cadence®

Allegro Design Entry HDL Utilities User Guide

Product Version 23.1
September 23.1

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

Contents

Preface

About This Manual

This manual describes the various utilities that are used with Allegro Design Entry HDL. This manual is for Design Entry HDL users.

Finding Information in This Manual

The following table summarizes the topics covered in this manual.
.

If you want to know more about	Read...
An overview of the various Design Entry HDL utilities.	Chapter 1, “Overview”
Archiving a design	Chapter 2, “Archiving a Design”
Generating BOM reports	Chapter 3, “Generating BOM Reports”
Cross-referencing a design	Chapter 4, “Cross-Referencing a Design”
The user interface of Archiver	Appendix A, “Archiver Dialog Boxes”
The user interface of BOM	Appendix B, “BOM-HDL Dialog Boxes”
The user interface of CRefer	Appendix C, “CRefer Dialog Boxes”
The user interface of IFF2HDL	Appendix D, “IFF2HDL Dialog Boxes”

Related Documentation

The following manuals give you information about other tools used during the digital simulation process:

If you want to know...	Read...
How to use Design Entry HDL to enter schematics	*Allegro Design Entry HDL User Guide*
More about Design Entry HDL digital libraries	*Allegro Design Entry HDL Libraries Reference*
More about properties supported by Cadence PCB design software.	*Allegro Platform Properties Reference*

1

Overview

About Design Entry HDL

Design Entry HDL is a design environment that supports behavioral and structural design descriptions captured in text and graphics. It incorporates block editing functions for quick architectural design.

Design Entry HDL organizes schematic information into pages. It captures and displays only one page of schematic information at a time. Design Entry HDL is a by-reference editor because it references all parts in the schematic from various libraries that reside at the reference or local area. A standalone, self-contained database of the libraries and the design can be created using the Archiver utility.

Design Entry HDL Features

The features of Design Entry HDL are listed below:

- A top-down (hierarchical) design that lets you quickly draw blocks and connect wires between blocks. A cross-view generator (Genview) to create blocks from HDL descriptions or automatically generate HDL text from high-level diagrams.
- A customizable user interface that lets you customize menus and toolbars, map keys to functions, and create new commands.
- A hierarchy editor lets you view the structure of your design.
- An attribute editor that lets you annotate properties on a design to drive the physical layout.
- Integration with the Design Synchronization toolset. This toolset lets you view differences between your schematic and the board layout and then synchronize them.
- Cross-probing between Design Entry HDL and other Cadence tools.

- Support for design reuse. You can associate logical components with a layout section to create reusable components. This component can be reused in other areas in your design and also in the designs you create later.
- Integration with CheckPlus, an advanced rule checking and rule development system.
- Integration with Allegro Constraint Manager tool that allows you to capture and manage electrical constraints as you implement logic.
- Support for importing Intermediate File Format (IFF) files that can be created for radio frequency (RF) designs. You can create radio-frequency (RF) designs using tools such as ADS or MDS by Agilent Technologies, Inc. The ADS tool supports the creation of Intermediate File Format (IFF) files. Once the RF design is ready, you can create IFF files for the schematic and layout of the design. You can import a schematic IFF file into Design Entry HDL to transfer the graphics and connectivity data of the RF design into Design Entry HDL and then use the RF design as a block in a larger Design Entry HDL design.
- Design Entry SKILL, the SKILL programming interface to Design Entry HDL.

Design Entry HDL Utilities

The utilities of Design Entry HDL are

- Archiver
- CRefer
- BOM
- IFF2HDL
- PIC
- Design Entry EDIF
- Variant Editor
- Design Synchronization toolset
- PXL
- VDD

This book covers Archiver, CRefer, and BOM in separate chapters.

This chapter briefly covers the following utilities:

- Design Entry EDIF 3 0 0
- Packager-XL
- Variant Editor
- Design Synchronization Toolset
- IFF2HDL

Design Entry EDIF 3 0 0

Design Entry EDIF 300 Schematic Reader translates schematic data in an EDIF 300 file into the Cadence database. When you translate EDIF 300 schematic data with EDIF 300 Schematic Reader, you preserve both the graphical representation of the schematic and the connectivity information.

The Design Entry EDIF 300 Writer is used to translate schematic design data files from the Cadence database into the EDIF 300 format. When you translate schematic data with EDIF

300 Writer, you can preserve both the graphical representation and the connectivity information of the schematic.

For more information, see the Cadence document *Allegro Design Entry EDIF 300 User Guide*.

Packager-XL

Packaging involves translating your logical design (schematic) captured using Design Entry HDL into a physical design ready for placement and routing using Allegro PCB Editor.

Packager-XL is an interface between the logical design and the physical layout for the Cadence Board Design Solution.

Packager-XL has two modes of operation:

- Forward mode—Packager-XL translates a logical design entered in Design Entry into a physical design ready for layout by PCB Editor.
- Feedback mode—Packager-XL receives changes made in PCB Editor and incorporates these changes into the logical design.

Packager-XL uses the standard Hardware Description Language (HDL) naming conventions to simplify interlude communication. The library structure used is based on the Library-Cell-View and is common across all Cadence solutions.

Packager-XL places HDL-based netlist files in a packaged view within the design cell view. For more information, see the Cadence document *Packager-XL Reference*.

Variant Editor

The Design Variance solution lets you create and manage designs that are different from each other by small differences. The different designs are called variants. Each variant includes a common base design consisting of a set of core elements. However, these variants have small differences caused by the presence or absence of a component, or because of minor differences in a component's properties, such as resistance or voltage.

A new GUI based tool called Variant Editor is available to simplify the process of managing design variants and generating Bill of Material (BOM) reports.

Variant Editor allows you to:

- Use the Physical Part table (PPT) driven data for defining variant component values.

- Generate the Bill of Material (BOM) that reflects the electrical stuff list for a variant.
- Generate a delta list of components from the base design for a variant.
- Generate a comparative BOM of different variants.
- Generate BOM reports in multiple formats, such as spreadsheet format and HTML format.
- Annotate variant data from the variant database into the schematic.
- Generate the interface file that is read by PCB Editor to create variant assembly drawings.
- Cross-probe with the Design Entry HDL.
- Filter components in the Component List.
- Support associated mechanical parts and culottes.
- Support global find of specific components.
- Synchronize the changes made in the variant database with the changes made in the original schematic.
- Replace an existing component with another component that has a different name or a non-compatible footprint.

For more information, see the Cadence documents *Design Variance User Guide* and *Design Variance Tutorial*.

Design Synchronization Toolset

The primary need for synchronization is caused by changes that occur either in the board or in the schematic after the initial transfer of packaged information to the board. The changes that occur in the board after the initial transfer of packaged information from the schematic are of the following four types:

Component changes

You may add new components in the design to handle signal integrity and electromagnetic compatibility problems. These components may include termination resistors, series or shunt buffers, and bypass capacitors.

Connectivity changes

You may make connectivity changes to facilitate routing after the initial placement of components. Connectivity changes may be caused by pin swaps, section swaps, and reference designator (refdes) swaps.

Reference designator changes

You may change reference designators to debug board problems.

Property changes

You may modify certain components in the board. These modifications will cause property changes.

The Design Synchronization toolset includes Design Differences and Design Association.

The Design Differences tool allows you to:

- View a list of differences between the logical (Design Entry HDL schematic) and physical (PCB Editor) databases.
- Customize the display to view differences of your interest by filtering out specific differences.
- View the objects in the entire logical and physical design as a hierarchical tree composed of components, nets, and parts.
- Query on a design by part name, reference designator, net name, and property name-value pairs.
- Cross-probe instances, nets, and pins on the Design Entry HDL schematic design or PCB Editor layout design to display the source of the difference.
- Synchronize either the logical or physical design after viewing and accepting the differences.
- Generate a Marker file to backannotate physical connectivity changes to the Design Entry HDL schematic using the Design Association tool.

The Design Association tool allows you to:

- Navigate the Design Entry HDL schematic. You can perform a design editing session to update and synchronize the logical Design Entry HDL schematic design with the corresponding physical layout drawing.
- Navigate to the individual pages of the schematic and prompt you with the connectivity changes and design changes that need to occur.

The Design Association tool communicates with the Design Entry HDL schematic through Design Entry HDL SKILL, executes the function related to each action, and updates the Design Entry HDL schematic design.

Note: For more information, see the Cadence document *Design Synchronization and Packaging User Guide*.

IFF2HDL

Design Entry HDL and PCB Editor support importing Intermediate File Format (IFF) files that can be created for radio-frequency (RF) designs. Design Entry HDL supports an IFF interface for importing a schematic IFF file that you can create for your RF schematic, and PCB Editor supports an IFF interface for importing a layout IFF file that you can create for the layout of your RF design.

You can create a radio-frequency (RF) design using tools such as ADS or MDS by Agilent Technologies, Inc. The ADS tool supports the creation of Intermediate File Format (IFF) files. Once the RF design is ready, you can create IFF files for the schematic and layout of the design. These files can then be used to transfer your design information into Design Entry HDL and PCB Editor.

You can get both graphics and connectivity data of the RF design into Design Entry HDL and then use the RF design as a block in a larger Design Entry HDL design. The interface also gives you the advantage of making concurrent changes in an RF design and then re-importing it into a larger design in Design Entry HDL and PCB Editor.

Note: For more information, see the chapter Importing Radio-Frequency Designs in the Cadence document *PCB Flows*.

2

Archiving a Design

About Archiver

Any project that you create can use two types of libraries: local and reference. Local libraries are stored in the project directory. Reference libraries are called from a central location. The project directory contains the `cds.lib` file, which includes links to the reference libraries used by the design.

If you need to send a design to someone in a different geographical location, you will have to send the design along with all the libraries that it references. Each reference library contains thousands of parts but your design may have only a few parts. Using Archiver, a Cadence tool, you can create a standalone design that includes only the relevant parts used in a design from the reference libraries. For example, if your project uses only the `ls00` part from the `lsttl` reference library, Archiver will copy only `ls00` to the archive and not the entire `lsttl` reference library.

Archiver is also useful because reference libraries are updated from time to time. This might cause situations where your design references parts that have changed and, thereby, impact the functionality of the design. To avoid this, you can use Archiver to refresh your design.

Archiver allows you to archive a design and all the parts that it uses without archiving entire reference libraries. Archiver identifies every physical part in a design and then creates a local library containing only those parts. Archiving only those parts used in the design significantly reduces the amount of data that is to be archived, and considerably simplifies archiving and restoration processes. An archived design can be used independently of reference libraries and other reference data. You can use this feature to store the design on a tape and use it later.

Creating an archive is a one-time task. You can later open the archive using Archiver and identify the differences in parts by comparing the archived and reference libraries. If differences exist, you can update the archive with the new reference library parts. Archiver also generates a list of unused parts in the archive. If required, you can delete these parts.

Creating a New Archive

To create a new archive, you must access the Archiver dialog box. The Archiver dialog box lets you specify the project file or directory to be archived, along with the output directory. By default, the root design specified in the selected `.cpm` file is archived. You can archive all designs in the library.

You can also define the views that Archiver should traverse to archive the design. You can select the *Schematic* option, and specify Verilog or VHDL as the view to be traversed, or select the *All* option to specify that all views need to be traversed.

You can open this dialog box in one of the following ways:

- Open a project that you want to be archived in Allegro Project Manager, and select *Tools – New Archive*.
- Double-click on the `archiver.exe` file in Windows Explorer.
- Enter `archiver -proj <project_file>` from the command prompt.

To create a new archive, do the following:

1. Open the Archiver dialog box.
2. Specify the name of the project you want to archive in the *Project File* field. Its location is automatically displayed in the *Location* field.
3. Specify the directory for saving the Archiver output in the *Output Directory* field.
4. Specify the views that Archiver should traverse to archive the design in the *Views To Traverse* field.

   Note: See "Using Views to Traverse Example" on page 21 to understand this option better.
5. Select the files and directories to be archived in addition to the specified project file in the *Other Files/Directories To Archive* list.
6. If you want the entire archive to be created as a single file, select the *Create Single File Archive* check box.
7. When you select the *Create Single File Archive* check box, the *Delete Archived Directory* check box becomes active. You can select this check box to delete the archived directory.
8. If you want to archive all your root designs, select the *Archive All Designs* check box.

9. When you select the *Create Single File Archive* check box, the *Compression Utility* field becomes active. It displays the zip or tar command based on the operating system on your computer.

   The compression command displayed in a Windows-based system is as follows:

   ```
   zip -r $archive $location
   ```

   The compression command displayed in a UNIX-based system is as follows:

   ```
   tar -cvf $archive $location
   ```

   Here, `$archive` represents the name of the zip file, while `$location` represents the location where the output file will be saved. This location is the same as the value stored in the *Output Directory* field.

   You can change the `$archive` or `$location` variables and customize the path where the compressed archive file will be created. However, you do not necessarily need to change the values in the *Compression Utility* field.

   Note: The zip or tar utility must be installed on your machine.

10. Click on the *OK* button to start archiving the design. Archiver will create the archived directory and a compressed single file archive (named `<root design>_archive.zip` or `<root design>_archive.tar`) for the selected design. The Archiver Progress window appears when you click on this button.

    Note: This button is disabled if you do not specify the project file or the output directory.

Note: For more details, see Archiver Dialog Box in Appendix A.

Note: A check box is included in the Archiver dialog box, which checks if you want to proceed with the archival of templates for PSpice-enabled projects. A command line switch is also provided for batch runs, to enable or disable archiving templates.

Using Views to Traverse Example

Assume that you have a root design `top`, which has two views: `sch_1` and `vlog_model`. In the `sch_1` view, assume the bottom cell is instantiated and in the `vlog_model` view, the `bottom1` cell is instantiated. This configuration now has two possible hierarchies: top->bottom (traversing sch_1) or top->bottom1 (traversing vlog_model).

By default, the *Views To Traverse* option is set to *All*. This selection will cause Archiver to traverse both `sch_1` and `vlog_model` views and the 3 cells: `top`, `bottom`, and `bottom1` are archived.

Now, if the *Views To Traverse* option is set to *Schematic*, only the `top` and `bottom` cells are archived. In all cases, all views of the cells are archived. Therefore, if `bottom` has a packaged view, it will always be archived. In short, the *Views To Traverse* option only determines the traversed cells. It does not control which views are archived.

Opening an Archive

The Archiver Open dialog box lets you specify the archive to be opened. It displays the list of archived and reference libraries in the archive. You can check the reference library locations to ensure whether they are correct. The Archiver Open dialog box also displays the location of the `cds.lib` file. You can change this location if the `cds.lib` file containing the reference libraries has changed. You can open this dialog box in one of the following ways:

- Open a project that you want to be archived in Project Manager, and select *Tools – Open Archive*.
- Double-click on the `archopen.exe` file in Windows Explorer.
- Enter `archopen -proj <project_file>` from the command prompt.

To open an existing archive, do the following:

1. Open the Archiver Open dialog box.
2. Specify the name of the project you want to archive in the *Project* field. Its location is automatically displayed in the *Location* field. A list of the libraries in the archive libraries and the corresponding reference libraries is also displayed.
3. If you want to change the reference libraries displayed in the list, you can specify the path to a different library in the `cds.lib` field. You can either type the path or use the *Browse* button on the right of the `cds.lib` field to select another `cds.lib` file.
4. If you want to list the reference cells that are different from the archived cells, select the *Compare archive and reference libraries (Diff)* button. The DIFF Utility dialog box appears. See DIFF Utility Dialog Box in Appendix A for more details.
5. Click the *Apply* button.
6. When you are done making changes to a design extracted from an archive, click the *Update Archive* button to commit your changes and recreate the archive. When you select this button, Archiver detects the changes made to the design and acts in the following ways:
    a. If there are new components in the design, Archiver copies these components from the reference libraries to the present archive.

b. If there are unused components in the archive that are not used in the design, Archiver displays the Delete Unused Components dialog box from where you can selectively delete the unwanted components. See Delete Unused Components Dialog Box in Appendix A for more information.

Update Archive does not change these components. To change the components, run the DIFF utility and selectively change the cells before you click the *Update Archive* button.

7. Click the *OK* button to close the Archiver Open dialog box.

Note: For more details, see Archiver Open Dialog Box in Appendix A.

Archiving a Design from the Command Prompt

You can create an archive of a design from the command prompt by using the `archcore` utility. The syntax for the `archcore` utility is as follows:

```
archcore
-proj <project>
-path <output_path>
[-alldesigns]
[-views <view1> <view2>...]
[-f <file_name>]
[-compresscmd <compress command>]
[-delarchivedir]
[-ignorefile <list of file names, folder names or file extentions separated by white spaces>]
```

`-proj <project>`	Specifies the name of the project file `.cpm` to be archived.
`-path <output_path>`	Specifies the path of the output directory where the design is to be archived.
`-alldesigns`	Specifies that all root designs should be archived. By default, Archiver traverses only the root design specified in the `.cpm` file. If all root designs need to be archived, the `-alldesigns` option needs to be specified.

`-views <view1> <view2>…`	Specifies the views to traverse for archiving the design. You can specify the view as any combination of Schematic, Verilog, or VHDL. If you do not specify any view, Archiver traverses all views.
`-f <file_name>`	Specifies the name of the file that contains the list of directories and files to be archived along with the design. The `file_name` should specify the fully qualified path of the file.
`-compresscmd <compress command`	Specifies the compress command to be used in order to compress the archive. Specify the compresscmd parameter as: `"cdszip -r $archive $location"` This parameter must be enclosed within double-quotes.
`-delarchivedir`	Indicates that the archive directory should be deleted after creating the tar file
`[-ignorefile <list of file names, folder names or file extentions separated by white spaces>]`	Specify the file names, file extensions, or folder names that you want Archiver to ignore while archiving. The values must be enclosed within double-quotes and separated by spaces. This parameter does not accept full paths to files. You can also use wild card characters. For example "`*.log *.mkr pxl.log physical`" are all valid values for this parameter.

Note: If you archive a design using the `archcore` utility with the `-f` parameter specifying a file containing a list of all the files and directories to be archived, the file or directory last file or directory is not archived.

For example

```
archcore -proj <cpm> -path <output path> -alldesigns -f file.txt
```

If `file.txt` includes a new line at the end of the file, the last defined file or directory listed in `file.txt` will be archived.

Archived Directory Structure

When you archive your design, the archived design is stored in the directory <Projectname>_archive. You can specify the path of the <Projectname>_archive directory in the Output Directory field in the Archiver dialog box.

The <Projectname>_archive directory contains the following files or directories:

- The .cpm file
- Archiver.log file
- Cds.lib file
- Directories containing parts used by the design. (If your project uses parts from reference libraries, Archiver creates a directory named archive_libs and stores all parts in it.)
- Any other files and directories specified by you to be copied with the design in the Other Files/Directories To Archive list.

With the 16.6 release, the signal model files are also included in the archived project. To archive signal model files, define the directive `ARCHIVE_SI_MODEL_FILES` in the project cpm. When this directive is defined, a new folder ‘arch_model_files’ is created in the archived project location and all the model files are archived into this folder.

Note:

In Solaris- or Unix- based systems, Archiver preserves the date-time stamp of all archived files with the exception of the files that it modifies. The files that Archiver modifies include cds.lib and *.cpm. The <Projectname>_archive directory also contains the new date-time stamp.

Note: Due to the limitations of Windows-based systems, the date-time stamps of all files are changed by Archiver.

3

Generating BOM Reports

After you have packaged a design, you require a report that can help you place an order for the required components. This report is popularly called the Bill of Materials (BOM). A BOM report lists all the components used in a design along with the part numbers and values of the different properties of each component. You can specify the properties to be displayed in a BOM report. If a particular property does not apply to a component, the field corresponding to that property for the component is left blank in the BOM report.

You can generate three types of BOM reports.

1. The BOM report for the base schematic

   The base schematic BOM report contains the listing of all the components used in the base schematic. All the property values including the part number correspond to the values chosen in the base schematic.

2. ABOM report for any of the variants that you have created

   A variant BOM report contains the listing of all the components used in a particular variant. All the property values including the part number correspond to the values selected in the particular variant.

3. A part-number based comparison BOM report

   The part-number based comparison BOM report provides a part number-based comparison between the components of the base schematic and all the variants. While generating the comparison BOM report, only the preferred values of components and alternate groups are considered.

The BOM-HDL tool lets you generate a bill of materials report for a design. Before running the BOM-HDL tool, you must first package your design. Packager-XL generates physical netlist (`pstchip.dat`, `pstxnet.dat`, `pstxprt.dat`) files that contain physical information about each part, the connectivity information about each net, and the information about how logical components on the schematic correlate to the physical reference designators and section assignments in the board.

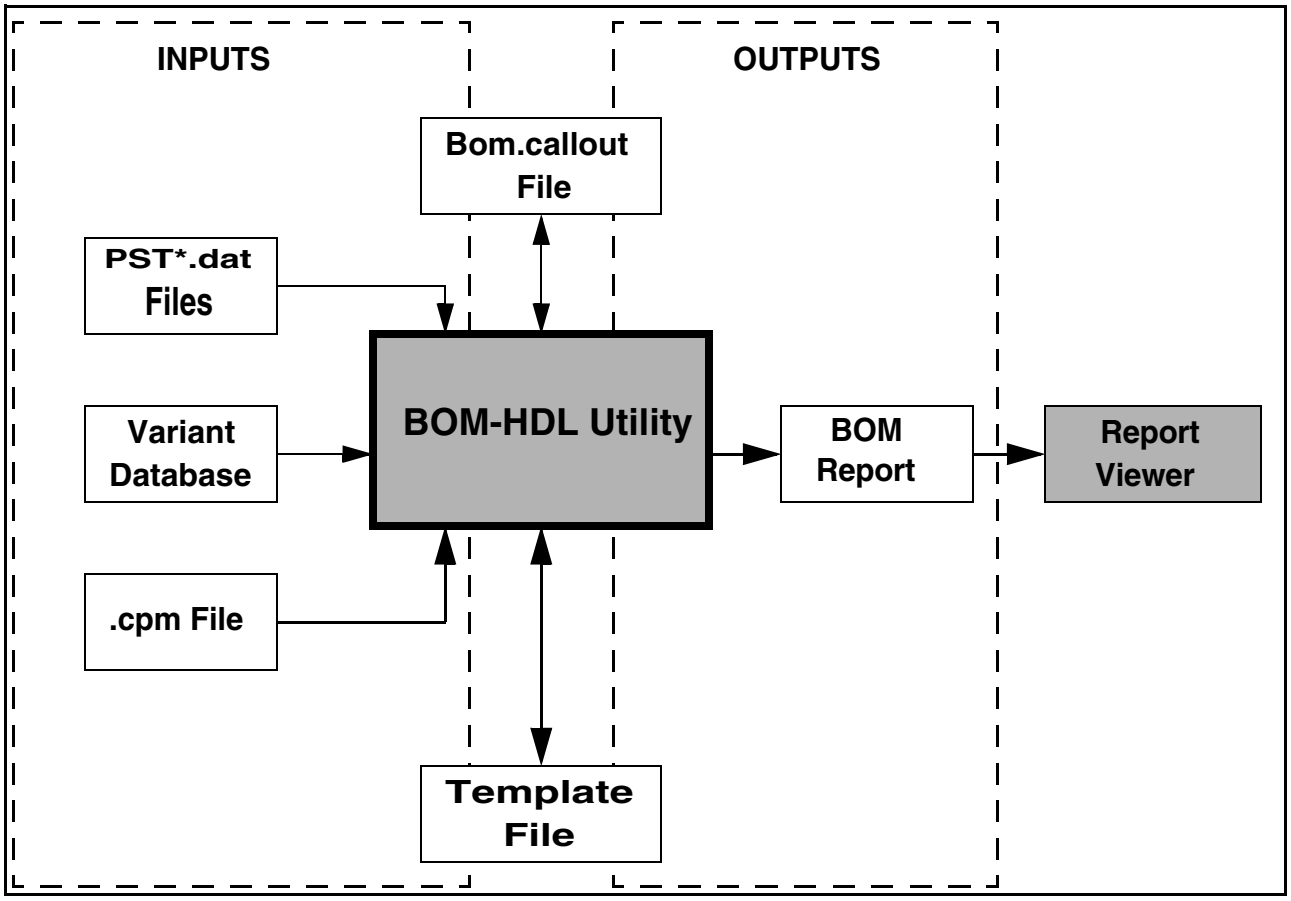
The BOM-HDL tool uses the information in the netlist files to generate BOM reports. The format of the BOM reports is specified by the BOM template file.

Understanding How BOM-HDL Works

BOM-HDL accepts inputs from five sources (project file, netlist files, the BOM template file, the variant database, and the callout file) to generate BOM reports.

The BOM-HDL: Functional Diagram figure on page 28 explains the working of BOM-HDL.

Figure 3-1 BOM-HDL: Functional Diagram



The Input Side

At the input side, BOM-HDL uses the following files to gather information:

1. Project file (`.cpm`) - BOM-HDL checks for the BOM-HDL section in the project file to find information about whether or not BOM-HDL had been previously used. The options used to generate BOM report are seeded in the project file. This file also stores the path to the template file.

   Note: If BOM-HDL is being used for the first time, the project file will not contain any

BOM-HDL section. The BOM-HDL section will be created in the project file only on first use of the tool.

2. Netlist (`pst*.dat`) files - BOM-HDL requires the netlist files (`pstchip.dat`, `pstxprt.dat`, and `pstxnet.dat`) to obtain property and connectivity information about the design. To generate netlist files, you need to package the design.
3. Variant database - If you want to generate BOM reports for variants, you need the variant database (`variant.dat`) as an input.
4. Template file - BOM-HDL uses the template file to determine the properties to be displayed in the BOM report. The template file also lets you manage the look and feel of the BOM report. To customize the look and feel of the BOM report, you need to change the information in the template file. The information that you can customize includes formatting information (the properties to be displayed), callout information, and report header information.
5. Callouts file - BOM-HDL uses the callouts (`bom.callout`) file to obtain information about the callouts. Callouts are mechanical parts, such as nuts and screws, which have no graphical representation in the schematic. The properties stored in the Callouts file are included in the BOM report.

   You can use BOM-HDL to edit the Callouts file. To edit the Callouts file, you need to specify it in the Customize Template dialog box, and then use the `Edit` button to edit it. You can add or remove mechanical parts, and specify their quantity.

The Output Side

After accepting inputs, BOM-HDL processes the information and generates BOM reports. The output formats supported by BOM-HDL are:

- An ASCII text file with information in a tabular format.
- Spreadsheet format with a user-specified delimiter such as comma or colon.
- HTML format

A BOM report lists all the properties mentioned in the template file. The values for the properties are obtained from the netlist and callout files.

If you have generated the BOM report in the HTML format, you need to have an Internet browser such as Netscape Communicator or Microsoft Internet Explorer to view the BOM report.

Prerequisites for Running BOM-HDL

Before you run BOM-HDL, you need:

- The BOM template file to determine the properties to be displayed in the BOM report. BOM-HDL searches for the BOM template file in the following order:
  - Checks for the path specified by the `-t` option - If you are running BOM-HDL for the first time, use the `-t` option to specify the path to the template file. If the relative path to the template file is specified, BOM-HDL searches the file in the specified path relative to the `bom` view of the design. If the path is invalid, BOM-HDL displays an error message.
  - If the template file is not found in the user-defined path, BOM-HDL searches for the template file in the project file. If you have run BOM-HDL once, the path to the template file is saved in the project file. BOM-HDL uses this path to load the template file.
  - If the template file is not found in the user-defined path, BOM-HDL searches for the template file in the locations specified in the `setup.loc` file and loads the template file from the location where it is found first.
- To package your design to ensure that you have the required netlist files: `pstchip.dat`, `pstxprt.dat`, and `pstxnet.dat`.

Launching BOM-HDL

You can open BOM-HDL in one of the following ways:

- From the *Tools* menu of Variant Editor, choose *Generate Report*.
- From the *Tools* menu of the Project Manager, first choose *Packager Utilities* and then choose the *Bill of Materials* command.
- From the *Tools* menu of Design Entry HDL, first choose *Packager Utilities* and then run the *Bill of Materials* command.
- Specify the `bomhdl -proj` ... command at the command prompt. For more information, see the topic Creating a BOM Report from the Command Prompt.

Creating the BOM for the Base Schematic

To manage inventory for creating designs, you might require a report that lists all the components in the base schematic. You can use BOM-HDL to generate the BOM report for

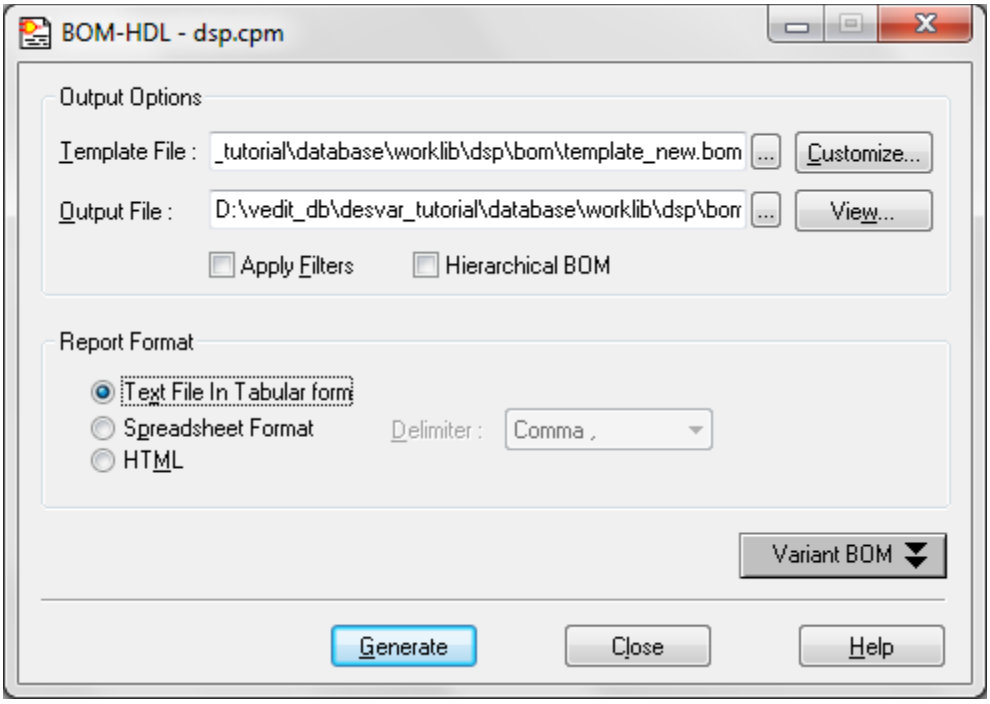
the base schematic, which lists all the components in the design along with their properties. The properties displayed in the report are determined by the BOM template file. You can customize the properties listed in the BOM report.

Creating the Base Schematic BOM

1. Open the BOM-HDL dialog box by selecting the *Generate Report* command from the *Tools* menu of Variant Editor.

   The BOM-HDL dialog box appears and the *Template File* and the *Output File* fields are seeded with the default file names. See Prerequisites for Running BOM-HDL on page 30 for more information about the template file.

Figure 3-2 BOM-HDL Dialog Box



Note: If you have used BOM-HDL to generate variant BOM reports in a previous run, BOM-HDL will display the variant options as in the BOM-HDL Dialog Box Displaying Variant Options figure on page 34.

2. If you want to customize the template file before generating BOM reports, click on the *Customize* button to display the Customize Template dialog box. You can use this dialog box to change the properties that will be displayed in the BOM report. You can also

change the general look and feel of the report. After making the required changes, save the template file and close the Customize Template dialog box.

3. The default path to the output file is seeded in the *Output File* field. If you want to change the path or file name, use the browse button to define the new path. You can also enter the new path.

4. By default, BOM-HDL generates base schematic reports. However, if you have generated a variant BOM report or a variant comparison BOM report in the last session, the corresponding radio button will be selected by default. If the *Base Schematic BOM* radio button is not selected, click on it to select it.

   All BOM reports are created in the `bom` view of the top-level design. BOM-HDL assigns default names to BOMs, which can be viewed and changed in the *Output File* field. Notice that the name assigned to the BOM report is `BOM.rpt`.

   Note: If your design does not have any variant data, the *Variant BOM* and *Variant Comparison BOM* options are disabled.

5. By default, the BOM report is generated as a text file. You can change the format of the BOM report to spreadsheet or HTML. If you choose the spreadsheet format, a letter such as comma or semicolon is used as delimiter. You can specify the delimiter by selecting it in the *Delimiter* field. To change the report format, you can choose the *Spreadsheet Format* or *HTML* radio button.

   Note: If you choose the HTML format, the name of the output file changes to `BOM.html`.

6. By default, the output file is generated in the BOM view of the project. To change the path of the output file, specify the new path in the *Output File* field. Alternatively, you can browse to the new path.

7. To generate the BOM report, click *Generate*.

   The BOM report is created in the `bom` view.

   A message box appears informing that the BOM report is generated. You are asked whether you want to view the report.

8. Click on the *Yes* button to view the BOM report.

   If you have selected the *Text File in Tabular form* or *Spreadsheet Format* option, the file appears in a text editor. If you have selected the *HTML* option, the file appears in the default HTML browser.

