

Connectivity-to-Schematic User Guide

**Product Version IC23.1
June 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.
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. 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

<u>Introducing Connectivity-to-Schematic</u>	5
<u>Licensing Requirements</u>	6
<u>Connectivity-to-Schematic — How it Works</u>	6
<u>Running the Connectivity-to-Schematic Tool</u>	7
<u>Graphical User Interface</u>	8
<u>SKILL Functions</u>	10
<u>Standalone Mode</u>	10

2

<u>Working with Connectivity-to-Schematic</u>	11
<u>Introducing the User Interface</u>	11
<u>Connectivity-to-Schematic Options Tab</u>	12
<u>Schematic Generation Options Tab</u>	14
<u>Analog Schematic Options Tab</u>	21
<u>Generating Schematics</u>	24
<u>Guidelines to Create and Edit Symbols</u>	24
<u>Generating Digital Schematics</u>	25
<u>Generating Analog Schematics</u>	29
<u>Setting Environment Variables</u>	31
<u>Using SKILL Functions</u>	34

3

<u>Working In Standalone Mode</u>	35
<u>Starting Connectivity-to-Schematic in Standalone Mode</u>	35
<u>Using the Command-line Options</u>	35
<u>Creating the Parameter File</u>	37

A

Log File Messages 43

Common Messages 43

Error Messages 44

B

Managing Index and Multi-Sheet Schematics..... 47

Managing Multi-Sheet Schematics 47

Managing Index Schematics 49

Introducing Connectivity-to-Schematic

The Connectivity-to-Schematic tool is used to generate digital and analog schematic views from netlist views. This topic describes how to use the Connectivity-to-Schematic tool. It is aimed at the designers of digital and analog circuits and assumes that you are familiar with:

- The Virtuoso Studio design environment and application infrastructure mechanisms designed to support consistent operations between all Cadence tools.
- The applications used to design and develop integrated circuits in the Virtuoso Studio design environment, notably Virtuoso Schematic Editor.
- Virtuoso technology data.

This topic is aimed at designers of integrated circuits and assumes that you are familiar with:

- The Virtuoso Studio design environment and application infrastructure mechanisms designed to support consistent operations between all Cadence tools.
- The applications used to design and develop integrated circuits in the Virtuoso Studio design environment, notably Virtuoso Layout Suite and Virtuoso Schematic Editor.
- Component Description Format (CDF), which lets you create and describe your own components for use with ADE.

The Connectivity-to-Schematic tool (conn2sch) is used to generate schematic views from netlist views. A netlist view contains only connectivity information for a design. The Connectivity-to-Schematic tool generates a fully placed and routed schematic view using the connectivity information. You can create both, digital and analog schematic views using this tool. The Connectivity-to-Schematic tool provides different placement and routing engines to generate digital and analog schematic views. A schematic view can be opened, viewed, and edited using Virtuoso® Schematic Editor.

This topic describes how the Connectivity-to-Schematic tool works. The topic also describes different methods of running the tool.

Licensing Requirements

Digital Schematic View

In this view, the Connectivity-to-Schematic tool searches for the following licenses in the specified order and checks out one of them:

- 95100 Virtuoso® Schematic Editor L
- 95300 Virtuoso® Layout Suite L
- 95115 Virtuoso® Schematic Editor XL
- 95310 Virtuoso® Layout Suite XL
- 4 tokens of 95320 Virtuoso® Layout Suite - GXL

Analog Schematic View

In this view, the Connectivity-to-Schematic tool searches for the following licenses in the specified order and checks out one of them:

- 95115 Virtuoso® Schematic Editor XL
- 95310 Virtuoso® Layout Suite XL
- 4 tokens of 95320 Virtuoso® Layout Suite - GXL

For information on licensing in the Virtuoso Studio design environment, see [Virtuoso Software Licensing and Configuration Guide](#).

Connectivity-to-Schematic — How it Works

The Connectivity-to-Schematic tool:

1. Accepts an OpenAccess (OA) netlist view as input and generates an OA schematic view by placing each component and wiring the connections with pins and nets.

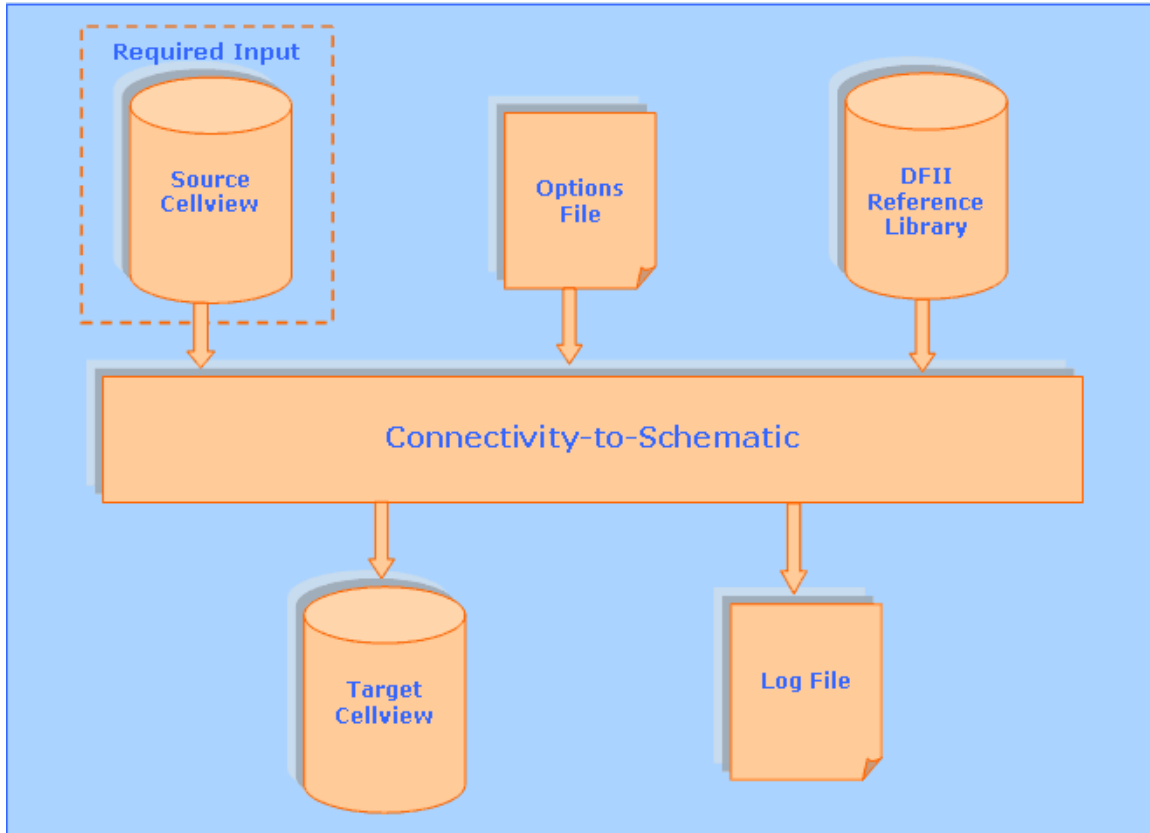
Note: This tool accepts only one cellview at a time. If you want to create a schematic view for more than one cellview or an entire library, you should call the `conn2Sch` function or `conn2sch` executable iteratively for each cellview.

2. Retains all cellview properties of source cellview in the destination schematic view.
3. Retains all effective CDF properties and other instance properties.

Connectivity to Schematic User Guide

Introducing Connectivity-to-Schematic

The following figure shows the files that the Connectivity-to-Schematic tool uses and creates while generating a schematic view:



Note: You need to specify a valid OA cellview as a source to generate a schematic view.

Running the Connectivity-to-Schematic Tool

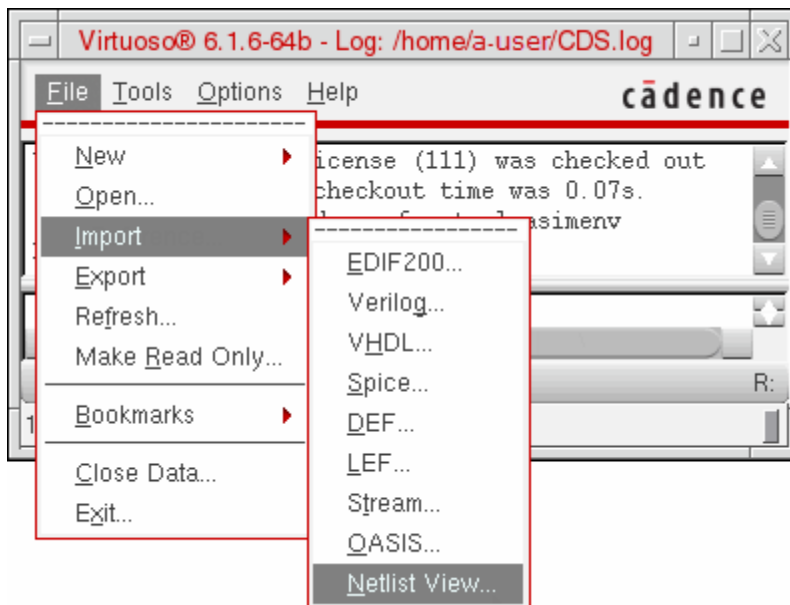
You can run the Connectivity-to-Schematic tool using:

- Graphical User Interface
- SKILL Functions
- Standalone Mode

Graphical User Interface

The graphical user interface (GUI) of the Connectivity-to-Schematic tool can be accessed from Virtuoso® Design Environment workbench. To access the Connectivity-to-Schematic GUI:

