OrCAD Capture to EDIF 200 Schematic Translator CAP2EDIF

User's and Reference Manual





Copyright © 2003 Cadence Design Systems, Inc.

All rights reserved. OrCAD and Cadence and the OrCAD and Cadence logos are registered trademarks.

All other are properties of their respective holders.

Copyright © 1999-2003 Electronic Tools Company.

Copyright © 2006-2011 Elgris Technologies, Inc.

All rights reserved. Confidential. May be photocopied by licensed customers of

Cadence for internal business purposes only



This document may not be stored in a retrieval system, reproduced or transmitted in any form or by any means, either in whole or in part, without express prior written permission of Electronic Tools Company or Elgris Technologies, Inc. Copying includes translating into another language or format. Notice to Government users: Use, duplication or disclosure by the Government is subject to the restrictions as set forth in subparagraphs (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013. Unpublished--all rights reserved under the copyright laws of the United States.

This document contains proprietary information of Electronic Tools Company or Elgris Technologies, Inc. Electronic Tools Company has prepared this document for use only by employees of Electronic Tools Company or Elgris Technologies, Inc. and use by customers of Electronic Tools Company or Elgris Technologies, Inc. under the current license and maintenance agreements of Electronic Tools Company or Elgris Technologies, Inc. The only undertakings of Electronic Tools Company or Elgris Technologies, Inc. respecting the information in this document are as contained in such contracts, and nothing contained in this document shall be construed as changing said contracts. Information in this document is subject to change without notice and does not represent a commitment on the part of Electronic Tools Company or Elgris Technologies, Inc. Any use of this information except as defined by a written contract with Electronic Tools Company or Elgris Technologies Inc., is not authorized and, with respect to any such unauthorized use, neither Electronic Tools Company, Elgris Technologies Inc, nor any of the contributors to this document makes any representation or warranty, nor shall any warranty be implied, as to the completeness, accuracy, or usefulness of the information contained in this document or that such use of information may not infringe privately owned rights, nor do they assume any responsibility for liability or damage of any kind which may result from such use of such information.

ELECTRONIC TOOLS COMPANY, 928 First Street West, Sonoma, California 95476, USA Phone: 707-996-3320; Fax: 707-939-0246; e-mail: sales@e-tools.com

ELGRIS TECHNOLOGIES, INC. 465 Stony Point Road, 236, Santa Rosa, California 95401, USA Phone: 707-237-2794; Fax: 707-573-0237; e-mail: sales@elgris.com



Overview

This section provides a general description of the EDIF 200 schematic applications and presents the basic concepts you must understand to use them.

EDIF, which stands for Electronic Design Interchange Format, is an industry standard to facilitate formatting and exchanging electronic design data between EDA (Electronic Design Automation) systems. It is designed to account for all types of electronic design information, including schematic design, symbolic and physical layout, connectivity, and textual information, such as properties.

EDIF was originally proposed as an industry standard by Mentor Graphics, Motorola, National Semiconductor, Texas Instruments, Daisy Systems, Tektronix, and the University of California at Berkeley, all of which collaboratively embarked on its development. Since that time, EDIF has been accepted by more and more companies. EDIF V 2 0 0 was approved as a standard by the Electronic Industries Association (EIA) in 1987, and by the American National Standards Institute (ANSI) in 1988.

Figure 1 illustrates the process of creating an EDIF 200 file from a Cadence OrCAD Capture design.

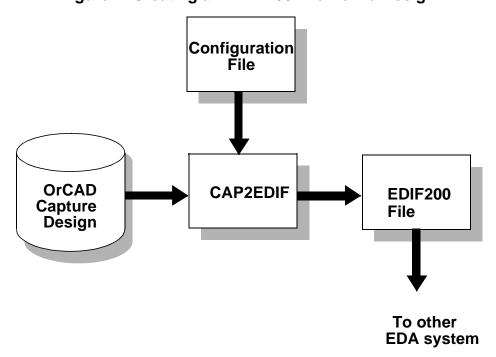


Figure 1: Creating an EDIF 200 File from a Design

Overview 1



The following subsections describe the process and elements involved in converting an OrCAD Capture design to an EDIF file using CAP2EDIF.