Creating Variant BOM Reports

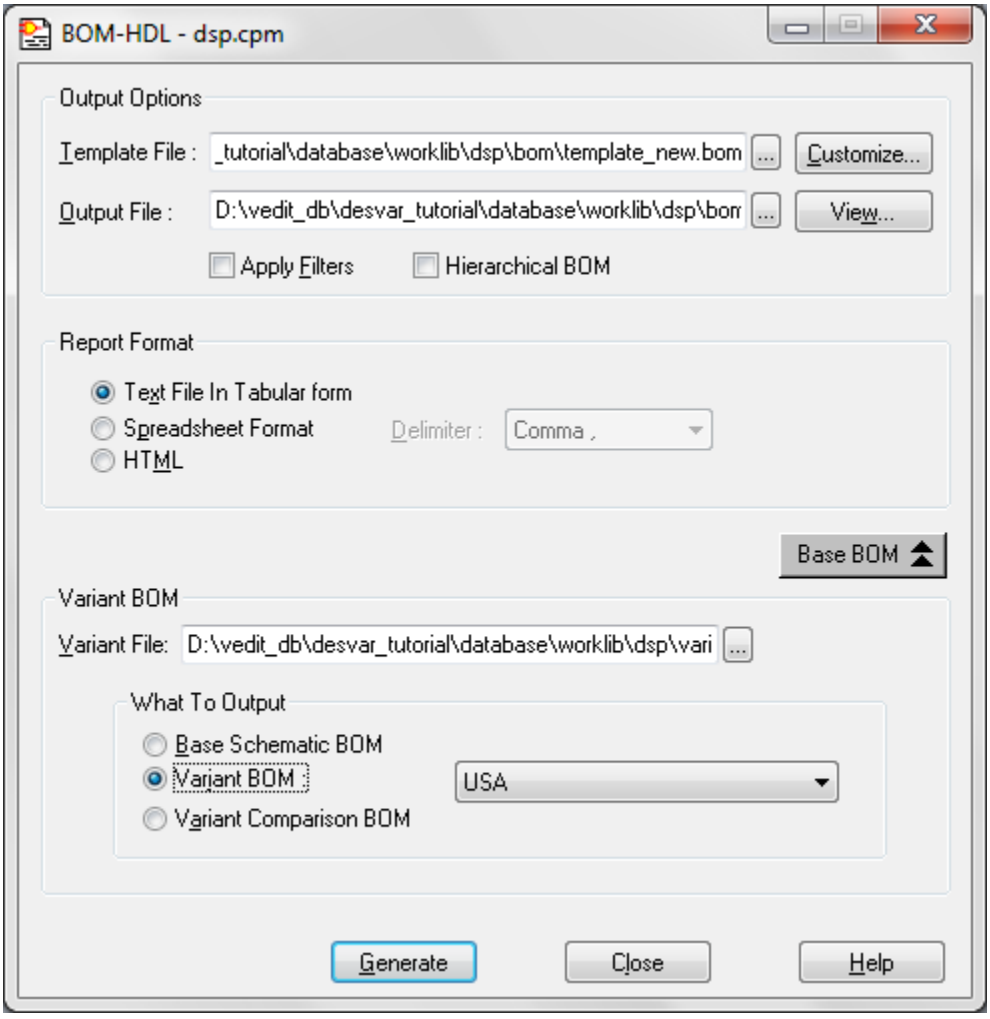
If you have a design with variants, you can use the *Variant BOM* radio button to generate BOM reports for individual variants. You can also create BOM reports for all variants simultaneously. These reports also let you know whether the preferred values of the components for a variant are the same as the values in the base schematic. You can also display the list of DNI components.

Creating a Variant BOM Report

1. If the variant options are not displayed in the BOM-HDL dialog box, click on the *Variant BOM* toggle button.

   The BOM-HDL dialog box expands to display the variant options as shown in the following figure.

Figure 3-3 BOM-HDL Dialog Box Displaying Variant Options



2. In *Variant File*, enter the path to the variant database. You can also use the browse button to display the Load Variant File dialog box, which can be used to locate the variant database.

   Note: You can specify an absolute or relative path to the variant file.

3. Click on the *Variant BOM* radio button and then choose the required variant from the list box. (To generate variant BOM reports for all variants, choose the *All Variants* option.)

4. Click on the *Customize* button to display the Customize Template dialog box. You can use this dialog box to change the properties that will be displayed in the BOM report and to change the general look and feel of the report. After making the required changes, save the template file and close the Customize Template dialog box.

5. To generate the BOM report(s), click on the *Generate* button in the BOM-HDL dialog box.

   The BOM report is created in the `bom` view.

   A message box appears with the information that the BOM report is generated and prompts you to view the report.

6. Click on the *Yes* button to view the BOM report.

   If you have selected the *Text File in Tabular form* or *Spreadsheet Format* option, the file appears in a text editor. If you have selected the *HTML* option, the file appears in the default HTML browser.

   The variant BOM report displays the property values for all the components in the selected variant. The report includes a new column named *Var Status*, which displays the status of each component in the design.

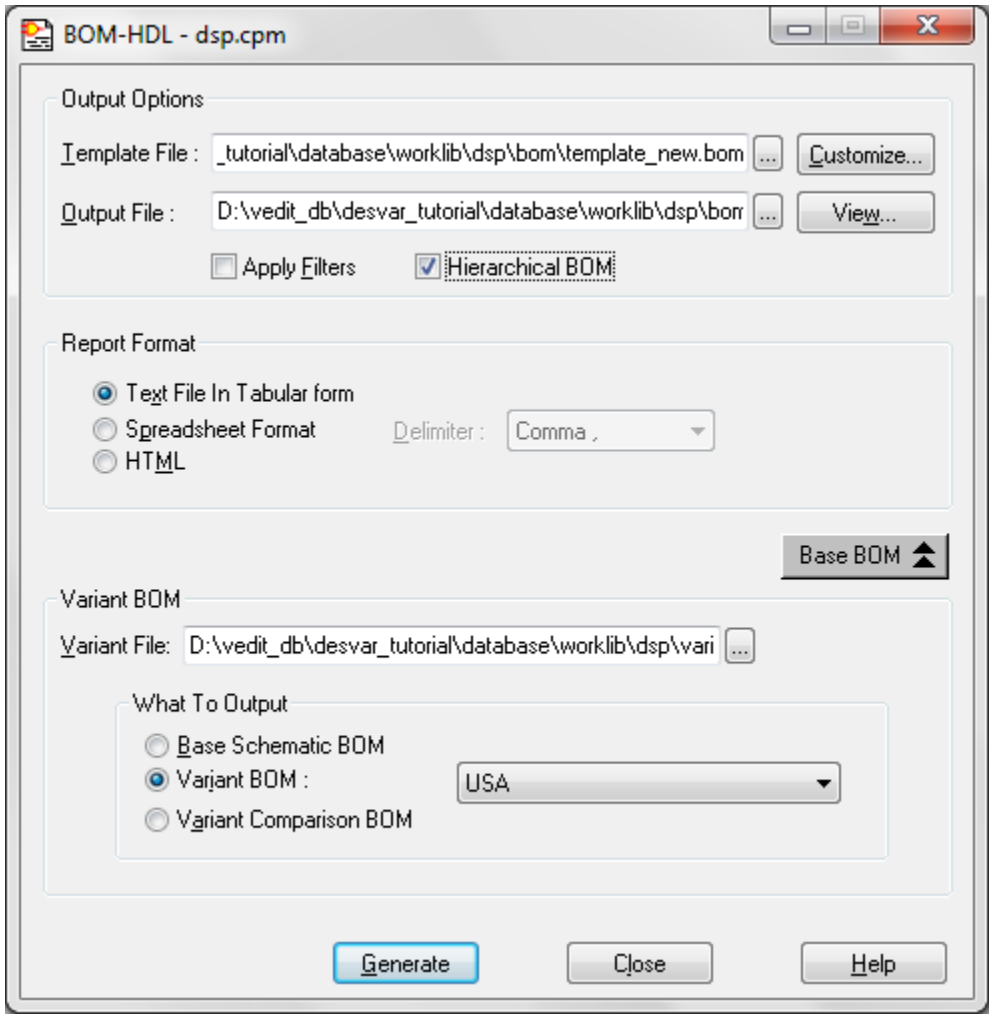
Creating Hierarchical Variant BOM Reports

If you have a design with hierarchical variants, you can generate hierarchical BOM reports for a design. The hierarchical BOM report has a single entry for each reuse block instance which has been identified for hierarchical variants. Components from block instances are excluded from the main BOM. Each of the block instances has its own BOM report available for the base as well as for each of the variants defined inside the block.

To generate a hierarchical BOM report, do the following:

1. Select the Hierarchical BOM check box.
2. If the variant options are not displayed in the BOM-HDL dialog box, click on the *Variant BOM* toggle button.

The BOM-HDL dialog box expands to display the variant options as shown in the following figure.



3. In *Variant File,* enter the path to the variant database. You can also use the browse button to display the Load Variant File dialog box, which can be used to locate the variant database.

   Note: You can specify an absolute or relative path to the variant file.

4. Click on the *Variant BOM* radio button and then choose the required variant from the list box. (To generate hierarchical variant BOM reports for all variants, choose the *All Variants* option.)

5. In the first box of the dialog, in Output Options, next to the Template File field, click on the *Customize* button to display the Customize Template dialog box. You can use this dialog box to change the properties that will be displayed in the BOM report and to

change the general look and feel of the report. After making the required changes, save the template file and close the Customize Template dialog box.

6. To generate the BOM report(s), click on the *Generate* button in the BOM-HDL dialog box.

   The BOM report is created in the `bom` view.

   A message box appears with the information that the BOM report is generated and prompts you to view the report.

7. Click on the *Yes* button to view the BOM report.

   If you have selected the *Text File in Tabular form* or *Spreadsheet Format* option, the file appears in a text editor. If you have selected the *HTML* option, the file appears in the default HTML browser.

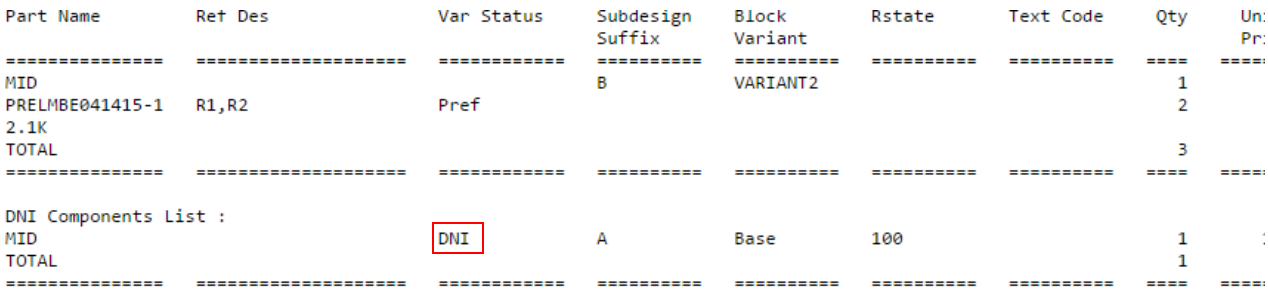
```
Part Name         Ref Des                Var Status     Subdesign     Block        Rstate       Text Code     Qty     Unit
                                                        Suffix        Variant                                       Price
===============   ====================   ============   ===========   ==========   ==========   ==========   ====   =======
MID                                                     A             VARIANT1     100                            1       100
MID                                                     B             VARIANT2                                    1
PRELMBE041415-1   R1,R2                  Pref                                                                     2
2.1K
TOTAL                                                                                                            4
===============   ====================   ============   ===========   ==========   ==========   ==========   ====   =======

DNI Components List :
TOTAL                                                                                                            0
===============   ====================   ============   ===========   ==========   ==========   ==========   ====   =======
```

The hierarchical variant BOM report lists the following:

- Each block instance as a single entry
- Components in blocks without variants; components in blocks with variants are not displayed
- The SUBDESIGN_SUFFIX property, as well as any other property specified for a block (for example, ref_des_pattern), is displayed in the hierarchical variant BOM report.

If you mark a block as DNI, variants of that block will not be displayed in the report, and *DNI* is displayed in the DNI Components List field. The Variant Name field will display *Base* instead of the variant names for this block.



Part Name	Ref Des	Var Status	Subdesign Suffix	Block Variant	Rstate	Text Code	Qty	Un: Pr:
MID			B	VARIANT2			1	
PRELMBE041415-1 2.1K	R1,R2	Pref					2	
TOTAL							3	
DNI Components List :								
MID		DNI	A	Base	100		1	
TOTAL							1	

Custom Property for Hierarchical Variants

If you have multiple reuse block instances with variants in a design, you may want to specify a specific set of instances for variant processing. For example, in a design called TEST, you have a block MID with two instances—MID1 and MID2. You want the option of enabling hierarchical variants for MID1, but not MID2.

To define the block instances that can be enabled for variant processing, you can use a custom property. The custom property name can be configured using a project (`.cpm`) file directive, HIER_BOM_PROP.

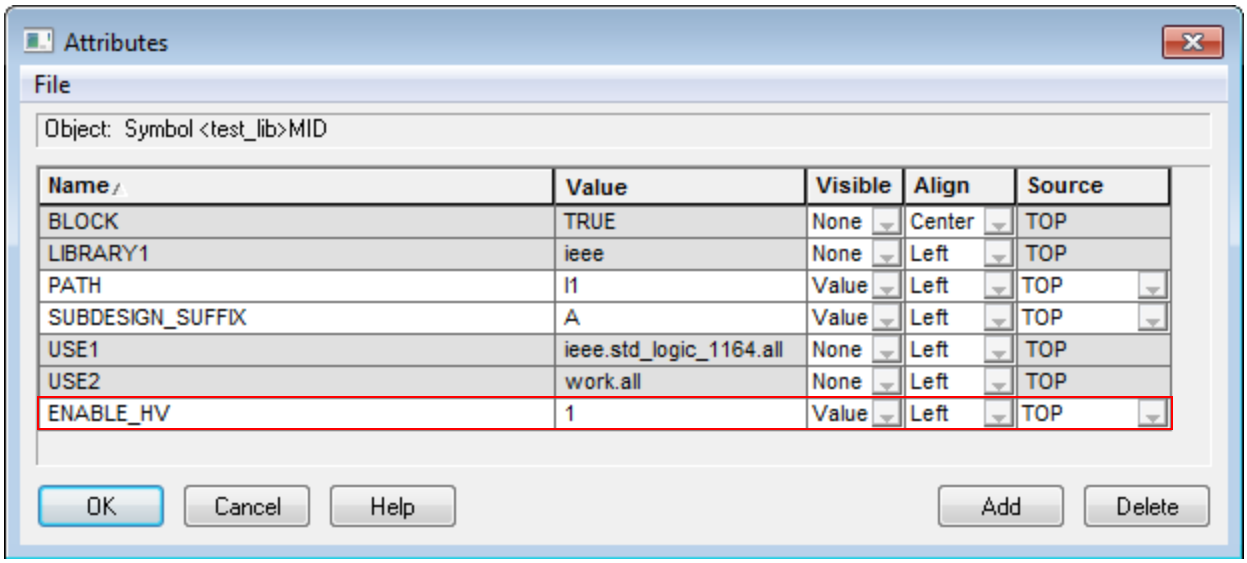
To define the block instance that can be enabled for variant processing, do the following:

1. Open the `.cpm` file where you want to specify the custom property.
2. Specify the `HIER_BOM_PROP` directive in the `START_BOMHDL` section.

   ```
   HIER_BOM_PROP '<custom property value>'
   ```

   The value for the directive can be anything that you want to use to identify a block instance that should not be enabled. For example, `ENABLE_HV`.

After you specify the directive in the `.cpm` file, you must specify an attribute for the block instance that you want to enable for variant processing. The attribute name must be the same as the value you specified for the `HIER_BOM_PROP` directive.



In this example, a custom property, `ENABLE_HV`, is added to the MID1 block instance. As a result, this variant will be listed in the hierarchical variant BOM report although components related to the MID1 instance will be excluded from BOM reports.

The MID2 block instance will not be listed in BOM reports, although its components will be included in BOM reports.

For more information about hierarchical variants, refer to the Working with Hierarchical Variants section in *Allegro® Design Entry HDL User Guide*.

Creating the Variant Comparison BOM Report

If you want a snapshot report that contains a part number-based comparison between the components of the base schematic and all the variants, generate the variant comparison BOM report. This report will list:

1. The components that have a different part-number for particular variant
2. The `DNI` components in individual variants
3. All the common components across the base schematic and all variants.

Creating the Variant Comparison BOM

1. Ensure that the *Variant BOM* section is displayed in the BOM-HDL dialog box, and the *Variant File* field is seeded with the path to the variant database.
2. Click on the *Variant Comparison BOM* radio button.
3. Choose a file format. To choose HTML file format, choose the *HTML* radio option.
4. Click on the *Generate* button.

The variant comparison report is generated in the HTML format with the values of the different variants displayed as columns. The default filename for the variant comparison report is `ComparisonBOM.rpt`. This report includes the `COMMON COMPONENTS LIST`, which includes the components that have not been customized in Variant Editor. These components have the same value for the base schematic and all variants.

Customizing the BOM Template

The BOM template defines the properties that appear in the BOM report. It also defines the look and feel of the BOM report. You can customize the BOM template to change the report header, the different properties listed in the report, the format style of the report, and variant specific settings.

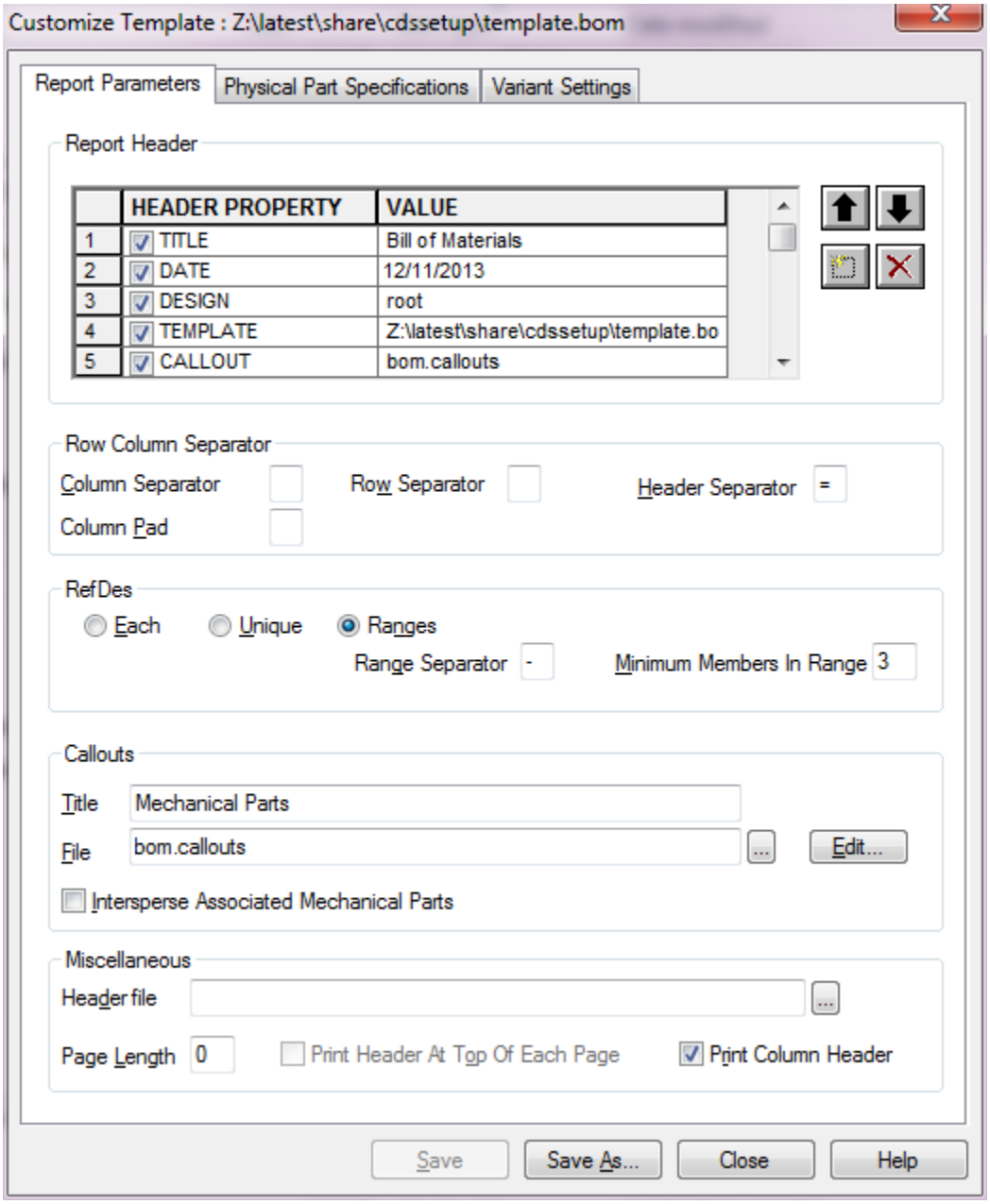
Displaying the Customize Template Dialog Box

To customize the BOM template, you make changes in the Customize Template dialog box.

➤ To display the Customize Template dialog box, click on the *Customize* button located to the right of the *Template File* field in the BOM-HDL dialog box.

The Customize Template dialog box appears with the Report Parameters tab selected. See the Customize Template - Report Parameters Tab figure on page 41.

Figure 3-4 Customize Template - Report Parameters Tab



Setting Report Parameters

To customize the report header and row column formatting, you need to set the report parameters by changing them in the Customize Template - Report Parameters tab. You can also add more properties, or delete existing properties or reorder them. The properties added

to the report header display at the top of the BOM report and are useful in categorizing information. If you delete properties, ensure that you have at least one property left in the report. For more information about the report parameters that you can customize, see Customize Template - Report Parameters on page 154.

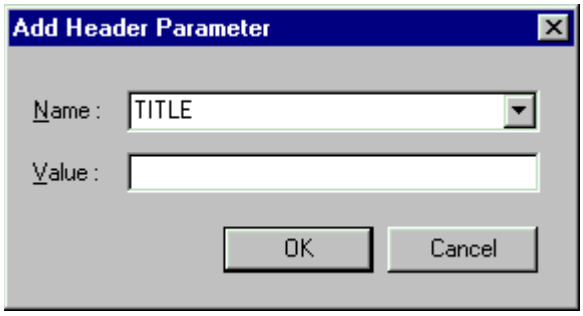
In the following section, you will learn how to set the report parameters.

Customizing Report Parameters

1. To add a new header property in the *Report Header*, click on the  button.

   The Add Header Parameter dialog box appears.

Figure 3-5 Add Header Parameter Dialog Box



2. Choose the *Name* field by clicking in it, and enter the new property name and press `Tab`.
3. The *Value* field is selected. Enter the value for the new property.
4. To save the new property and close the Add Header Parameter dialog box, click on the *OK* button.

   Note: You may like to reorder rows and move the new property to the top, thereby ensuring that the new property appears first in the BOM report. If you reorder properties, the BOM report will reflect the order of properties that you have set. To reorder a property, choose the row corresponding to the property by clicking on it, and then press the *Up* or *Down* arrow button the desired number of times.

5. If you do not want to display any header property in the BOM report, clear the check box corresponding to that property.

   BOM-HDL will list only those properties in the report header that re selected in the *Report Header* section.

   Note: You can change the style in which date is displayed in the BOM report by double-clicking on the *VALUE* column for the `DATE` property. You can display date in one of the

following formats.

6. The default BOM template file uses a space as the column separator, and it does not have any row separator defined. To change the column and row separators, enter the new value in the *Column Separator* and the *Row Separator* fields.

   - The delimiter is a single character. In *Column Separator*, the default delimiter is a space. To custom define the separator, first delete the space then specify your delimiter.
   - The row and column separators will work only if you are generating the BOM report as a text file in ASCII format. If you set values for the row column separators, and set the report format as HTML, the values for the row and column separators will be overwritten. The HTML report will display in a matrix structure, without any row or column separators. If you set the values for the row and column separators, and set the report format as spreadsheet, the values for the row separator and the column separator will be overwritten by the letter that you have defined as a delimiter.

7. You can also define the *Header Separator* and the *Column Pad*. The letter that you define as the *Header Separator* is used to separate the heading row from the data rows in the BOM reports. The letter that you specify as the *Column Pad* is used to pad the column; that is, the letter appears on both sides of the column boundary. By default, the *Header Separator* is assigned the value = and the *Column Pad* separator is a space.

   Note: The *Header Separator* and the *Column Pad* options will work only if you are generating the BOM report as a text file in tabular format. Do not customize these options if you plan to generate BOM report in HTML or as a spreadsheet.

8. By default, if there are three or more consecutive reference designators for the same part name (defined by the `BOM_PART` property), they appear in the BOM report as ranges. You can list the reference designators on separate rows by selecting *Unique* under *RefDes*. You can select the *RefDes Ranges* check box to display all reference designators in a range. For example, a part, `74LS00`, has reference designators `U1` to `U5`. If *Ranges* is selected, they will be listed as:

   ```
   74LS00 U1-U5  5
   ```

   But if *Unique* is selected, they will be listed as:

   ```
   74LS00 U1 1
   74LS00 U2 1
   ```

```
...
74LS00 U5 1
```

You can also define a different range separator by entering a new letter in the *Range Separator* field or increase the minimum members in the range by entering a new number in the *Minimum Members in Range* field. By default, - is used as the range separator and the minimum number of members that you can define in the range is `3`.

Note: SPB release 15.5 onwards you have the option of displaying each column of a BOM report with unique values. To use this feature, you set the cpm directive, `UNIQUE_FEATURE` in the project `cpm` file. You can set this directive by adding the following line under the BOMHDL section of the `project.cpm` file: `UNIQUE_FEATURE 'ON'`. If you do not set this directive, the unique listing is displayed only for the refdes column provided you have selected the Unique option under RefDes in the BOM-HDL dialog box.

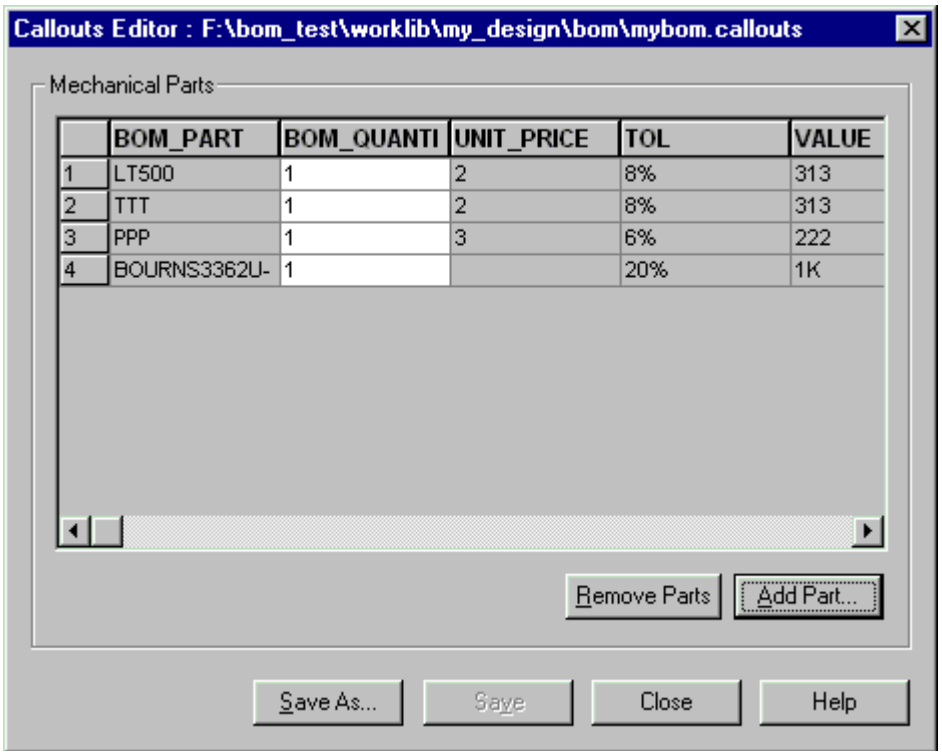
9. To change the path of the callout file, click on the browse button to the right of the *Callouts File* field.

The Load Callouts File dialog box appears where you can choose the path to the callouts file.

You can make changes to the callout information. For this, choose the *Edit* button to the right of the *Callout File* field.

The Callouts Editor Dialog Box figure on page 45 appears.

Figure 3-6 Callouts Editor Dialog Box



You can make changes to the callouts information. See Callouts Editor on page 169 for more information about changing the callout information. After making the changes, click on the *Save* button.

Note: If you want to add a new part, you can click on the *Add Part* button and choose any available part. Only those parts in ptf files that are assigned the `CLASS=MECH` property are available for selection.

Note: Callouts are displayed as the last section in a BOM report. This section lists the serial number and the values of those properties the `MECH` check box for which are selected in the Customize Template - Physical Part Specifications tab. If you want to display values for any property in the Callouts section of the BOM report, select the `MECH` check box corresponding to that property in the Customize Template - Physical Part Specifications tab. Ensure that you have at least one `MECH` check box selected for any property to display its value in the BOM report.

10. The default header printed on the BOM report comes from the selection in the *Report Header* box. If you want to place a customized header, you can create the header in an ASCII text file and enter the path to that file in the *Header File* field.

In the BOM report, first the header properties in the *Report Header* box will be displayed and then the header file will be displayed. The list of components and their properties will follow thereafter.

11. By default, the BOM report is printed on a single page. If you have many components, this report can become too long and difficult to read. You can divide the BOM report into multiple pages, and declare that each page must contain a fixed number of lines. To define the number of lines to be displayed in each page, type a number in the *Page Length* field. (This number must be equal or greater than `20`.)

When you define a value in the *Page Length* field, the *Print Header At Top Of Each Page* check box becomes available for selection.

12. To print the header at the top of each page, choose the *Print Header At Top Of Each Page* check box.

Note: If the header file is too long and it cannot be displayed within the page length restriction on each page, an error message will display asking you to truncate the header file.

13. By default, the BOM report lists a column header that lists the names of properties displayed in the report. To remove the display of the column header, clear the selection from the *Print Column Header* check box.

Note: The *Header Separator* will be displayed in the BOM report only if you are printing the column header.

14. To save the template file, click on the *Save* button.

You can also save the template file with a new name. For this, click on the *Save As* button and define the name of the new file.

Selecting the Physical Parts

A BOM report, by default, lists only five properties: `BOM_INST` (Ref Des), `BOM_QUANTITY` (Qty), `COST` (Unit Price), `COST` (Cost), and `BOM_PART` (Part Name)). You can customize the information in the BOM report. For this, you can include or exclude properties to be displayed for each part by customizing the information in the Physical Part Specifications tab of the Customize Template dialog box. Each property that you choose in the Physical Part Specifications tab appears as a column in the BOM report. You can choose any number of available properties to be displayed in the BOM report. The only restriction is that you must have at least one property selected for display in the BOM report.

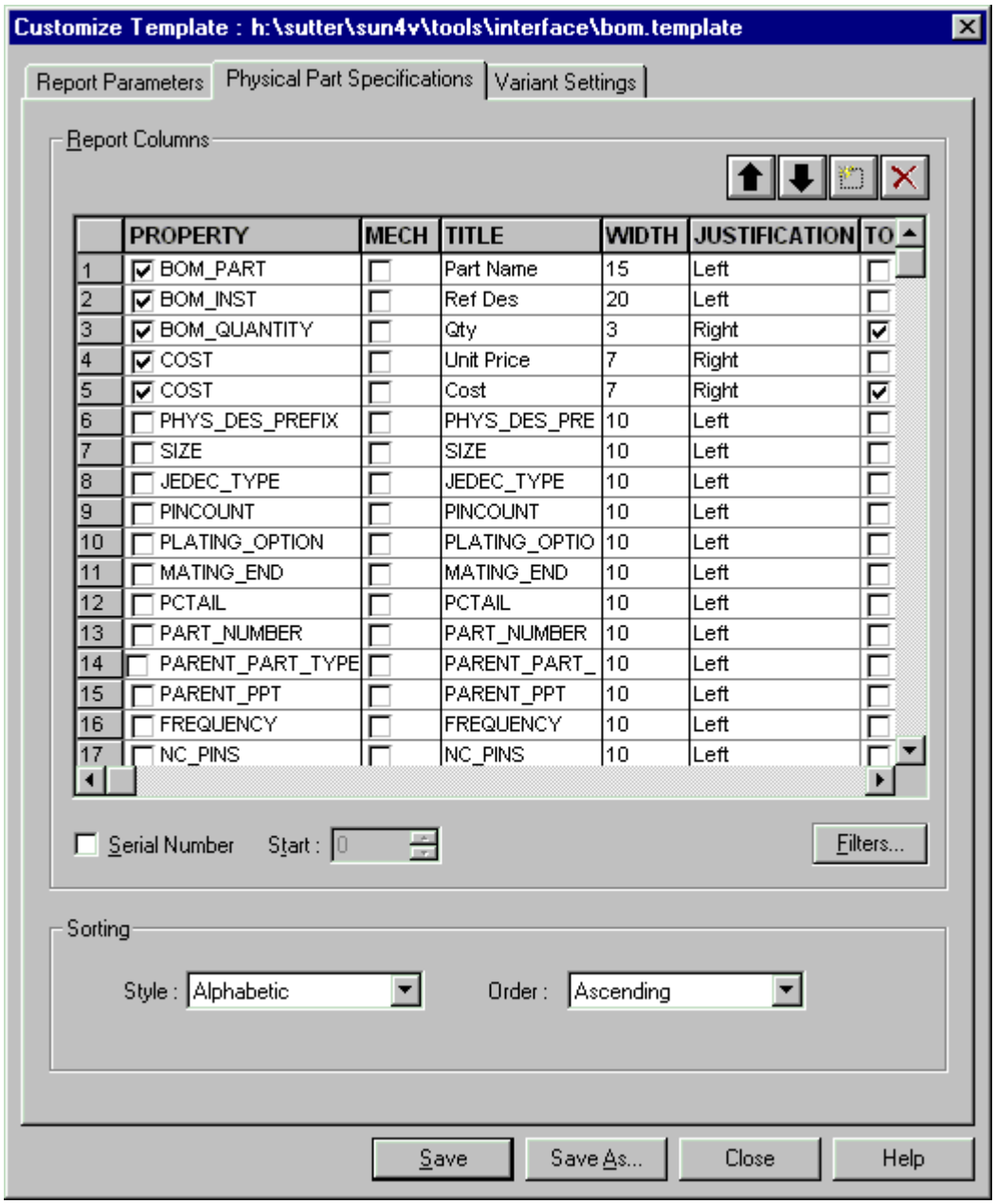
In the following section, you will learn how to choose physical parts and display them in the BOM report.

Customizing Report Columns

1. The default tab in the Customize Template dialog box is *Report Parameters*. To customize the list of properties that can be displayed in the BOM report, click on the *Physical Part Specifications* tab.

   The Physical Parts Specification tab is selected in the Customize Template dialog box. See Customize Template - Physical Part Specifications Tab figure on page 48.