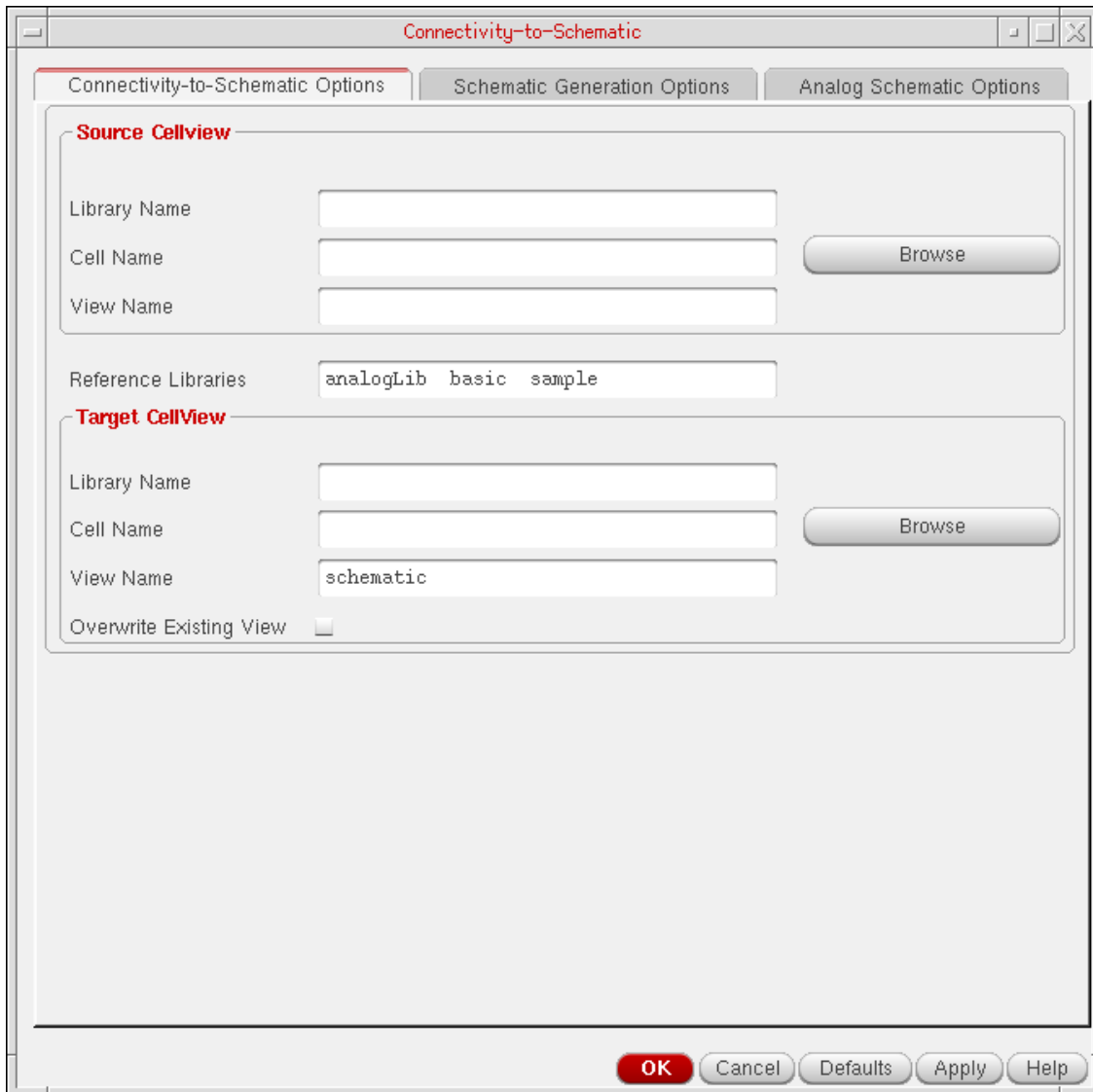
1. Click *File* and point to *Import*. A list of import tools is displayed as shown in the following figure.



Connectivity to Schematic User Guide

Introducing Connectivity-to-Schematic

2. Click *Netlist View* to launch the Connectivity-to-Schematic tool. The following figure shows the GUI of the Connectivity-to-Schematic tool:



Alternatively, you can use the `Alt+f i n` menu access keys (press `Alt` and `f` keys simultaneously and then press the `i` key followed by the `n` key) to run the Connectivity-to-Schematic tool from CIW.

For information about using the Connectivity-to-Schematic GUI, see [Chapter 2, “Working with Connectivity-to-Schematic”](#).

SKILL Functions

The Connectivity-to-Schematic tool supports SKILL functions that enable you to create schematic views and launch GUI of the tool. These functions are run in Virtuoso® Design Environment workbench.

To know more about these SKILL functions, see *HDL Import and Netlist-to-Schematic Conversion SKILL Reference*.

Standalone Mode

You use the user interface of Connectivity-to-Schematic to specify the options and parameter settings. However, while working in the standalone mode, you can create files that contain all the options and parameter settings used by the tool to generate schematics. To know how to work in the standalone mode, see Chapter 3, “Working In Standalone Mode”.

Working with Connectivity-to-Schematic

The Virtuoso Design Environment workbench enables you to access the Connectivity-to-Schematic tool using both, GUI and SKILL functions. This chapter explains the GUI of the Connectivity-to-Schematic tool. The chapter also explains how to generate digital and analog schematic views. In addition, the chapter describes environment variables and SKILL functions supported by the Connectivity-to-Schematic tool.

Introducing the User Interface

The user interface of Connectivity-to-Schematic consists of the following tabs:

- Connectivity-to-Schematic Options
- Schematic Generation Options
- Analog Schematic Options

In addition, the user interface contains the following buttons, which are common for all tabs:

- *OK*: Passes the information that you provide to generate a schematic to conn2sch executable and closes the user interface of Connectivity-to-Schematic.
- *Cancel*: Closes the user interface of Connectivity-to-Schematic.
- *Defaults*: Restores default settings on the tabs. If you change the default settings on any tab, the changes are retained during the workbench session, by default.
- *Apply*: Passes the information that you provide to generate a schematic to conn2sch executable. Unlike the *OK* button, the user interface of Connectivity-to-Schematic does not close when you click the *Apply* button.
- *Help*: Access help on how to use Connectivity-to-Schematic and get online support.

Connectivity-to-Schematic Options Tab

This tab contains options to specify source and target cellviews required to generate a schematic view. The following figure shows the *Connectivity-to-Schematic Options* tab:

The screenshot shows a dialog box titled "Connectivity-to-Schematic" with three tabs: "Connectivity-to-Schematic Options" (selected), "Schematic Generation Options", and "Analog Schematic Options".

Source Cellview

Library Name: [text box]
Cell Name: [text box] [Browse button]
View Name: [text box]

Reference Libraries: analogLib basic sample

Target Cellview

Library Name: [text box]
Cell Name: [text box] [Browse button]
View Name: schematic
Overwrite Existing View: ☐

At the bottom are buttons: OK (highlighted in red), Cancel, Defaults, Apply, and Help.

Source Cellview

Contains fields to specify a source cellview, which you need to convert to a schematic view. A source cellview consists of an existing library, cell, and view. You can either type the library,

cell, and view names in the *Library Name*, *Cell Name*, and the *View Name* fields or select these using the *Browse* button.

Reference Libraries

Specifies a list of libraries that contain the symbol views for the instances present in the source cellview. The analogLib, basic, and sample libraries are displayed as reference libraries, by default. If a master instance of all other instances in the source cellview is not of the *dbcSchematicSymbol* type, then a symbol from the reference libraries is used. If a symbol is not found in any of the reference libraries, then a new symbol view is created in the destination library and this is used as an instance master.

If you want to generate a multi-sheet schematic, you can specify a reference library containing the sheet border and the index sheet symbols, such as the *US_8ths* library.

Target Cellview

Contains fields to specify the destination library name, cell name, and view name where you need to generate the schematic view. You can either type the library, cell, and view names in the *Library Name*, *Cell Name*, and the *View Name* fields or select these using the *Browse* button.

If you do not specify the target library name or the cell name, the source library name or the source cell name is used by default. The target view name is schematic, by default. However, you can change the view name to any other view name.

Browse

Opens Library Browser to enable you to select the target library, cell, and view names. If the target cell and view do not exist, you can use the *Browse* button to select only the library name. To specify new target cell and view names, you need to type them in the *Cell Name* and *View Name* fields.

The *Browser* button is available for the source and target cellviews.

Overwrite Existing View

Overwrites the target cellview if it exists. If you do not select this option and a target cellview of the same name exists, then the Connectivity-to-Schematic tool will exit without creating the schematic.

Connectivity to Schematic User Guide

Working with Connectivity-to-Schematic

Default value: Off

Schematic Generation Options Tab

This tab contains options to manage the appearance of the schematic view, which Connectivity-to-Schematic generates. For example, you can specify if the schematic should be fully routed or extracted. The following figure shows the *Schematic Generation Options* tab:

The screenshot shows the 'Connectivity-to-Schematic' dialog box with the 'Schematic Generation Options' tab selected. The dialog has three tabs: 'Connectivity-to-Schematic Options', 'Schematic Generation Options', and 'Analog Schematic Options'. The 'Schematic Generation Options' tab contains the following settings:

- Sheet Symbol: none
- Maximum Number Of Rows: 1024
- Maximum Number Of Columns: 1024
- Font Height: 0.0625
- Line To Line Spacing: 0.2
- Line To Component Spacing: 0.5
- Component Density: 0 (slider from Lowest to Highest)
- Pin Placement**
 - ☒ Left and Right Sides
 - ☐ All Sides
 - ☐ Pin Placement File
 - Pin Placement File: [empty text box]
- Text to Symbol Generator Files: [empty text box]
- Full Place and Route: ☒
- Extract Schematics: ☒
- Generate Square Schematics: ☒
- Verbose: ☐
- Minimize Crossovers: ☐
- Analog Schematic Generation: ☐
- Optimize Wire Label Locations: ☐
- Fast Schematic Generation**
 - Generate Fast Schematic: ☒
 - Instances Greater Than: 20000
 - Ports Greater Than: 5000
- CellView To Connect Floating Ports/Nets**
 - Library: [empty text box]
 - Cell: [empty text box]
 - View: [empty text box]

At the bottom of the dialog are buttons for OK, Cancel, Defaults, Apply, and Help.



To create the symbol of the top cell of your input design, set the environment variable `generateTopCellSymbol` to `t`.

Sheet Symbol

Specifies the symbol that controls the size and page orientation of the sheet on which the schematic is to be made. Valid entries are the names of customer-designed sheet symbols and sheet symbols offered by Virtuoso® Design Environment.

A selection of sample sheet symbols is shipped with Virtuoso Design Environment in the `<install_dir>/tools/dfII/etc/cdslib/sheets/US_8ths` library. These symbols include metric sheet symbols A0 through A4, and sheet symbols for the traditional A, A.book (vertical orientation), B, C, D, E, and F sizes.

The library containing the sheet symbol entered in this field must be in your `cds.lib` file for the Connectivity-to-Schematic tool to use the correct information.

If the *Sheet Symbol* is set to `none`, a single sheet schematic is generated. If a specific sheet symbol size is provided, and the schematic cannot fit on a single sheet, a multi-sheet schematic is generated. A multi-sheet schematic has one index sheet and many schematic sheets.