CAP2EDIF and the Configuration File. The configuration file is an ASCII file containing commands that you use to control the operation of CAP2EDIF. Though it is an optional item, the configuration file is usually involved in a design or library translation. When designs need special setup or mapping, you must specify these configuration commands in the configuration file before you read the design. You create a configuration file as you would any other text file.

The OrCAD Capture Design. A schematic design in OrCAD Capture.

CAP2EDIF always uses the current version of the design to produce the output EDIF representation. Consult the OrCAD Capture User's manual for additional information on how to create a design in the OrCAD Capture environment.

CAP2EDIF performs three main tasks. First, it reads the configuration for your design translation from a file. Next, it reads the OrCAD Capture design. Finally, it writes out the EDIF representation of the OrCAD Capture design.

EDIF 200 File. The output of CAP2EDIF is a file that contains an EDIF 200 schematic view of a design. This file conforms to the constructs defined in the *Electronic Design Interchange Format Version* documents, available from the Electronics Industries Association (EIA).

Requirements

Windows systems running OrCAD Capture 10.0 and above.

Major Features of the CAP2EDIF Schematic Translator

The following lists the major capabilities of the CAP2EDIF:

- CAP2EDIF supports EDIF Version 2 0 0.
- CAP2EDIF can translate all design including symbols and schematic pages;

2 Requirements



- CAP2EDIF can translate only symbols when symbol library is translated
- CAP2EDIF translates all levels of Hierarchy of the design
- CAP2EDIF translates all design data including connectivity and properties that reside on the objects
- CAP2EDIF translates occurrence data if special configuration command is provided
- CAP2EDIF is configured with the help of the configuration file
- CAP2EDIF issues warnings and error messages, helping you to debug your translation.

Running Capture to EDIF

Below are instructions on how to translate an OrCAD Capture schematic or library file into an EDIF 2 0 0 file

- 1. Select a schematic or library file in OrCAD Capture.
- 2. Click File -> Export Design
- 3. Click on EDIF tab.
- 4. In the "Save As" field, type the name of the EDIF 2 0 0 file you wish to save your design to. You could use the "Browse" button to locate the file you want to save to.
- 5. In the "Configuration file" field, type path to the CAP2EDIF configuration file, or use the "Browse" button to locate it.
- 6. Press on the OK button.
- 7. The LOG file with the name "cap2edi.log" will be created in the directory where the created EDIF 200 file resides. The Session log will contain the contents of the cap2edi.log file.
- 8. Please look at the EDIF 2 0 0 file that was created.



Key Concepts

Translation of designs and libraries

Open design or Library in OrCAD Capture and launch the Export dialogue to translate it to the EDIF 2 0 0 file.

With the default configuration file translator will convert only root Schematic View and all its contents if the design is translated or it will convert all Library contents if Library is translated.

Specify **ConvertAll = 1** in the [OrCAD Reader] section of the configuration file if you want to translate Schematic Views of the design that are not part of design's root Schematic hierarchy.

Translation of occurrence data

As Capture documentation points out an occurrence property is a user property applied to multiple occurrences of placed instances of parts or symbols in a design. This is the same as the user properties displayed and editable from the Capture v7.2 Physical view.

If your design has complex hierarchy (please consult OrCAD Capture on-line help for complex hierarchy definition) translation of occurrences become critical.

The translator will not create Occurrence data in the EDIF 2 0 0 file if default configuration file is used. Please, use OutputBackAnnotation = 1 in the [OrCAD Reader Section] of the configuration file if you want to translate occurrence data to the EDIF 2 0 0 file.

Mapping properties and names while writing to EDIF 2 0 0

Substituting names

CAP2EDIF provides a configuration file command that lets you substitute characters between the OrCAD Capture and EDIF 200 systems.

For example, to substitute the pin name "ACIA CLK" from OrCAD Capture design for name "SPECIAL_PIN" enter the following command in the [Mapper Section] of your configuration file:

setup name substitution "SPECIAL_PIN" "ACIA CLK" -PIN

4 Key Concepts



Substituting special characters

CAP2EDIF provides a configuration file command that lets you substitute characters between the OrCAD Capture and EDIF 200 systems. The **Setup** command lets you specify the OrCAD Capture characters and the substitution set which should replace them in the EDIF 200 file.

