related Topics

For information about...	Click this topic...
How to translate a design from within Capture	Translating from Within Capture

Translating designs without using Capture

[Translating in Stand-Alone Mode](#)