The cell name of the index sheet for the multi-sheet schematic has the same name as the target cell name. The schematic sheets are named as `<target_cell_name>@sheet<nnn>` where `nnn` is the sheet number. For example, if the target cell name is `top`, the index sheet is also named as `top`. If the `top` index sheet has three schematic sheets, these are named as `top@sheet001`, `top@sheet002`, and `top@sheet003`.

Default value: `none`

Maximum Number Of Rows

Specifies the maximum number of rows on each sheet. This option is used only on a multi-sheet schematic. The value must be an integer from 1 to 1024. The maximum number of rows that can fit on a sheet is 1024.

The number of rows that are displayed on a sheet might be less than the specified number of rows depending on the size of the instance symbols and the size of the sheet.

Default value: 1024

Maximum Number Of Columns

Specifies the maximum number of columns on each sheet. This option is used only on a multi-sheet schematic. The value must be an integer from 1 to 1024. The maximum number of columns that can fit on a sheet is 1024.

The number of columns that are displayed on a sheet might be less than the specified number of columns depending on the size of the instance symbols and the size of the sheet.

Default value: 1024

Font Height

Controls the size of the font used for pin, wire, and instance labels. The value must be a real value between 0.0375 and 0.125. The wire and instance labels use the font size specified in *Font Height*. Pin labels are scaled down to 75 percent of the specified size.

Default value: 0.0625

Line To Line Spacing

Specifies the spacing in inches between nets flowing in a channel for each sheet. The spacing for net segments connected to instance pins depends on the pins placed on the symbols of the instances. *Line To Line Spacing* is used for all other net segments. The value must be a decimal number in the range of 0.19 to 0.5 inches.

Default value: 0.2

Line To Component Spacing

Specifies the spacing in inches between a component and the nearest net flowing in a channel. The value must be a decimal number in the range of 0.19 to 0.5 inches.

Default value: 0.5

Component Density

Controls the density of a schematic. The value must be an integer from 0 to 100, where 100 is the most dense and 0 is the least dense.

Default value: 0

Pin Placement

Specifies the placement of pins on a symbol. Pin placement can be defined as any of the following two options:

- *Left and Right Sides*: Places pins on the left and right sides of a symbol, usually with input pins on the left and inout pins and output pins on the right.
- *All Sides*: Places pins on any side of a symbol.
- *Pin Placement File*: Places pins on specific sides of modules as defined in a file called pin placement file. When you select this option, you must provide the path of the pin placement file in the *Pin Placement File Name* field.

Pins and their placement on modules are defined in the pin placement file in the following format:

```
pin_placement := {moduleName, {left|right|top|bottom}, comma-separated-pin-name-list}
```

For example:

```
pin_placement := inv, left, a, b, c, d
pin_placement := inv, top, e, f
pin_placement := test4, bottom, c, d
pin_placement := test4, left, e
```

The first line in the example given above specifies that module `inv` has pins `a`, `b`, `c`, and `d` oriented on the left side.

Default value: `Left and Right Sides`

Text to Symbol Generator Files

Specifies a space-separated list of tsg files to be used by Connectivity-to-Schematic to generate symbols in the target library. Connectivity-to-Schematic internally runs the Text-to-Symbol generator, a tool that reads the symbol descriptions given in the tsg files to create symbol views.

For more details about the Text-to-Symbol generator and the tsg files, see [Text-to-Symbol Generator](#) in Virtuoso Schematic Editor L User Guide.

Important

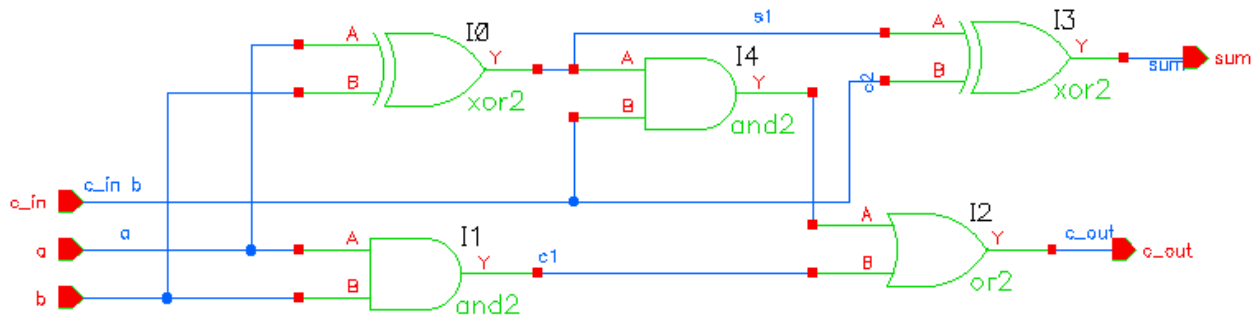
If the tsg files are given, the direction of pins specified by the [Pin Placement](#) option is ignored. Pins are placed on symbols as defined in the tsg files.

Connectivity to Schematic User Guide

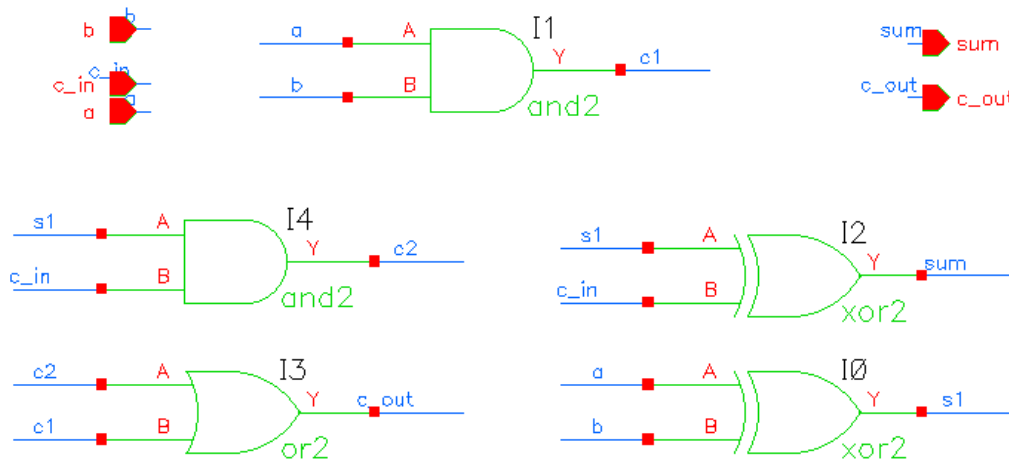
Working with Connectivity-to-Schematic

Full Place and Route

Specifies whether all the connections should be made by wiring. The following figure shows a fully routed schematic.



Turn this option off to generate a schematic in which the nets are not explicitly connected. In such a schematic the nets are not wired and the connectivity is indicated by name. This type of schematic is useful for importing large designs where the primary focus is in creating a netlist and simulation and not viewing the design. In addition, generating such a schematic is much faster than generating a completely routed schematic as a significant amount of run-time is spent during routing. The following figure shows a schematic in which the nets are not explicitly routed:



Default value: On

Note: You can also use the [Fast Schematic Generation](#) options to determine how nets must be connected when the design has a large number of instances and ports.

Generate Square Schematics

Specifies whether to square the schematic. Turn this option off if you do not want the rows and columns of devices modified to make a rectangular schematic into a square one.

Default value: `On`

Minimize Crossovers

Specifies whether to minimize crossover of nets. Turn this option `on` to minimize crossovers of nets.

Default value: `Off`

Optimize Wire Label Locations

Specifies whether to override default label placement. Turn this option on to override default label placement, which keeps overlap of segments or labels to a minimum, in favor of fast placement, which places labels of segments at the midpoint without checking for minimum overlap.

Default value: `Off`

Extract Schematics

Specifies whether to extract the target cellview in order to report errors and warnings in the schematic. An extracted schematic has blinking markers for the errors or warnings in the schematic view.

Default value: `On`

Verbose

Specifies whether to print detailed status messages while the schematic is being partitioned and routed. Turn this option on to print detailed messages.

Default value: `Off`

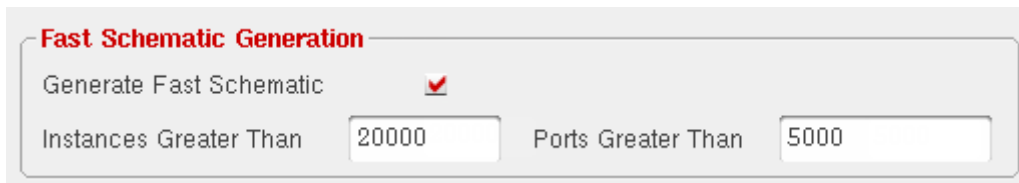
Analog Schematic Generation

Specifies whether to generate an analog schematic using analog placement and routing engine. The options on the *Analog Schematic Options* tab become active only when you turn this option on. The Connectivity-to-Schematic tool generates digital schematic by default.

Default value: Off

Fast Schematic Generation

The options in this group box enable fast generation of the schematic when the design being imported contains a large number of instances or ports.



The following table lists the fields in the *Fast Schematic Generation* group box, with their default values and environment variables.

Field	Default Value	Environment Variable
<i>Generate Fast Schematic</i>	On	generateFastSchematic
<i>Instances Greater Than</i>	20000	fastSchematicMaxInst
<i>Ports Greater Than</i>	5000	fastSchematicMaxPort

For details on the environment variables, see [“Setting Environment Variables”](#) on page 31.

To use the fast schematic generation feature, select the *Generate Fast Schematic* check box and specify the number of instances and ports in their respective fields. If the number of instances or ports in the design exceeds the specified number, the tool generates a schematic in which the instances are placed in a two-dimensional array without any routing, and the connectivity of the nets is indicated by names.

Notes:

- To enable the fast schematic generation feature without considering the number of instances or ports in the designs being imported, select *Generate Fast Schematic* and type 0 in the *Instances Greater Than* and *Ports Greater Than* fields.

- If you disable the fast schematic generation feature, and the design has a large number of instances and ports, schematic generation can take significant time to place and route the instances and nets.

Cellview to Connect Floating Ports/Nets

The fields in this group box connect the floating nets ports in the schematic with a cellview when the source cellview is a netlist view.



The image shows a dialog box titled "CellView To Connect Floating Ports/Nets" in red text. It contains three input fields labeled "Library", "Cell", and "View" on the left. To the right of these fields is a "Browse" button. The fields are currently empty.