Figure 3-7 Customize Template - Physical Part Specifications Tab



2. To include any new properties in the BOM report, such as the `JEDEC_TYPE` and `PART_NUMBER` properties, choose the `JEDEC_TYPE` and `PART_NUMBER` check boxes.

3. If you want to display the mechanical parts associated with any property, choose the *Mech* check box corresponding to that property. Choose the *Physical Part Specifications* Tab in the *Customize BOM Template* option and then choose the *PROPERTY* and *MECH* check boxes against the `MECH_PART1,MECH_PART2` properties

When you generate the BOM report, the mechanical parts in the callouts file will be displayed with the values for the selected properties.

Note: BOM-HDL will list only those components from ptf files (as mechanical parts) that have been assigned the `CLASS=MECH` property.

4. If you want, you can change the title for any property by typing the new title in the *Title* field for that property.

5. Each property has a default width associated with it. For example, the default width of the `JEDEC_TYPE` property is 10 characters. To change the width to 15, choose the *Width* field for the `JEDEC_TYPE` row and type `15`.

   Note: When you choose the *Width* field, a spin box appears. You can use this box to increase or decrease the value in the *Width* field.

   Note: When you change the column width for a property to a value lower than the width of a property value, the property value will be displayed in multiple lines. For better readability, set the column width for properties to a value greater than the width of the property value.

6. You can change the justification for the text in any column. For example to change the justification of the `JEDEC_TYPE` property, click in the *Justification* field for the `JEDEC_TYPE` property and choose *Right* in the list box.

7. If you want to add the total for all properties values and display it at the end of the property column in the BOM report, choose the *Total* field corresponding to that property.

8. If you have a property with a numerical value (for example, `COST`) and want to display the product of that property value and the quantity, choose the *Subtotal* field corresponding to that property.

9. If you are generating a BOM report in Spreadsheet format and you have defined a delimiter, say a colon, choose the *Quote* field corresponding to the properties that contain that delimiter.

   Selecting the *Quote* field ensures that for all properties where the delimiter is part of the property's value, a quotation mark is used to separate the value of that property from other delimiters.

   For example, if you are using a semi colon (:) as the delimiter and have a property such as `D:E`, the property value will be displayed as `"D:E"` when the Quote check box is selected.

   The *Quote* option is also useful if you have a property value with the following format: xxx-xx-xx, say `PART-NUMBER = 9011-05-11`.

In such cases, the spreadsheet program modifies the format and changes the data type to date and displays the data as `5/11/9011`.

When you select the *Quote* check box, the BOM utility adds a leading space and ensures that the spreadsheet program does not modify the data type.

The *Quote* option is also useful if the property value has a space in it. Without the quote option, the value might be split in the output report file.

You can also use the `-forcequotes` argument as "`bomhdl -proj *.cpm -forcequotes`" in the command line for column values. If you use this argument, quotes are added to all column values regardless of the *Quote* check box selection.

10. By default, all properties that have `NULL` values in the BOM report are displayed with the letter `?`. This letter is defined by the *Missing* field. If you want to change the letter that defines the *Missing* field, enter the new text in the missing field. For example, you can enter the word `Missing` to represent missing values.

11. To add the serial numbers for all parts in the BOM report, choose the *Serial Number* check box. This check box corresponds to the BOM_ITEM_NUM property, which like other properties can be modified. For example, you can change the position, header or width of the serial number column or selectively check on or off serial numbers for electrical and mechanical parts.

The *Start* field becomes active and displays the value 0. The parts in the BOM report will begin with the default serial number 0 or any other serial number that you specify in the *Start* field.

12. To sort the BOM report based on any property, you have to make that property the key property. BOM-HDL treats the first property that has the check box for the electrical components selected as the key property and uses it to sort all parts in the BOM report.

For example, to make the `JEDEC_TYPE` property the key property, select the row corresponding to the `JEDEC_TYPE` property and use the *Up* button to move the row to the top of the property list.

Note: If you have mechanical parts, BOM-HDL treats the first property listed in the Mechanical Parts section as the key property for generating the BOM report for mechanical parts. This property can be different from the key electrical property specified in the Physical Part Specifications tab of the Customize Template dialog box.

13. By default, all properties are sorted in alphabetic style. To change the style to alphanumeric, choose the *Alphanumeric* option in the *Style* list.

The style changes to alphanumeric. Notice that the sorting order is ascending. You can change the order to descending, if required.

Note: In alphanumeric sort, if you have two parts A2 and A12, part A2 will be sorted

before the part A12 (when sorting order is ascending).

14. To save the template file, click on the *Save* button.

Setting Filters on Parts

You might sometimes want only those parts in a BOM report that fulfill a particular condition to be listed. Using filters, you can define conditions for selecting parts. For example, you can define a condition to include only resistors in the BOM report. To define such a filter, you need to use the following condition:

```
ref des LIKE R*
```

where, `ref des` means reference designators, `LIKE` denotes that the search will be on wild card entries, and `R*` denotes that the search will return all parts that have their reference designators starting with the letter `R`.

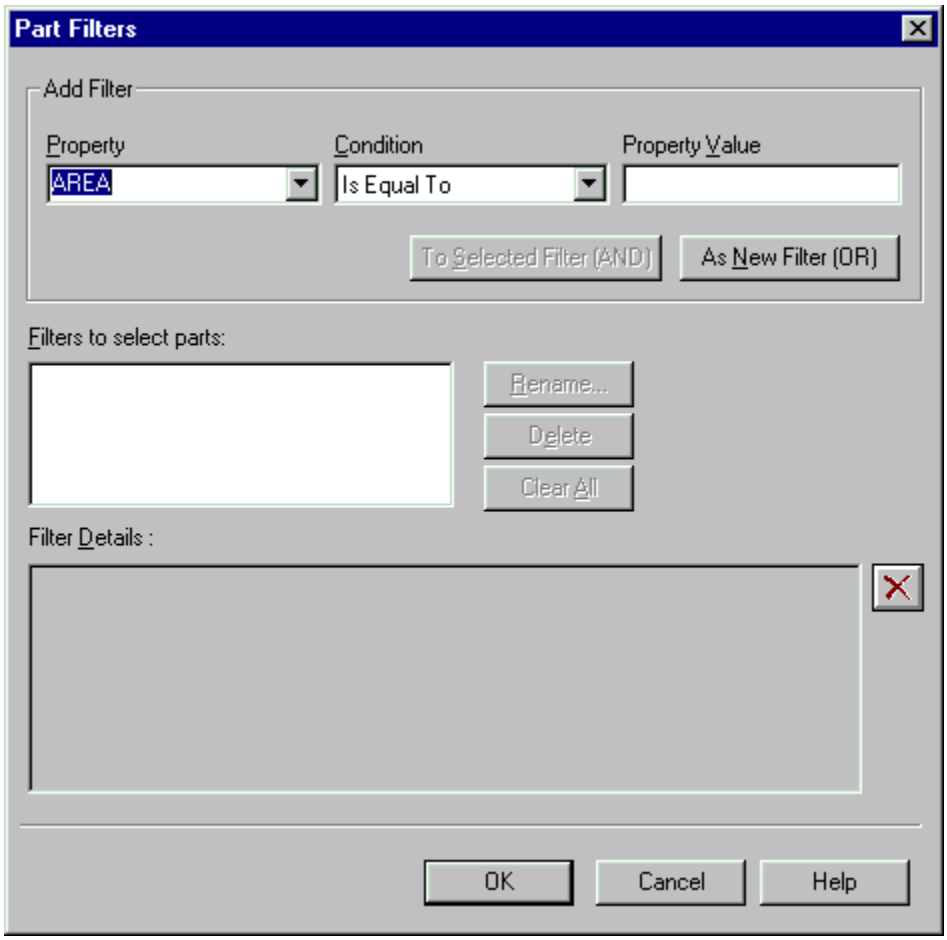
You can also define more complex conditions where the BOM report lists only those parts that satisfy a combination of properties. In the following section, you will learn how to filter parts in a BOM report.

Setting Filters on Parts

1. To set conditions on parts that are displayed in a BOM report, click on the *Filters* button in the Physical Part Specifications tab of the Customize Template dialog box.

The Part Filters Dialog Box figure on page 52 appears. You can use this dialog box to set filters on parts. You will set three filters. The first filter is an `AND` condition between the `SPEED` property and the `JEDEC_TYPE` property. To set this filter, define a condition on the `SPEED` property and then add to that condition another condition on the `JEDEC_TYPE` property)

Figure 3-8 Part Filters Dialog Box



2. To set a condition on any property, type the name of the property in the *Property* field or choose the property from the *Property* list button.

3. Specify the condition for filtering properties. The default condition is *Is Equal To*. You can change this condition to `Is Not Equal To`, `Is Greater Than`, `Is Less Than`, `Is Greater Than Equal To`, `Is Less Than Equal To`, or `Like`. The `Like` condition is used to choose wildcard entries. For example, to define parts with speed less than `15ns`, choose `Is Not Equal To as the condition.`

4. Enter the value of the property in the *Property Value* field. For example, to define that only parts with a speed less than `15ns` are listed in the BOM report, type `15ns` in the *Property Value* field and click on the *As New Filter* button.

   Notice that a new filter named *New Filter 1* is displayed in the *Filters List*. You can rename it to a more intuitive name such as `Speed`. Use the *Rename* button for this purpose.

When you add a filter, the *To Selected Filter (AND)* button in the Part Filters dialog box becomes active. You use this button to add conditions to an existing filter, which is selected in the *Filters List*.

5. To add another condition to the selected filter, define a new property, condition, and value, and then choose the *To Selected Filter (AND)* button. For example, to add another condition to the Speed filter, define a new property `JEDEC_TYPE`, condition *Is Less Than*, and property value as `DIP15`. After defining the condition, click on the *To Selected Filter (AND)* button.

   The *Filters List* is unchanged, but the *Filter Details* list displays the details of the filter conditions.

6. You can define more filters. At any given time, you may delete any filter or all filters. To delete one filter, select it, and choose *Delete*. To delete all filters, choose them, and click on *Clear All*.

7. Choose *OK* to close the Part Filters dialog box.

   The *Physical Part Specifications* tab of the Customize Template dialog box appears. You can save the changes by clicking on the *Save* button. Alternatively, you can use the *Save As* button to save the template file with a different filename. The new name is automatically seeded in the *Template File* field of the BOM-HDL dialog box.

8. The filters you have defined are saved in the template file. To generate BOM reports that use these filters, click on the *Apply Filters* check box in the BOM-HDL dialog box.

9. To generate the BOM report, click on the *Generate* button.

Tip

If you want to generate a BOM report that filters on a particular property and displays all components that have that property with any value, define a filter that includes the condition:

```
<property_name> Like ?*
```

Limitations in Applying Filters

BOM-HDL might not be able to filter:

- Properties in the callouts section of the BOM report.
- Properties that have values in digits. For example, properties such as `BOM_QUANTITY`, `VOLTAGE`, `RATED_POWER`, and `TOL` have values in digits and these properties are not applied to the BOM report.

Changing Variant Settings

By default, a variant BOM report displays only the components that have `Pref` as their status. You might like to change the variant BOM settings to display the alternates and DNI components.

The default status of preferred, alternate and DNI components are displayed with the designators `Pref`, `Alt`, and `DNI`. You can also assign different designators to represent the status for these components.

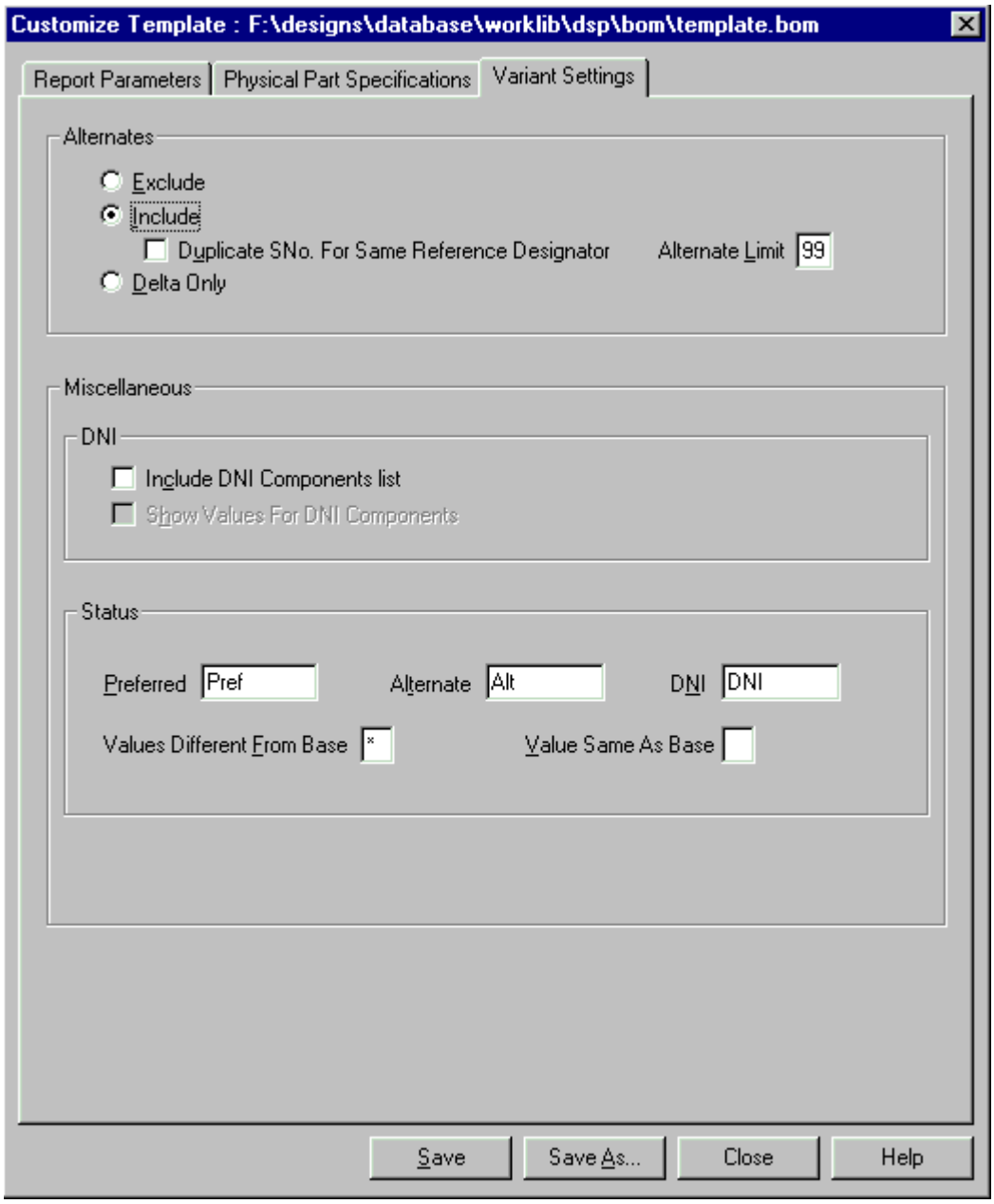
Note: You can use a different status designator for the preferred component.

Changing the Variant Settings

1. The default tab in the Customize Template dialog box is *Report Parameters*. To customize the variant report settings, click on the *Variant Settings* tab.

   The Variant Settings tab is selected in the Customize Template dialog box. See Customize Template - Variant Settings Tab figure on page 55.

Figure 3-9 Customize Template - Variant Settings Tab



2. To include alternates in variant BOM reports, choose the *Include* radio button.

   Note: When you choose the *Include* radio button, the *Duplicate SNo. For Same Reference Designator* check box and the *Alternate Limit* field becomes active.

   You may select the *Duplicate SNo. For Same Reference Designator* check box to duplicate the serial number in the BOM report for same reference designators. However,

you can choose this check box only if `RefDes` is the key property in the Physical Part Specifications tab.

By default, the *Alternate Limit* field has the value 99. You may reduce this limit by entering another number between 0 and 99.

3. To generate a variant report that lists the differences between the variant and the base schematic, select the *Delta Only* radio button.
4. To include DNI components in the BOM report, choose the *Include DNI Components List* check box.

   When you choose the *Include DNI Components List* check box, the *Show Values for DNI Components* check box becomes active for selection.
5. To display the property values for DNI components, choose the *Show Values for DNI Components* check box.
6. To change the status designator for the Preferred, Alternate or DNI components, enter the new designator in the respective fields.
7. To choose a different status designator for values different from base schematic or values same as the base schematic, specify the new designator in the *Values Different From Base* or *Values Same As Base* fields. For example, to use the + sign as the status of the variant values that are different from the base schematic, enter + in the *Values Different From Base field.*

   You can save the changes by clicking on the *Save* button. Alternatively, you can use the *Save As* button to save the file with a different file name. The new name is automatically seeded in the *Template File* field of the BOM-HDL dialog box.

Managing Mechanical Parts

A schematic consists of electrical and mechanical parts. In BOM-HDL, you can manage callouts and associate mechanical parts using the Callouts Editor dialog box to add, remove, or edit mechanical parts. You can also change the quantity of mechanical parts. BOM-HDL translates the mechanical part data and adds them as lines into the BOM reports.

Besides the use of Callouts Editor dialog box for associating mechanical parts, you can create zero-pin count devices for adding stand-alone mechanical parts in Design Entry HDL and Packager-XL. These parts are handled in the same way as electrical parts and are listed in the BOM reports.

Associating Mechanical Parts

To assign mechanical parts to a component, you need to first add the logical or physical part to the design via the PPT. Next, you can associate mechanical parts to electrical parts by defining MECH_PART as mechanical properties in the PPT rows for electrical parts in the PPT files. You can define mechanical parts in the PPT files by defining the CLASS property as MECH. The mechanical parts with the CLASS property as MECH can be added using the Callouts Editor dialog box.

When you change the PPT row for a component in a variant, the mechanical parts associated with that component also change. The mechanical parts associated with the new PPT row are associated with the component. Therefore, the BOM report for the variant lists the part associated with the new PPT row. You can even add or create a new mechanical part at the time of BOM generation using the Callouts Editor.

Note: An associated mechanical part does not inherit PTF properties from the parent electrical part. The property columns in the BOM report are therefore based on the *Missing* field in the BOM template. In case no customization is done, the default value of "?" is picked.

How Mechanical Parts are Associated to Electrical Parts

To associate mechanical parts to electrical parts, the latter are injected with properties such as MECH_PART1, MECH_PART2, and MECH_PART3. The value of MECH_PART* properties maps to the mechanical part PPT rows. Following examples demonstrate how mechanical parts are associated to electrical parts.

Example1 shows how a ptf file stores information about associated mechanical parts (The mechanical part in example is PRES):

Example 1

```
PART 'PRES'
CLASS=DISCRETE
{================================================================================
========}
:VALUE(OPT='1K')    | RTOL(OPT='5%')    | PART_NUMBER     | JEDEC_TYPE | COST  |
DESCRIPTION | MECH_PART1          ;
{================================================================================
========}
 '10K'              | '10%'             | '24709-001-87'  | 'RES400'   | '.90' |
'GGGGH'      | 'LIFT:LT300:3'
 '100K'             | '5%'              | '27777-999-888' | 'RES400'   | '.90' |
'GGGG'       | 'LIFT:LT500:5'
END_PART
```

In Example 1, the mechanical part property has the following syntax:

```
'<Part_Name>:<Row_Name>:<Quantity>'
```

where, Row_Name uniquely identifies a PPT row for a mechanical part with the Part_Name in the mechanical part ptf file. *<Row_Name>* is represented as:

```
<Part_Name>-A,B,C...
```

where, A, B, C ... represent a comma separated list of key properties.

It is a good practice to specify the *<Row_Name>* in its expanded form in PPT rows. Expanded Row_Name is written as <Part_Name>-<Row_Name>. Example 2 shows a mechanical part property that has an expanded Row_Name. Notice that this syntax is different from Example1, where the *<Row_Name>* is not expanded.

Example 2

```
PART 'PRES'
CLASS=DISCRETE
{================================================================================
========}
:VALUE(OPT='1K')    | RTOL(OPT='5%')    | PART_NUMBER     | JEDEC_TYPE | COST  |
DESCRIPTION | MECH_PART1          ;
{================================================================================
========}
 '1K'               | '5%'              | '24707-001-49'  | 'RES400'   | '.67' |
'ABCD'       | 'LIFT:LIFT-LT400:5'
END_PART
```

Notice that the part table of the part PRES has MECH_PART1 as an injected property. Therefore, adding the part with the PART_NUMBER = '24707-001-49' property will by default add the following associated mechanical part to the board:

```
PART_NAME=LIFT: Row_Name=LIFT-LT400 : QUANTITY=5
```

By analyzing the mechanical ptf file, you can see how a mechanical property maps to a row in the file. For example, let's check how the MECH_PART property value 'LIFT:LT500:5' covered in Example 1 maps in the mechanical PPT (section) displayed in Example 3.

Example 3

```
PART 'LIFT'
CLASS=MECH
{=============================================================================}
:PART_NUMBER                    = VALUE | TOL | POWER | UNIT_PRICE ;
{=============================================================================}
```

```
 'LT500'(~LT500)            = '313' | '8%' | '1010' | '2'
 'LT300'(~LT300)            = '222' | '6%' | '1030' | '3'
 'LT400'(!)                 = '223' | '58%' | '1020'| '4'
END_PART
```

To determine the row in the mechanical ptf file that maps to the `MECH_PART` property value `'LIFT:LT500:5'`, BOM-HDL searches for `PART_NAME = LIFT`, and `ROW_NAME = LT500`. The PPT row returned is:

```
'LT500'(~LT500)            = '313' | '8%' | '1010' | '2'
```

Notice that the third row in Example3 contains (!) in `ROW_NAME`. This notation signifies that the `ROW_NAME` is preceded by the `PART_NAME` and a dash (-) sign. Therefore, the value of the `ROW_NAME` is `LIFT-LT400`. Check Example2 where `ROW_NAME = LIFT-LT400`.

Defining Mechanical Kits

Mechanical kits are useful in situations where sets of mechanical parts are always included together. For example, assume a connector requires the following mechanical parts: 4 nuts, 5 washers, and 7 screws. To specify this requirement, you can define a mechanical kit and add it to the schematic. BOM-HDL treats a mechanical kit like a mechanical part, except that while associating the kit to the electrical part, it recursively associates each mechanical part in the mechanical kit with the electrical part.

Example 4 shows how a ptf file stores information about mechanical kits, `KIT001` and `KIT002`.

Example 4

```
PART 'KIT'
: PART_NUMBER   =  MECH_PART1          | MECH_PART2        | MECH_PART3
KIT001(~KIT001) = 'STANDOFF:HW0012:10' | 'BRACKET:HW0370:5' | 'SOCKET:SK0009,24:4'
KIT002(~KIT002) =  'STANDOFF:HW0013:4' | 'BRACKET:HW0372:2' | ""
END_PART
```

The `KIT001` mechanical kit contains the following parts:

```
10 number STANDOFF HW0012
5  number BRACKET HW0370
4 number  SOCKET SK0009
```

If you associate `KIT001` to an electrical part, all 19 parts in the kit are associated with the electrical part.

Example 5 shows a section of ptf file with PPT rows for part CONN20 with the associated mechanical kit 'KIT:KIT001:1'.

Example 5

```
PART 'CONN20'
:PLATING_OPTION | MATING_END = PCTAIL        | PART_NUMBER |  MECH_KIT1 ;
 'TIN'          | '6.09mm' (!)    = '3.18mm' | '22-28-4200'|  'KIT:KIT001:1'
 '15u,gold'     | '6.09mm' (!)    = '3.18mm' | '22-28-4203'|  ""
END_PART
```

In Example 5, the value of property MECH_KIT1 is KIT: KIT00: 1, signifies that the part name is KIT, the row name is KIT0001, and the quantity is 1. Therefore, if you add one quantity of connector CONN20 for this PPT row, by default, BOM-HDL adds one quantity of KIT: KIT001 to the board. Consequently, all 19 parts in the kit KIT001 are added to the board.

How Mechanical Parts are Listed in BOM Reports

There are two ways in which you can display mechanical parts in the BOM report:

- Display the mechanical parts along with the electrical part with which they are associated in the same section of the BOM report.

  To display mechanical parts along with the electrical part with which they are associated, select the *Intersperse Associated Mechanical Parts* check box in the Report Parameters tab of the Customize Template dialog box.

  All mechanical parts with the status Pref, Alt, and DNI are listed in the BOM report along with the electrical part with which they are associated. The format in which a mechanical part appears in the BOM report is:

  <*PART_NAME*> (<*RefDes*>) (MECH)

- Display the mechanical parts as a separate heading.

  If you do not select the *Intersperse Associated Mechanical Parts* check box, the associated mechanical parts are listed with the callouts table under the heading Mechanical Parts. The BOM report lists the part names of all mechanical parts.

  The BOM report includes all mechanical parts with the status as Pref merged together. However, mechanical parts with the status as ALT appear in the following syntax:

  <*PART_NAME*> (ALT))

  Note: Mechanical parts with the status as DNI are not included in the BOM report.

Creating a BOM Report from the Command Prompt

You can generate BOM reports both from the command prompt or by using the BOM-HDL dialog box. It is recommended that when generating the BOM report for the first time, use the BOM-HDL dialog box. This will help you customize different options. For generating subsequent BOM reports, you may work from the command prompt.

Syntax of the bomhdl Command

The syntax for generating a BOM report from the command-line prompt is as follows:

```
bomhdl.exe -proj <Project_File_Name> [-nographic]  [-t <Template_File_Name>] [-f
     HTML | SS|TEXT] [-delim <delimeter>] [-o <outfile name>] [-a YES | NO]  [-var
     <Variant_Database> [variant1 variant2...] [COMPARE] [ALL]]
```

To generate a BOM report from the command-line prompt, you provide inputs to the BOM-HDL tool by passing different switches. All the switches that are displayed within brackets are optional while other switches such as `-proj` are mandatory.

The description of the different switches is as follows:

Switch	Description
-nographic [optional]	Specifies that BOM-HDL will run in the silent (nographic) mode (where no dialog boxes are displayed). The BOM report will be generated based on the settings that you define using the different switches or, if no particular switch is defined, the BOM report will be generated using the settings stored in the project file. **Note:** You will not get any message of successful completion. You can check the output in the location specified by the `-o` switch.
-proj *<project_path_name>* [required]	Specifies the path of the project file (`.cpm` file name). **Note:** You need to specify the full path to the project file. An incomplete path or a wrong path will generate the error message, 'Either -proj switch absent or incorrect project path name'. **Note:** You can specify the relative path from the current directory.

Switch	Description
-t or -T <*template_file_name*> [Required if you are working in the silent mode and the template file is not present in the default directories]	Specifies the name of the template file. Depending on whether you specify the full or relative path of the template file, BOM-HDL searches the template file at the specified location and loads the file. If the file is not present at the specified location, BOM-HDL searches for the template file in the `bom` view. Therefore, if your template file is present in the `bom` view, specify only the file name; otherwise, specify the complete path to the template file. If the template file is not found in the specified path or if the *<Template_File_Name>* is not specified, BOM-HDL searches for the locations as specified in the `setup.loc` file. It searches locations in the order specified and loads the template file from the location it finds first. If the `-t` or `-T` switch is not used then BOM-HDL first searches for the template file settings in the project file. If no information for the template file is stored in the project file then BOM-HDL searches for locations as specified in the `setup.loc` file and loads the template file from the location it finds first.

Switch	Description
-o or -O <*output_file name*> [optional]	Defines the name of the output BOM report file. In the graphic mode, BOM-HDL by default picks the value of the *Output File* field from the project (`.cpm`) file. If the project file does not have information specified about the output file, the *Output File* field displays: ■ The `BOM.rpt` file in the `bom` view of the design as the default output filename if the report format is text file or spreadsheet ■ The `BOM.html` file in the `bom` view of the design as the default output filename if the report format is HTML. Depending on the report format type you choose and whether you are generating the BOM report for the base schematic or a variant, the name of the file in the *Output File* field automatically changes. For example, if you are generating a variant BOM report for the `INDIA` variant in the HTML format, the name of the file will be `INDIA.html`. If you specify an output file but do not specify its path, BOM-HDL creates the output file in the `bom` view of the current project. If the path information is available, that information is used. **Note:** The value of the output file specified in the graphics mode will be seeded in the project file. In the silent (nographic) mode, BOM-HDL will generate the BOM report but will not seed the output file path in the project file.
-f [TEXT][HTML][SS] [optional]	Specifies the format of the BOM output file. `HTML` signifies the HTML file format, while `SS` denotes the spreadsheet format. The default format is ASCII text type. This is the format assigned when the `HTML` or `SS` format is not specified or no other formatting option is specified in the project file.
[-delim <delimeter>]	Specifies the delimiter to be used in report generation.

-a [YES|NO] [optional]	Specifies whether or not filters be applied to BOM reports.

Variant BOM-Specific Switches

Switch	Description
-var <Variant_Database> [variant_name] [ALL] [COMPARE] [optional]	Specifies the path of the variant database filename. If the path of the variant database is not specified, BOM-HDL searches for the variant file in the `variant` view of the selected design. You can also provide the name of the variant for which the BOM report is to be generated. Alternatively, you can use the `ALL` switch to generate BOM report for all variants or the `COMPARE` switch to generate the variant comparison BOM report. **Note:** If you use multiple switches in the `bomhdl` command, ensure that the `-var` switch is used as the last switch.

BOM Usage Examples

The following examples describe how to open the variant BOM report from the command prompt. For all the examples, BOM reports are being created for the project `hexcounter.cpm` located in the `share` directory of the `d:` drive.

Base Schematic BOM Usage Examples

1. To open the BOM-HDL dialog box for the project `hexcounter.cpm` located in the `d:/share` directory, type:

   ```
   bomhdl -proj d:/share/hexcounter.cpm
   ```

   Note: The template file is first searched in the project file. If no entry for the template file is present in the project file, the `bom` view of the current project is searched. If the template file is not found in the `bom` view, BOM-HDL searches for the locations as specified in the `setup.loc` file and uses the location that it finds first.

2. To generate the base schematic BOM report for the project `hexcounter.cpm` in the silent mode, type:

   ```
   bomhdl -nographic -proj d:/share/hexcounter.cpm
   ```

   The report will be generated and a message will be displayed asking whether or not you want to view the report.

3. To open the base schematic section of the BOM-HDL dialog box for the project `hexcounter.cpm` using the `MyTemplate.bom` file as the template file, type, do the following:

```
bomhdl -proj d:/share/hexcounter.cpm -t  d:/share/templates/MyTemplate.bom
```

Note: If you use the `-nographic` switch in the above example, then the base schematic BOM report is generated.

Variant BOM Usage Examples

1. To open the BOM-HDL dialog box for the project `hexcounter.cpm` located in the `d:/share` directory and the variant database `variant.dat`, type:

   ```
   bomhdl -proj d:/share/hexcounter.cpm -var variant.dat
   ```

If you use multiple switches in the* `bomhdl` *command, then ensure that the* `-var` *switch is used as the last switch.

2. To generate variant BOM reports for all variants using the `MyTemplate.bom` as the template file, type:

   ```
   bomhdl -proj hexcounter.cpm ALL -t  MyTemplate.bom -var variant.dat
   ```

   Note: When you open the BOM-HDL dialog box, it comes loaded with the options that you have specified from the command prompt and for all the optional fields for which you did not enter any information, the default settings are obtained from the project file.

BOM-HDL FAQ

This section lists answers to the frequently asked questions (FAQ) on BOM-HDL.

- How should I use BOM-HDL to ignore parts?
- How can I create a partial BOM report?
- How can I customize the header of a BOM report?
- How can I suppress the report viewing dialog box in the nographic mode?
- How can I import a BOM report into a Spreadsheet Program?
- How do I exclude testpoints from BOM?
- How can I remove BOM_PART or some other property from the BOM report?

How should I use BOM-HDL to ignore parts?

If you want to ignore certain parts in your BOM report, you need to perform the following steps:

1. In the schematic, assign the `BOM_IGNORE = True` property to all parts that need to be ignored in the BOM report.
2. Run Export Physical to update the packager output files.

Note: If the packager output files are not updated, the BOM report might not show you the accurate information.

3. In BOM-HDL, create a filter named `BOM_IGNORE` that has the property `BOM_IGNORE`, condition `Is Not Equal To`, and property value `True`. To create the filter perform the steps:
    a. Choose *Tools > Packager Utilities >Bill of Materials* to open the BOM-HDL dialog box.
    b. Click on the *Customize* button.
    c. Click on the *Physical Part Specifications* tab.
    d. Click on the *Filters* button.
    e. Select the property as `BOM_IGNORE`, condition `as Is Not Equal To`, and property value as `True`, and click on the *As New Filter* button.
    f. Click *OK* to save the filter.
    g. Click on the *Save* button to save the template.
    h. Click on the *Close* button to close the Customize Templates dialog box.
4. Apply the filter and generate the BOM report by performing the following steps:
    a. Select the *Apply Filters* check box in the BOM-HDL dialog box.
    b. Click on the *Generate* button to generate the BOM report.

The BOM report ignores all parts that have the `BOM_IGNORE = True` property.

How can I create a partial BOM report?

You can create a partial BOM report in two ways:

- Create a variant named partial—For this, place all components you want in a function A and the rest components in another function B and then include the desired function A in a variant named `Partial`. You can then generate the report for that variant.
- Use a (negative) filter—For this, seed a particular property or its particular value common across the components desired in the partial BOM. Then use a negative filter for that property and generate the BOM report. For example, if a property `GROUP_NO = 564` is present on a certain set of components in schematic, filtering on `GROUP_NO (NOT_EQUAL_TO) 564` will generate the desired partial BOM report.

How can I customize the header of a BOM report?

By default, a BOM report displays the following five rows of information in its header.

- **Title**—You can enter the value in the Value column of the Report Header section in the Customize template dialog box.
- **Date**—Unless you change it, the default value is the current system date. You can select a default style for date display by clicking on the *Value* field of the *Date* row.
- **Design**—Represents the name of the design as mentioned in the project file.
- **Template**—Represents the path to the BOM template file.
- **Callout**—Represents the path to the callouts file.

If you do not want any of these rows to be displayed in the BOM header, clear the check box corresponding to that row in the Header property column of the *Report Header* section. For example, to remove the *Callouts* row from the BOM header, clear the check box corresponding to the *Callouts* row in the *Report Header* section.