For example, to substitute the slash (/) character found in the names of OrCAD Capture design with the underscore (_) character in an EDIF 200 file, enter the following command in the [Mapper Section] of your configuration file:

setup character substitution "_" "/" -Names

See more about substitution of names, characters and strings in the [Mapper Section] of the configuration file.

Mapping Properties

OrCAD Capture properties can be mapped to EDIF 200 properties. This section explains how and when you would want to perform this task.

If your target system has a property whose purpose is equivalent to an OrCAD Capture property with a different name, you can map these properties using the **Property** command. For example, if your target system layout expects a property named "PKG_TYPE" to tell it the type of footprint to use, and the OrCAD Capture design has the property "PCB Footprint" which contains the information; you could map the two properties in this way:

property "PKG_TYPE" "PCB Footprint" INST STRING

The EDIF 200 file that is output will contain a property called "PKG_TYPE" in place of every occurrence of the OrCAD Capture "PCB Footprint" property.



Capture to EDIF 200 Translator Configuration file

OrCAD Capture to EDIF 200 Translator Configuration file contains of two sections:

- [OrCAD Reader Section] contains Translator options (arguments).
- [Mapper Section] contains commands to add/remove/map properties and names.

Default name of configuration file used with EXPORT from Capture is **cap2edi.cfg**

OrCAD Reader Section

<u>SuppressWarnings</u>

Full name: SuppressWarnings

Syntax: SuppressWarnings = 0 or SuppressWarnings = 1

Default: SuppressWarnings = 0

By default, Warnings are output to the log during Capture database conversion. To disable output of Warnings set *Suppress Warnings* = 1.

IniFilePath

Full name: Capture INI-file Path

Syntax: *IniFilePath* = "<path to capture INI-file>"

Default: *IniFilePath* = ""

Capture.ini file contains information about Capture default colors and fonts. Specify path to Capture INI-file to save default colors and fonts into the EDIF file, otherwise information from INI will be ignored. In the absence of the INI file, default fonts and colors of Capture database are used.

Default value is *IniFilePath* = "", do not use Capture INI-file.

Example: *IniFilePath* = "capture.ini"

FullLibraryNamePath

Full name: Use full path names for libraries

Syntax: FullLibraryNamePath = 0 or FullLibraryNamePath = 1

Default: FullLibraryNamePath = 0

Specifies how the translator will output Library name to an EDIF file. If **FullLibraryNamePath** is enabled (equal to 1) full path names for



libraries will be output.

Disable **FullLibraryNamePath** (equal to 0) to output only file names (without extension) for libraries.

For example: The Capture library c:\capture\library\ttl.olb is saved in EDIF as library with the name "c:\capture\library\ttl.olb" (when **FullLibraryNamePath** is 1) or ttl (when **FullLibraryNamePath** is 0).

Note: Full path name for library will be generated if the design is using libraries with the same file names, but with different full paths (even if **FullLibraryNamePath** is 0).

ConvertAll

Full name: Convert All Views

Syntax: ConvertAll = 0 or ConvertAll = 1

Default: ConvertAll = 0

If **ConvertAll** is enabled (equal to 1) translator will read all schematic views of a Capture Design, otherwise it will read only the root schematic view and all its contents.

MultipleLibs

Full name: Generate Multiple Libraries

Syntax: MultipleLibs = 0 or MultipleLibs = 1

Default: MultipleLibs = 1

Enable **MultipleLibs** on if you want the EDIF file to contain as many libraries as were referenced by Library Parts and Symbols instantiated in the design. If **MultipleLibs** is off all LibraryParts and Symbols instantiated in the design will be placed into single library in the EDIF file.

UniquePins

Full name: Unique pin names

Syntax: UniquePins = 0 or UniquePins = 1

Default: UniquePins = 0

OrCAD Capture allows to have multiple pins with the same name on one Library Part.

Set **UniquePins** to 1 to disable same port names in one symbol in the EDIF file.

To keep original duplicated OrCAD Capture pin names use default value.



PackagePinNumbersToDesignator

Full name: Package Pin Numbers in Symbol Pin

Syntax: PackagePinNumbersToDesignator = 0 or PackagePinNum-

bersToDesignator = 1

Default: PackagePinNumbersToDesignator = 0

Set **PackagePinNumbersToDesignator** to 1 to output pin numbers used by one Pin in the Package into the Port Designator construct in the EDIF file.