The fields specify the library name, cell name, and view name of the cellview that you want to use for connecting to the floating ports and floating nets. For example, to use the `noconn` cellview to connect to the floating ports and nets, you can specify the library as `basic`, cell name as `noconn` and view names as `symbol`. You can either type the library, cell, and view names in the *Library Name*, *Cell Name*, and the *View Name* fields or select these using the *Browse* button.

Note: This feature will work only when the source cellview is a netlist view.

Alternatively, you can use the command-line option `noconn_symbol` or the `.cdsenv` variable `noconn_symbol`.

Analog Schematic Options Tab

This tab contains options to generate an analog schematic from an imported netlist. The information provided on this tab is passed to the `conn2sch` executable only if the *Analog*

Connectivity to Schematic User Guide

Working with Connectivity-to-Schematic

Schematic Generation option, on the *Schematic Generation Options* tab, is turned on. The following figure shows the *Analog Schematic Options* tab:

The screenshot shows a dialog box titled "Connectivity-to-Schematic" with three tabs: "Connectivity-to-Schematic Options", "Schematic Generation Options", and "Analog Schematic Options". The "Analog Schematic Options" tab is selected. It contains three sections:

- CellView To Be Used For Power Cell**:
 - Library Name:
 - Cell Name:
 - View Name:
 - Library Manager Browser button
- CellView To Be Used For Ground Cell**:
 - Library Name:
 - Cell Name:
 - View Name:
 - Library Manager Browser button
- Global Net**:
 - Power Net Name:
 - Ground Net Name:

At the bottom of the dialog are buttons: OK, Cancel, Defaults, Apply, and Help.

Cellview To Be Used For Power Cell

Contains options to specify a cellview that you need to use as a master of a power cell instance. Select the values provided in the *Library Name*, *Cell Name*, and the *View Name* cyclic fields to specify a cellview. Alternatively, you can type the library name, cell name, and view name in these fields. The default values are:

Library Name: analogLib

Cell Name: vdd

View Name: symbol

If multiple instances of a power cellview exist, only a single instance is created while importing a design to ensure that the final schematic contains only one symbol instance of a power cell.

Cellview To Be Used For Ground Cell

Contains options to specify a cellview that you need to use as a master of a ground cell instance. Select the values provided in the *Library Name*, *Cell Name*, and the *View Name* cyclic fields to define a cellview. Alternatively, you can type the library name, cell name, and view name in these fields. The default values are:

Library Name: analogLib

Cell Name: gnd

View Name: symbol

If multiple instances of a ground cellview exist, only single instance is created while importing a design to ensure that the final schematic contains only one symbol instance of a ground cell.

Global Net

Contains options to specify power net and ground net names. Power and ground nets are used to vertically align the instances. Instances connected to a power net are placed towards top of a schematic and instances connected to a ground net are placed towards bottom of a schematic.

You can specify the power net and ground net names in the *Power Net Name* and the *Ground Net Name* fields. Default values for these fields are:

Power Net Name: vdd!

Ground Net Name: gnd!

Generating Schematics

Guidelines to Create and Edit Symbols

You must follow certain guidelines to create and edit symbols, which a schematic uses. If you do not follow these guidelines, the Connectivity-to-Schematic tool might not generate the schematic, or might generate a schematic, which is incorrect or off-grid. Also, nets might overlap symbols and symbol labels might overlap nets or net labels. The guidelines to create and edit symbols are:

- To prevent extraction errors, make sure the snap spacing is 10 database units (dbu) or greater.

When creating schematics, the Connectivity-to-Schematic tool takes the snap spacing for schematics and symbols from the property, `xSnapSpacing` on the `viewType`, schematic in the target library. Ideally, snap spacing should be an even number of dbu.

- To ensure that schematics are on-grid, check that the snap spacing of the target library and that of the symbols in the reference libraries are the same.

If this is not possible, make sure the symbols in the reference libraries have a snap spacing that is a multiple of the snap spacing of the target library.

- To ensure that schematics are correct, check that the outer edges of all pin figures on each side of the symbol are on the same line (pin figures cannot be recessed from other pin figures) and are abutting the bounding box.

The bounding box is the smallest box enclosing all pin figures and all shapes on the Device-Drawing Layer Purpose pair. Also, make sure that no symbol figures are beyond the pin figures on either side of the symbol shape. You should also specify a unique pin access direction on the pins of the symbol; otherwise, a net might be connected to a pin from a wrong direction.

- Snap spacing is set lower than 10 dbu in the `schematic` view type in the target library.
- Snap spacing in the symbol is not a multiple of the snap spacing in the target.

Note: If the pins of a symbol are too close, the input/output pins of the cellview in which the symbol is placed may overlap. In such cases, the input/output pins are displaced and connected by their names to the connecting net.

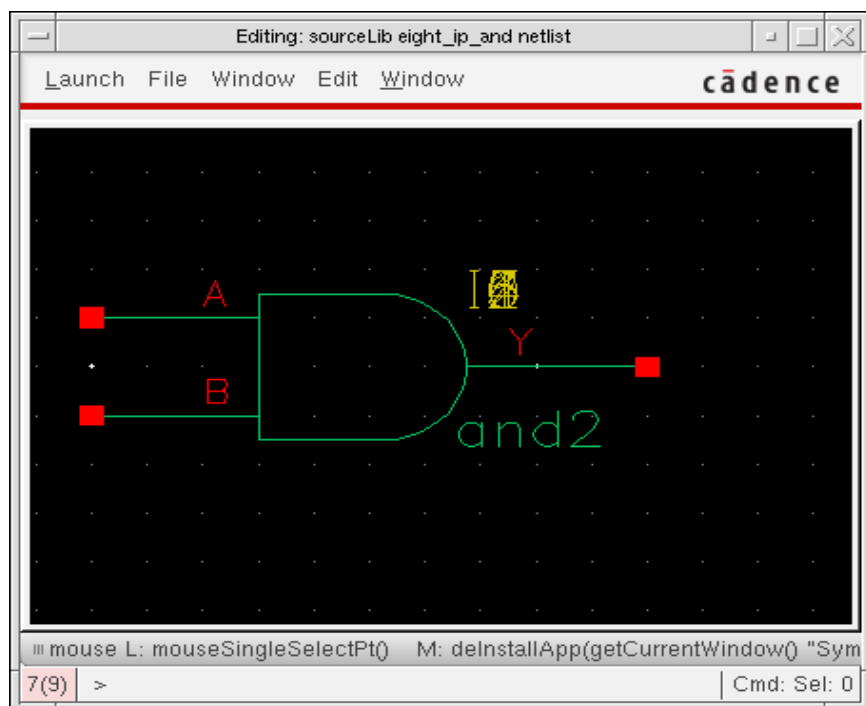
- To ensure that schematics are on-grid, make sure all marked lengths are a multiple of the snap space, including the pin-to-pin distances and the origin-to-pin distance.

The Connectivity-to-Schematic tool places the origin of the Composer Symbol on the snap grid. If all marked lengths are not multiples of the snap space, the generated schematic will

be off-grid. Usually this is not a problem because the Symbol Editor automatically imposes this restriction. Problems might occur only if the snap space of the symbol has been changed during or after editing the symbol.

Generating Digital Schematics

The following figure shows the netlist view of the `eight_ip_cell` cell existing in the `sourceLib` library:



To generate a digital schematic view from the netlist view:

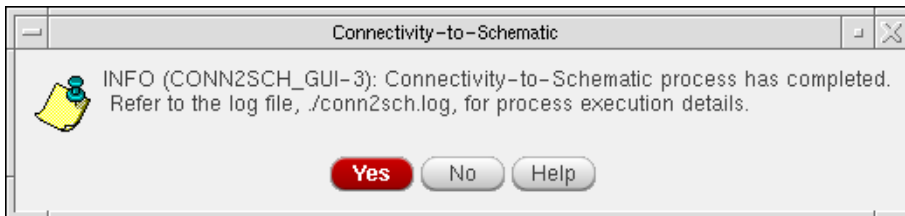
1. Click *File*, point to *Import*, and then click *Netlist View* to run the Connectivity-to-Schematic tool.
2. Specify the names of a source library, source cell, and source view in the appropriate fields on *Connectivity-to-Schematic Options* tab.
3. Specify the names of target library, target cell, and target view in the appropriate fields on the *Connectivity-to-Schematic Options* tab.

If you do not specify the target library name, the source library is used as the target. Similarly, if you do not specify the target cell name, then the source cell name is used as the target cell name.
4. Click the *Schematic Generation Options* tab to modify schematic generation options.

Connectivity to Schematic User Guide

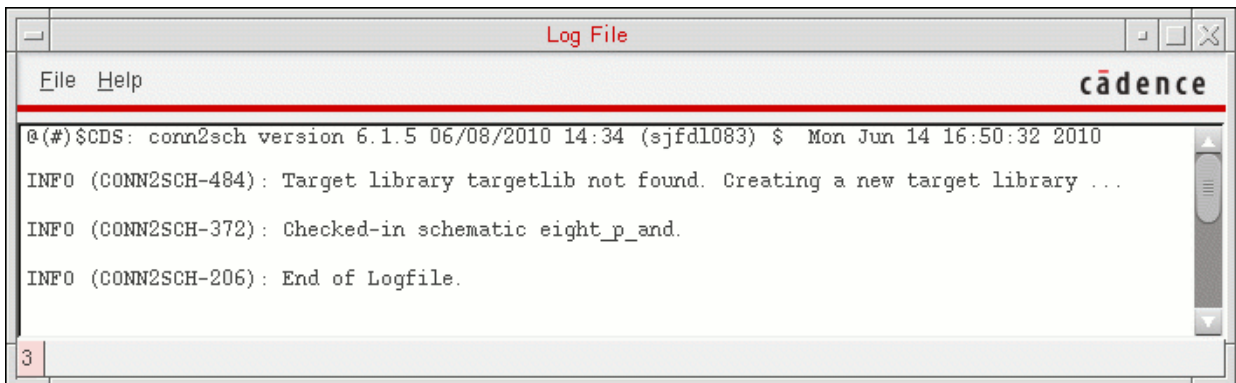
Working with Connectivity-to-Schematic

5. Click *OK* or *Apply* on the *Connectivity-to-Schematic* tab to generate a digital schematic.
6. Wait until Connectivity-to-Schematic displays a completion message as shown in the following figure:



The completion message specifies that a schematic is generated in the specified target library.