If you want to add any other row of information in the *Report Header* section of the BOM report:

➤ Click on the *Add* button in the *Report Header* section.

The Add Header parameter dialog box appears. You can select any property in the *Name* list. The Name list contains 10 properties. The first 5 are the same as mentioned above and are displayed by default in the BOM report. The remaining properties, if selected, will include the following information in the Report Header:

- **Product**—Displays the default name is BOM-HDL.
- **Version**—Displays the product version number.
- **Description**—If you are generating a variant comparison BOM, BOM-HDL displays Part Number based Comparison BOM as description. For Base schematic BOM, BOM-

HDL displays the description as Base schematic BOM. If you have defined a description for variant and are generating a variant BOM report, the description is listed in this row.

- **Project Path**—Displays the path to the project file.
- **Variant**—Displays the name of the variant.

Besides these properties, you can even add a custom property in the *Report Header* section. For this, type the name of the property in the *Name* field and enter its value in the *Value* field in the Add Header Parameter dialog box.

How can I suppress the report viewing dialog box in the nographic mode?

You can set the following environment variable to suppress the report viewing dialog box:

```
DISABLE_VIEW_REPORTS_DIALOG
```

To suppress the dialog box, enter the following command before using the `bomhdl` command:

```
setenv DISABLE_VIEW_REPORTS_DIALOG true
```

How can I import a BOM report into a Spreadsheet Program?

To import a BOM report into a spreadsheet program, do the following:

1. Generate the BOM report in the Spreadsheet format by selecting the *Spreadsheet Format* radio button and select Semicolon (;) as the delimiter.
2. Assuming that the `BOM.rpt` report is generated, open the spreadsheet program and open the `BOM.rpt` file in it.

   The Text Import Wizard displays. The wizard has detected that the data is delimited. Notice that the *Delimited* radio button is selected by default.
3. Click on the *Next* button.

   The *Tab* delimiter is selected by default.
4. Click on the *Comma* check box to select comma as the delimiter.
5. Click on the *Next* button.

   You may now select each column and customize it or click Next to complete the operation.
6. Click on the *Next* button to display the BOM report properly in the spreadsheet program.

How do I exclude testpoints from BOM?

To exclude testpoints from BOM, perform the following steps:

1. Copy the `bom.tempate` file from <*your_install_dir*>`\tools\fet\interface` to your local project.
2. Rename `bom.template` to `custom.template`.
3. Edit the `custom.template` file to include the following entry:

   ```
   BEGIN_PHYS_PARTS;
   BEGIN_EXCLUDE;
   NAME = TESTPOINT END_EXCLUDE;
   ```

   Note: `NAME= directory/cell name` of the part.
4. Run Packager-XL in the forward mode. For this, perform Export Physical.
5. Backannotate the design.
6. Run *Tools > Packager Utilities > Bill Of Materials*
7. In the *Template File* field select the `custom.template` file.
8. Enter the output file name in the *Output File* field. *
9. Click on the *Generate* button.

How can I remove BOM_PART or some other property from the BOM report?

All properties that are listed in a BOM report are the properties that have check boxes corresponding to them selected in the *Report Column* section of the Customize template - Physical Part Specifications tab. If you want to remove any of these properties from the BOM report, clear the check box corresponding to that property. For example, to remove the `BOM_PART` property from the BOM report, clear the check box corresponding to the `BOM_PART` property and save the template file.

You must have at least one property selected in the BOM report.

4

Cross-Referencing a Design

This chapter includes the following:

Overview

When you view a plot of a schematic, it is often difficult to trace a signal. The CRefer tool traces the signals in a schematic drawing and annotates their locations. Annotations by CRefer are called cross-references.

CRefer places the signal cross-references next to each signal and creates schematic reports that contain the list of signal and part cross-references.

Depending upon the nature of a signal, CRefer does the following:

- For each output signal in the schematic, CRefer lists all input locations where that signal appears.
- For each input signal, CRefer lists the locations of the sources of that signal.
- For each interface signal in a design, CRefer lists all the nets that are connected to it across the hierarchy.

Besides creating schematic reports, CRefer also creates text reports that contain cross-referencing information about signals and parts in a design.

About Cross-References

Types of Cross-References

CRefer places two types of cross-references: flat and hierarchical.

- **Hierarchical Cross-References** - When two signals in different blocks are connected to each other, CRefer creates hierarchical cross-reference for them.
- **Flat Cross-References** - When two signals in the same block are connected to each other, CRefer creates flat cross-reference for them.

  Note: If you select the *Distinguish Between Ports and Offpages* check box in the Cross Referencer Options - Content Tab, CRefer distinguishes between flat and hierarchical cross-references.

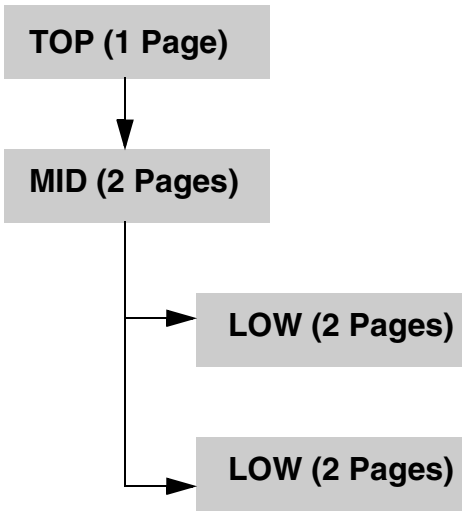
  - Flat cross-references are placed on ports or offpage symbols that do not have the `HDL_PORT` property on the pin.
  - Hierarchical cross-references are placed on ports or offpage symbols that have the `HDL_PORT` property on a pin.

  Note: For more information about assigning properties to ports and offpage symbols, see Adding Ports or Offpage Symbols to Signals on page 95.

How the Nature of a Design Influences Cross-Referencing

A design is either flat or hierarchical. The primary difference between a flat design and a hierarchical design is that all the pages in a flat design are sequentially placed as a continuous structure. A flat design is like a tall building; you cannot reach the fourth floor from the second floor without passing through the third floor. Similarly, to process page 4 in a flat design, you need to understand the contents of page 3. There is no way to bypass it although pages can be non-contiguous.

On the other hand, a hierarchical design is like a tree with multiple branches. A tree has multiple levels of hierarchy. The trunk divides into branches. Each branch has boughs and each bough has twigs, and each twig has leaves, flowers or fruits. Like a tree, a hierarchical design has multiple levels of hierarchy. Each level of hierarchy consists of a block, which can have a single page or multiple pages. Within the same block, the design behaves like a flat design. The following figure represents a hierarchical design.



In this hierarchical design, `TOP` is the parent block, which instantiates another block, `MID`. `TOP` and `MID` have a parent-child relationship: `TOP` is at the highest level, `0`. `MID` is at the next level, level `1`. `MID` is parent to two instances of the block `LOW`. And these two instances are at the same level, `2`. All these blocks have a different number of pages.

If you cross-reference signals in the same block, the cross-references generated are flat. For more information about the format of flat cross-references, see Flat Cross-References. If you cross-reference signals across multiple blocks, the cross-references generated are hierarchical. For more information about the format of hierarchical cross-references, see Hierarchical Cross-References.

Flat Cross-References

Flat cross-references are marked in the schematic on those ports or offpage symbols that link to signals on the same block. The format of a flat cross-references is:

```
Sheet# Ygrid Xgrid [Type]
```

where:

Sheet#	Represents the drawing sheet number.
Ygrid and Xgrid	Represents the grid coordinates where the signal appears. The grid coordinates are determined using the page border grid. If you select the *Omit Zone Information* check box in the Cross Referencer Options - Content Tab, the grid coordinates are not displayed.
Type	Represents the signal's I/O type. The I/O type is blank if it cannot be determined or if you have selected the *Omit Input/ Output Arrows* check box in the Cross Referencer Options - Format Tab. For more information about signal types, see I/O Types on page 4-116.

Example of a Flat Cross-Reference

A signal on page 1 that is input on page 3 receives a cross-reference label similar to

```
3A4<
```

This cross-reference specifies that the signal appears as input at the zone A4 on page 3.

Hierarchical Cross-References

You can direct CRefer to generate hierarchical cross-references either on the original schematic view (*sch_1*) or in a new view (*schcref_1)*. By default, hierarchical cross-references are generated on the original schematic view.

To specify that cross-references should be generated in the *schcref_1* view, which is created under the top-level cell of the current project, you can select the *Generate Flattened Schematic* check box in the Cross Referencer Options - Content Tab. The design is flattened, copied to the *schcref_1* view, and CRefer adds cross-references.

The format of a hierarchical cross-reference is:

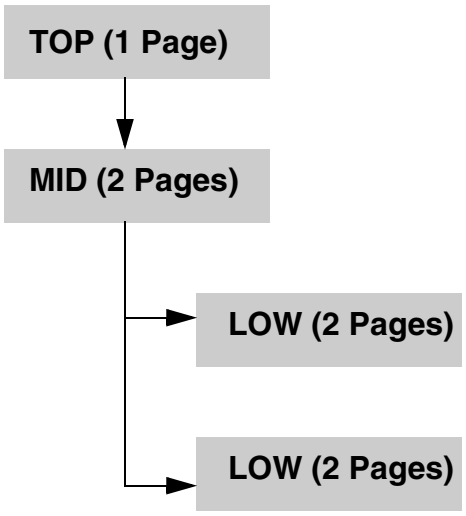
```
[Signal_Name@][_][Block_Name_@] Sheet# Ygrid Xgrid [Type]
```

Where:

Signal_Name@	If the signal is going up in the hierarchy, `Signal_Name` represents the name of the signal connected to the pin in the parent block. If the signal is going down in the hierarchy, `Signal_Name` represents the name of the signal in the child block to which it is connected. **Note:** The `Signal_Name@` is displayed if the *Show Signal Names in Hierarchical Cross References* check box is selected in the Cross Referencer Options - Content tab.
Block_Name_@	Denotes that the signal comes from the specified block. For the parent block, the cross-reference will include the name of the child block. However, for the child block, the block name will be the name of the child itself. **Note:** The `Block_Name@` is displayed if the *Show Block Names in Hierarchical Cross References* check box is selected in the Cross Referencer Options - Content tab.
Sheet#	Denotes the drawing sheet number. See Support for Design Entry HDL Custom Variables on page 4-118 for details about assigning design sheet variables to your design.
YGrid and Xgrid	Represents the grid coordinates where the signal appears. The grid coordinates are determined using the page border grid.
Type	Represents whether the signal is going up or down in the hierarchy. The type is blank if you have selected the *Omit Hierarchical Arrows* check box in the Cross Referencer Options - Format Tab.

Example of a Hierarchical Cross-Reference

The following figure represents a hierarchical design.



In the hierarchical design above, if the signal CLK in TOP is connected to the pin A in MID, the cross-reference of the pin A in MID will be:

```
1C7^
```

This cross-reference indicates that the CLK signal connected to the A pin in MID is from the C7 zone (Ygrid=C, Xgrid=7) of page 1 of TOP. The ^ character indicates that CLK is a hierarchical signal.

However, if you have selected the *Show Signal Names in Hierarchical Cross References* and the *Show Block Names in Hierarchical Cross References* check boxes, the cross-reference for the signal CLK would be as follows:

```
CLK@_TOP_@1C7^
```

CRefer will also append a cross-reference to the CLK signal in TOP. This cross-reference will be:

```
2B8v
```

This cross-reference indicates the CLK signal is connected to the pin corresponding to zone B8 on page 2.

Note: If the signal name used in a hierarchical block does not match with the pin name used in the symbol for that block, CRefer inserts a cross-reference with the following syntax:

*<signal_name_at_top_level>-<Page_number_at_top_level>[path_name_of_block]^*

For example, the following cross-reference is placed when the signal name used in a hierarchical block does not match with the pin name used in the symbol for that block:

```
B<0>-1[I1]^
```

Note: The *schcref_1* view is an output only view created by CRefer. Do not make any design changes to it as CRefer cross-references the design based on the *sch_1* view. Each time you cross-reference the design, the *schcref_1* view is created again. Therefore, any changes you make in the *schcref_1* view will be lost.

Note: The `PAGE_NUMBER` directive is added with reference to the `lowerleft` directive in the `cref.dat` file. The origin of the page is calculated as:

```
(-lowerleft_x, -lowerleft_y)
```

The `PAGE_NUMBER` directive value in the `cref.dat` file is offset from the origin calculated from the `lowerleft` directive in the `cref.dat` file. For example, if you need the `PAGE_NUMBER` directive at location `(0,0)` when the `lowerleft` directive is defined as `(1000,200)`, you need to define the `PAGE_NUMBER` directive as `(1000,200)`.

How CRefer Cross-References a Design

Cross-Referencing a Design

When you cross-reference a design, CRefer first parses the design and differentiates blocks based on the following:

- Replicated versus non-replicated—CRefer treats a block as replicated if it is a hierarchical block and if it is being used at two or more places in the design. If a particular block is being replicated, the entire subtree under it, that is, all the blocks within it, are also treated as replicated.
- Read-only versus writable—Based on whether a block is read-only or writable, CRefer places the cross-references on the schematic, or in the OPF file (`cref.opf`). Design Entry HDL reads the `cref.opf` file and displays cross-references on the schematic.

  For replicated hierarchies, and read-only blocks in the schematic, CRefer creates cross-references in the `cref.opf` file in the sch_1 view of the top-level cell.

Important

For cross-references to be read from the `cref.opf` file, you must add offpage bodies on all the nets that you want cross-referenced.

Regardless of whether a block is writable, read-only, replicated, or non-replicated, adding offpage symbols to nets that you want cross-referenced is helpful for the following:

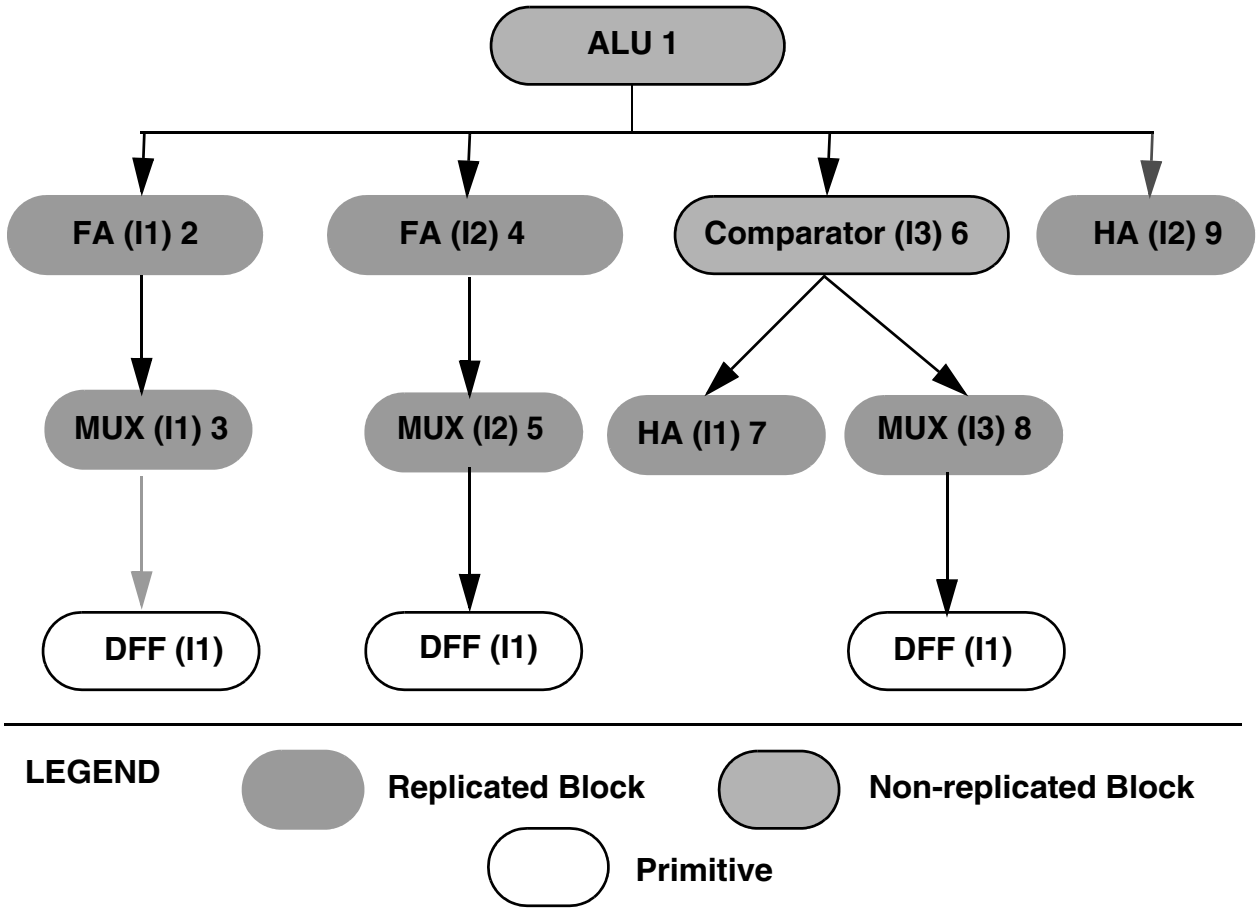
- You can specify the placement of the cross-references being generated with the help of placeholders and edit the offpage symbols as required.
- You can control the visibility of cross-references on the schematic by setting offpage properties.

Depending upon your design, you may have multiple scenarios. For example, consider the design in the ALU Hierarchical Design figure on page 79. Note that the design is named `ALU`, and that this design includes multiple levels of hierarchical blocks.

The `ALU` block contains four blocks, which include two instances of `FA`, one instance of `Comparator`, and one instance of `HA`. The `FA` block is replicated. CRefer will treat the entire hierarchy under both the `FA` blocks as replicated.

Note that the `MUX` block is also replicated. There is one instance of `MUX` under the `FA (I1)` block and another instance under the `Comparator` block. CRefer will treat both instances of `MUX` as replicated hierarchy. This means that the `MUX (I1)` block and its child block `DEF (I1)` are both treated as replicated hierarchies.

Figure 4-1 ALU Hierarchical Design



Naming Convention in Block: Block name (instance name) Design current sheet value

Note that the `HA` block is also replicated. One `HA` block exists below the `Comparator` block, while another instance of the `HA` block exists below the `ALU` block.

In summary, CRefer will treat the `ALU` and `Comparator` blocks as non-replicated hierarchies and all other blocks as replicated hierarchies.

Cross-References for a Hierarchy

For non-replicated blocks, that is `ALU` and `Comparator`, CRefer annotates the properties on the placeholders specified on ports or offpage symbols. If you have not defined any

placeholders on the ports or offpage symbols on the schematic, CRefer creates placeholders and attaches cross-references to the ports or offpage symbols.

Create placeholders on the ports or offpage symbols to ensure that cross-annotations are placed as desired on the schematic. The placeholder should be a property name-value pair.

If there are any `XR` or `$XR` properties on the schematic that are not attached to any ports or offpage symbols, CRefer deletes those properties.

If you have not used ports or offpage symbols on the signals that need to be cross-referenced, select the *Cref Signals Not Connected to Flag Bodies* check box in the Cross Referencer Options - Content tab so that CRefer cross-references these signals.

If a net is in the parent block and in two child blocks, while cross-referencing, the parent block is assigned two XR properties while each child block is assigned only one XR property for the parent. From one child design module, to identify other child design modules of the same parent, you can use the XR properties of the parent module.

If a net occurs on two pages, for example `page 1` and `page 2` of a block, and also down the hierarchy from `page 1`, there will be XR properties on `page 1` for the hierarchy below it and one for `page 2`.

Cross-References for a Replicated Hierarchy

For all replicated hierarchies, such as `FA` (see ALU Hierarchical Design on page 4-79), CRefer creates cross-references in the `cref.opf` file in the `sch_1` view of the top-level cell (`ALU`). The `cref.opf` file is a binary file.

Important

You need to add ports or offpage symbols to the schematic for all the nets that you want cross-referenced.

When CRefer runs out of placeholders, it places all the remaining cross-references on top of the last placeholder. The following message is also generated:

```
Signal <signal_name> at <location_of_SIG_NAME> required
<number_of_required_placeholders> placeholders on
<name_of_the_attached_plumbing_body>.
```

If there is no placeholder attached to a port or offpage symbol, CRefer creates placeholders and annotates cross-references.

Cross-References for Read-Only Blocks

If you have read-only blocks in the schematic, CRefer will add cross-references in the `cref.opf` file in the `sch_1` view of the top-level cell (`ALU`).

Important

You must add ports or offpage symbols to the schematic for nets that you want cross-referenced.

Controlling CRefer Annotations Using UI Options

- **Cref Signals Not Connected To Flagbodies -** By default, CRefer ignores signals that are not connected to flag bodies (offpage or port). If you want CRefer to read and process such signals, select the *Cref Signals Not Connected to Flagbodies* check box. CRefer will attach cross-references to the `SIG_NAME` property of signals.

  For example, you have a signal A<2..0> synonym-ed to another signal B<2..0> at the top level. The A<2..0> signal is connected to a flag body while signal B<2..0> is not. If you do not select the *Cref Signals Not Connected to Flagbodies* check box, neither of the two signals will be cross-referenced.

  Note: If you select the *Cref Signals Not Connected To Flagbodies* check box, CRefer will not be able to correctly cross-reference signals in a replicated hierarchy and read-only blocks in the `sch_1` view.

  To cross-reference a design with replicated or read-only blocks, you must place flag or port bodies on all signals that need to be cross-referenced.

Caution

CRefer is optimized to generate cross references for signals with input/ output type specified as a flag body. It is strongly recommended that you use flag or port bodies for cross- referencing purposes and not use the Cref Signals Not Connected to Flagbodies check box. Using flag or port bodies will help you avoid placement problems. You should define all flag bodies in the Cref data file.

If you do not add flag bodies to replicated or read-only blocks and check the *Cref Signals Not Connected to Flagbodies* box, CRefer will add empty placeholders, such as `$XR0=?`, to these signals. CRefer will also not annotate cross references in such cases.

- **Redo Placement of Crefs -** By default, CRefer retains the previous placement of cross-references. If you select the *Redo Placement of Crefs* check box, the previous

placement is lost, and CRefer uses the original placeholders on the ports or offpage symbols to place cross-references.

- **Add Crefs as Hard Properties -** All placeholders created by CRefer are soft properties (`$XR`). However, if you select the *Add Crefs as Hard Properties* check box in the Cross Referencer Options - Format tab, all placeholders are converted into hard properties (`XR`).

Summarizing In-Place Cross-Referencing and Place Holder Support

CRefer uses the algorithm displayed in the In-Place Cross-Referencing of Signals figure on page 83 and In-Place Cross-Referencing for Blocks figure on page 84 to cross-reference any design. Note that the action performed by CRefer is based on the following factors:

- Does a signal have offpage symbols or ports?
- Does a signal have placeholders?
- What is the nature of the block (replicated, non-replicated, or read-only)?

The behavior of CRefer for various types of designs will be as follows:

Legacy designs with XRs attached to SIG_NAMEs and no placeholders in the library

CRefer searches all `XR` properties attached to signals or ports and retains their values. However, CRefer attaches `XR` properties to the ports or offpage symbols instead of `SIG_NAMEs`.

Legacy designs with XRs attached to SIG_NAMEs. The library has been updated with placeholders, and the hier_write operation has been performed in Design Entry HDL.

By default, CRefer honors the existing placement, but attaches the `XR` properties to offpage symbols. If the *Redo Placement of Crefs* check box is selected, CRefer treats the design as a new design and puts `XR`s in the placeholders provided in the library.

Legacy designs with a few new pages added. The library in the design has placeholders.

By default, CRefer honors existing placement and uses placeholders wherever `XRs` were not originally present. If a signal had two `XRs` initially, and is assigned 3 `XR`s in the next run due to changes in the design, CRefer places the new `XR` using its own algorithm. It will attach the cross-references to ports.

New designs with placeholders

CRefer always uses placeholders for placement and honors any changes made in the placement in the later runs.

Figure 4-2 In-Place Cross-Referencing of Signals

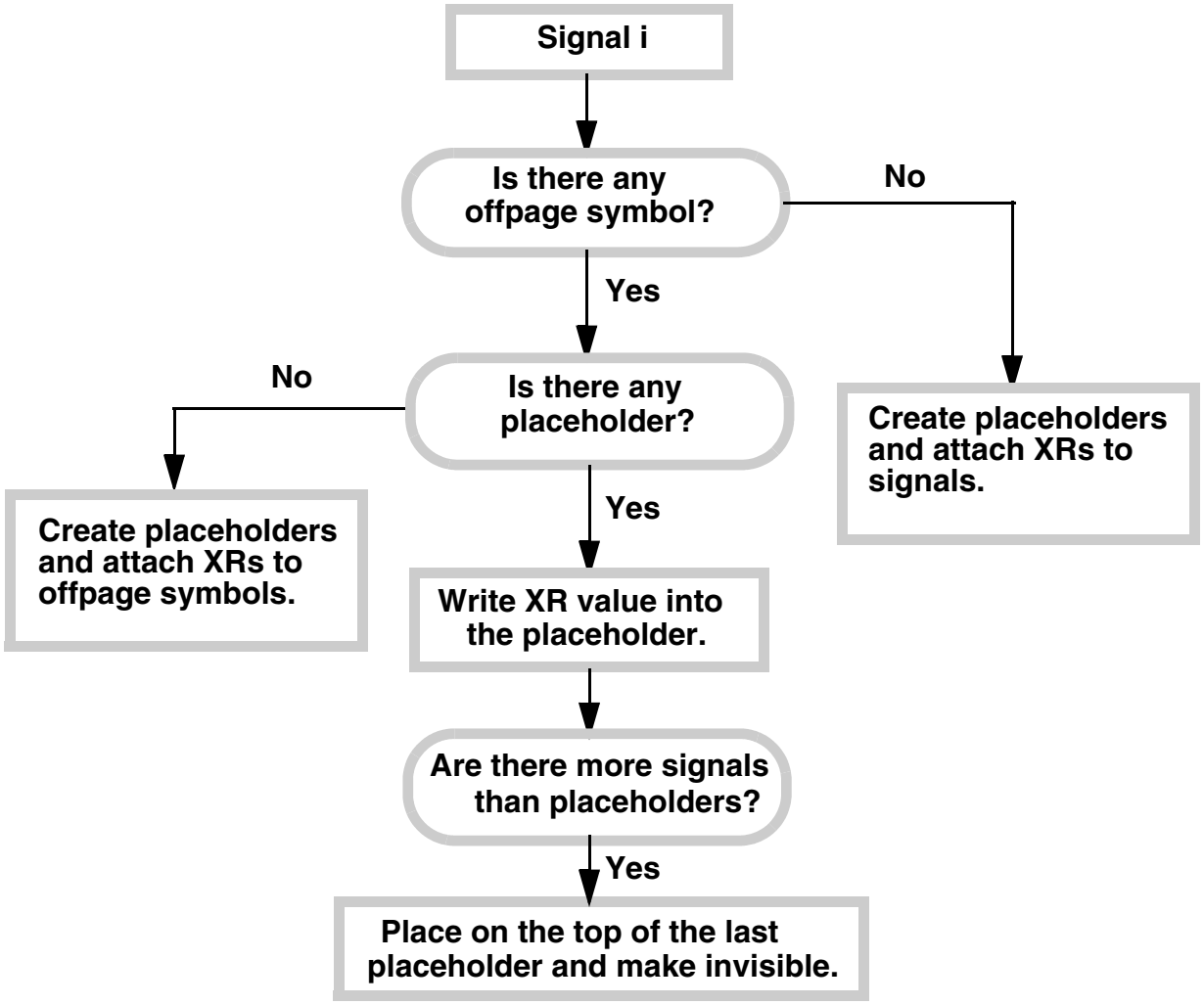
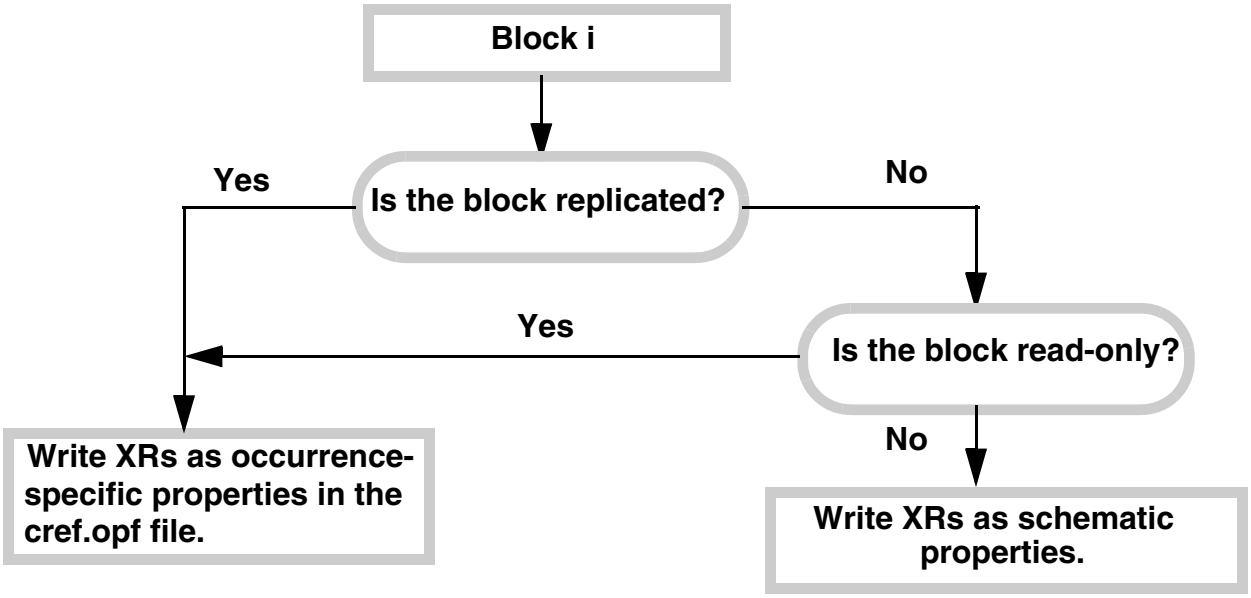


Figure 4-3 In-Place Cross-Referencing for Blocks



Getting Started with CRefer

Before you cross-reference a design, you need to do one or all of the following procedures:

- Preparing the Design for Cross-Referencing on page 84
- Determining the Right Cross-Referencing Options on page 86
- Working with the Cref Data File on page 88
- Creating the Cref Data File for Page Borders on page 91
- Creating Custom Offpage I/O Flag Bodies on page 92
- Making Cross-References Permanently Visible on page 94
- Adding Ports or Offpage Symbols to Signals on page 95

Preparing the Design for Cross-Referencing

To prepare a design for cross-referencing, ensure the following:

1. The quality of cross-references generated by CRefer is directly proportional to the quality of the design entered in the Design Entry HDL schematic. You should ensure that the

schematic has enough space for CRefer to place cross-references. Therefore, design the schematic with cross-referencing in mind. For all signals that you want cross-referenced, leave enough space.

In particular, ensure that you leave enough room around flag/port/offpage symbols so that CRefer annotations are placed properly. The wire-to-wire spacing between nets where flag/port/offpage symbols are attached should be increased as necessary.

2. It is recommended that you add a `$XR<n>` placeholder on the port or offpage symbol that you want to be cross-referenced. This will ensure that you get cross-references at the desired place on the schematic. For more information about adding placeholders, see Adding Placeholders on Ports or Offpage Symbols on page 125.

3. Ensure that the `OFFPAGE=TRUE` property is attached to all ports and offpage symbols.

4. CRefer determines whether a signal needs to be cross-referenced if it has an offpage body, port, or offpage symbol attached to it. Therefore, ensure that you have attached an offpage body, port, or offpage symbol to every signal that you want cross-referenced.

   You can, however, direct CRefer to cross-reference all signals whether or not they are connected to flag bodies or ports. For this, select the *Cref Signals Not Connected to Flag Bodies* check box in the Cross Referencer Options - Content Tab.

   Note: CRefer has a different signal processing algorithm for signals connected to pins and wires. It is recommended that you connect signals to wires to allow optimum cross referencing.

5. Ensure that all the schematic pages in your design use one of the page borders available in the standard library. You can also use custom page borders in your design.

6. You can add custom text for CRefer variables to for clearer annotations. For example, you can place custom text such as “This block goes to page <*CREF_TO_LIST*>” in the schematic. To add custom text, use the Design Entry Options and Custom Text dialog boxes in Design Entry HDL.

7. If you want to change the order of the modules (that is, hierarchical blocks), use the Hierarchy Viewer window in Design Entry HDL. Similarly, if you want to exclude certain modules from the hierarchical design, exclude those modules in the Hierarchy Viewer window.

8. Ensure that you have packaged the design before cross-referencing it. This is important if you want to generate the Parts Cross Reference report (crefparts.txt). For more information about packaging a design, see *Packager-XL Reference*.

9. If you want to see OPF properties as visible in the *schcref_1* view, backannotate the design.

Determining the Right Cross-Referencing Options

After you have prepared the design for cross-referencing, you must set the right cross-referencing options for your design depending on your needs:

1. If your drawings use page borders other than those provided in the standard library, create the cref data file (`cref.dat`), which defines your custom page borders. Specify the path to the cref data file in the *Cross Referencer Options – Cref Data File* tab. This path will be written into the project file, and will be automatically available the next time you cross-reference your design.

2. Ensure that you have added a description about all offpage flag bodies and ports in the cref data file for the project (even if the same description also exists in the site/Cadence-level Cref data file). For more information about adding custom offpage I/O flag bodies, see Creating Custom Offpage I/O Flag Bodies on page 4-92.

3. If you want to suppress cross-referencing of specific signals or generate cross-references for power signals, you should list them in the cref data file. For more information about suppressing cross-referencing of certain signals, see Suppressing Cross-Referencing of Signals on page 4-92 or Creating Cross-References for Power Signals on page 4-92.