OutputBackAnnotation

Full name: **Output Back Annotation**

Syntax: $Output\ Back\ Annotation = 0\ or\ Output\ Back\ Annotation = 1$

Default: $Output\ Back\ Annotation = 0$

Set **OutputBackAnnotation** = **1** to save occurrence data to the EDIF file. Otherwise no occurrence data will be output to the EDIF file.

UnixPaths

Full name: Path Style in Library Name

Syntax: UnixPaths = 0 or UnixPaths = 1

Default: UnixPaths = 1

Set this option to 0 to use Windows-style path in Library naming. By default, library extension is removed and Unix-style path will be used. This option is effective only if Full Library path is used as the name of the library (see **FullLibraryNamePath** option).

Example:

In the source capture design library "C:\Capture\Libraries\testlib.olb" is used.

Set UnixPath = 1 to name this library "C/Capture/Libraries/testlib" or set UnixPath = 0 to name this library "C:\Capture\Libraries\testlib.olb" in the EDIF file.



Mapper Section

Mapping Commands:

Property

Usage

property propertyName_edif propertyName_capture owner property-Type

Description

This command enables mapping property name in the object.

Arguments

<Owner>: The object in which you want to map the Property.

<PropertyType>: Type of Property: INTEGER, REAL, BOOLEAN,
STRING.

Example:

property "PART_NAME" "Source Package" INST STRING

The preceding example substitutes the name of "Source Package" property on a Capture Instance, with name "PART_NAME" on the corresponding EDIF Instance.

Substitution of Names or Strings

Usage

setup name substitution "edif_name" "capture_name" owner setup string substitution "edif_name" "capture_name" owner setup character substitution "edif_string" "capture_string" mode

Description

The "setup name substitution" command lets you replace names of pins, nets, symbols, and instances.

The "setup character substitution" command lets you specify the character (in most cases illegal for EDIF format character set) and substitution set, which should replace them in the EDIF being created. This command supports the substitution of (as example) illegal characters in name strings. Two argument strings (capture_string and edif_string) specify character mapping. Characters in these strings will be mapped by position. So the first character in the capture_string will be replaced by the first character in the edif_string, second by second etc. If the capture_string has a length greater than the edif_string the characters in



the tail of the capture_string string will be removed from the names in the generated design. If the edif_string is longer the tail of the edif_string will be ignored.

The "setup string substitution" command allows you to change the value of the properties attached to pins, nets, instances and symbols.

Arguments

- <edif_name>: Object name you want to replace with (in the EDIF file)
- <capture_name>: The name of the object you want to find and replace
 in the source capture OrCAD design
- <owner>: specifies the type of object that owns the name or string:
- -PIN allows you to change the name or property of a pin.
- -NET allows you to change the name or property of a net.
- -INST allows you to change the name or property of an instance.
- -SYM allows you to change the name or property of a symbol (this corresponds to the EDIF cell).
- <capture_string>: The string of characters that must be replaced with
 characters specified in the edif_string string
- <edif_string>: The string of characters that must replace characters specified in the capture_string
- <mode>:
- -Names: The mode switch specifies the type of objects for which the substitution applies.

Examples:

1) When writing an EDIF file, if you want to replace Capture's Instance name "I789" with the EDIF Instance name "MyInstance", use the following command:

```
setup name substitution "MyInstance" "I789" -INST
```

2) This command replaces all '[' and ']' characters with '<' and '>'. As example Bus name A[1..3] will be replaced by A<1..3>.

```
setup character substitution "<> " "[]"
-Names
```

forgetproperty

Usage

forgetproperty prop>

Description

A general mechanism to remove properties from parts, instances, nets, pins and other objects.



Arguments

>: The Name of the property to remove

Examples

This command removes property Value from objects in the EDIF file. forgetproperty "Value"

<u>author</u>

Usage

author <author_string>

Description

A general mechanism to change the author in the EDIF file.