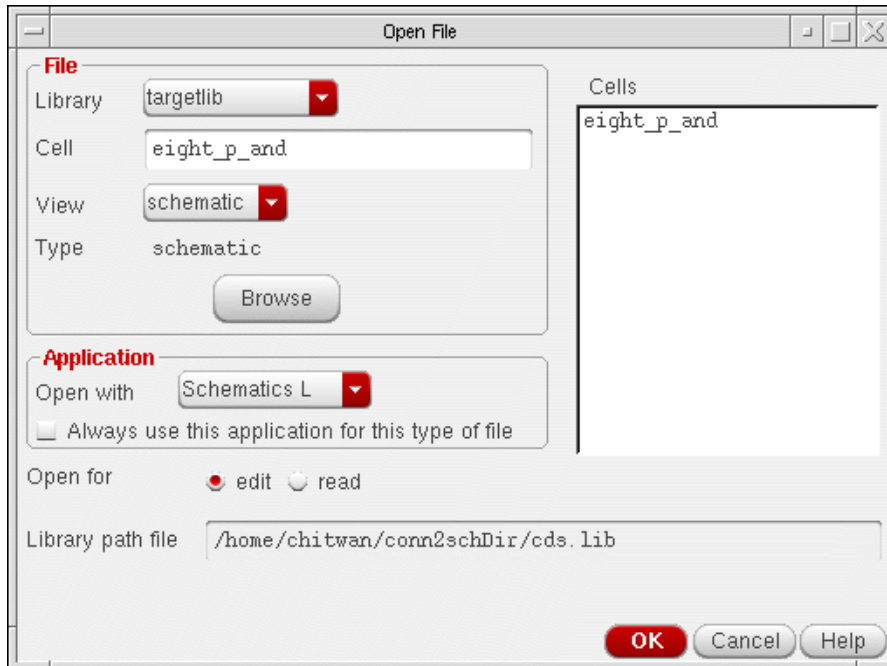
7. Click *Yes* to view the log file. The following figure shows the log file contents:



Connectivity to Schematic User Guide

Working with Connectivity-to-Schematic

8. Select *File* — *Open* to open the *Open File* dialog box as shown in the following figure:



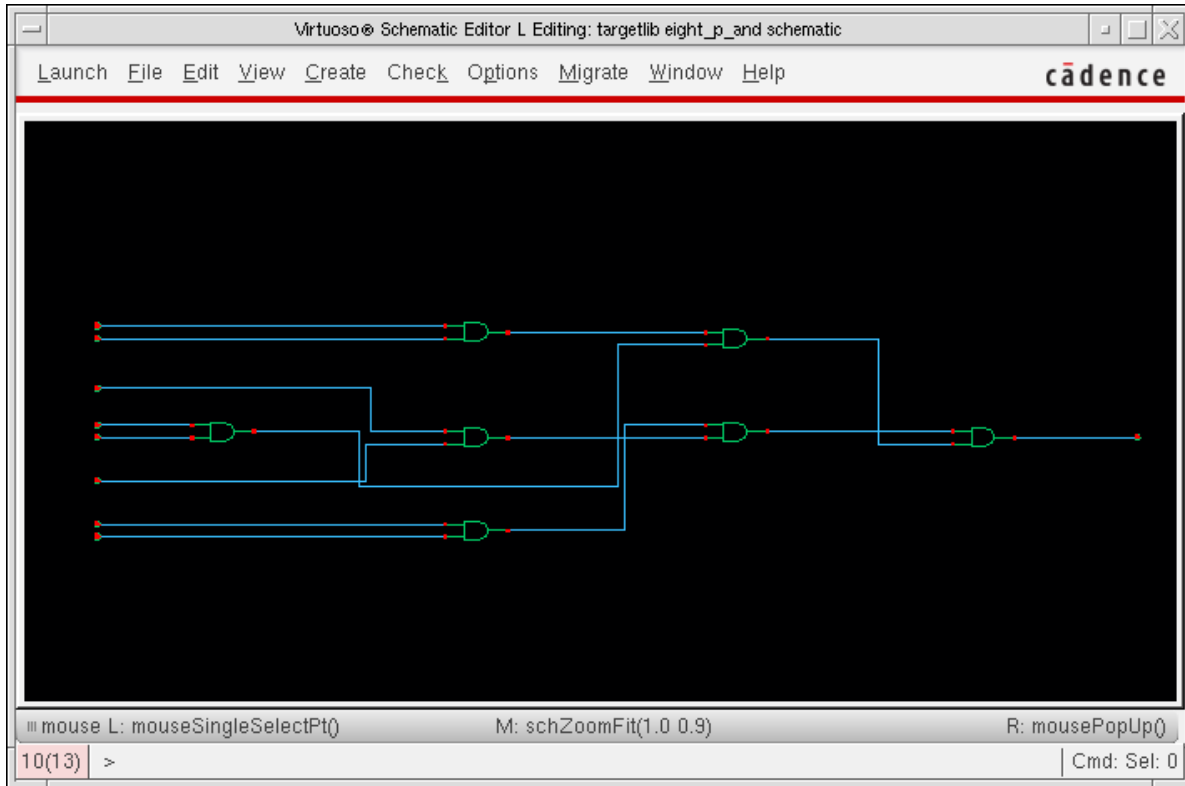
Alternatively, you can open the schematic view from Library Manager.

9. Select *digital_sch* from the *View* drop-down list box on the *Open File* dialog box. The schematic view generated by Connectivity-to-Schematic is displayed in the Virtuoso Schematic Editor L window.

Connectivity to Schematic User Guide

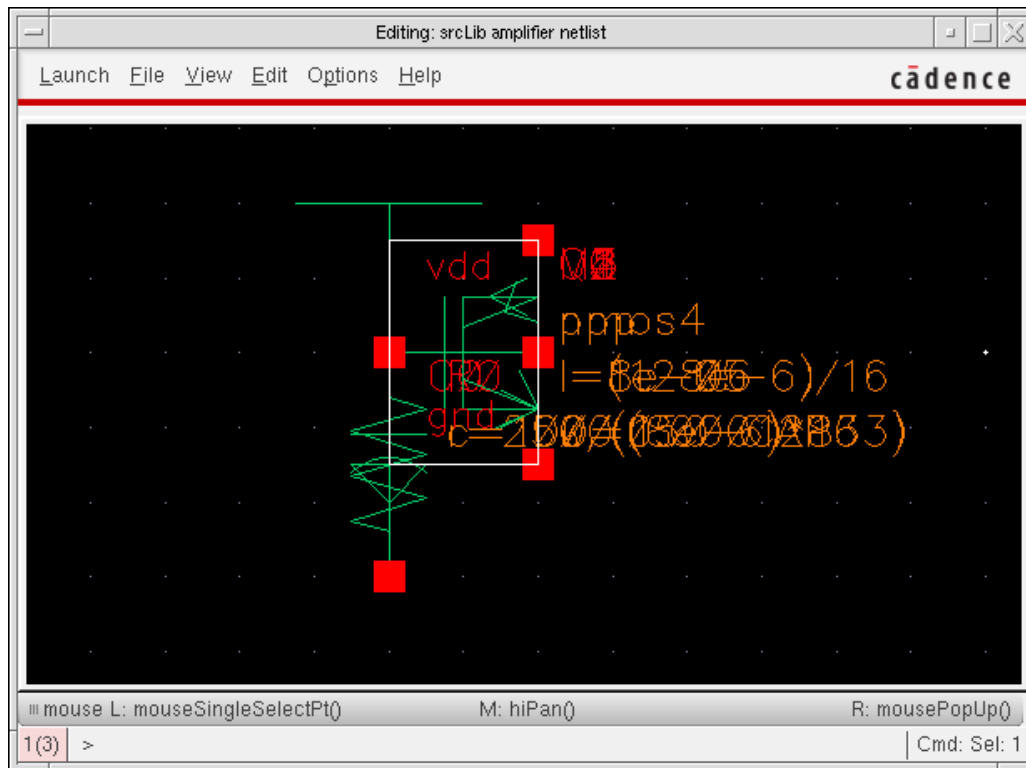
Working with Connectivity-to-Schematic

The following figure shows the digital schematic view for the `eight_ip_cell` cell in the `targetlib` library:



Generating Analog Schematics

The following figure shows the netlist view of the `amplifier` cell existing in the `srcLib` library:



To generate an analog schematic from the netlist view:

1. Click *File*, point to *Import*, and then click *Netlist View* to run the Connectivity-to-Schematic tool.
2. Specify the names of a source library, source cell, and source view in the appropriate fields on *Connectivity-to-Schematic Options* tab.
3. Specify the names of target library, target cell, and target view in the appropriate fields on the *Connectivity-to-Schematic Options* tab.

If you do not specify the target library name, the source library is used as the target. Similarly, if you do not specify the target cell name, then the source cell name is used as the target cell name.