4. If you have cross-referenced a design once and want to retain the placement of cross-references during repeat cross-referencing, do not make changes to the cross-reference format. For example, do not change the display of zone, signal, or block information, or the text size or spacing. If you do make changes to the cross-reference format, select the *Redo Placement of Crefs* check box in the Cross Referencer Options - Format Tab to ensure that the cross-references are placed again.

5. If you need to split the bus at the lower level in the hierarchy, it is recommended that you use the complete bus for the signal name at the lower level and tap its individual bits. This will provide better cross-reference reports.

6. If you want to write the cross-referencing options in the project file, use the CRefer dialog box. Changes made using this dialog box are stored in the project file. However, changes in cross-referencing options using the command-line prompt are only used for that CRefer run. These changes will not be stored in the project file.

7. If you want to find the sheet number of current page or find the total number of pages present in the schematic, use sheet numbering.For more information about adding any CRefer custom variable (which includes sheet numbering variables), see Adding CRefer Custom Variables on page 4-119.

8. If you want to distinguish between hierarchical and flat cross-references, select the *Distinguish Between Ports and Offpages* check box in Cross Referencer Options -

Content tab. This will ensure that hierarchical cross-references are placed on ports and flat cross-references are placed on offpage symbols.

9. If you have multiple users who use the same cross-referencing option, you can create a site project file and save the default cross-referencing options in it. To create a `site.cpm` file, use an existing projects project file, or create a dummy project and use its project file to define your site settings. For more information about creating a site project file, see *Allegro Design Entry HDL User Guide*.

Note: You can cross-reference a schematic from the command-line prompt. However, it is recommended that you use the CRefer dialog box to make all cross-referencing settings, and then, if required, use the command-line prompt to cross-reference the design.

To cross-reference a design from the command-line prompt, use the following syntax:

```
creferhdl -proj <project_file> [-d] [-e] [-i] [-expand][-l] [-o] [-p] [-q][-r] [-s] [-z]
```

`-proj`	specifies the path to the project file you want to cross-reference
`-d`	Deletes all existing cross references in the design
`-e`	Retains duplicate entries
`-i`	Omits input output arrows in cross references
`-expand`	Creates a separate view for this run
`-l`	Inserts Block Names in hierarchical cross references
`-o`	Sorts by page number only
`-p`	Specifies a cross reference - flag body/wire spacing, in1/200 inch
`-q`	Specifies a cross reference - cross reference spacing, in 1/200 of an inch
`-r`	Creates all cross references again
`-s`	Scales text
`-z`	Omits zone information from cross references

Use the `-d` option to delete all the existing cross-references in the design.

Note: After cross-referencing a design from the command-line prompt, it is recommended to close the design, and open it again to see the cross references in the design.

Working with the Cref Data File

Creating a Cref Data File

The Cref data file (`cref.dat`) is a hand-edited file. You can use the default (standard) Cadence-supplied `cref.dat` as a template to create a custom `cref.dat` file. The default `cref.dat` file is stored in the following location:

`<your_install_dir>/share/cdssetup/creferhdl/cref.dat`.

The Default Cref Data File

The following is an excerpt from the Cadence default `cref.dat` file for the A SIZE PAGE Border:

```
pagename "A SIZE PAGE"
version 1
lowerleft (-3750, 0)
upperright (0, 5000)
xmark "1" -500
xmark "2" -1500
xmark "3" -2500
xmark "4" -3425
ymark "A" 950
ymark "B" 2200
ymark "C" 3350
ymark "D" 4475
excludearea "1" (-1700,0) (0,400))
schrep “basenets” (1,22) (3,30)
```

Table 4-1 on page 89 provides a description of the directives used in the `cref.dat` file. You can create a copy of this file, use it as a template, and set the values of various directives per your requirements.

Table 4-1 Cref.dat Directives

Cref.dat Directive	Description
`pagename`	Identifies the page border symbol name (the name that you would use with the `ADD` directive in Design Entry HDL).
`version`	Identifies the symbol version of the page symbol; you can omit this directive if the version number is 1.
`lowerleft`	Sets the lower left corner coordinate, in default units. To determine the coordinate, use the `SHOW COORDINATE` directive in Design Entry HDL and click at that point.
`upperright`	Sets the upper right corner coordinate, in default units. The coordinates for `lowerleft` and `upperright` should define the largest possible rectangle that fits on the sheet, taking into account the space occupied by the title box.
`xmark`	Specifies the horizontal coordinate of the cross-reference key letters or numbers located along the top or bottom of the page. When you use the `SHOW COORDINATE` directive, click the mouse directly on the top of the number or letter; do not click on the boundary lines.
`ymark`	Specifies the vertical coordinate of the key letters or numbers along the left or right side of the page. Click the mouse directly on the top of the number or letter when you use the `SHOW COORDINATE` directive; do not click on the boundary lines.
`pagenumber`	Specifies the coordinate where CRefer prints a page number for the signal and part cross-reference summary sheets. You can omit this directive if you do not want the pages numbered.
`pagenote`	Prints a text note anywhere on the page. You can use as many notes as required.
`excludearea`	Specifies keep-out areas for page borders while writing schematic reports. You can specify one or more exclude areas for a single page border. To specify the area, you use Design Entry HDL coordinates. For example, in the following line, (-2500,0) (0,375) represent the lower left and upper right coordinates: `excludearea "1" (-2500,0) (0,375)` At runtime, if this entry is incorrect, the `cref.log` file displays a non-fatal error but CRefer will complete the run. This directive is optional and its absence in the `cref.dat` file does not show up as an error.

schrep	Allows report customization by letting you specify the name of the report, the column numbers to be printed, and the maximum number of characters to be printed in each column.
	
	You can customize basenets, netsbypage, synonyms, and crefparts report using the schrep directive. However, you cannot customize CreferHDL text reports using the schrep directive.
	
	The default values for the schematic reports are listed below:
	
	`schrep "basenets" (1,20) (2,35) (3,25)`
	`schrep "netsbypage" (1,7) (2,15) (3,20) (4,15) (5,21)`
	`schrep "synonyms" (1,15) (2,20) (3,25) (4,21)`
	`schrep "crefparts" (1,10) (2,20) (3,50)`
	
	You need to make separate entries per page border. If an entry for a report does not exist for the page border being used, the defaults for that report will be used.
	
	Note: The schrep directive must be specified at the page border level. If you define multiple page borders in the cref.dat file, you need to specify the schrep directive for each page border.
	
	If the given column widths cause the total number of characters for a row to exceed the width of the page border, CRefer uses its own defaults and give an error in the schrpt.log file. If any column entry exceeds the specified width for the column, it will continue into the next line.
	
	In the following example, 'basenet' is the name of the report, the first of the coordinates (1 and 3) is the column number and the second (22 and 30) is the column width. Each column information is stored within parenthesis. Column two has not been specified and will not be generated in the output.
	
	`schrep "basenets" (1,22) (3,30)`
	
	You can also add comments in the cref.dat file using the curly braces:
	
	`{ schrep "basenets" (1,25) (2,30) }`
	
	Note: Anything within curly braces will be treated as a comment line.
	
	Note: For more information about CRefer reports, see "Defining Output Reports" on page 103.

The Cadence B SIZE PAGE border is as follows:

```
pagename B SIZE PAGE
version 1
```

```
lowerleft (-8125, -125)
upperright (125, 5125)
xmark 1 500
xmark 2 1500
xmark 3 2500
xmark 4 3500
xmark 5 4500
xmark 6 5500
xmark 7 6500
xmark 8 7500
ymark A 1000
ymark B 2200
ymark C 3325
ymark D 4500
pagenumber (-425, 50)
pagenote (-2000, 50) Eng Name
pagenote (-500, 250) date
excludearea "1" (-8000,0) (0,400)
```

Creating the Cref Data File for Page Borders

To create cross- references, you must use a page border for each page of the design. The Cadence *standard* library provides six standard page borders—`A SIZE PAGE` to `F SIZE PAGE`. These page borders are the first versions and can be identified by their version numbers (1 in these cases). The details of these page borders are available in the cref.dat file.

You can use the same or different page borders for the pages of the design. To use different page borders for various pages of the design, you must define the page borders in the `cref.dat` file in a sequential order.

You can:

1. Use a standard page border.
2. Modify a standard page border and use it.
3. Create and use your own page border.
4. Specify exclude areas for page borders.
5. Customize schematic reports.

Creating Custom Offpage I/O Flag Bodies

You can create and use your own offpage bodies. The body shape does not matter, but the `COMMENT_BODY=TRUE` property must be attached to the body drawing. Version 1 of the `OFFPAGE` flag body is shown below as an example.

To identify your custom offpage bodies; you must specify their I/O type in the `cref.dat` file. Each version of the body must be declared separately. Typically, you need six versions of an I/O flag: input, output, and bi-directional flags facing both left and right.

The syntax for declaring offpage bodies is:

```
INFLAG "flag name " VERSION number
OUTFLAG "flag name " VERSION number
BIFLAG "flag name " VERSION number
```

For example, you can declare six versions of an offpage body named CROSSFLAG as follows:

```
INFLAG "CROSSFLAG" VERSION 1
OUTFLAG "CROSSFLAG" VERSION 2
BIFLAG "CROSSFLAG" VERSION 3
INFLAG "CROSSFLAG" VERSION 4
OUTFLAG "CROSSFLAG" VERSION 5
BIFLAG "CROSSFLAG" VERSION 6
```

Creating Cross-References for Power Signals

By default, the power signals—VCC, NC, VSS, GND, 0, and 1—are not cross-referenced. To create cross-references for the power signals, define the `NONTRIVIALNET` directive in the `cref.dat` file:

```
NONTRIVIALNET "Signal name in quotes"
```

Suppressing Cross-Referencing of Signals

You can suppress cross-referencing of any signal by specifying the signal name in the cref.dat file. By default, the power signals-VCC, NC, VSS, GND, 0, and 1-are not cross-referenced. To suppress cross-referencing of signals other than those listed, use the `TRIVIALNET` directive in the *cref.dat* file:

```
TRIVIALNET "Signal name in quotes"
```

Note: The `TRIVIALNET` directive is useful to ignore global signals while cross-referencing a design.

Determining Coordinates

To determine the xmark and ymark coordinates in a page, do the following:

1. Open the schematic in Design Entry HDL.
2. Choose *Display — Coordinate*.

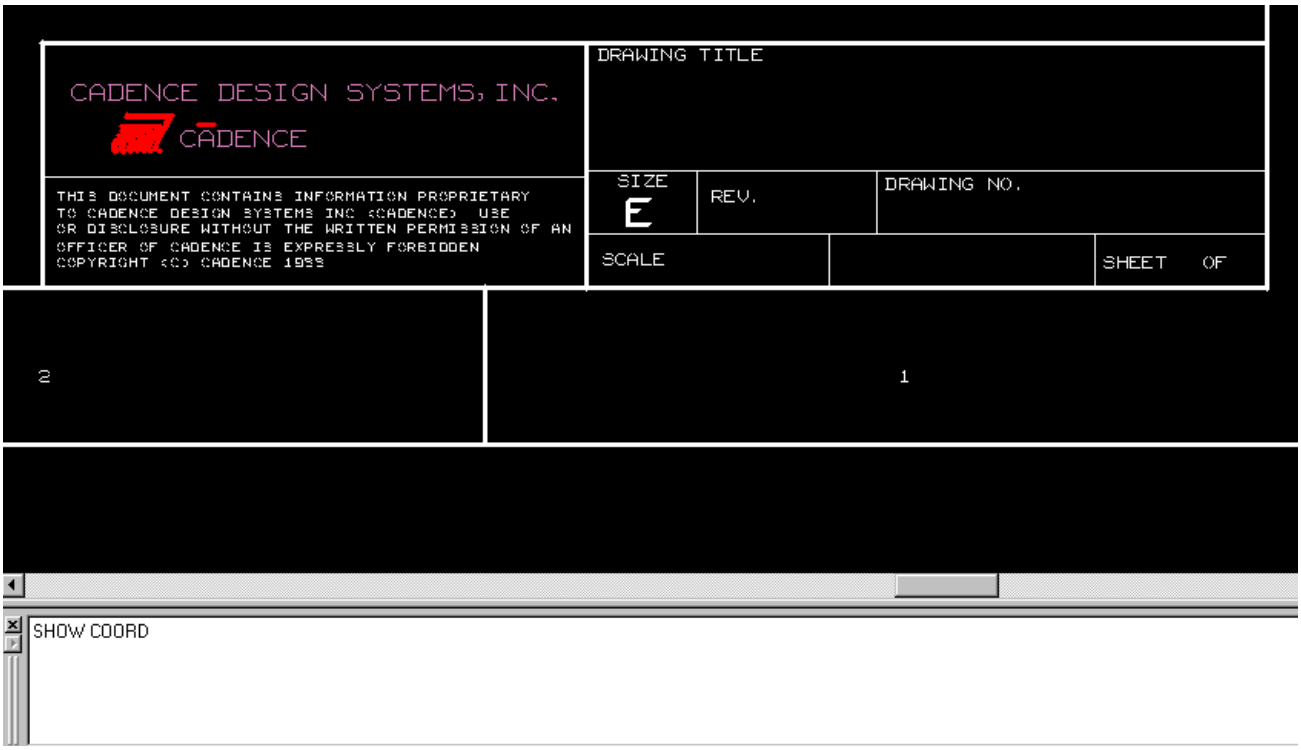
   Note: You can also use the Console Window to display coordinates. For this, type `SHOW COORDINATE` (or SHOW COORD) in the Design Entry HDL console and press Enter.
3. Click on the number or letter of the Zone (For example, 1 or A). The coordinate value is displayed in the Design Entry HDL status bar.

   Note: Click the mouse directly on the top of the number or letter; do not click on the boundary lines.
4. If the Zone number or Zone letter is to the left of the Origin, subtract the X coordinate from the Origin X coordinate.
5. If the Zone number or Zone letter is to the right of the Origin, add the X coordinate to the Origin X coordinate.

The resulting number is the value of xmark. The same calculation also applies to ymark.

To understand how to determine coordinates, consider the following example:

Part of a page border is displayed below. The origin of the page border is located at the center of the page. All zone numbers and letters displayed are to the right of the origin.



The `Show COORD` command is entered. You can click on any Zone letter or Zone number in the graphic to determine its coordinate. To find the value of the coordinates for the Zone number “1”, click on the number 1 in the above graphic. Design Entry HDL will return the xmark and ymark for the Zone number “1”.

Making Cross-References Permanently Visible

To make cross-references permanently visible, do the following:

1. Start Design Entry HDL and edit the schematic.
2. Group the invisible `$XR` properties.

   ```
   FIND $XR*
   ```

3. Make the property values visible.

   ```
   DISPLAY VALUE “A”
   ```

   ‘A’ is the group name assigned by the *Find* command. If you have already created some groups in the current Design Entry HDL session, the group you created with the *Find* command may have another name.

4. Use the *Next* command in Design Entry HDL to automatically zoom in on each property.

Note: It is recommended that you make the XR placeholders on offpage connectors visible. The placeholders will become visible on the schematic only when the XR properties are annotated on the schematic.

Adding Ports or Offpage Symbols to Signals

Adding offpage symbols

Add offpage symbols to all signals that will have flat (offpage/onpage) cross-references.

To add an offpage symbol, assign the following property set for the `SRC`, `ONSRC`, `ONDST`, and `DST` symbols in the existing libraries:

```
OFFPAGE=SRC|ONSRC|ONDST|DST
```

Note: Ensure that this property is added on the original symbols and not on the pins.

Adding Ports

Add ports to all signals that will have hierarchical cross-references by doing the following:

1. Assign the property `OFFPAGE=TRUE` to the port.
2. Assign the following property set to the port:

```
HDL_PORT=IN|OUT|INOUT
```

Using CRefer

You can directly cross-reference a design or you can first customize the cross-referencing options and then cross-reference the design. To cross-reference a design properly, you need to do one or all of the following:

- Deleting Cross-References on page 107

Cross-Referencing the Design

To cross-reference a design, do the following:

1. Ensure that the librarian has made the necessary changes as mentioned in Placeholder Support on page 124.
2. Make sure that you have added ports or offpage symbols to all signals that require cross-annotation. This will ensure that cross-references assigned by CRefer will move along with ports allowing for predictable placement in the schematic.
3. Open the schematic and perform the `hier_write` operation (*File — Save Hierarchy*) on the schematic of the root drawing. This operation will assign the `PATH` property to all the offpage symbols.
4. Close the schematic by selecting *File — Exit* in Design Entry HDL.
5. Cross-reference the design.

After the design is cross-referenced, you can open Design Entry HDL and view cross-references. You will see that CRefer has made annotations for the entire design. CRefer has also substituted the variable values for all custom text that includes CRefer-specific custom variables.

Note: It is recommended that you should not edit the cross references until you have performed final cross-referencing.

Generating Cross-References for a Design

You can cross-reference a design using the *CRefer* dialog box (that is, by using the user interface) or by running `creferhdl` command from the command prompt.

To open the *CRefer* dialog box, do one of the following:

- Select *Tools — CRefer* from the *Project Manager* menu bar.

  The *CRefer* dialog box appears with the name of the project file selected in the Project Manager.

- You can also double-click on the `creferui.exe` file from `<your_install_dir>/tools/bin`. CRefer displays the file browser. Select the project file that is to be cross-referenced.

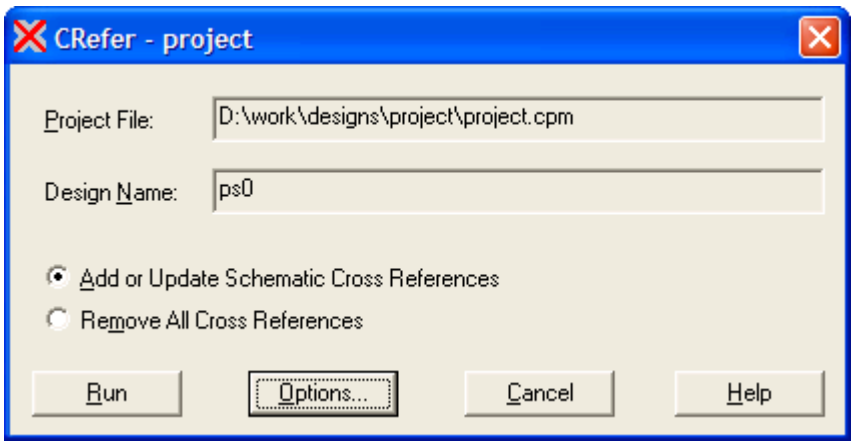
To cross-reference a design using the `creferhdl` command, use the following syntax:

```
creferhdl -proj <project_file>
```

where, `-proj <project_file>` is the path to the project file you want to cross-reference. Use the `-d` option to delete all the existing cross-references in the design.

Note: Before you cross-reference a design from the command prompt, set all cross-referencing options using the *Cross Referencer Options* dialog box.

CRefer Dialog Box



After you have opened the CRefer dialog box, use the following procedure to cross-reference the design.

1. If you want to change the Cross Referencer settings, click on the *Options* button.

   The Cross Referencer Options dialog box is displayed. You can change the Cross Referencer settings here. When you have changed the settings, click on the *OK* button to return to the CRefer dialog box.

2. To cross-reference a design, select the *Add or Update Cross References* radio button.

3. To start the cross-referencing process, click on the *Run* button.

   The CRefer Progress Window is displayed indicating the progress of the design cross-referencing.

   Note: CRefer does not generate hierarchical XRs for nets whose base net does not appear in the given block. To generate XRs for a top-level block, CRefer looks for all the nets in the lower-level block that have the same name. For a bus, it checks whether the net(s) lies within the range.

If the complete bus is aliased with scalars, the scalars are recognized as base nets in the netlist for the top-level block. Therefore, no XRs are generated for the bus.

If any interface net is aliased to another net in the same block, such that the base net is not the interface net, the hierarchical XR is not generated in the block where it is aliased.

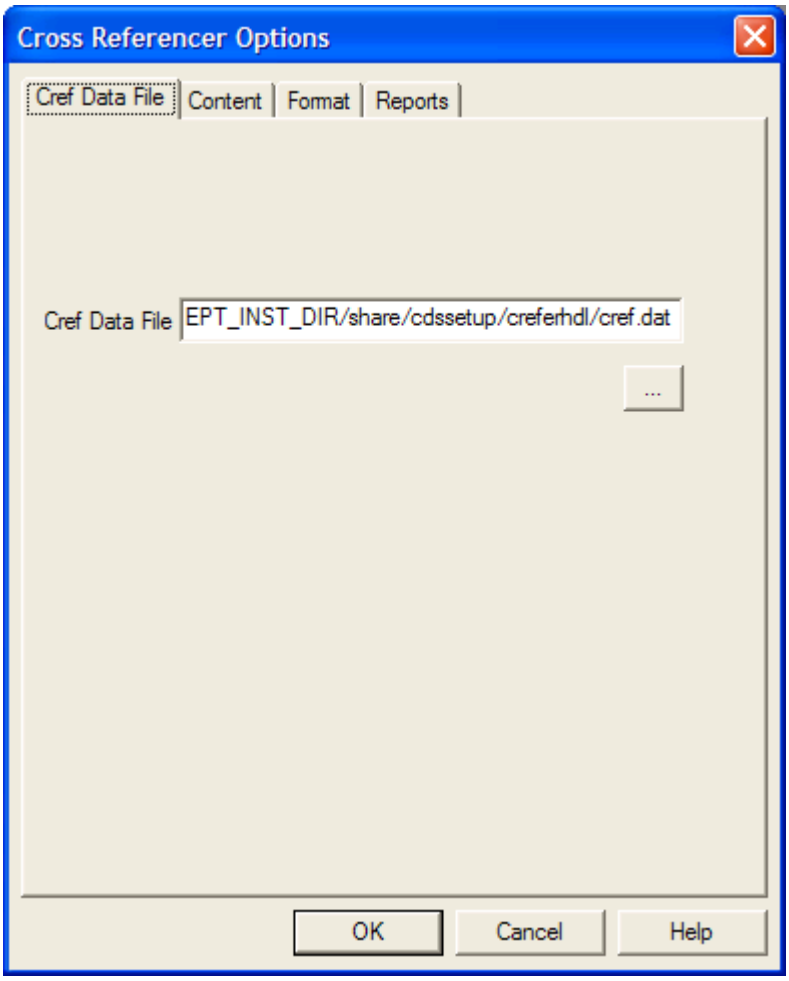
Changing the Cref Data File

By default, CRefer uses the CSF search to locate the Cref data file specified in the project file to obtain the default page border, offpage bodies, and information about signals that need be cross-referenced.

You can change the Cref data file in the Cross Referencer Options - Cref Data File Tab by:

- entering the path of the cref data file in the *Cref Data File* field
- browsing to the new file by using the *browse* button.

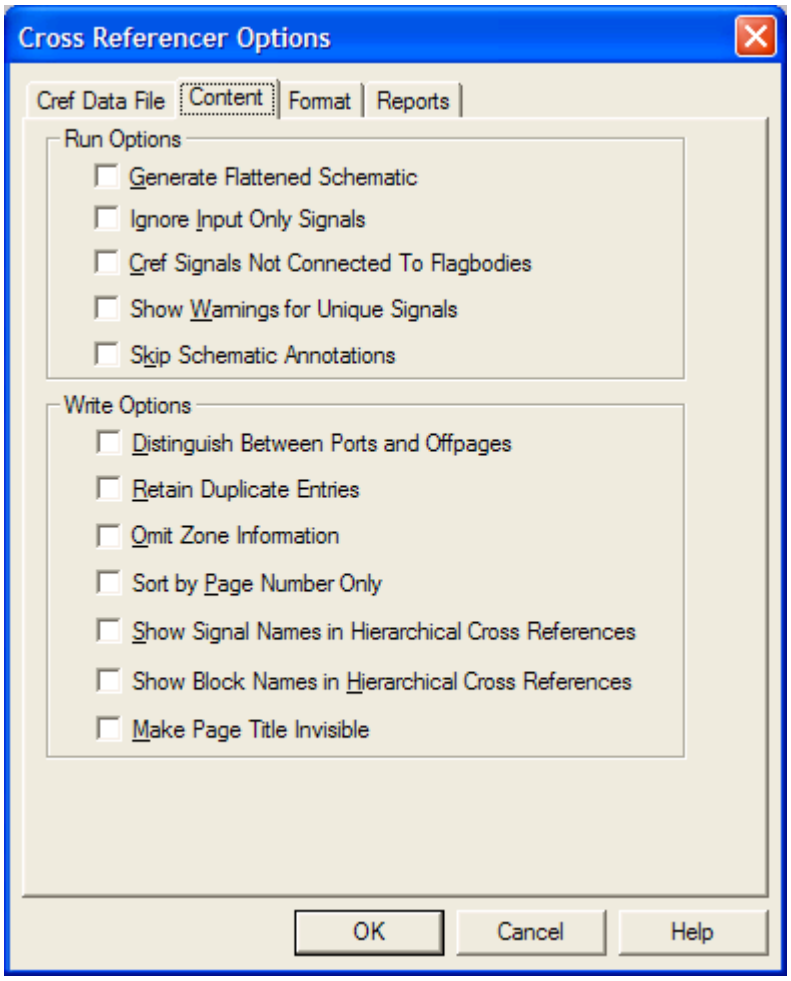
Cross Referencer Options - Cref Data File Tab



Configuring Run and Write Options

To configure the default run and write options, use the Cross Referencer Options - Content tab. You can select the signals that CRefer can ignore while running. You can also define how CRefer should write the cross-references in the project (.cpm) file.

Cross Referencer Options - Content Tab



1. To create a new flattened view (schcref_1) view in the top-level cell for the current project) for the cross-referenced design, select the *Generate Flattened Schematic* check box.
2. To ignore input-only signals, select the *Ignore Input Only Signals* check box. These signals will also be ignored in the schematic reports.
3. By default, CRefer ignores signals that do not have flag bodies or ports. If you want CRefer to read and process signals that do not have flag bodies or ports attached, select the *Cref Signals Not Connected to Flag Bodies* check box. These signals will also be placed in the schematic reports.
4. To display warnings for signal names that occur only once in the design, select the *Show Warnings for Unique Signals* check box.
5. To skip the placement of cross-references on a schematic, select the *Skip Schematic Annotations* check box.

6. To specify that CRefer places hierarchical cross-references on ports and flat cross-references on offpage symbols, select the *Distinguish Between Ports and Offpages* check box.
7. To retain the duplicate entries in the signal and part cross-references, select the *Retain Duplicate Entries* check box.
8. To annotate information only about page numbers and not about zones, select the *Omit Zone Information* check box.
9. To sort the signal cross-references only by page number, and not by the input/output type, select the *Sort by Page Number Only* check box.
10. To display the signal names in hierarchical cross-references, select the *Show Signal Names in Hierarchical Cross References* check box.
11. To write the name of the block where the signal originates in the cross-reference information for hierarchical designs, select the S*how Block Names in Hierarchical Cross References* check box.

    The cross-reference will appear as:

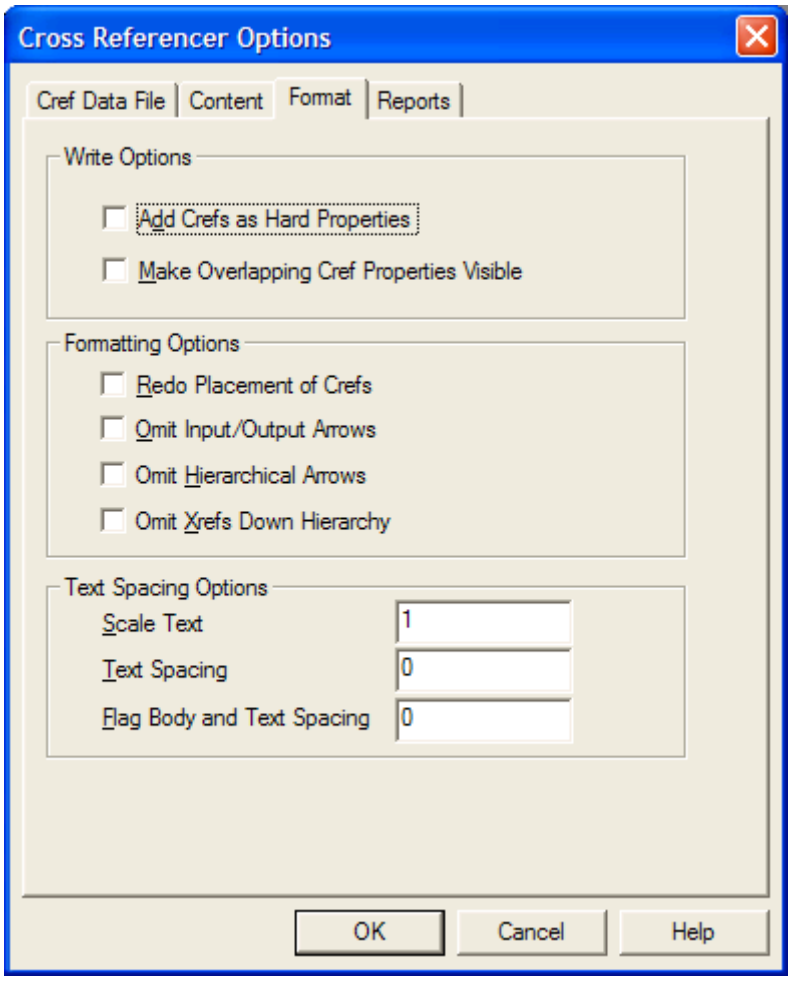
    ```
    [Signal_Name@][_BlockName_@]Page#Ygrid Xgrid[Type]
    ```

12. To make the XR page title invisible, select the *Make Page Title Invisible* check box.

Configuring Formatting Options

The Cross Referencer Options - Format tab allows you to configure CRefer formatting options. You can use this tab to specify whether or not existing crefs will be used. You can also specify that input, output, and hierarchical arrows be ignored. Further, you can define the text size of annotated CRefer properties and the space between each property.

Cross Referencer Options - Format Tab



To configure formatting options, do the following:

1. To add Cref properties as hard properties (XR), select the *Add Crefs as Hard Properties* check box. By default, CRefer add cross-references as soft properties ($XR).
2. To make the properties visible (that cannot be displayed on the schematic due to lack of space) on the schematic, select the *Make Overlapping Properties Visible* check box.
3. To override the previous cross-reference placement in a design (this information is available if the design has been cross-referenced in the past.), select the *Redo Placement of Crefs* check box.

   By default, CRefer reuses the existing cross-reference information.
4. To omit the writing of the characters (that is, "<", ">", and "<>") for I/O types, select the *Omit Input/Output Arrows* check box.

5. To omit characters being used (that is, “^" and “v") for I/O types, select the *Omit Hierarchical Arrows* check box.
6. To omit cross-references down a hierarchy, select the *Omit Xrefs Down Hierarchy* check box.
7. To increase or decrease the text size of cross-references, type the scale factor (the text size in relation to the default display) in the *Scale Text* field.

   Example - To scale the text size to half of the original size, enter 0.5 in the Scale Text field. To scale the text size to twice the original size, enter 2 in the Scale Text field. The setting 0.5 can be used when the schematic is densely packed.

   Note: When you specify the Scale Text, keep in mind the smallest pages in the design.
8. To increase or reduce the space between two cross-references, type a number in the *Text Spacing* field.

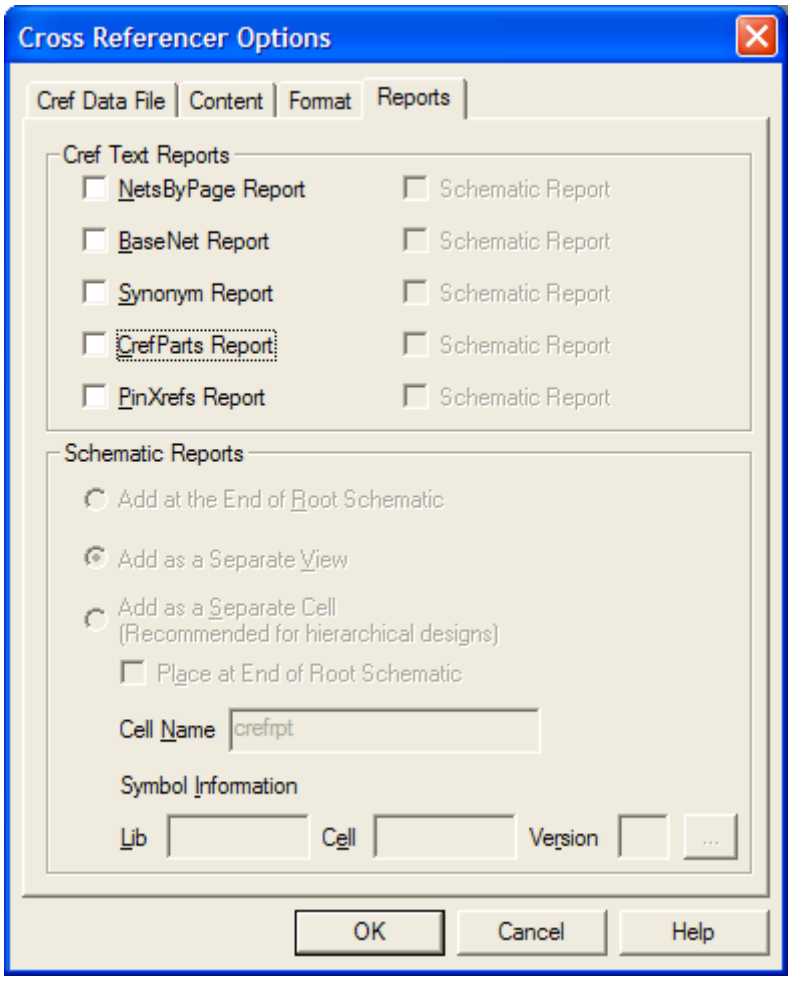
   Note: CRefer uses default Design Entry HDL coordinates for text spacing.
9. To increase or reduce the space between the flag body and the cross-reference text, type a number in the *Flag Body and Text Spacing* field. To reduce the spacing, use a negative value; otherwise, use a positive value.

   Note: By default, CRefer reuses the existing cross-reference information and visibility information.

Defining Output Reports

To define the types of reports to be generated as outputs, use the Cross Referencer Options - Reports tab. For example, you can generate Synonym reports.