Arguments

< author_string >: Name of author

Examples

This command sets the author in the EDIF 200 to be "electronic tools company".

author "electronic tools company"

OrCAD Capture to EDIF 200 Mapping

This section explains the similarities and differences between terminology in the OrCAD Capture and EDIF 200 systems. It is not intended to be a comprehensive mapping, but rather an aid to familiarize the OrCAD Capture user with EDIF 200 terminology, and vice versa. For more information on any of the EDIF concepts presented in this section, refer to the *Electronic Design Interchange Format* documents, which can be obtained from the Electronic Industries Association. For more information on the OrCAD Capture design concepts presented in this section, refer to the OrCAD Capture user's manual.

OrCAD Capture to EDIF Identifier Renaming

EDIF identifiers are restricted to only alphanumeric characters and the underscore (_) symbol. OrCAD Capture names, on the other hand, are not. So, as not to violate EDIF conventions, while at the same time preserving the name used by the original application, EDIF V 2 0 0 provides the "rename" construct. When an illegal identifier (in this case, a OrCAD Library pathname), such as *c:\capture/ttl.off* is brought into EDIF, EDIF stores the old name and gives it a new, acceptable name using the proper construct.



The CAP2EDIF application reads the OrCAD Capture name and substitutes illegal characters with an acceptable EDIF identifier. Also, names that do not begin with a letter have an ampersand (&) added as the first character.

Libraries

Libraries, in both EDIF and OrCAD Capture environments, are groupings of parts and designs based on a set of common characteristics. A single EDIF file may contain descriptions of many libraries. A single design can reference parts in several different libraries.

Cells, Parts, Schematics and Instances

The EDIF 2 0 0 cell represents OrCAD Capture cell. An EDIF 2 0 0 view represents OrCAD Capture Library Part, contents of the EDIF 2 0 0 view represent OrCAD Capture Schematic. EDIF 2 0 0 cell can have multiple views that can be used for different purposes. Sometimes the views of the cell are used to represent alternative symbols (e.g. DeMorgan equivalents).

In OrCAD Capture only 2 Views of the Library Part are allowed. They are called "Normal" and "Convert". The "Normal" and "Convert" views of OrCAD Capture Library part will be translated to the corresponding views of an EDIF 2 0 0 cell. OrCAD Capture instance will be translated to EDIF 2 0 0 instance.

Attributes and Properties

Both EDIF and OrCAD Capture systems use objects called *properties*, which give more information about an object than just basic connectivity or graphics. Properties have two parts: a *name* and a *value*.

Additionally, EDIF has a set of commonly used properties called *attributes*. The EDIF attribute names are special reserved words (or keywords). As with any property, each EDIF attribute has an owner-either instance, net, or port (pin)--for which it is valid.

Because of the many types of properties design objects can have, only certain types of properties have EDIF attributes. For example, the OrCAD Capture instance property Part Reference has an equivalent EDIF *attribute*, named *designator*, whose owner is an instance.



Table 1 lists and briefly explains each of the EDIF attributes.

Table 1: EDIF Attributes Summary

Attribute	EDIF Type	Description
acload	miNoMax	An attribute of a port used to express external load capacitance.
criticality	integer	A positive or negative integer value used to describe the relative importance of a net to other nets for routing purposes.
dcfaninload	number	An attribute used to compute the fan-in load of an output or input port.
dcfanoutload	number	An attribute used to compute the fan-out load of an output or inout port.
dcmaxfanin	number	An attribute that specifies the maximum allowed fanin of input or inout ports.
dcmaxfanout	number	An attribute that specifies the maximum allowed fanout of output or inout ports.
designator	string	An attribute which specifies a pin number or a reference designator for a cell instance.
direction	INOUT INPUT OUTPUT	An attribute used to specify direction of a port, such as input, output, or inout.
unused	N/A	An attribute of ports, port instances, and off-page connectors indicating the object is not used in the interface of this particular view.

Ports and Pins

The EDIF term *port* encapsulates the OrCAD Capture term *pin* and term *Hierarchical Port*. *Ports* are connection points to cells.

Nets

In both OrCAD Capture and EDIF terminologies, a net is used to connect instances in a design. In EDIF V 2 0 0, the "net" construct together with the "joined" construct, describes how a net is connected to ports of instances in a design.

Net Buses

OrCAD Capture buses map to EDIF V 2 0 0 "net array" constructs.