4. Click the *Schematic Generation Options* tab to modify schematic generation options.

Connectivity to Schematic User Guide

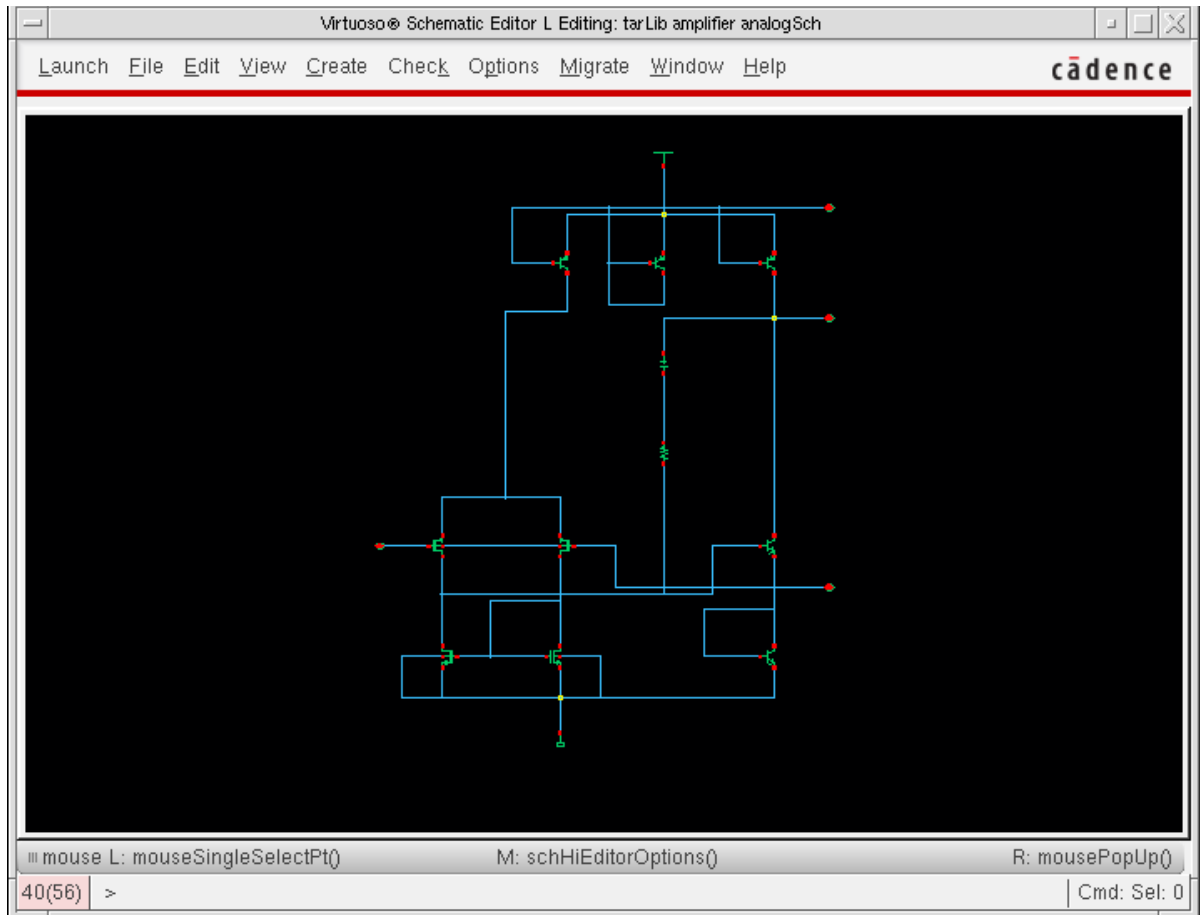
Working with Connectivity-to-Schematic

5. Select the Analog Schematic Generation option to activate the options on the *Analog Schematic Options* tab.
6. Click the *Analog Schematic Options* tab to modify analog schematic options.
7. Click *OK* or *Apply* on the *Connectivity-to-Schematic* tab to generate an analog schematic
8. Wait until Connectivity-to-Schematic displays the completion message that a schematic is generated in the specified target library.
9. Click *Yes* on the Connectivity-to-Schematic message box to view the log file.
10. Select *File — Open* to open the *Open File* dialog box.
11. Select `analogSch` from the *View* drop-down list box on the *Open File* dialog box. The schematic view generated by Connectivity-to-Schematic is displayed in the Virtuoso Schematic Editor L window.

Connectivity to Schematic User Guide

Working with Connectivity-to-Schematic

The following figure shows the analog schematic view of the `amplifier` cell in the `tarLib` library:



Video

For information on the salient features of Analog Schematic Generator and how it is integrated with Spice In and Connectivity-to-Schematic, see [Analog Schematic Generation](#) multimedia demonstration.

Setting Environment Variables

Environment variables are the global values or settings that determine the default behavior of the Connectivity-to-Schematic tool (`conn2sch`). You can change the values of these variables to customize the operation and behavior of the tool. Environment variables are specified in the `.cdsenv` file, which can be accessed from the following path:

```
<inst_dir>/tools/dfII/etc/tools/conn2sch/.cdsenv
```

Connectivity to Schematic User Guide

Working with Connectivity-to-Schematic

The following table shows the entries in the `.cdsenv` file and the default values of the variables. Notice that each entry in the `.cdsenv` file begins with the tool name, followed by the environment variable name, variable type, the value, and the range of the variable.

Environment Variable Name	Type	Value	Range
<code>asg_options</code>	boolean	<code>nil</code>	-
<code>density_level</code>	int	<code>0</code>	<code>(0 100)</code>
<code>dest_cell_name</code>	string	<code>" "</code>	-
<code>dest_sch_lib</code>	string	<code>" "</code>	-
<code>dest_symbol_view_name</code>	string	<code>"symbol"</code>	-
<code>dest_view_name</code>	string	<code>"schematic"</code>	-
<code>extract_schematic</code>	boolean	<code>t</code>	-
<code>fastSchematicMaxInst</code>	int	<code>20000</code>	-
<code>fastSchematicMaxPort</code>	int	<code>5000</code>	-
<code>full_place_and_route</code>	boolean	<code>t</code>	-
<code>generate_square_schematic</code>	boolean	<code>t</code>	-
<code>generateTopCellSymbol</code>	boolean	<code>nil</code>	-
<code>generateFastSchematic</code>	boolean	<code>t</code>	-
<code>ground_net</code>	string	<code>"gnd!"</code>	<code>nil</code>
<code>ground_symbol</code>	string	<code>"analogLib gnd symbol"</code>	-
<code>import_if_exists</code>	int	<code>0</code>	-
<code>label_height</code>	float	<code>0.0625</code>	<code>(0.0375 0.125)</code>
<code>line_component_spacing</code>	float	<code>0.500000</code>	<code>(0.19 0.5)</code>
<code>line_line_spacing</code>	float	<code>0.200000</code>	<code>(0.19 0.5)</code>
<code>log_file_name</code>	string	<code>"./conn2sch.log"</code>	-
<code>minimize_crossovers</code>	boolean	<code>nil</code>	-
<code>optimize_wire_label_locn</code>	boolean	<code>nil</code>	-
<code>page_col_limit</code>	int	<code>1024</code>	<code>(1 1024)</code>
<code>page_row_limit</code>	int	<code>1024</code>	<code>(1 1024)</code>
<code>pin_placement</code>	cyclic	<code>left_and_right_sides</code>	<code>nil, left_and_right_sides, all_sides, file, <file-name></code>

Connectivity to Schematic User Guide

Working with Connectivity-to-Schematic

Environment Variable Name	Type	Value	Range
power_net	string	"vdd!"	nil
power_symbol	string	"analogLib vdd symbol"	-
ref_lib_list	string	"analogLib basic sample"	-
sheet_symbol	string	"none"	-
src_cell_name	string	" "	-
src_sch_lib	string	" "	-
src_view_name	string	" "	-
verbose	boolean	nil	-
noconn_symbol	string	"basic noConn symbol"	-

Using SKILL Functions

The Connectivity-to-Schematic tool provides various SKILL APIs for tasks such as the following:

- Generating a schematic view from an imported netlist view.
- Invoking the graphical user interface of the tool.

For details, see *HDL Import and Netlist-to-Schematic Conversion SKILL Reference*.

Working In Standalone Mode

The Connectivity-to-Schematic tool can be run either from the Virtuoso® Design Environment workbench or in the standalone mode. This chapter describes the command to run the Connectivity-to-Schematic tool in the standalone mode, the parameter file options, and the command-line options.

Starting Connectivity-to-Schematic in Standalone Mode

The standalone executable, `conn2sch` can be accessed from the following directory:

```
<inst_dir>/tools/dfII/bin
```

The command to start the Connectivity-to-Schematic tool is:

```
conn2sch -lib <libName> -cell <cellName> -view <viewName>  
[-param <paramFileName>] [-destlib <destLibName>]  
[-destview <destViewName>] [-cdslib <cdsLibFileName>  
[-f <optionsFileName>] [-log <logFileName>] [+place_only]  
[-fast_labels] [-help] [-version] [-min_crossovers]  
[-nosquare] [-asg] [+noxtrschr] [-overdense] [-verbose]
```

The `-lib`, `-cell`, and `-view` are mandatory command-line options. The command-line options are not case-sensitive.

Using the Command-line Options

The following table describes the command-line options for the Connectivity-to-Schematic tool:

Option	Description
<code>-lib <libName></code>	Specifies the source library name.

Connectivity to Schematic User Guide

Working In Standalone Mode

Option	Description
<code>-cell <cellName></code>	Specifies the source cell name.
<code>-view <viewName></code>	Specifies the source view name.
<code>-param <optionsFileName></code>	Specifies the name of a <code>conn2sch</code> parameter file.
<code>-destlib <libName></code>	Specifies the destination library name.
<code>-destview <viewName></code>	Specifies the destination view name.
<code>-log <logFileName></code>	Specifies the log file to which the warning and error messages are written. The <code>-log</code> option takes precedence over the <code>log_file_name</code> variable used in the parameter file.
<code>-f</code>	<p>Specifies the file containing all the command line options.</p> <p><code>conn2sch -f conn2sch_cmd_opts</code></p> <p>where <code>conn2sch_cmd_opts</code> contains all the valid command line options.</p>
<code>-help</code>	Prints a description for the command line options.
<code>-version / -v</code>	Prints the version number.
<code>-over_dense</code>	Generates high-density schematics.
<code>+place_only</code>	Generates a quick unrouted schematic with connectivity indicated by name.
<code>-nosquare</code>	Does not square the schematic; that is, does not manipulate rows and columns of devices to convert a rectangular schematic into a square schematic.
<code>-min_crossovers</code>	Minimizes crossover of nets.
<code>-fast_labels</code>	Enables faster placement of labels. When this option is on, the Connectivity-to-Schematic tool labels segments at the midpoint and does not check for minimum overlap.
<code>+noxtrsch</code>	Does not extract the schematic; that is, does not find errors and warnings that have been written to the OpenAccess databases.
<code>-cdslib <fileName></code>	Specifies the <code>cds.lib</code> file to be used.

Connectivity to Schematic User Guide

Working In Standalone Mode

Option	Description
-verbose	Option to print detailed status messages during the schematic placement and routing process.
-asg	Specifies that an analog schematic is to be generated using analog place and route engine.

Creating the Parameter File

You can create a parameter file containing a list of all the options and parameter settings that are used by the Connectivity-to-Schematic tool to generate the schematics. This file can be specified with the *-param* command-line option while invoking the tool. The command to use a parameter file is:

```
conn2sch -param paramFile -lib testLib -cell top -view netlist
```

A sample parameter file, `parameter_file.txt`, is available in the `<install_directory>/doc/conn2schuser/example` directory.