Cross Referencer Options - Reports Tab



To generate different reports, do the following:

1. To create a report that contains the list of nets and their base nets grouped by page, select the *NetsByPage Report* check box.
2. To create a report that contains the signal cross-reference information grouped according to the design cells, select the *BaseNet Report* check box. The Basenets report also includes the direction characters with the signal and synonym information.
3. To create a report that traces a net across a hierarchical design, select the *Synonym Report* check box.
4. To create a report that contains information about all unit cross-references for the entire design, select the *CrefParts Report* check box. The Crefparts report includes information about the path property attached to the cell, the symbol name, and the cross-references.

5. To generate a pin cross-references report, select the *PinXrefs Report* check box. This report contains the part name, body name, pin number, zone, and physical net name.

Note: To append a selected report to the schematic, select the *Schematic Report* check box next to it. The *Schematic Reports* group box is activated when you select an *Schematic Report* check box.

6. Select the *Add at the end of Root Schematic* option to add signal and part cross reference reports at the end of the root schematic. This radio button is selected by default, signifying that CRefer appends all Cref reports at the end of the root schematic.

   Note: While adding extra pages to the schematic, CRefer uses the default page border specified in the project file. If the default page border is not specified in Design Entry HDL, CRefer searches for the page border in the `cref.dat` file. If information is not available about any custom page borders, CRefer uses the page border from the last page of the schematic to create new pages.

   Note: This option is grayed out for hierarchical designs.

7. Select the *Add as a Separate View* option to create a separate view for the schematic reports. The crefout view is created. This view contains two reports, one each for signal and part cross reference.

   Note: If you select the *Create Separate View for Schematic Reports* radio button, the reports added by CRefer at the end of the schematic during any previous run are automatically deleted.

8. Select the *Add as a Separate Cell* option create a separate cell structure, into the `sch_1` view of which the CRefer report pages are added. The remaining fields appear grayed until this option button is selected. The `generate_separate_cell` directive in the `cpm` file corresponds to this option. Select *Place at the End of Root Schematic* to specify that CRefer should append a page to the root schematic and place the new symbol on it. Specify a *Cell Name* by which the report page cell will be added to the schematic. The default name is `CrefRpt`. Use the browse button to open the View Open dialog box to get information about the library, the cell, and the version of the symbol that you want to use.

   Note: This is recommended for hierarchical designs.

Note: The *PinXrefs Report* and *Add as a Separate Cell* options take considerably more time than the other options do.

Adding a Cross-Reference Property to Appear on Page Border by Default

You can attach the $XR_PAGE_TITLE property on the page border so that it appears on the page borders of all the pages of a design. Subsequently, this property will appear at the specified position, by default.

1. In Design Entry HDL, choose *Tools — Options*.
2. In the *Custom Variables* tab, define a custom variable, `XR_PAGE_TITLE`.
3. Specify the value as *$XR_PAGE_TITLE*.
4. Click *OK*.
5. Choose *Text — Custom Text*.
6. In the Custom Text dialog box, scroll down the Variables list to the entry for `XR_PAGE_TITLE`.
7. Add this variable to the Symbols of all the page borders that are used in design:
   a. Open the desired page border symbol.
   b. Click the *Text — Custom Text* tab.
   c. Select `XR_PAGE_TITLE` from the variable list.

      Note that the entry appears in the Format string field and as an unsubstituted value in the DISPLAY string field.
   d. Click *Apply*.

      The variable is attached to the cursor.
   e. Click the origin of the page border symbol and place the text wherever you want it to appear on the page border.
   f. Save the page border symbol.

      In the same way, you can attach this variable to all the page border symbols used in the design and save them.
8. In Project Manager, choose *Tools — Crefer.*
9. In the Crefer dialog box, click *Options*.
10. Click the *Content* tab.
11. Select the *Generate Flattened Schematic* and *Make Page Title Invisible* options.

12. Click *OK*.

13. Click *Run.*

14. View the schcref_1 view to verify the substitution of the $XR_PAGE_TITLE.

Deleting Cross-References

To delete the cross-references from a design, use the CRefer dialog box.

1. Select the *Remove All Cross References* radio button in the CRefer dialog box.

2. Click on the *Run* button.

The CRefer Progress window displays the progress of cross-references being deleted.

Note: To delete the cross-reference for a design from the command-line prompt, use the following syntax:

```
creferhdl -proj <project_file> -d
```

where, `-proj <project_file>` specifies the path to the project file you want to cross-reference, and the `-d` option is used to delete all the existing cross-references in the design.

How CRefer Deletes Cross-References

When deleting cross-references, CRefer will:

- Delete all properties, whether hard or soft. Cross references for `XR` and `$XR` properties will be deleted.
- Delete the `XR_PAGE_TITLE` from the page borders.
- Delete the extra pages added to the schematic by earlier cross-referencing.
- Delete the values of custom variables used in custom text.
- Delete the `cref.opf` file.

When deleting cross-references, CRefer will not:

- Delete reports from the *rptcref_1* view.
- Delete properties from the *schcref_1* view. Since the *schcref_1* view remains unchanged, all custom text `CREF` variable values stored in it remain intact.

  Note: If you want to delete only soft or hard properties, you can write a custom SKILL

routine to override the CRefer default settings.

Understanding CRefer Output

Identifying Inputs and Outputs

There are two different ways in which you may find the direction information about a signal in the schematic:

1. **Input/Output Arrows** – If you have already cross-referenced a design, CRefer adds direction characters (the input signals are represented by the < sign, and the output signals are represented by > sign) to the cross-references. This is a quick way to identify inputs and outputs. If you select the *Omit Input/Output Arrows* check box, CRefer will not add the arrows.
2. **Hierarchical Arrows –** In a hierarchical design, a signal may be going up or down a hierarchy. If the signal is going up in the hierarchy, CRefer assigns the signal ^ as a direction signal. If the signal is going down, CRefer assigns the signal v as a direction signal. If you select the *Omit Hierarchical Arrows* check box, CRefer will not add the hierarchical arrows.

CRefer makes it easy for you to identify the input or output information about a signal, but it also needs the input/output information to display the signal direction, and to place cross-references at the correct locations. Based on whether the signal is input or output, CRefer will place an XR string at the left or the right end of wire, respectively.

Note: It is strongly recommended that the direction information about a symbol be supplied at the time of schematic entry for accurate cross-referencing.

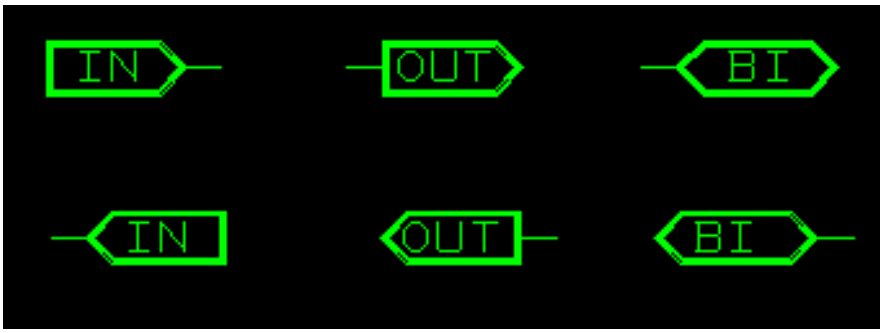
You can define the direction of a signal by attaching an OFFPAGE flag body to the signal. Flag bodies and ports are commonly used in schematic standards to indicate when the origin or destination of a signal is on another page of the schematic. Different versions of symbol bodies are used to define the signal as input, output, or bi-directional. If a signal has a flag body or port, CRefer uses the information contained in it to determine the direction of the signal.

When CRefer encounters a signal that does not have a flag body or port, it reads the *chips.prt* file for the component to which the signal is connected to obtain information about the direction of the component's pins. Based on the direction of the pin to which the signal is connected, the signal is marked as input, output, or bi-directional.

By default, CRefer understands the OFFPAGE, PORT, and FLAG bodies from the Cadence *standard* library. You can also create custom offpage flag bodies. However, if you add any flag

body or port in the schematic, specify it in the `cref.dat` file to ensure proper cross-referencing.

The six standard Cadence supplied versions of the OFFPAGE flag body are shown below:



You can also add the OFFPAGE flag body to another library on your system.

Note: If you change the parts in the *standard* library, or add parts to it, be sure to keep a copy of the flag bodies at another location. Future Cadence library updates may delete any new or changed bodies in the library.

Sample Signals Labeled with Cross-References

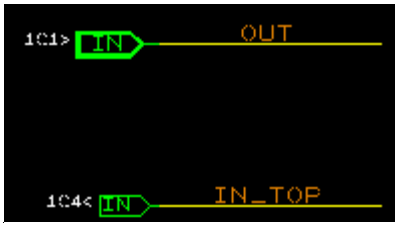
Example 1

The following figure displays a drawing that has two signals annotated by CRefer.

1. The `OUT` signal appears twice, once as input in the C4 zone on page 2, and once representing a hierarchical signal going down in the D2 zone on page 4.
2. The `OUT_TOP` signal is a hierarchical signal, which appears in the C2 zone on page 4.

Example 2

The following figure displays a drawing that has two signals annotated by CRefer.



1. The `OUT` signal appears as output in the C1 zone on page 1.
2. The `IN_TOP` signal appears as input in the C4 zone on page 1.

CRefer reads your schematics and then rewrites them to include the cross-reference information. The cross-reference information is generated as properties attached to the nets or pin names. By default, CRefer assigns soft properties. You can specify that CRefer assign hard properties.

Note: By default, CRefer tries to place cross-references for all signals. However, if CRefer does not find enough space for the cross-references, it makes the cross-reference values invisible to avoid the overlapping of CRefer properties. You can use the `show properties` directive in Design Entry HDL to temporarily make the invisible cross-references visible.

Signals that are not Cross-Referenced

CRefer may not assign a cross-reference to a signal if:

- The signal is unique:
  - The flag bodies to which the signal is connected is not listed in cref data (`cref.dat`) file.
  - The signals are not connected to any flag (`OFFPAGE`) bodies and the *Generate Xrs for signals not connected to flagbodies* option is deselected.
  - The signal is connected to an unnamed signal at the top-level design.
- The input/output type of the signal cannot be determined because it is not attached to an OFFPAGE flag body or a port, or if the signal is attached to a horizontal wire with an unattached end.
- Unnamed signals are not cross-referenced and cross-references to signals connected to unnamed signals are also not passed on to them.

When a signal cannot be cross-referenced, it is assigned an invisible property named `$XRERR` with the value as `IOTYPE?`.

If the *Ignore Inputs Only Signals* check box is selected, the `$XRERR` property may also be assigned the `NODRIVE?` value.

If the *Show Warnings for Unique Signals* check box is selected, the `$XRERR` property may also be assigned the `UNIQUE?` value.

Note: If there is an invisible signal in the schematic, an invisible cross-reference is created for it.

To check why a signal was not cross-referenced, verify its property value by doing the following:

1. Start Design Entry HDL and edit the schematic.
2. Group the invisible `$XRERR` properties by doing one of the following:
   - Select *Group — Create — By Expression*. This will display the Pattern dialog box. Enter `$XRERR` and click *OK*.
   - Enter the `FIND $XRERR` command in the console window.

   The properties are grouped and assigned the name ‘A’. If you have already created some groups in the current Design Entry HDL session, the group might have a different name.
3. Make the property values visible by doing one of the following:
   - Select *Group — Show Contents [A]*.
   - Enter the `DISPLAY VALUE "A"` command in the console window.

   Note: A is the group name assigned by the `FIND` command.

Use the `Next` command to automatically zoom in on each property.

Cross-References as Links

Cross-references (XRs) in a design are converted to hyperlinks called Cref links, which help you quickly navigate to a target net by clicking a Cref link on the source net. You can also move backward and forward by clicking Cref links. This new functionality makes it easier for you to trace a signal across a design. It helps you review the connectivity of a net quickly and enables you to retrace a signal if required.

By default, the cross-references in your design are active links. The directive that converts cross-references to hyperlinks is the HYPERLINKS directive. For more information about the directive, refer to the ***HYPERLINKS*** section of the *Allegro Front-End CPM Directive Reference Guide*.

Important

Cref links work only if the correct `cref.dat` file is available to the design. If this file is not available, the link will navigate to the linked page, but will not zoom to the appropriate zone.

Note: In case of legacy designs, first run Crefer to record the path of the `cref.dat` file in the CPM file. This will ensure that the Cref links work fine.

Using Cref Links

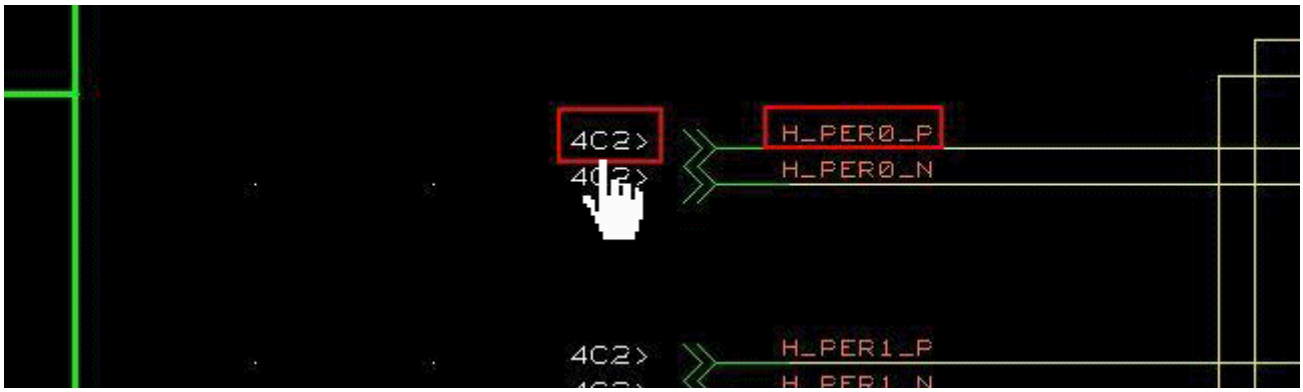
After activating the Cref links, press the Ctrl key and the left mouse button simultaneously on an XR and Design Entry HDL displays the target location on the schematic. As you move the mouse pointer over a Cref link, the following tool tip appears:

CTRL + click to follow link

As you press the Ctrl key, the mouse pointer changes to a hand pointer indicating that it is a hyperlink. When you click the XR on the target net, it takes you back to the original location.

Example

To trace the signal name `H_PER0_P` on page 3 of the sample design, you need to Ctrl+LMB on the Cref link `4C2`.



1. Keeping the CTRL key pressed, click the left mouse button.

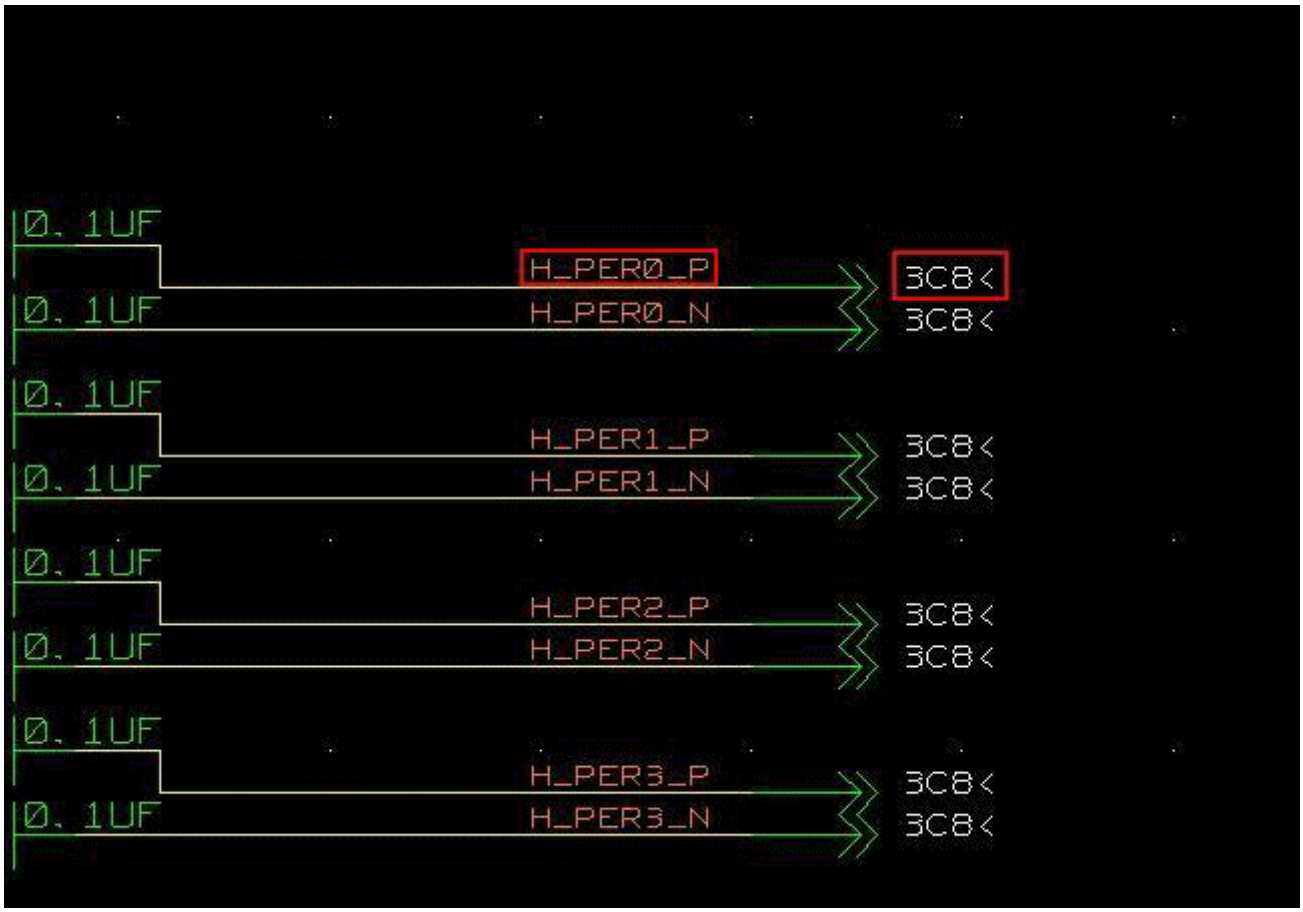
Note: The only syntax supported for Cref Links is `<page_no><zone><I/O type>`. where *zone = YgridXgrid.*

For example, `4C2>`, where `4` represents the page number, `C2` represents the zone (Ygrid = C, Xgrid = 2), and `>` represents the i/o type, output signal.

Important

Zone must consist of only two letters, each representing ygrid and xgrid, respectively. Only one character is allowed to represent a grid (y or x). Also, these characters must only be alphanumeric in nature.

The target location on page 4 is displayed.



1. Click 3C8 on page 4.

   It takes you back to the C8 zone on page 3.

Handling Out-of-Sync Cross References

A Cref link might lead to an incorrect location on the schematic if the target of a CRef Link goes out-of-sync when pages are added or removed, components are moved, or other connectivity changes take place.

To fix the issue, exit Design Entry HDL and run CRefer to regenerate cross-references. When you launch Design Entry HDL again, the Cref link points to the appropriate location.

Reference Information

CRefer Text Reports

CRefer generates five text reports: BaseNets, NetsByPage, Synonyms, CrefParts, and PinXrefs report. In addition, a CRefer error report, `creferror.dat`, is generated. These reports are added in a separate view named *rptcref_1* view in the top-level cell for the current project. You can view the reports in a text editor.

NetsByPage Report

This report contains the list of nets and their base nets grouped by page. To view the report, open the `netsbypage.txt` file located in the *rptcref_1* view.

Note: The BaseNets and NetsByPage reports include the direction characters with the signal and location information.

Note: If you have a bus at the top level that is split at the lower level, the Basenets and NetsByPage reports will display only the Most Significant Bit (MSB) in the synonym list of the bus at the top level. For example, the bus `A<2..0>` at the top level will show only the `A<2>` bit as its synonym. At the lower level however, the cross-referencing information will be generated for all bits.

BaseNets Report

This report contains signal cross-reference information grouped according to each occurrence of the design cells. To view the report, open the `basenets.txt` file located in the *rptcref_1* view.

Note: CRefer puts the location of each signal and its synonyms in the reports. However, if a design has buses split into multi-bit vectors, the report will list both the signal and its super-set as its synonyms.

Synonym Report

This report contains information about the synonyms corresponding to each base signal along with their location, zone, and direction information. It groups the nets and the basenets in a design by the design sheets.

CrefParts Report

This report contains information about all unit cross-references for the entire design. The information includes the location property attached to the cell, the symbol name, and the cross-references.

CRefer reads all the instances of the cells in both, the replicated and non-replicated hierarchy, to generate complete reports.

PinXrefs Report

This is a pin cross-references report, which contains the part name, body name, pin number, zone, and physical net name.

Cref Error Report

This report is generated when CRefer encounters any warning, and is useful for debugging purposes. You can check this report by opening the `creferror.txt` file in the *rptcref_1* view under the top-level design.

Note: The CRefer error report is a subset of the Cref log file.

Schematic Report

You can also append a selected report to a schematic. To do this, select the *Schematic Report* check box next to the report. The Schematic Reports group box is activated when you select the *Schematic Report* check box.

I/O Types

By default, CRefer writes the characters indicating the I/O type of the signals. The following table lists the different characters used along with their descriptions.

Character	Description
<	The signal is an input Signal
>	The signal is an output signal
<>	The signal is an inout (Bidirectional) signal
^	The signal is going up in the hierarchy.

v The signal is going down in the hierarchy.

Page Numbering

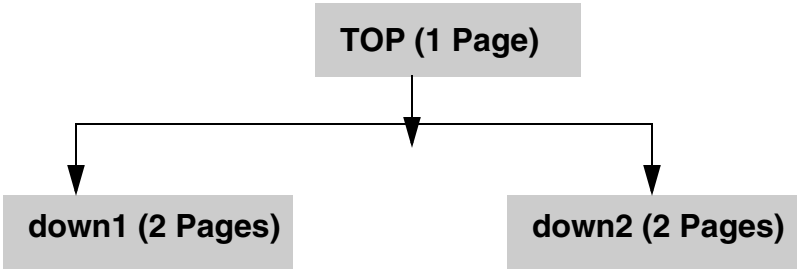
CRefer assigns a page number to each schematic sheet in the design hierarchy, and writes these numbers at the upper left corner of the sheet. The page numbers used in cross-references are based on these page numbers.

This is in sync with the CURRENT_DESIGN_SHEET variable provided by Design Entry HDL. This is also consistent with the order in which pages are plotted. CRefer page 5 implies that the corresponding page will be the 5th one to be plotted in a hierarchical plot.

To map CRefer sheet numbers in schcref_1 to schematic page numbers in sch_1, you use the CREF_ORIG_PAGE custom variable. This variable is substituted with the actual page number of the schematic so that it can be used to map the schcref_1 pages to sch_1 pages.

Further, in case of in-place cross-referencing (sch_1), you can traverse through pages easily using the CURRENT_DESIGN_SHEET variable and the gotosheet command in Design Entry HDL. For example, a CRef annotation to page 4 implies that the corresponding signal is on sheet 4 of the design. This can be reached by using 'gotosheet 4' in Design Entry HDL, or by checking CURRENT_DESIGN_SHEET values while viewing plots.

For a flat design that has pages 1-3 and 7-9, the total number of sheets remains 6, and they are numbered from 1 to 6. For a hierarchical design, consider the following example:



In sch_1, TOP, down1 and down2 will have page numbers as 1 and 2. In schcref_1, page numbers are from 1 to 6 and CRefer follows these sheet numbers (which are always in sequence starting from 1) and not the original page numbers. Similarly, when you have pages 1, 2, 3, 10, 11, 12, CRefer again follows sheet numbers 1, 2, 3, 4, 5, 6. Cross-references are numbered according to the sheet numbers, which are in sequence even though the original design might have skipped pages, just as with a hierarchical design. If down1 has two

pages numbered as `1` and `5`, even then the sheet numbers will be from `1` to `6` because sheet numbers are independent of skipped pages in the design.

Support for Design Entry HDL Custom Variables

You can use custom variables in a default page border and provide a default value, if required. Custom variables are special variables that are supported by Design Entry HDL. You can use these variables for intelligent plotting of cross-referenced schematics. For example, using these variables, you can place page information such as 'This is page 1 of 24' on cross-referenced schematics. You can also use custom variables to store information such as the company name and author name.

The list of available CRefer variables is displayed in the following table:

Variable Name	Description
CREF_TO_LIST	Defines where the pages for the blocks are located in a cross-referenced flattened design.
CREF_FROM_LIST	Defines where the pages in a flattened design came from in the original design.
CREF_ORIG_DESIGN_NAME	Defines the original design name
CREF_ORIG_PAGE	Defines the original page number
CREF_ORIG_VIEW	Defines the name of the original view
TOTAL_DESIGN_SHEETS	Lists the total number of pages in the Design Entry HDL schematic. While calculating the `TOTAL_DESIGN_SHEETS` value, the number of pages in all modules of the design is taken in account. The `TOTAL_DESIGN_SHEETS` value is the same as the total number of pages generated by CRefer when it cross-references a design by creating a flattened view of the base schematic.

Variable Name	Description
CURRENT_DESIGN_SHEET	Lists the sheet number of the current page in the schematic. **Note:** Whenever you add or delete pages, or perform module ordering, the value of the TOTAL_DESIGN_SHEETS and CURRENT_DESIGN_SHEET variable changes. However, each time a change occurs in the schematic, Design Entry HDL does not reevaluate the value of the TOTAL_DESIGN_SHEETS and CURRENT_DESIGN_SHEET variables. Therefore, it is important that you cross-reference the design to get the updated values of the TOTAL_DESIGN_SHEETS and CURRENT_DESIGN_SHEET variables. **Note:** You can easily search for any custom text variable by selecting it in the *Variables drop-down* list in the Custom Text dialog box in Design Entry HDL.

Adding CRefer Custom Variables

To add any CRefer variable, you must first define custom text and include the variable. After defining custom text, attach it to an object on the schematic.

If you are adding the CREF_TO_LIST variable or the CREF_FROM_LIST variable, the ideal place to add the variable is at the top or near the block for which you want to find cross-referenced page data. For other CRefer variables, the ideal location is the page border.

Note: CRefer specific custom variables will be substituted when you run CRefer with the *Generate Flattened Schematic* check box selected.

Example of Adding a CRefer Variable

You will add the CREF_TO_LIST CRefer variable to the top of a module named DOWN in the CREF1 design. The custom text that uses the CREF_TO_LIST variable will display this message: “This block goes to page <CREF_TO_LIST>”.

Steps

1. Choose *Text — Custom Text* in Design Entry HDL to display the Custom Text dialog box.
2. In the *Format string* field, enter the following text: “This block goes to page”.
3. Select the *<CREF_TO_LIST>* variable in the *Variables* list.

   The *Display string* is automatically appended.
4. Click *OK*.

   The custom text is attached to a cursor.
5. Move the cursor at the top of the block `down` and click to attach the custom text to this block.
6. Click again to place the custom text on the schematic.
7. Right-click and press *Done* to complete the operation.

The custom text ‘This block goes to page `<CREF_TO_LIST>`’ is attached to the block `DOWN` and when you cross-reference the design, the block `DOWN` displays the following text: “This block goes to page 3” where 3 is the value of the `<CREF_TO_LIST>` variable.

Performing To/From Property Annotation

CRefer performs to/from property annotation to specify where the pages in a flattened design come from the original design and where the pages from the original hierarchical design are included in the cross-referenced flattened design.

To perform to/from property annotation, CRefer uses the `CREF_TO_LIST` and `CREF_FROM_LIST` custom variables.

Note: To/from property annotation is performed only when a new flattened view (*schcref_1*) is created.

Steps for Performing To/From Property Annotation

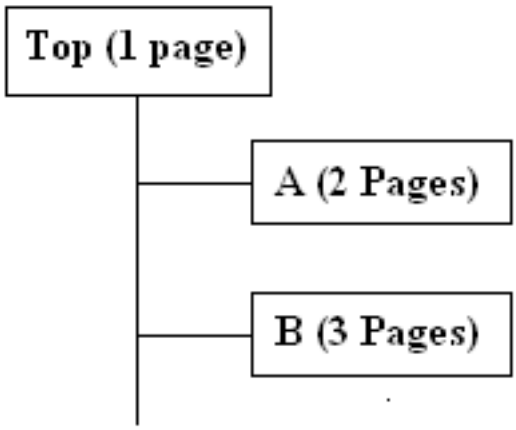
Define and attach custom text for the `CREF_TO_LIST` and `CREF_FROM_LIST` variables in the schematic through Design Entry HDL. For more information about attaching custom text, see Design Entry HDL Help.

1. CRefer calculates the values of custom variables in the custom text and provides it to Design Entry HDL.

2. Design Entry HDL updates the custom text with the values received from CRefer and annotates the properties on the schematic.

Example

Assume you have the following hierarchical design named `Sample`:



The `Sample` design has one page at the top level (`TOP`). The `TOP` page contains two blocks, A and B, of pages 2 and 3 respectively. After cross-referencing, the design will consist of six pages, where Page 1 corresponds to the block `TOP`, pages 2 and 3 correspond to the block A, and pages 4 to 6 correspond to the block B.

Under (attached to) the symbol on page 1 for block A is annotated a property:

```
CREF_TO_List = Pages 2, 3
```

While on the page border of pages 2 and 3 is annotated another property:

```
CREF_FROM_List = 1B3 - 1P
```

Where 1P is the instance name (for block A) and 1B3 is the cross-reference for the hierarchical symbol.

Similarly under the symbol on page 1 for block B is annotated a property:

```
CREF_TO_List = Pages 4, 5, 6
```

While on the page border of pages 4, 5 and 6 is annotated another property:

```
CREF_FROM_List = 1C7 - 2P
```

Where 2P is the instance name (for block B) and 1C7 is the cross-reference for the hierarchical symbol.

Note: For replicated and read-only blocks, CRefer calculates the value of the CREF_TO_LIST and CREF_FROM_LIST custom variables and annotates them to the cref.opf file as CDS_CREF_TO_LIST and CDS_CREF_FROM_LIST variables, respectively.

Note: If there are CRefer-specific custom variables annotated to the page border, CRefer annotates that variable to the canonical name of the schematic in the cref.opf file.

Example

Assume you have the following custom text

```
CRefer from list is <CREF_FROM_LIST>
```

on page 1 of the schematic MUX, then CRefer will substitute the CREF_FROM_LIST property on the canonical name for page 1 of the schematic MUX. The property will have the following value:

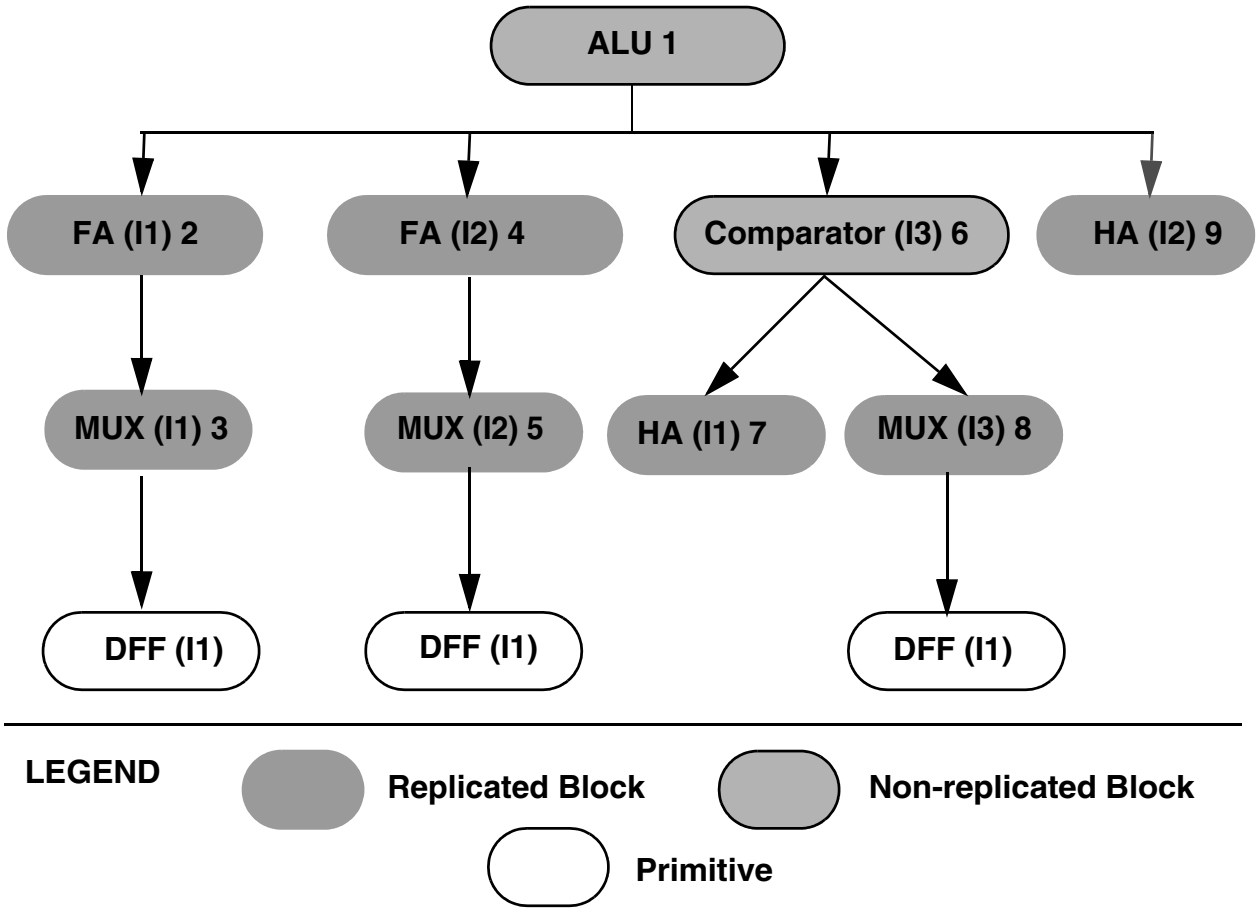
```
I1-3C5
```

where, I1 represents the first instance of the MUX block, 3 represents the page number of the instantiating block, and C3 represents the zone coordinate of the instantiating block.

Example of Design Sheet Variables

Assume you have assigned the custom text ‘This is sheet <CURRENT_DESIGN_SHEET>’ on the MUX schematic in the DFF (I1) block as shown in the following figure.

Figure 4-4 ALU Hierarchical Design



Naming Convention in Block: Block name (instance name) Design current sheet value

Also assume that all blocks in the `ALU` design are single-page blocks that have not been reordered. As the design is traversed in the depth-first order, the custom text 'This is sheet <CURRENT_DESIGN_SHEET>' on the `MUX` schematic in the `DFF` (`I1`) block will result in the following OPF properties at the `@alu_lib.alu(opf)`.

```
@alu_lib.alu(sch_1):page1_i1@alu_lib.fa(sch_1):page1_i1@alu_lib.mux(sch_1):page1_
i1@alu_lib.dff(sch_1):page1
This is sheet 3

@alu_lib.alu(sch_1):page1_i2@alu_lib.fa(sch_1):page1_i2@alu_lib.mux(sch_1
):page1_i1@alu_lib.dff(sch_1):page1
This is sheet 5
```

If you have assigned the TOTAL_DESIGN_SHEETS variable as part of custom text to any block in the schematic, that variable would have the value 9.

Important

When working with custom variables, Design Entry HDL displays properly substituted custom variables on the schematic. For a non-replicated hierarchy, Design Entry HDL always substitutes the CRefer-specific custom variables.

Updating Custom Text Variables for Page Numbers

You need to update the CURRENT_DESIGN_SHEET and TOTAL_DESIGN_SHEETS custom text variables for page numbers to ensure that the schematic page displays the correct page number. This updating needs to be done in the following cases:

- If you have modified the design by adding or deleting pages or blocks.
- When you add custom text variables for page numbers on a schematic page, the name of the variable is substituted by the page number. If the schematic page continues to display the variable name instead of the page number, you need to update the variable. For example, if you add the CURRENT_DESIGN_SHEET custom text variable in the 20th page in a hierarchical design, the schematic page may display <CURRENT_DESIGN_SHEET> instead of 20.
- When you renumber schematic pages using the page renumbering commands, the schematic pages might not display the correct page number.

Note: When you perform module ordering, cross-referencing, or plotting of the design, the custom text variables are updated automatically.

To update the custom text variables for page numbers, choose *Text > Update Sheet Variables*.

The custom text variables for page numbers on all pages in the design are updated to display the correct page number.

Placeholder Support

To ensure that CRefer can optimize the placement of cross-references in a design, you can add placeholders on the schematic and perform the following procedures:

- Adding Placeholders on Ports or Offpage Symbols on page 125
- Editing of Invisible Placeholders on page 127

- Managing Changes to the standard Library on page 4-128

Note: In many firms, librarians may be responsible for creating placeholder support for symbols.

Adding Placeholders on Ports or Offpage Symbols

CRefer will only place annotations for signals that have a `$XR<n>` placeholder attached on the port or offpage symbol. Therefore, add placeholders on all ports and offpage symbols. To add a placeholder to a port or offpage symbol:

1. Open the Attribute form.
2. Assign the following placeholder property to the port or offpage symbol:

   ```
   $XR0=?
   ```

When assigning placeholders, start from the `$XR0` property. Do not use any other property for defining placeholders.

3. Repeat step 2 to assign more placeholders. Add the placeholders in a contiguous manner as `$XR0`, `$XR1`, `$XR2`, and so on. For example to create the second placeholder, define the following property to the port or offpage symbol:

   ```
   $XR1=?
   ```

   Note: The number of placeholders in a design depends upon the nature of your design. Since different designs may have a different number of cross-annotations on these ports or offpage symbols, it is recommended that `version 1` of your symbols has enough placeholders to allow proper cross-referencing. When CRefer runs out of placeholders, it assigns all the remaining cross-references to the last place holder. The following message is also generated:

   ```
   Signal <signal_name> at <location_of_SIG_NAME> required
   <number_of_required_placeholders> placeholders on
   <name_of_the_attached_plumbing_body>.
   ```

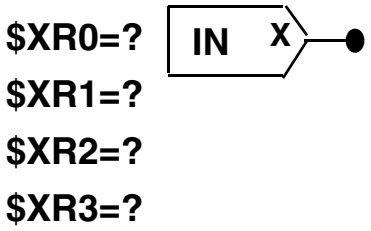
   Note: If you have both ports, and offpage symbols, connected to the same point of a signal and if you have selected the *Distinguish Between Ports and Offpages* check box in the Cross Referencer Options - Content Tab, cross-references will be attached to either the port or the offpage symbol, but not to both. To ensure that cross-references are attached to both ports and offpage symbols, connect the port and offpage symbol to the signal at different points.

Recommended Steps

1. Add all ports and offpage symbols to the `cref.dat` file located at `<your_install_dir>/share/cdssetup/creferhdl`. For more information about the `cref.dat` file, see Working with the Cref Data File on page 4-88.

2. Create different versions of symbols that allow both vertical and horizontal stacking. This will ensure that a designer can switch between vertical and horizontal stacking by versioning the instances in the schematic. You can also assign different number of placeholders in different versions of symbols.

Figure 4-5 Vertical Stacking of Placeholders



3. Use the appropriate alignment. CRefer will use the standard Design Entry HDL alignment for writing cross-annotations on placeholders. It is recommended that you left-align the placeholders for all `OUT` ports and right-align the placeholders for all `IN` ports. You should select a suitable alignment for other ports and offpage symbols. Figure 4-6 on page 126, Figure 4-7 on page 127, and Figure 4-8 on page 127 provide an idea about why appropriate alignment is necessary. Note that the left alignment of placeholders is causing the cross-references to overlap on the `IN` symbol. However, right alignment of placeholders causes proper placement of cross-references.

Figure 4-6 IN Port Connected to MUX Block: Before Property Substitution

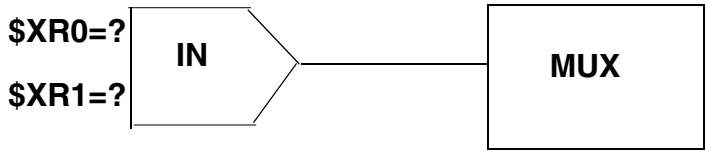


Figure 4-7 IN Port Connected to MUX Block: After Left-aligned Property Substitution

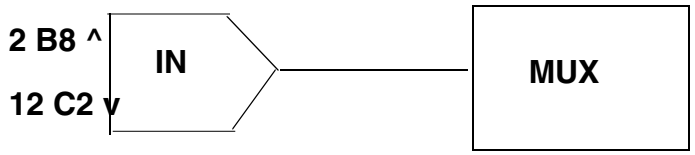
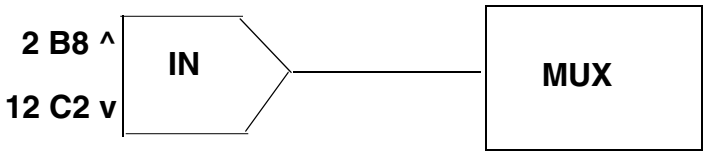


Figure 4-8 IN Port Connected to MUX Block: After Right-aligned Property Substitution



4. Make the placeholders invisible - If you defined the placeholders on ports, or offpage symbols as visible, on cross-referencing the design, you will see the `?` sign in all placeholders that are not substituted by CRefer. To avoid this problem, you should make placeholders on ports or offpage symbols invisible by selecting *Group — Property Display — Invisible* in Design Entry HDL.

Editing of Invisible Placeholders

To edit an invisible placeholder, do the following:

1. Create a group of all placeholders by selecting *Group — Create — By Expression* in Design Entry HDL.
2. Type `$XR*=?` and press Enter.

   The group is created.
3. Make the placeholders visible by selecting *Group — Property Display — Value* command in Design Entry HDL.
4. Edit the placeholders and move them as required.
5. Select the group of placeholders and make them invisible by selecting *Group — Property Display — Invisible* in Design Entry HDL.

Managing Changes to the standard Library

If you make changes to any symbols in the Cadence `standard` library, such as adding placeholders to ports or offpage symbols, you may lose those changes when you install quarterly incremental releases or updates to the `standard` library.

It is recommended that you make a local library of all the symbols that you customize in the `standard` library, and use that library. Avoid making changes to symbols in the `standard` library directly.

Formatting Reports

If you want CRefer reports to confirm to a particular style, you can set some directives in the `START_CREFERHDL` and `END_CREFERHDL` section of the `cpm` file. The following section details these directives.

Using the OMIT_CELL_FROM_CREFPARTS Directive

CRefer outputs the design name in the Crefparts report as shown below:

```
C11                                    CAP                  dashhf[2B2]
```

Typically, hierarchical designs have better formatting but flat designs have extra column space, which invariably adds extra pages in the Crefparts report for a large-sized design.

To suppress the design name and the white space from the `crefparts` report, add the following directive to the `START_CREFERHDL` and `END_CREFERHDL` section in the `cpm` file:

```
OMIT_CELL_FROM_CREFPARTS 'ON'
```

If you cross-reference the design, the design name will not be written in the report, thereby making it compact. The output of the report will look like this:

```
C11   CAP   [2B2]
```

Note that the `OMIT_CELL_FROM_CREFPARTS` directive eliminates the extra white space due to the removal of the design name.

Using the OMIT_CREFPARTS_HIERARCHY Directive

You can use the `OMIT_CREFPARTS_HIERARCHY` directive to omit higher level cells from the parts by page report (`crefparts.txt` file), when the flattened schematic (`schcref` view) is generated.

By default, a parts by page report may have the following output:

```
%############################################################
Title: Cref Part Report
Design: laptop
Date: Nov 16 11:51:03 2004
%############################################################
C1          CAPACITOR             serial_port[ 5C5 ]alu[ 4C7 ]laptop[ 1B2 ]
...
```

Note that the parts by page report a capacitor named `C1` is present in the `serial_port` design. The report mentions that the capacitor exists in multiple (3) pages, while it exists in only one page in the flattened design. The extra page/zone information is relevant for a hierarchical page but not for a flat page.

To avoid extra page/zone information for flattened designs, you can add the `OMIT_CREFPARTS_HIERARCHY` directive with value `ON` to the `START_CREFERHDL` and `END_CREFERHDL` section in the `cpm` file. If you now cross reference the design, the parts by page report will have following content and format:

```
%############################################################
Title: Cref Part Report
Design: laptop
Date: Nov 16 11:51:03 2004
%############################################################
C1          CAPACITOR             serial_port[ 5C5 ]

...
```

Using the FORMAT_CREF_REPORTS Directive

The `FORMAT_CREF_REPORTS` directive is used to perform:

- Zone wrapping—In schematic reports, the Location[zone][direction] column is wrapped in such a manner that the zone information appears together in the report. The zone information will be wrapped based on space character.

  If `FORMAT_CREF_REPORTS` is not set, then the Location[zone][direction] column might get wrapped at the exact point when the limit of characters for the column is crossed and make reading location data difficult.

  ```
  FS2_TMS 7C6 7D7 7F9 10D
  2
  ```

  If `FORMAT_CREF_REPORTS` is set, the Location[zone][direction] column will have proper value formatting, and wrapping will be done on space characters.

  ```
  FS2_TMS 7C6 7D7 7F9
  10D2
  ```

- Double line spacing problem—The text reports for basenets, netsbypage, and synonyms display all the columns (such as net names and location values) in the same row, instead of starting on the next row. As these reports are pasted into the schematic, the schematic reports also reflect the same changes resulting in reduced size of schematic reports.

Note: The FORMAT_CREF_REPORTS directive is set by default to ON. If you set this directive to OFF, CRefer will create text reports for basenets, netsbypage, and synonyms with all columns starting on different rows causing an increased number of schematic report pages. Further, setting the FORMAT_CREF_REPORTS directive to OFF will prevent zone wrapping. Cadence recommends that you avoid changing the value of the FORMAT_CREF_REPORTS directive.

Using the BASENET_OMIT_SYNONYM Directive

You can use the BASENET_OMIT_SYNONYM directive to omit the synonym column in the basenet report and schematic reports.

By default, the BASENET_OMIT_SYNONYM directive is set to OFF, which results in the display of synonyms information corresponding to base signals. Depending upon the number of synonyms attached to a base signal, multiple rows appear in the basenet report. For each base signal, as displayed below:

```
Base Signal     Synonyms                                    Location([Zone][dir])

T_IN            T_IN - @cref_lib.TOP_CREF_TEST               1C4< 2C4<
                L_IN - @cref_lib.CREF_TEST(i1_page2)         8A4< 8B4<
                M2_IN - @cref_lib.MID2_CREF_TEST(i2_page1)   5C4<
```

If you set the BASENET_OMIT_SYNONYM directive to ON and then cross reference the report, the report will display as follows:

```
Base Signal         Location([Zone][dir])

T_IN                 1C4< 2C4<  8A4< 8B4<  5C4<
```

Note that the synonym columns are not displayed and all location values are listed in the same row as the base signal name. This formatting is especially beneficial in schematic reports as it saves space.

CRefer Error Messages

This section includes information about the following:

Overview

While cross-referencing a design, CRefer may encounter errors. The warning messages are logged in the CRefer error data (`creferror.dat`) file generated in the `rptcref_1` view under the top-level design. The `cref.log` and `creferhdl.lst` files in the `temp` directory under the root design record both fatal and non-fatal error messages. Another file, `creferhdl_pinxref.lst`, is also created when the PinXrefs Report is generated. Depending upon the criticality of the error, the error messages may have one of the following three levels of severity:

Fatal Error Messages—These errors signify the non-completion of a critical step necessary for CRefer to cross-reference the design. CRefer cannot continue its operation till this error is rectified. As a result, CRefer terminates cross-referencing the design at the point it encounters the error. Any cross-references made during the current run of CRefer are deleted.

Non-Fatal Error Messages—These errors are not as critical as fatal errors, and therefore CRefer will not terminate cross-referencing the design. However, CRefer might not be able to cross-reference the design properly till you rectify these errors.

Warning Messages—When CRefer does not find any part, instance, or symbol in the schematic, or is unable to save the cross-referenced schematic files, it displays warning messages. After logging the warning message, CRefer continues to cross-reference the design.

Whether or not you encounter any error while cross-referencing the design, you should check the error report. You may then fix all errors, save the schematic, and perform another cross-referencing on the design. This procedure will ensure that you get an error-free cross-referenced design.

Fatal Error Messages

Fatal error: On line *<line_number>* of *<file_name>*. Two borders in drawing.

Description: This error message occurs when two page borders are used in the drawing sheet. CRefer displays the error and exits.

Solution: Delete one of the two page borders in the schematic and save it. Run CRefer again to cross-reference the design.

Fatal error: On line *<line_number>* of *<file_name>*. Page border should not be rotated.

Description: This error message occurs when the page border is rotated in the drawing sheet. As a result of the rotation, CRefer gets confused while calculating the cross-references. It does not cross-reference the design and exits.

Solution: Please ensure that you have not rotated the page border. If the current page border is not suitable, create a new page border and define it in the `cref.dat` file. Use the new page border to prepare the schematic and cross-reference it.

Fatal error: On line *<line_number>* of *<file_name>*. Wireowner: Design Entry HDL binary file is corrupt.

Description: This error message occurs when the Design Entry HDL binary file is corrupt. After displaying the error message, CRefer exits.

Solution: Write the drawing sheet specified by the *<file_name>* to generate the correct binary file. Run CRefer again to cross-reference the design.

Fatal error: Can't find body *<body_name>* version *<version_number>* in libraries. Write your design, as the schematic is out of sync with the parent-child database.

Description: This error message occurs when the parent-child database is out of sync with the schematic.

Solution: Write your design. This will synchronize the parent-child database and the schematic. You may now run CRefer again to cross-reference the design.

Fatal error on line *<line_number>* of *<path_of_the_cref.dat_file>*: String too long.

Description: This error message occurs while parsing the Cref data file (`cref.dat`) when any string has length greater than 300 characters.

Solution: Ensure that no string in the Cref data file has length greater than 300 characters.

Note: If you are using a `tscrpage.dat` file from release 13.6, make changes to the page border version in that file.

Fatal error: Number too long.

Description: This error message occurs while parsing the Cref data file (`cref.dat`) when any number has more than 300 digits.

Solution: Ensure that no number in the Cref data file has more than 300 digits.

Fatal error: String too long *<string_name>*.

Description: This error message occurs when any signal name has length more than 4096 characters.

Solution: Ensure that no signal in the schematic has length more than 4096 characters.

Fatal error: Vectored signal has '<' but no '>' .

Description: This error message occurs for buses that do not have both angle brackets, that is < and >. For example, you may have a bus defined as `A<3`.

Solution: Ensure that the signal has both direction signs. For example, if you have defined a bus as `A<3`, ensure that it is defined as either `A<3..0>` or `A<3>`.

Fatal error: Must specify lowerleft before upperright.

Description: This error message occurs when the `upperright` directive is defined before the `lowerleft` directive in the page border section of the Cref data file.

Solution: Ensure that you define the `lowerleft` directive before the `upperright` directive in the Cref data file.

Fatal error: Width of the page is too small.

Description: This error message occurs when the width of the drawing sheet is defined as less than 1000 Design Entry HDL coordinates.

Solution: Ensure that you have defined the width of the drawing sheet to be greater than 1000 Design Entry HDL coordinates. One quick way to check this width is to check the difference between the upperright x coordinate and the lowerleft x coordinate. Ensure that the difference between the two x coordinates is greater than 1000.

Fatal error: Height of page is too small.

Description: This error message occurs when the height of the drawing sheet is defined as less than 1000 Design Entry HDL coordinates.

Solution: Ensure that you have defined the height of the drawing sheet to be greater than 1000 Design Entry HDL coordinates. One quick way to check this width is to check the difference between the upperright y coordinate and the lowerleft y coordinate. Ensure that the difference between the two y coordinates is greater than 1000.

Fatal error: Must specify lowerleft before xmark.

Description: This error message occurs when the `xmark` directive (which defines the horizontal coordinate of the cross-reference key letters or numbers located along the top or bottom of the page) is defined before the `lowerleft` directive in the page border section of the Cref data file.

Solution: Ensure that you have defined the `lowerleft` directive before the xmark directive in the Cref data file.

Fatal error: Must specify lowerleft before ymark.

Description: This error message occurs when the `ymark` directive (which specifies the vertical coordinate of the key letters or numbers along the left or right side of the page) is defined before the `lowerleft` directive in the page border section of the Cref data file.

Solution: Ensure that you have defined the lowerleft directive before the ymark directive in the Cref data file.

Fatal error: Must specify lowerleft before pagenumber.

Description: This error message occurs when the `pagenumber` directive (which specifies the coordinate where CRefer prints a page number for the signal and part cross-reference summary sheets) is defined before the `lowerleft` directive in the page border section of the Cref data file.

Solution: Ensure that you have defined the `lowerleft` directive before the `pagenumber` directive in the Cref data file.

Fatal error: Too many symbol views for *<body_name>*.

Description: This error message occurs when a signal body has more than 100 views.

Solution: Ensure that no signal body is assigned more than 100 views. Reduce the number of views assigned to bodies exceeding 100 views and save the schematic.

Fatal error: Can't open *<body_file>* for read.

Description: This error message occurs when CRefer is not able to open the `symbol.css` file for reading.

Solution: Ensure that you have the correct `*.css` file and that you have at the least Read permissions on it.

Fatal error: Error in scanning symbol view *<symbol_view>*.

Description: This error message occurs when the symbol view *<symbol_view>* is not beginning with the `sym_` or `SYM_` characters.

Solution: Rename the symbol view to start with either `sym_` or `SYM_` characters.

Fatal error: - <option_name> requires argument.

Description: This error message occurs in one of the following cases:

1. You may not have specified the <project_name> after the `-proj` option while running CRefer from the command-line prompt.
2. You may not have specified any value for the `-q` option.
3. You may not have specified any value for the `-p` option.
4. You may not have specified any value for the `-s` option.

Solution: Ensure that you have specified the required arguments after the *<option_name>*. It is recommended that you avoid using the command-line option and configure the cross-referencing options in the Cross Referencer Options Dialog Box.

Fatal error: Illegal option: *<option_name>* Reference `CRefer -help`.

Description: This error message occurs when you have used an option that is not supported by CRefer.

Solution: Ensure that you have specified the required arguments after the *<option_name>*.

Fatal error: *<component_name>* not found in libraries.

Description: This error message occurs when you have used a component *<component_name>* in the schematic that is not available in the libraries specified by the `LIBRARY` directive in the `cds.lib` file.

Solution: Ensure that the library containing the component *<component_name>* is included in the list of libraries specified by the `LIBRARY` directive.

Fatal error: Could not open `<file_name>` for writing.

Description: This error message occurs when CRefer is unable to create any of the following reports: Basenets, Netsbypage, Synonyms and Crefparts.

Solution: Ensure that you have at least Write permissions on the disk where CRefer will create reports. Also check that the disk has enough space for the new reports.

Fatal error: Can't open `config.dat` for read.

Description: This error message occurs when the file *<your_install_dir>*`/tools/language/config.dat` is not found in the specified path.

Solution: Ensure that the `config.dat` file is available in the *<your_install_dir>*`/tools/language` directory.

Fatal error: Could not open project *<project_name>*.

Description: This error message occurs when the project file could not be opened for reading

Solution: Ensure that the path to the project file *<project_name>* is correct. If the path to the project file is correct, check whether all sections in the project file are valid. You might have an invalid section such as missing end section identifier causing the project file not being opened properly.

Non-Fatal Error Messages

Error: Failed to write *<file_name>*.

Description: This error message occurs when CRefer is unable to add signal and part cross-reference reports to the end of the schematic.

Solution: If you do not select the *Create Separate View for Schematic Reports* radio button, CRefer adds the reports to the end of the root design. Ensure that you have write permissions on the schematic. You may otherwise select the *Generate Flattened Schematic* check box and run CRefer. This will force CRefer to create a new view instead of appending the reports at the end of the schematic.

Internal error: backpagecref: Not enough space for signal xref summary. Please reduce the xref scaling factor.

Description: This error message occurs while writing the reports at the end of the schematic if your scale text is large and the CRefer text cannot be accommodated in the available space.

Solution: Reduce the scale text in the Cross Referencer Options - Format Tab to a value that allows CRefer text to fit in the available space.

Internal error: dourefs: Not enough space for unit xref summary. Please reduce the xref scaling factor.

Description: This error message occurs while writing the reports at the end of the schematic if your scale text is large and the CRefer text cannot be accommodated in the available space.

Solution: Reduce the scale text in the Cross Referencer Options - Format Tab to a value that allows CRefer text to fit in the available space.

Internal error: Unable to find border in page.

Description: This error message occurs when a non-standard page border is used in the schematic and the same page border is not defined in the Cref data file.

Solution: Define the page border in the Cref data file.

Warning Messages

Warning: Part *<part_name>* not found.

Description: This warning message occurs when you are using a special name for the part *<part_name>*, which CRefer cannot map properly. This name may include a weird combination of colons, semi commas, and percentage signs.

Solution: Use alphanumeric characters and the underscore character to name all parts in the schematic. This will to ensure that CRefer recognizes all parts properly.

Warning: Instance *<instance_name>* not found.

Description: This warning message occurs when you are using a special name for the instance *<instance_name>*, which CRefer cannot map properly. This name may include a weird combination of colons, semi commas, and percentage signs.

Solution: Use alphanumeric characters and the underscore character to name all parts in the schematic. This will to ensure that CRefer recognizes all instances properly.

Warning: No pin found for net *<net_name>* in the hierarchical symbol.

Description: This warning message occurs when the hierarchical symbol and the lower level associated schematic contain variable syntax.

Solution: The warning message is for informational purpose. You may decide whether or not you want to split the buses at the lower level in the schematic. You may also use a vectored signal and tap it to get different bits at the lower level associated schematic. This will provide better cross-referencing.

Warning: Path not found for *<component_name>*.

Description: This warning message occurs when CRefer does not find any PATH property for a component.

Solution: Add the PATH property to the component *<component_name>* and save the schematic. You may cross-reference the design again to ensure that CRefer properly places cross-reference information for the component *<component_name>*.

Warning: Could not open *<file_name>* to write.

Description: This warning message occurs when the disk is full or CRefer does not have the appropriate permissions to write into the binary file of the design (`page*.csb`). This file is a temporary file where CRefer writes data before appending it to the schematic being cross-referenced.

Solution: Ensure that there is enough disk space for CRefer to complete its operation. Also check that you have minimal Write permissions on the folder where you want CRefer to complete its operation.

Note: Warning messages 1 to 5 may occur multiple times in a design depending upon how many instances, parts, nets, or bodies have the problems as described above.

Warning: Signal *<signal_name>* at (*<ymark><xmark>*) has unknown input/output type.

Description: This warning message occurs when standard input/output types are not defined for the signal *<signal_name>* located at the grid coordinates (*<ymark><xmark>*).

Solution: Use offpage connectors to specify the input/output type for all signals and specify then in the cref.dat file.

Warning: Signal *<signal_name>* is labeled only once in the design.

Description: This warning message occurs when the signal *<signal_name>* appears only once in the design.

Solution: This warning is for information purpose. By default, this warning is not displayed. If you want to display the warning, select the *Show Warnings for Unique Signals* check box.

Warning: Signal *<signal_name>* at (*<ymark><xmark>*) is not driven anywhere.

Description: This warning message occurs when the signal *<signal_name>* located at the position (*<y mark><x mark>*) appears only as an input in the design. By default, this warning is not displayed. If you want to display the warning, select the *Ignore Input Only Signals* check box.

Solution: Ensure that the signal uses default outputs.

Warning: Signal *<signal_name>* at (*<y mark><x mark>*) has too little display space.

Description: This warning message occurs when there is too little display space near the signal *<signal_name>* to put the cross-reference on the schematic. In such cases CRefer will make them invisible.

Solution: While designing the schematic, try to leave sufficient space near signals to ensure proper cross-referencing. However, you may not always have sufficient space to leave for cross-referencing. You may select the *Make Overlapping Cref Properties Visible* check box to direct CRefer to place the cross-references for all overlapping signals (for example, the signal *<signal_name>*) as visible properties.

Warning: Flag body *<flag_body_name>* at (*<y mark><x mark>*) has no signals attached to it.

Description: This warning message occurs when you have not attached any signal to the flag body *<flag_body_name>*.

Solution: While designing the schematic, ensure that you attach a signal to each flag body or port, and leave sufficient space near the body to ensure proper cross-referencing.

Note: Error numbers 6 to 10 appear only once in a CRefer run. These errors appear at the first occurrence of the instance causing the error. If you leave proper space for CRefer properties and follow the basic rules in preparing a schematic, you will not encounter any of these errors.

Warning: Multiple versions of sheet border of design *<design_name>* specified in the cref.dat file using version: *<version_number>*.

Description: This error message occurs when there exists more than one version of page border in the `cref.dat` file. The warning is for information purpose as CRefer picks the first available version of the page border in the `cref.dat` file.

Solution: Check the version of the page border that you want to use and ensure that it is the first version in the `cref.dat` file. If you have a local `cref.dat` file for your project, it may be a good idea to avoid using multiple versions.

Note: If you are using a `tscrpage.dat` file from release 13.6, make changes to the page border version in that file.

A

Archiver Dialog Boxes

Archiver Dialog Box

Procedures Related Info

The fields in the Archiver dialog box are described below.

Project File	Specify the name and path of the project or click *Browse* to browse to the location where the project exists. If you leave this field blank, the *OK* button at the bottom of the dialog box is disabled.
Location	Displays the location of the specified project file.
Output Directory	Specify the directory path where you want to save the Archiver output. Alternatively, you can click *Browse* to select the directory. If you leave this field blank, the *OK* button at the bottom of the dialog box is disabled.
Views To Traverse	Specify the views that Archiver should traverse to archive the design by selecting one of the following radio buttons: *All*—By default, Archiver traverses all views. If you do not want to archive all the views, select the *Schematic* option button. *Schematic*—Select this option if you want Archiver to traverse only the schematic view. The *Verilog* and *VHDL* check boxes are enabled when you select this option. Select the *Verilog* or *VHDL* check box to traverse the Verilog or VHDL view, respectively, of your design.

Other Files/ Directories To Archive	Displays the list of all the files and directories in the project directory associated with the project file that you specified. For example, if you specify <.../.../projdir/projfile.cpm> as the project file, Archiver displays all the files in the <.../.../projdir/> directory in the Other Files/ Directories to Archive list. You can select the files and directories to be archived in addition to the specified project file. By default, Archiver creates a new directory named <root design>_archive. This directory contains the root design, the project file, the `cds.lib` file, the part table files, the files corresponding to all components in the design, and any other files or directories that you select here.
Create Single File Archive	Select this check box to create a single file of the entire archive. When you select this check box, the *Delete Archived Directory* and *Compression Utility* fields are enabled.
Delete Archived Directory	Select this check box to mark the archived directory for deletion.
Archive All Designs	Select this check box to archive all your designs. The design can contain multiple root designs. Archiver traverses only the root design specified in the project (.cpm) file. If all root designs need to be archived, select the *Archive All Designs* check box.

Compression Utility

Displays the zip or tar command based on the operating system on your computer.

The compression command displayed in a Windows-based system is as follows:

```
zip -r $archive $location
```

The compression command displayed in a UNIX-based system is as follows:

```
tar -cvf $archive $location
```

Here, `$archive` represents the name of the zip file, while `$location` represents the location where the output file will be saved. This location is the same as the value stored in the *Output Directory* field.

You can change the `$archive` or `$location` variables and customize the path where the compressed archive file will be created. However, you may not need to change the values in the Compression Utility field.

The zip utility or tar utility must be installed on your machine.

Ignore Files

Specify the file names, file extensions, or folder names that you want Archiver to ignore while archiving. Multiple entries must be separated by spaces. This field does not accept full paths to files.

You can also specify wild card characters in this field. For example `*.log *.mkr pxl.log physical` are all valid entries for this field.

OK

Starts archiving the design. Archiver creates the archived directory and a compressed single file archive (named `<root design>_archive.zip` or `<root design>_archive.tar`) for the selected design. The Archiver Progress window appears when you click on this button.

This button is disabled if you do not specify the project file or the output directory.

Cancel

Closes the Archiver dialog box.

Procedures

- Creating a New Archive
- Archiving a Design from the Command Prompt

Related Info

- Archived Directory Structure

Archiver Open Dialog Box

Procedures Related Info

The fields in the Archiver Open dialog box are described below:

Project

Helps you specify the project you want to open by typing its path. Alternatively, you can use the *Browse* button on the right of the *Project* field to select the project file. When you specify a project file, two things that happen are

a. The location of the project file is automatically displayed in the *Location* field.

b. A list of libraries present in the archive libraries and the corresponding reference libraries is displayed.

cds.lib

Helps you change the reference libraries displayed in the list by specifying the path to a different library in the `cds.lib` field. You can either type the path or use the *Browse* button on the right of the `cds.lib` field to select another `cds.lib` file.

Compare archive and reference libraries (Diff)

Helps you specify that you want to list the reference cells that are different from the archived cells. The DIFF Utility dialog box appears. See"DIFF Utility Dialog Box" on page 147 for information about this dialog.

Apply

Applies the changes you have made without closing the dialog.

Update Archive

Commits your changes to a design extracted from an archive and recreates the archive. Archiver detects the changes made to the design and acts in the following ways:

a. If there are new components in the design, Archiver copies these components from the reference libraries to the present archive.

b. If there are unused components in the archive that are not used in the design, Archiver displays the Delete Unused Components dialog box from where you can selectively delete the unwanted components. See"Delete Unused Components Dialog Box" on page 148 for more information.

Update Archive does not change these components. To change the components, run the DIFF utility and selectively change the cells before you click the *Update Archive* button.

Close

Closes the Archiver Open dialog box.

Procedures

Opening an Archive

Related Info

Archived Directory Structure

DIFF Utility Dialog Box

Procedures

The DIFF Utility dialog box appears when you select the *Compare archive and reference libraries (Diff)* check box in the Archiver Open dialog box. The DIFF utility compares the cells in reference and archive libraries and displays a list of cell-level differences. After reviewing the list, you can replace the cells in the archive library with the ones found in the reference libraries.

Replace Local	Marks a cell to be replaced by its referenced version.
Replace All Locals	Marks all cells to be replaced by their referenced versions.
OK	Closes the Diff Utility and displays a list of cell-level differences between cells in the reference and archive libraries.
Cancel	Cancels the operation.

Procedures

Opening an Archive

Delete Unused Components Dialog Box

Procedures

This dialog box lists components present in the archive but not used in the design. You can delete some or all of these components here.

Delete	Marks the selected component for deletion.
Delete All	Marks all unused components for deletion.
OK	Deletes selected components.
Cancel	Cancels the operation.

Procedures

Opening an Archive

B

BOM-HDL Dialog Boxes

BOM-HDL

Procedures

The first dialog box is displayed when you run the BOM tool is BOM-HDL. This dialog box is divided in three main sections:

1. Output Options
2. Report Format Options
3. Variant BOM Options

These are described below:

Output Options group box

Template File

Enter the path to the template file in this field. Alternatively, you can use the *browse* (...) button to browse to the file. You can also define the template file by using the `-t` or `-T` switch in the `bomhdl` command.

You can enter the full path or the relative path. If you specify the full path, BOM-HDL searches for the template file in that path. If you specify the relative path, BOM-HDL searches for the template file relative to the `bom` view of the current project. Therefore, if your template file is present in the BOM view, specify only the file name; otherwise, specify the complete path to the template file. If the template file is not found in the specified path, an error message is displayed.

Note: When the BOM-HDL dialog box is displayed for the first time, the *Template File* field is filled with the template file path in the following order: (1) the last saved template file in the project file; (2) the `bom` view of the current project; and (3) the locations as specified in the setup.loc file.

Customize

Launches the Customize Template dialog box. The information specified in the template file is used to fill the initial entries in the different tabs. After customizing the values, you can save the template file (with the same name or with a new name) and use it for future runs.

Output File

Enter the path of the output file that contains the BOM report in this field.