Parameter File Format

Each line in the Connectivity-to-Schematic tool parameter file has a parameter specification statement. The syntax of the parameters follows the Backus Naur format:

```
parameter_file ::= {parameter_specification_statement}
```

Each parameter specification statement begins with the parameter name followed by the field separator, and then by the value or set of values for the parameter. Following is an example of a parameter specification statement:

```
parameter_specification_statement ::=  
parameter_name ps_seperator parameter_value
```

Parameter name could be any of the following:

```
parameter_name ::=  
ref_lib_list |  
dest_view_name |  
dest_symbol_view_name |  
generate_top_cell_symbol |  
log_file_name |  
import_if_exists |  
sheet_symbol |  
page_row_limit |  
page_col_limit |  
label_height |  
line_line_spacing |  
line_component_spacing |
```

Connectivity to Schematic User Guide

Working In Standalone Mode

```
density_level |
pin_placement |
power_symbol |
ground_symbol |
power_net |
ground_net |
noconn_symbol
```

The field separator is ‘:=’

```
ps_seperator ::=      :=
```

You can specify either one value or a set of values separated by a comma for the parameters, as follows:

```
parameter_value ::=      value, {value}
```

The data type of the value can be a string, an integer, or a real number.

```
value ::= string_value | integer_value | real_value
```

In addition, the following rules are applicable for the value:

- Any value that is not an integer or real is string.
- String does not require quotes. Trailing and preceding white spaces are automatically removed.
- White spaces within a string are preserved if the string is enclosed in quotes.

Following is a sample parameter file:

```
ref_lib_list := basic, sample, US_8ths
dest_view_name := schematic
dest_symbol_view_name := symbol
generate_top_cell_symbol := 1
log_file_name := ./conn2sch.log
import_if_exists := 0
sheet_symbol := none
page_row_limit := 1024
page_col_limit := 1024
label_height := 9
line_line_spacing := 0.200000
line_component_spacing := 0.500000
density_level := 100
power symbol := analogLib vdd symbol
ground_symbol := analogLib gnd symbol
pin_placement := all_sides
power_net := vdd!
ground_net := gnd!
```

Connectivity to Schematic User Guide

Working In Standalone Mode

```
noconn_symbol := basic noconn symbol
```

The default log file name is `conn2sch.log`. The log file name can be specified as an absolute path. By default, the log file is created in the current working directory. If a log file already exists with the same name, it overwrites the existing file.

Note: The command-line option `-log` takes precedence over the log file name specified in the parameter file.

Parameter File Options

The options used in the schematic generation parameter file are divided into three categories:

- General Parameter Options
- Schematic Generation Parameter Options
- Analog Schematic Generation Parameter Options

General Parameter Options

Option	Description
<code>ref_lib_list</code>	Specifies the reference libraries. Use blank space or a comma (,) to separate library names. <code>ref_lib_list := basic, sample, US_8ths</code>
<code>dest_view_name</code>	Specifies the view name to be used for the generated schematic cellview. The default view name is <code>schematic</code> . <code>dest_view_name := schematic</code>
<code>dest_symbol_view_name</code>	Specifies the view name to be used for the generated symbol view. The default view name is <code>symbol</code> . <code>dest_symbol_view_name := symbol</code>
<code>generate_top_cell_symbol</code>	Specifies if the symbol of the top cell must be created. The default value is 0. Set the value to 1 to generate the symbol of the top cell. <code>generate_top_cell_symbol := 1</code>

Connectivity to Schematic User Guide

Working In Standalone Mode

Option	Description
log_file_name	<p>Specifies a file that lists all error messages and log messages.</p> <pre>log_file_name := ./conn2sch.log</pre>
import_if_exists	<p>Specifies whether to import a cellview for which the schematic view already exists in the target library. When you set this value to 1, the cellview is imported. When you set this value to 0 (default), the cellview is skipped and is indicated in the log file.</p> <pre>import_if_exists := 0</pre>

Schematic Generation Parameter Options

Option	Description
sheet_symbol	<p>Specifies which sheet border size the Connectivity-to-Schematic tool applies when creating a multi-sheet schematic. The sheet borders reside in the <code>US_8ths</code> library. When this value is none, the Connectivity-to-Schematic tool generates an infinite sheet schematic.</p> <pre>sheet_symbol := none</pre>
page_row_limit	<p>Specifies the maximum number of rows on each sheet. Usually, the maximum number of rows is expressed as a number between 1 and 1024. If missing, the limit is automatically set to 1024.</p> <pre>page_row_limit := 1024</pre>
page_col_limit	<p>Specifies the maximum number of columns on each sheet. Usually the maximum number of columns is expressed as a number between 1 and 1024. If missing, the limit is automatically set to 1024.</p> <pre>page_col_limit := 1024</pre>
label_height	<p>Specifies the size of the font used for pin, wire, and instance labels. The value must be an integer greater than or equal to 1.</p> <pre>label_height := 9</pre>

Connectivity to Schematic User Guide

Working In Standalone Mode

Option	Description
line_line_spacing	<p>Specifies the spacing in inches between nets flowing in a channel. The value must be a real number in the range of 0.125 to 0.625.</p> <pre>line_line_spacing := 0.2</pre>
line_component_spacing	<p>Specifies the spacing in inches between a component and the nearest net flowing in a channel. The value must be a real number in the range of 0.125 to 0.625.</p> <pre>line_component_spacing := 0.5</pre>
density_level	<p>Specifies the density of the schematic. The input must be an integer from 0 to 100, where 100 is the most dense and 0 is the least dense.</p> <pre>density_level := 100</pre>
pin_placement	<p>Specifies the placement of pins on a symbol. Pin placement can be defined in the parameter file with any of the two values: <code>left_and_right_sides</code> or <code>all_sides</code>.</p> <p>For example:</p> <pre>pin_placement := left_and_right_sides</pre> <p>Alternatively, you can define placement of specific pins on specific sides of a module in a pin placement file, as described in the Pin Placement section. The pin placement file is referred in the parameter file as:</p> <pre>pin_placement := file, pinConn.pl</pre>
-noconn_symbol	<p>Connects the floating nets ports in the schematic with the <code>noconn</code> instance.</p> <p>For example:</p> <pre>noconn_symbol:= basic noconn symbol</pre>

Connectivity to Schematic User Guide

Working In Standalone Mode

Analog Schematic Generation Parameter Options

Option	Description
power_symbol	<p>Specifies a cellview, which you need to use as a master of a power cell instance.</p> <pre>power_symbol := analogLib vdd symbol</pre>
ground_symbol	<p>Specifies a cellview, which you want to use as a master of a ground cell instance.</p> <pre>ground_symbol := analogLib gnd symbol</pre>
power_net	<p>Specifies a power net, which is used to vertically align the instances.</p> <pre>power_net := vdd!</pre>
ground_net	<p>Specifies a ground net, which is used to vertically align the instances.</p> <pre>ground_net := gnd!</pre>

Log File Messages

All messages, whether they are warnings or error messages, are directed to the log file. You can specify the log file name with the *-log* command line option or specify it in the parameter file. The default log file name is `conn2sch.log`.

Common Messages

The following are some common messages that may appear in the log file:

- If you have specified all the arguments correctly, but the target library does not exist before starting `conn2sch`, then the following messages appear in the log file:
**conn2sch -lib src_lib -cell src_cell -view schematic -destLib dest_lib
-param conn2sch_params**

```
@(#) $CDS: conn2sch.exe version 6.1.2 08/02/2007 16:54 (cic612sun)
$ Tue Aug 7 10:36:21 2007

INFO (CONN2SCH-484): Target library dest_lib not found. Creating
a new target library ...

INFO (CONN2SCH-372): Checked-in schematic src_cell.

INFO (CONN2SCH-206): End of Logfile.
```
- If the *import_if_exists* variable is set to 0 in the parameter file, then the following messages appear in the log file.
**conn2sch -param conn2sch_params -lib src_lib -cell src_cell -view schematic
-destLib dest_lib**

```
@(#) $CDS: conn2sch.exe version 6.1.2 08/02/2007 16:54 (cic612sun)
$ Tue Aug 7 10:55:59 2007

ERROR (CONN2SCH-186): Cell src_cell and view schematic already
exists in the target library. Not importing because the option,
overwrite-if-exists, is not set to true.

INFO (CONN2SCH-206): End of Logfile.
```

Connectivity to Schematic User Guide

Log File Messages

- If the *import_if_exists* variable is set to 1 in the parameter file, then the following messages appear in the log file.

```
conn2sch -param conn2sch_params -lib srcLib -cell src_cell -view schematic  
-destLib destLib
```

```
@(#) $CDS: conn2sch.exe version 6.1.2 08/06/2007 15:12 (cic612sun)  
$ Tue Aug 7 11:18:12 2007
```

```
WARNING (CONN2SCH-185): Cell src_cell view schematic already  
exists in the target library. Overwriting this cell view because  
the option, overwrite-if-exists, is set to true.
```

```
INFO (CONN2SCH-372): Checked-in schematic src_cell.
```

```
INFO (CONN2SCH-206): End of Logfile.
```

Error Messages

The following are some common errors and the corresponding messages that appear in the log file.

- If the specified source cellview does not contain the instances and connectivity information, the following messages appear in the log file and Connectivity-to-Schematic exits:

```
conn2sch -lib srcLib -cell srcCell -view srcView -destLib destLib
```

```
@(#) $CDS: conn2sch.exe version 6.1.2 08/06/2007 15:12 (cic612sun)  
$ Tue Aug 7 11:22:58 2007
```

```
INFO (CONN2SCH-199): Source cell view NewCell has no instances.
```

```
ERROR (CONN2SCH-200): The cell view srcCell does not have  
connectivity information. Therefore unable to create schematic  
for this cell view. Exiting ...
```

```
INFO (CONN2SCH-206): End of Logfile.
```

- If you specify invalid arguments for the *-lib*, *-cell*, or *-view* options then the following messages appear in the log file.

```
conn2sch -param conn2sch_param -lib srcLib -cell srcCell -view schematic  
-destLib destLib
```

```
@(#) $CDS: conn2sch.exe version 6.1.2 08/06/2007 15:12 (cic612sun)  
$ Tue Aug 7 11:26:58 2007
```

```
ERROR (CONN2SCH-188): Could not open cell src_cell_new view  
schematic in library srcLib_new.
```

Connectivity to Schematic User Guide

Log File Messages

INFO (CONN2SCH-206): End of Logfile.

- If you do not specify the parameter file name with the `-param` option, then the following message appears on the terminal window.

conn2sch -param -lib src_lib -cell src_cell -view schematic -destLib dest_lib

```
@(#) $CDS: conn2sch.exe version 6.1.2 08/06/2007 15:12 (cic612sun)
```

```
$: (c) Copyright 1994-2006, Cadence Design Systems, Inc.
```

ERROR (CONN2SCH-3): Parameter file name has not been specified.
You must specify a valid parameter file name after the `-param` option.

Connectivity to Schematic User Guide

Log File Messages

Managing Index and Multi-Sheet Schematics

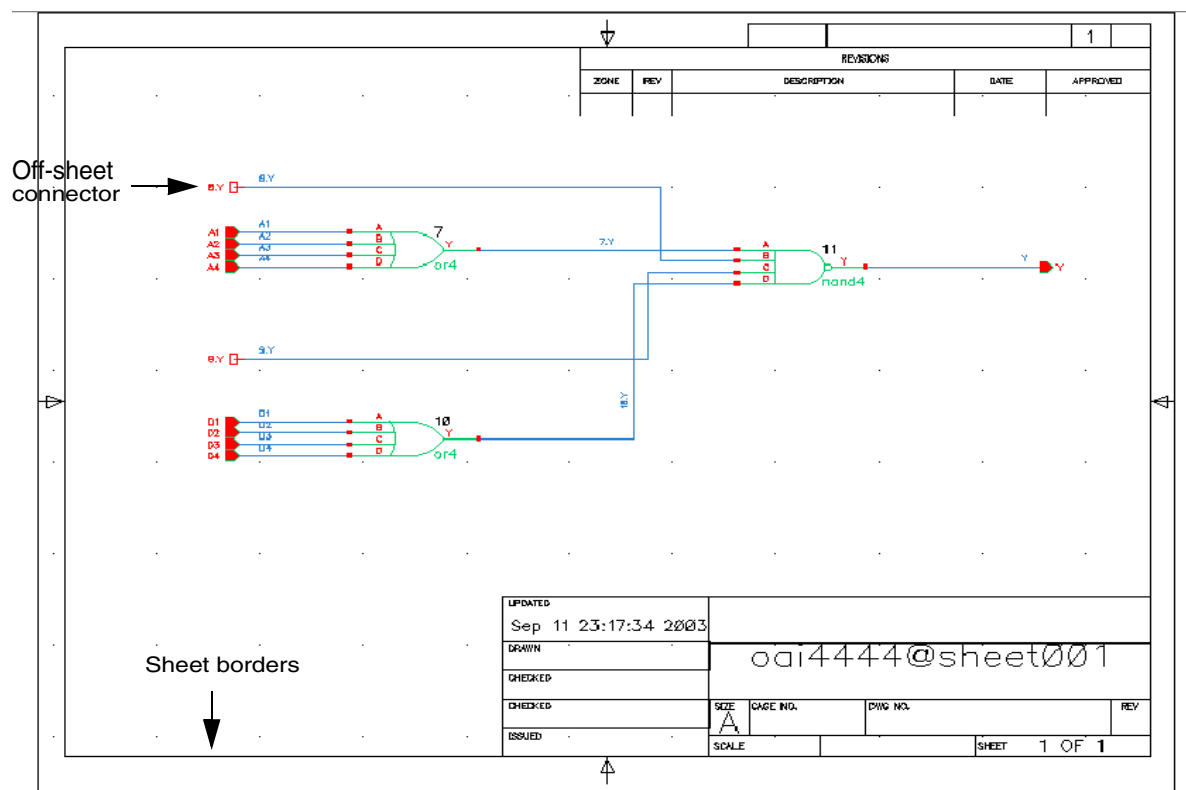
Managing Multi-Sheet Schematics

Connectivity-to-Schematic does not create multiple sheets when the source cellview is a multi-sheet schematic cellview. Therefore, all off-sheet connectors in the input are converted to cellview terminals. For example, if you provide a sheet of a multi-sheet schematic as input, then although you get a valid single-sheet schematic, the off-sheet connectors are converted to cellview terminals. Also, the generated schematic does not have any sheet border.

Connectivity to Schematic User Guide

Managing Index and Multi-Sheet Schematics

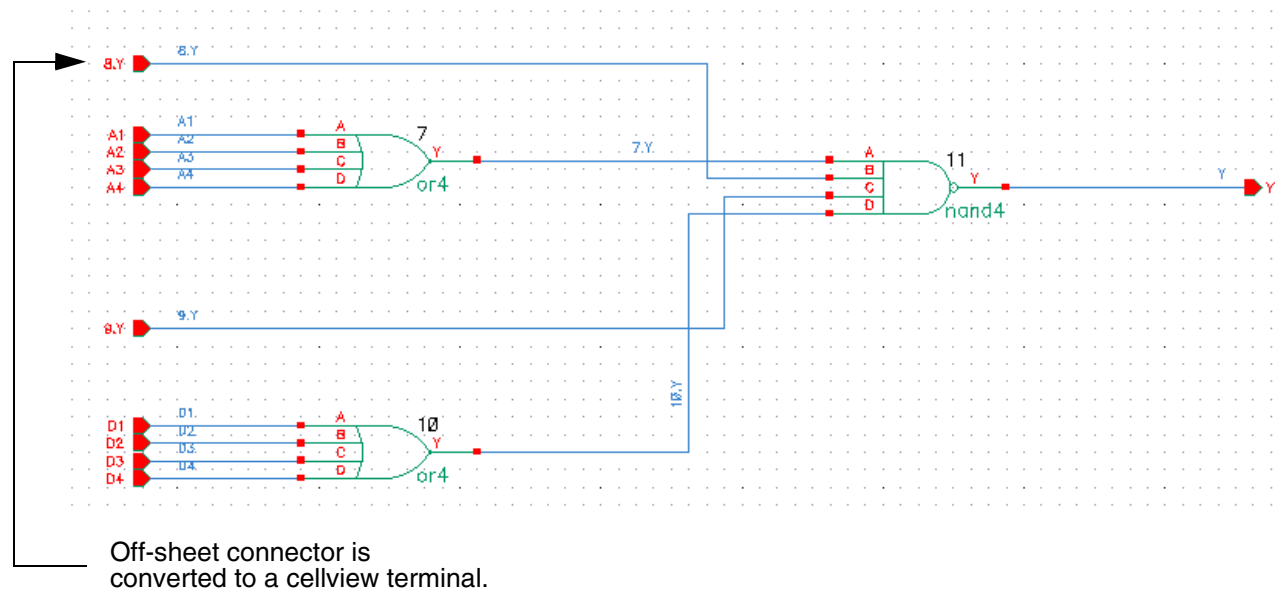
The following figure shows a sheet from a multi-sheet schematic that is provided as input to Connectivity-to-Schematic:



Connectivity to Schematic User Guide

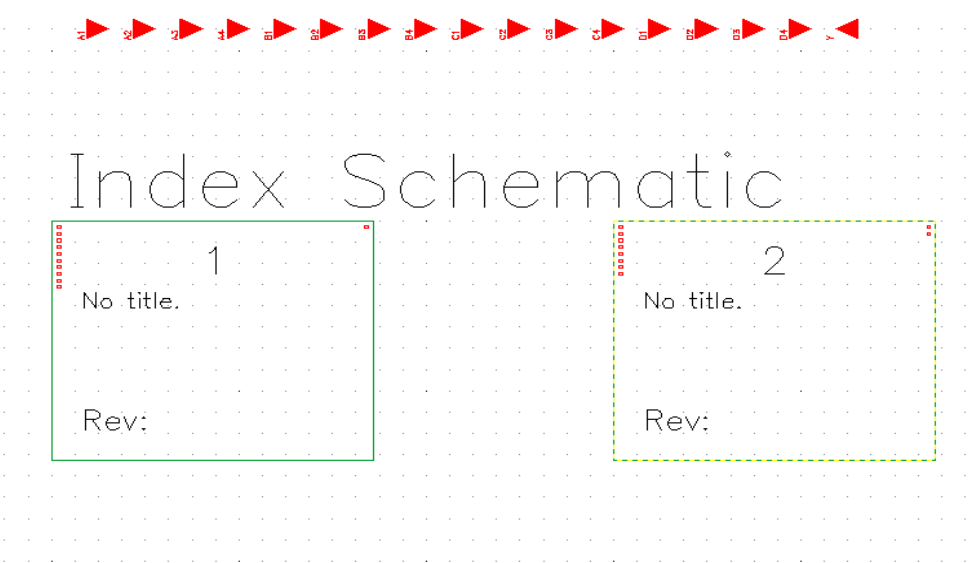
Managing Index and Multi-Sheet Schematics

The following figure shows the output that Connectivity-to-Schematic generates for the multi-sheet schematic:



Managing Index Schematics

If you provide an index schematic as input, then all the m-symbols are converted into a schematic representation with wires and pins. The following shows the index schematic that is provided as input to Connectivity-to-Schematic:

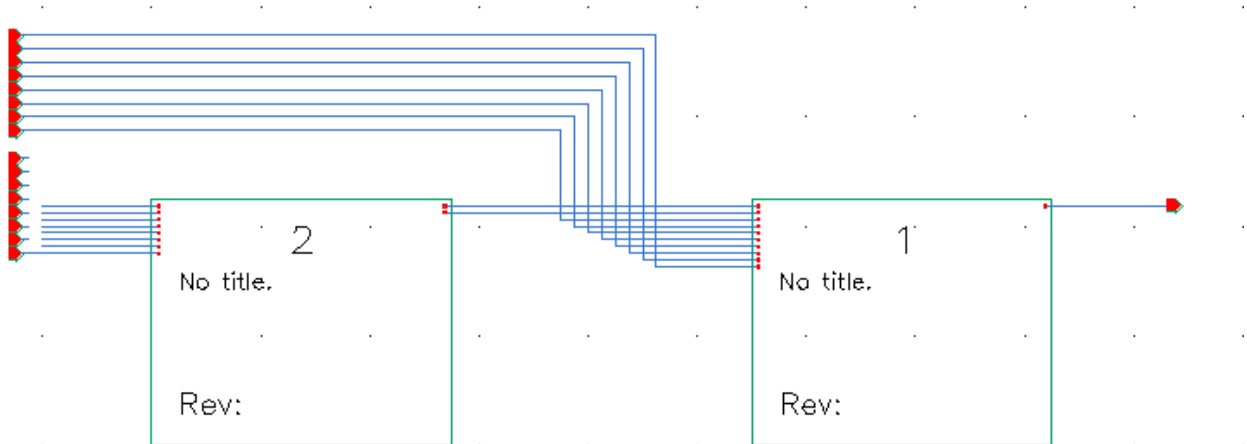


Index sheet as input to the Connectivity-to-Schematic tool

Connectivity to Schematic User Guide

Managing Index and Multi-Sheet Schematics

The following shows the output that Connectivity-to-Schematic generates for an index schematic:



m-symbols of an Index sheet are wired in the generated schematic.