By default, the value of the *Output File* field is picked from the project (.cpm) file. If the project file does not have information specified about the output file, the *Output File* field displays the `BOM.rpt` file in the bom view of the design as the default output filename.

Note: Depending on the report format type you choose and whether you are generating the BOM report for the base schematic or a variant, the name of the file in the *Output File* field automatically changes. For example, if you are generating a variant BOM report for `INDIA` variant in HTML format, the name of the file will be `INDIA.html`.

Note: If you specify an output file but do not specify its path, the BOM-HDL tool saves the output file in the `bom` view of the project. If you specify the full path of the output file, BOM-HDL saves the output file in your specified path.

View
Choose this button to view the BOM report (as defined in the *Output File* field) in the default viewer. The viewer will be a text viewer if the output file is in the text or spreadsheet format, or an HTML viewer if the output file is in the HTML format.

Apply Filters
Choose this check box if you want to define conditions on any property. To define a condition, use the *Filters* button in the Customize Template - Physical Part Specifications tab.

Note: The project file stores information about whether you have applied filters in the last run. If you have applied filters, the *Apply Filters* check box appears selected in subsequent runs.

Text File in Tabular Form
Choose this radio button to create the output file in the ASCII text format.

Spreadsheet Format
Choose this radio button to create the output file in the spreadsheet format, which can be imported in an application such as MS Excel. If you choose the *Spreadsheet Format* radio button, you can also specify the delimiter in the *Delimiter* list box. The default delimiter is comma (,). You can also use semicolon, colon, space, dot, tab, or hash as the delimiter.

HTML
Choose this radio button to create the output file in the HTML format. The file is stored in a tabular format as defined by the settings in the template file.

Variant BOM
By default, the options in this group box are not visible. However, you can choose the *Variant BOM* button to display the different options for generating the BOM reports. When you choose the *Variant BOM* button, the button toggles its name to *Base BOM*. Choose the *Base BOM* button to hide the variant BOM report options.

Variant File
Enter the name of the variant database file. You can use the *browse* button to browse to the file. Alternatively, you can type the path to the file. You can specify absolute path or relative path from the project directory. If you specify relative path, dot (.) is taken as the project directory and not the working directory.

Note: To generate a BOM report for a variant, you must have the variant database file (`variant.dat`), which is generated by using Variant Editor.

Note: If you do not specify the report format, the format is picked from the project file. If the report format is not specified in the project file, the BOM report is generated in the text format.

Base BOM	The *Base BOM* button is displayed only when variant options are expanded by selecting the *Variant BOM* button.
What to Output	Choose the type of the BOM report to be created by clicking on one of the following three options.
Base Schematic BOM	Choose this radio button to generate the normal BOM report for the base schematic with no variant information. The report generated contains the listing of all the components used in the base schematic. All the property values including the part number correspond to the values chosen in the base schematic.
Variant BOM	Choose this radio button to generate the variant BOM report. This report contains the listing of all the components used in a particular variant. All the property values including the part number correspond to the values selected in the particular variant. When you choose the *Variant BOM* radio button, the list box located to its right becomes available for selection. You can choose the variant for which you want to create the BOM report. You can also choose the *All Variants* option. This will generate variant BOM reports for all the variants. For example, if there are three variants: `INDIA`, `USA`, and `EUROPE` and you choose the *All Variants* option, BOM-HDL will generate the following BOM reports in the bom view of the project: `INDIA.rpt`, `USA.rpt`, and `EUROPE.rpt`. **Note:** If you want to customize the template file for variant specific options, use the Customize Template - Variant Settings on page 164.
Variant Comparison BOM	Choose this radio button to generate a number-based comparison between the components of the base schematic and all the variants. **Note:** While generating the comparison BOM report, only the preferred values of the components and alternate groups are considered.
Generate	Choose this button to generate the BOM report.
Close	Choose this button to close the BOM-HDL dialog box.
Help	Choose this button to display Help about the BOM-HDL dialog box.

Procedures

- Prerequisites for Running BOM-HDL
- Creating the BOM for the Base Schematic

- Creating Variant BOM Reports

Customize Template - Report Parameters

Procedures

The Customize Template dialog box can be opened by clicking the *Customize* button in the BOM-HDL dialog box. The default tab displayed is Report Parameters. This dialog box is divided into five main sections:

- Report Header
- Row Column Separator
- RefDes
- Callouts
- Miscellaneous

These sections are described below:

Report Header	The *Report Header* grid box displays the header properties and their values. You can enter new properties or delete the existing properties. Notice that a check box precedes each header property. If you clear the check box, the property is not displayed in the BOM report. By default, the Report Header list has five properties.
Title	Specifies a title to appear in the BOM report. The default value of the title is Bill of Materials. You can edit this value. If you want to blank the value of the title then type: TITLE = "".
Date	Specifies the current date as obtained from your system's clock to appear in the BOM report. **Note:** If you click in the *Value* column of the *Date* property, a combo box displays. You can choose this box to display different date formats. You can choose any of the available formats. If the check box to the left of the *Date* property is checked, the current date is included in the BOM report. A value unchecked omits the date entry in the report.
Design Name	Specifies the name of the design to appear in the BOM report. You cannot edit this option, as the design name is picked up from the project file.

Template Name	Specifies the name of the BOM template to appear in the BOM report.
Callout File Name	Specifies the name of the callout file to appear in the BOM report. The default template file stores the callout file name value as `bom.callout`. Besides the above 5 properties, which are selected by default, there are 5 optional properties that you can select and include in the Report Header.
Product	Displays the default name is BOM-HDL.
Version	Displays the product version number.
Description	If you are generating a variant comparison BOM, BOM-HDL displays Part Number based Comparison BOM as description. For Base schematic BOM, BOM-HDL displays the description as Base schematic BOM. If you have defined a description for variant and are generating a variant BOM report, the description is listed in this row.
Project Path	Displays the path to the project file.
Variant	Displays the name of the variant. **Note:** The *Variant* property will be visible only when you are generating variant reports.
Up	Moves a selected property up in the *Report Header* list.
Down	Moves a selected property down in the *Report Header* list. **Note:** If you move the mouse over the number to the left of *Header Property* column, an arrow will display. You can click at the number and choose the property. You need to choose a property, to move it up or down in the *Report Header* list or to remove it. You can even choose a property by clicking anywhere in the *Header Property* or *Value* field. If you have selected one property and want to choose more properties, keep the `Shift` key pressed and choose the required property.
Remove	Deletes the selected property. The deleted property is removed from the template file and will not be available in future runs, unless added again using the *Add New* button.
Add New	Choose this button to add a new property. When you choose the *Add New* button, the Add Header Property dialog box is displayed. You can use this dialog box to add a new property and define its value and title.

Column Separator Enter the character to be used to separate columns in the BOM report. The value of the column separator character in the default template is a space (" ").

Column Pad Enter the character that appears on both sides of the column separator character. The value of the column pad character in the default template is a space (" ").

Row Separator Enter the character to be used to separate rows in the BOM report. The value of the row separator character in the default template is a null character ("").

Header Separator Enter the character to be used to separate the header row from the part rows. The value of the header separator character in the default template is an equal sign ("=").

Note: The row and column separators will work only if you are displaying the report in a tabular format such as an ASCII file. If you set a value for the row separator or the column separator, and set the report format as *Spreadsheet* or *HTML*, the values for the row separator and the column separator are overwritten by standard values. The Spreadsheet report will be separated by the *Delimiter* defined in the BOM-HDL dialog box. The HTML report will be displayed in a standard table format with grid-lines creating a matrix structure.

Unique Select this to list each reference designator for a part on a separate row.

Ranges Select the *Ranges* check box to indicate that ranges, such as `U1-U5`, can be used for reference designators. The default template file has the *Ranges* check box as selected.

Note: When the *Ranges* check box is not selected, the *Range Separator* and *Minimum Members in Range* fields become inactive.

Range Separator Enter the character you want to use as range separator for reference designators. The default value of the range separator in the default template file is dash (-). You can change this default value and specify a different character. For example, if you have four consecutive reference designators `U1`, `U2`, `U3`, and `U4`, and if you have specified colon (`:`) as the range separator, the BOM report will display the range as `U1:U4`.

Minimum Members in Range	Sets the minimum number of members in a range. The default template file defines the value of the minimum number of members in a range as 3. You cannot define a number less than `3` or greater than `1024` as the *Minimum Members in Range* value. **Note:** The `BOM_PART` property must be the first property in the Physical Part Specifications tab of the Customize Template dialog box for the ranges function to work.
Title	Defines the title that will be used as the header in the callouts section of the report. The title value in the default template file is `Callouts`.
File	Defines the name of the callout file to be used. The value of the callouts file in the default template file is *<your_inst_dir>*/`share/cdssetup/bom.callouts`. You can change the filename by specifying the new path in the *File* field or by using the browse (...) button.
Edit	Choose this button to open the Callouts Editor dialog box where you can change the quantity of each part, or add or delete mechanical parts. If no callout file is specified, the default callout file *<cds_inst_dir>*/`share/cdssetup/bom.callouts` is opened for editing.
Intersperse Associated Mechanical Parts	Choose this check box to list the mechanical parts next to the physical parts to which they are associated. If the *Intersperse Associated Mechanical Parts* check box is not selected, the associated mechanical parts are listed in the Callouts section of the BOM report. The default template file defines that the *Intersperse Associated Mechanical Parts* check box is selected by default.
Header File	Defines the name of the file that contains the header to be used in the BOM report. This file is appended to the standard header as specified in the *Report Header* section of the Customize Template dialog box. The contents of the Header file are included whenever the header is printed. The default template file defines the value of the *Header File* field as "" (null string). **Note:** The header file must be an ASCII text file.

Page Length	Defines the number of lines per page. By default, the column headers and page numbers are printed at the top of each page. **Note:** By default, the BOM report is of a single page. By using the *Page Length* field, you can divide the report into multiple pages, where the length of each page is defined by the value in the *Page Length* field. Each of these pages will display the column header and the page number. You can also print the report header at the top of each page. For this, choose the *Print Header at Top of Each Page* check box. **Note:** You cannot set the value in the *Page Length* field below 20 because that will display too less information/ page. If you set any number below 20, when you save the file, the page length value will automatically revert to 20.
Print Header at Top of Each Page	When you change the value in the *Page Length* field, the *Print Header at Top of Each Page* check box becomes active. If you choose this check box, the header information will be printed at the top of every page. **Note:** If you choose Spreadsheet or HTML as the format for the BOM report, the report will be displayed in a single page. The values in the *Page Length* field and the *Print Header at Top of Each Page* check box will be ignored.
Print Column Header	By default, the *Print Column Header* check box is selected signifying that BOM report will contain a column header that lists the names of properties displayed in the report. To remove the display of the column header, clear the selection from the *Print Column Header* check box. **Note:** The *Header Separator* will be displayed in the BOM report only if you are printing the column header.
Save	Saves the changes in the template file as defined in the *Template File* field of the BOM-HDL dialog box.
Save As	Saves the changes in the template file with another name. This will display the Save BOM Template As dialog box, where you can specify the new path to the BOM template file.
Close	Closes the dialog box

Procedures

Setting Report Parameters

Customize Template - Physical Part Specifications

Procedures

The Physical Part Specifications tab in the Customize Template dialog box allows you to specify the properties that need to be extracted from the schematic and how to format the extracted information in the BOM report. This dialog box is divided in two main sections:

1. Report Columns group box
2. Sorting group box

These are described below:

Report Column This group box allows you to determine which schematic properties needs to be displayed as columns in the BOM report. T

Property This column specifies the properties that will be displayed in the BOM report. All the properties used in the schematic design and some BOM related properties (`BOM_PART`, `BOM_INST`, and `BOM_QUANTITY`) are listed in the *Property* column. There is a check box to the left of each property. If you choose the check box corresponding to any property, the property appears as a column in the BOM report. If the check box corresponding to any property is not selected, that property is not displayed in the BOM report.

Note: The first property listed in the *Report Columns* grid box is treated as the key property and it is used to sort the BOM report. By default, the first property in the *Report Columns* grid box is `BOM_PART`. This property represents the primitive name used for the part in the `pstchip.dat` file. The BOM report is, therefore, sorted on the `BOM_PART` property. To make another property as the key property, move it to the first row in the *Property* column. However when you change the key property, ensure that all components to be listed in the BOM report must have some non-null value against that property.

Note: See *Packager-XL Reference* in `cdsdoc` for more information about properties.

Mech

Mech is a short-form of Mechanical. If you choose the *Mech* check box for any property row, that property is extracted in the callouts (mechanical parts) table.

Note: You must have at least one property with the *Mech* check box selected. If you try to deselect all *Mech* check boxes then an error message appears stating that you must select at least one column.

Title

Defines the column title. A default title is provided for all properties. The title for the special BOM related properties is available in the default template file. For example, the `BOM_PART`, `BOM_INST`, and `BOM_QUANTITY` properties have `Part Name`, `Ref Des`, and `Qty` as the title. The title for all properties that are obtained from the netlist files is the same as the property name. You can edit the title for any property.

Width

Defines the number of characters that will be used as the width for the column. If an entry is longer than the column width, it continues in the next line. Therefore, the report always appears in a tabular structure. To change the value in the *Width* cell, click in the cell and either enter the new value or use the spin box to choose the new value.

The maximum width that you can define is 1023 characters while the default width is obtained from the template file. Two columns are separated by a space (or any other character specified in the *Column Pad* and *Column Separator* fields in the tab).

Note: It is advisable to set the column width to allow the maximum possible property value. If the column width is less than the property value, the BOM report will be cluttered.

Justification

Specifies whether the column text will be left justified, right justified, center justified, or not justified (none). To choose a value for *Justification*, click in the cell. A list box appears with four values: `Left`, `Right`, `Center`, and `None`. You can choose any of these values.

Total

Choose the check box to specify that the column total for the selected property must be displayed in the BOM report. This total is displayed at the end of the report.

Subtotal

Choose the check box to specify that the value displayed for the column is a product of the property value and the `BOM_QUANTITY` for that row.

Quote

When you generate the BOM report as a spreadsheet, select the *Quote* check box to ensure that the property value is enclosed in double quotes if the value has a delimiter.

For example, if you are using a semi colon (:) as the delimiter and have a property such as `D:E`, the property value will be displayed as `"D:E"` when the Quote check box is selected.

The *Quote* option is also useful if you have a property value with the following format: xxx-xx-xx, say `PART-NUMBER = 9011-05-11`.

In such cases, the spreadsheet program modifies the format and changes the data type to date and displays the data as `5/11/9011`.

When you select the *Quote* check box, the BOM utility adds a leading space and ensures that the spreadsheet program does not modify the data type.

The *Quote* option is also useful if the property value has a space in it. Without the quote option, the value might be split in the output report file.

You can also use the `-forcequotes` argument as "`bomhdl -proj *.cpm -forcequotes`" in the command line for column values. If you use this argument, quotes are added to all column values regardless of the *Quote* check box selection.

Missing

Enter a string value that will be displayed if a property is missing for a component. The default value is "?". You can change this value.

Note: The columns in the BOM report will appear in the order specified in the grid rows. To reorder report columns, you need to first choose a column by clicking at its serial number and then change the order of the grid row using the `Up` and `Down` arrow buttons.

Note: An associated mechanical part does not inherit PTF properties from the parent electrical part. The property columns in the BOM report are therefore based on this field. In case no customization is done, the default value is used.

Report Columns

group box

Serial Number

Choose this check box to assign a serial number to all the parts displayed in the BOM report.

Start	This field becomes active when the *Serial Number* check box is selected. You can change the starting serial number to one of your choice by typing the new number or by using the spin control.
Filters	Choose this button to launch the Part Filters dialog box. This dialog box allows you to include specific parts from the BOM report by defining the conditions on part names and properties.
Up	Used to move a row up in the *Report Columns* grid box.
Down	Used to move a row down in the *Report Columns* grid box.
Delete	Choose this button to delete the selected row in the *Report Columns* grid box. Therefore, if you want to remove any report column, choose the row corresponding to that column and click on the *Delete* button. BOM-HDL will remove the report column from the BOM report and the BOM template. **Note:** The *Delete* button works only for the properties that you have added. If you try to delete a row corresponding to a property seeded by default in the Physical Part Specifications tab, that row will re-appear the next time you load the Customize Template dialog box.
Add New Column	Choose this button to display the Add New Column dialog box. You can use this dialog box to add a new property and define its name and title.
Style	Determines the sorting style. The sorting style in the default template file is alphabetic. You can change this style to alphanumeric. If you have three parts with reference designators U1, U2, and U11, then, in the alphabetic sorting style, the result will be U1, U11, and U2 while in the alphanumeric sorting style, the result will be U1, U11, and U2.
Order	Determines the sorting order. The sorting style in the default template file is ascending. You can change this style to descending.

Procedures

Selecting the Physical Parts

Customize Template - Variant Settings

Procedures

The *Variant Settings* tab in the Customize Template dialog box allows you to customize the variant section in the template file. This tab will display only if you are generating variant BOM reports (that is, you have selected the *Variant BOM* button in the BOM-HDL dialog box). This dialog box is organized in two group boxes:

1. Alternates group box

2. Miscellaneous group box

These are described below:

Alternates	This group box allows you to define three types of reports based on the inclusion or exclusion of information about alternates.
Exclude Alternates	Choose the *Exclude Alternates* radio button to generate a variant report that does not include alternates. The report lists only the preferred value of components. **Note:** When a function is excluded from a variant, the components which are part of the function are DNI for that variant. While generating a variant BOM, you can show DNI components by selecting the Include DNI Components option. This will show all the components of the excluded function (DNI and non-DNI) as DNI.

Include Alternates

Choose the *Include Alternates* radio button to generate a variant report that contains information about all preferred and alternate values of components. By default, all DNI components are displayed as separate list at the bottom of the report.

When you choose the *Include Alternates* radio button, the *Duplicate SNo. For Same Reference Designator* check box and *Alternate Limit* check field are displayed.

By default, the BOM report does not repeat the serial number for the same reference designator in case of alternate values. If you want to display the reference designators for alternate values, choose the *Duplicate SNo. For Same Reference Designator* check box. You can choose this check box only if `refdes` is the key property in the Physical Part Specifications tab.

The default limit for the number of alternates that will display in the BOM report is 99. You can reset this limit to any number between 0 and 99 by typing the new number in the *Alternate Limit* field.

Delta Only

Choose the *Delta Only* radio button to generate a variant report that lists the differences between the variant and the base schematic. All the alternate values of components for the variant appear as DNI in this report.

Miscellaneous

This group box allows you to customize the formatting of the BOM report. You can specify whether DNI components be displayed in the report. You can also specify your own designators to specify the status of different components.

Include DNI Components

If you want the variant report to contain the list of DNI components, choose the *Include DNI Components* check box. This will display the list of DNI components but not their property values.

When the *Include DNI Components* check box is selected, the *Show Values For DNI Components* check box displays. Choose this check box to display the property values for DNI components.

Status

This group box contains five fields, which you can use to customize the information in the BOM report.

Preferred

Enter the value that you want to use to designate the preferred status in the BOM report. By default, the value is `Pref`.

Alternate	Enter the value that you want to use to designate the alternate status in the BOM report. By default, the value is `Alt`. For the first alternate, this value is `Alt1` and for the second alternate this value is `Alt2`. For example, if you specify `ABC` as the alternate value, the first alternate will be assigned the status `ABC1` and the second alternate will be assigned the status `ABC2`.
DNI	Enter the value you want to use to designate the DNI status in the BOM report. By default, the value is `DNI`. **Note:** You cannot have a blank value in the *Preferred*, *Alternate*, or *DNI* fields.
Values Different From Base	The value in this field is displayed next to a component when it has a value, which is not the same as the value in the base schematic. The default template file uses `*` to represent component values different from the value in the base schematic. You can change this value.
Value Same as Base	The value in this field is displayed in comparison BOM reports when the preferred value of a components is same as the base schematic values. The default template file uses a blank space to represent the component values that are same as the base schematic values. You can change this value.

Procedures

Changing Variant Settings

Add Header Parameter

Procedures

The Add Header Parameter dialog box is used to add a header property in the *Report Header* list. You access this dialog box by clicking on the *Add New* button in the Physical Part Specifications tab. The dialog box contains the following options:

Name

You can enter the name of the new property that you want to include. To display the list of properties that are currently placed in the *Report Header* list, choose the list button to the right of the *Name* field.

Value

By default, this field is unavailable and is seeded by the values from the template file or the project file. For example, the `DESIGN`, `TEMPLATE`, and `CALLOUT` properties obtain their value from the template file. The default values of the `VARIANT` and `DESCRIPTION` property are obtained from the variant database while the date of the system clock is used as the value for the `DATE` property.

Note: The `TITLE` property does not have any default value displayed in the Add Header Parameter dialog box. You can, therefore, specify any value for the title.

When you enter a new property name in the *Name* field, the *Value* field becomes active. You can now edit the value. After you have added a new property name and value, you can choose the *OK* button to add the property. If you do not want to save the new property, click *Cancel*.

Procedures

Customizing Report Parameters

Add New Column

Procedures

The Add New Column dialog box is used to add a property in the *Report Columns* list. You access this dialog box by clicking the *Add New* button in the Customize Template dialog box. The dialog box contains the following options:

Name	You can enter the name of the new property that you want to include. To display the list of properties that are currently placed in the *Report Header* list, choose the list button to the right of the *Name* field.
Value	Enter the value of the new property that you defined in the *Name* field. When you have added a new property name and value, you can choose the *OK* button to add the property. If you do not want to save the new property, click *Cancel*.

Procedures

Customizing Report Columns

Callouts Editor

Procedures

The Callouts Editor dialog box displays all mechanical part, all properties corresponding to them, and their quantity. The information about the callout properties is picked from the template file or the callouts file that you can specify using the *Callouts Title* field in the Customize Template dialog box. Changing callouts information

You can use the Callouts Editor dialog box to add or remove mechanical parts. You can even modify the quantity for each mechanical parts. To edit the `BOM_QUANTITY` property, choose that cell and enter the new value. The changed value will be effective only when you choose the *Save* button.

Note: Callouts are displayed as the last section in a BOM report. This section lists the serial number and the values of those properties the `MECH` check box for which are selected in the Customize Template - Physical Part Specifications tab. If you want to display values for any property in the Callouts section of the BOM report, select the `MECH` check box corresponding to that property in the Customize Template - Physical Part Specifications tab. Ensure that you have at least one `MECH` check box selected for any property to display its value in the BOM report.

The Callouts Editor dialog box contains the following buttons:

Add Part	Choose this button to display the Mechanical Part browser from where you can add more mechanical parts. You can choose the PPT rows in the Mechanical Part browser to add to the callouts file. The default quantity of the part is `1`. If you need to change this value, you can later edit it. You can even assign a non-integer value to the part. **Note:** To include mechanical parts in the ptf files, ensure that they are assigned the `CLASS=MECH` property. BOM-HDL will list only those components from ptf files (as mechanical parts) that are assigned the `CLASS=MECH` property. You can add these parts using the *Add Parts* button.
Remove Parts	Choose this button to remove the mechanical parts from the list. To remove a mechanical part, first choose it and then click on the *Remove Parts* button.
Save	Choose this button to save the changes to the callout file.

Close	Choose this button to close the Callouts dialog box and return to the place from where you opened the dialog box.

Procedures

Customizing Report Parameters

Part Filters

Procedures

The BOM-HDL dialog box supports improved filter options using which you can filter parts in a variety of ways.

The dialog box fields are described below:

Add Filter	This group box allows you to choose a property and define a condition. All parts that match this condition are listed in the BOM report.
Property	The Property list box displays the list of component properties. You can choose a property to define a condition on it.
Condition	The Condition list box displays a list of conditions that you can use to filter parts. The conditions available are: `Is` Equal To `Is` Greater Than `Is` Less Than `Is` Not Equal to `Is` Greater Than Equal to `Is` Less than Equal to `Like` - This condition applies for wildcard entries
`Property Value`	You define the value of the property in this field.
`To Selected Filter`	Choose this button to add the condition to the selected filter in the filter list. The new condition will be added to the selected filter with an AND condition.
As New Filter	Choose this button to add the filter as a new filter to the list of filters.
Filters List	This group box displays the filter list where all the defined filters are displayed.
Rename	By default, the first filter is created with the name `Filter1` and the second filter with the name `Filter2`. To rename the filter, use the *Rename* button.

Delete	Choose this button to remove the selected filter from the *Filters List*.
Clear All	Choose this button to remove all the filters from the *Filters List*.

Procedures

Setting Filters on Parts

Physical Part Filter

When you choose the `Add Part` button to add a mechanical part in the Callouts Editor dialog box, the Physical Part Filter dialog box displays. This dialog box is used to include mechanical parts from the Physical Part Table (PPT) files. The Physical Part Filter dialog box contains the following fields:

These are described below:

Part Names	The *Part Names* box displays the part names. When you choose any part name, all the available PPT rows for that part are displayed in the *Property List*.
Property List	The *Property List* displays all physical property names from left to right in the order you specify in the Property Options dialog box. The *Filter* field on the top of each column heading allows you to filter physical property values based on the string you enter. For example, if you have ten rows appearing in the Physical Part Filter dialog box and you want to filter out and use only those physical parts with part numbers starting with 1, enter `1*` in the *Filter* field and press enter. The Physical Part Filter dialog box displays only those physical part table rows with part number starting with 1.
Use Case Sensitive Filtering	Choose the *Use Case Sensitive Filtering* check box to use case sensitive filtering of physical properties from a PPT. For example, consider that the PPT for the selected part has two rows and the values for the `JEDEC_TYPE` property are `C123` and `c123`. If you choose case sensitive filtering, type `C*` in the filter, and press Enter. The Physical Part Filter dialog box will display only that row, which has `C123` as the value for the `JEDEC_TYPE` property.
Reset Filters	Choose this button to reset all applied filters to default values. By default, no filter is applied for any property.
Options	Choose this button to display the Property Options dialog box. The Property Options dialog box allows you to define the settings for annotating physical properties on the schematic.

Property Options

When you choose the *Options* button in the Physical Part Filter dialog box, the Property Options dialog box displays. This dialog box is used to define settings for annotating the physical properties on the schematic.

The Property Options dialog box contains the following fields:

These are described below:

Property Order

The *Property Order* list box displays the properties from the Physical Part Filter dialog box. You can use this option to re-arrange the order in which the physical properties are annotated in the schematic and displayed in the Physical Part Filter dialog box.

Filter

The *Filter* field allows you to filter physical property values based on the string you enter. For example, if you want the Physical Part Filter dialog box to display only that row of the PPT which has the value of `VOLTAGE` as `63V`, choose the `VOLTAGE` property in the Property Order list box, enter `63` in the *Filter* field, and choose *Apply*.

Annotate

Choose a suitable option to decide how physical properties are annotated on a schematic.

`Invisible`–Prevent annotation of physical properties on the schematic.

`Name`–Annotate only the names of the physical properties on the schematic.

`Value`–Annotate only the values of the physical properties on the schematic.

Note: If you choose `Invisible`, physical properties are added on the schematic and are read by all tools, but are not visible on the schematic.

Note: You cannot choose the `Invisible` option for a key property.

Numeric Sort

Choose the *Numeric Sort* check box to do numeric sorting. In numeric sorting, the lesser numbers are placed initially followed by the greater numbers. For example. 50 is placed before 150. If you do not choose the *Numeric Sort* check box, sorting is based on strings. In String sorting, the first characters of two values are taken and compared. If they are the same, the second characters are compared. For example, if you compare 150 and 50,150 is declared less than 50 because 1, the first number in 150, is less than 5, the first number in 50.

Hide Column

Choose the *Hide Column* check box to suppress the display of this column in the Physical Part Filter dialog.

Options Sets>>

Choose the *Options Sets* button to expand the Physical Options dialog box. You can define option sets using this part of the dialog box. What is an Option set? When you place a part with physical information in Design Entry HDL, you can specify property options to define the format and visibility of the properties. For example, you can define the property options in such a way that the properties such as `PART_NUMBER` and `TOL` will not appear on the schematic. These settings and definitions can be stored in a text file, named `pptoptionset.dat`, and reused by a group for a given project. The settings for a particular part that is stored in the `pptoptionset.dat` file is called an option set.

Name/Pattern

The Name/Pattern field displays the name of the currently displayed option set. To copy an option set:

1. Choose a name in the *Option Set* list.
2. Change the name in the *Name/Pattern* field.
3. Click *Add.*

Option Sets

The *Option Sets* box displays the current list of option sets in the list box.

Add

Choose the *Add* button to add a new option set with the name displayed in the *Name/Pattern* field. If the name already exists, decide whether to replace the current set or not.

Remove

Choose the *Remove* button to remove the current option set.

Load Options

Choose the *Load Options* button to load a completely new set of option sets from any file.

Save Options

Choose the *Load Options* button to save the current option sets to a file. The default location is a file named `ppt_optionset.dat`. This file is saved in the current project directory.

C

CRefer Dialog Boxes

CRefer Dialog Box

Procedures

When you select the *Tools > CRefer* option in the Project Manager or open CRefer from the command-line prompt, the CRefer dialog box displays. This dialog box allows you to create cross references for a schematic or update the cross references for a schematic. You can also use this dialog box to delete the existing cross references.

The CRefer dialog box includes the following options:

Project File	Specifies the path to the project that you are running using the Project Manager or the Flow Manager or the project which you have specified using the `-proj` option from the command-line prompt. **Note:** *Project File* is a read-only field.
Design Name	Specifies the name of the top-level cell of your design. **Note:** The title bar of the CRefer dialog box displays the current project name.
Add or Update Schematic Cross References	Select this option button to create new cross references or to update the existing cross references in your schematic. By default, the *Add or Update Schematic Cross References* radio button is selected.
Remove All Cross References	Select this radio button if you want to remove cross references from your schematic.

Run	Select this button to execute the selected options in the CRefer dialog box. If you have made any changes in the Cross Referencer Options dialog box, the changed options are also stored in your project (.cpm) file. The next time when you load the CRefer dialog box, you will find all options that you saved in the last CRefer run.
Options	Select this button to display the Cross Referencer Options dialog box, which allows you to modify the default cross-referencing options.
Cancel	Select this button to return to the place from where you opened CRefer. Any changes you made to the cross reference options will be preserved.

Procedures

- Generating Cross-References for a Design
- Deleting Cross-References

Cross Referencer Options Dialog Box

Procedures

You can use the CRefer default options to cross reference a design. However, if you need to customize cross-referencing options, you need to change the settings in the Cross Referencer Options dialog box. To access the Cross Referencer Options dialog box, click on the *Options* button in the CRefer dialog box.

The Cross Referencer Options dialog box allows you to change the following types of settings:

Change the cref data file	If you are using non-standard page borders or custom off-page bodies then you need to define them in the cref data file. For more information about selecting the cref data file, see the Cross Referencer Options - Cref Data File Tab.
Change the default run and write options	You can define how CRefer writes the cross references on the schematic by changing the default run and write options. For more information about the default run and write options, see the Cross Referencer Options - Content Tab.
Change CRefer formatting options	You can specify whether or not to use the existing cross-references. You can specify that the input/output arrows should be ignored while cross referencing a design. You can define the text size of annotated CRefer properties and the space between each property. For more information about these options, see the Cross Referencer Options - Format Tab.
Change CRefer reporting options	You can define the types of reports to be generated as outputs. For example, you can generate Basenets, Netsbypage, Synonyms, and Crefparts reports. For more information about these options, see the Cross Referencer Options - Reports Tab.
Change the settings for hierarchical designs	You can define whether or not to create a separate flattened view for a hierarchical design. When you flatten the hierarchical design, CRefer creates a new flattened view for the cross referenced design. You can change the order of modules in the new flattened view. For more information about these options, see the Cross Referencer Options - Content Tab.

Procedures

- Changing the Cref Data File
- Configuring Run and Write Options
- Configuring Formatting Options
- Defining Output Reports

Cross Referencer Options - Cref Data File Tab

Procedures

The Cross Referencer Options – Cref Data File tab is the first tab in the Cross Referencer Options dialog box. It allows you to define the path to the cref data file, which defines the page border and offpage bodies. You can either use the standard cref data file or customize it.

The Cross Referencer Options - Cref Data File tab contains the *Cref Data File* field. By default, the <*your_install_dir*>/share/cdssetup/creferhdl/cref.dat file is used as the cref data file.

You can change the path to the cref.dat file by entering the new path in the *Cref Data File* field. Alternatively, you can use the browse button to browse to the file.

Important

The default cref data file is seeded by the following logic:

a. CRefer looks for the file defined by the entry for the directive CREF_DATA_FILE in the project file.

b. If the cref data file is not found in step 1, CRefer UI uses the CSF search to find the cref data file.

Note: After you have run CRefer, the path to the cref data file is saved in the project file.

Procedures

Changing the Cref Data File

Cross Referencer Options - Content Tab

Procedures

The Cross Referencer Options - Content tab is the second tab in the Cross Referencer Options dialog box. It allows you to configure the default run and write options. You can specify the signals that CRefer should ignore while running. You can also define how CRefer should write the cross references in the project (`.cpm`) file.

The Cross Referencer Options - Content tab is organized in two group boxes, *Run Options* and *Write Options*.

Generate Flattened Schematic	Select this check box to create a new flattened view (`schcref_1`) view in the top-level cell for the current project for the cross referenced design.
Generate Cross References for all nets	The cross references generated after selecting this option will contain data considering nets from all the levels of the hierarchy. You can therefore view and navigate to the nets from all the levels of the Hierarchical design.
For primitive connected nets only	Select this check box to generate cross references for only those nets that are connected to primitives. This check box is enabled when you select *Generate Cross References for all nets*.
Ignore Input Only Signals	Select this check box to specify that the signals that are input only are not cross referenced. Only the signals used as outputs elsewhere in the design will be cross-referenced. These signals will also be ignored in the schematic reports. If you have a signal that is used both as input and output, CRefer does not cross reference any input signals but cross references the output or bi-directional signals. For example, consider the scenario where you have a signal at 4 places such as `1A1<`, `2B1<`, `3C1<>`, and `4D3<>`. If you select the *Ignore Input Only Signals* check box, CRefer will not cross reference the `1A1<` and `2B1<` signals, but it will cross reference the `3C1<>` and `4D3<>` signals.

Cref Signals Not Connected to Flagbodies

By default, CRefer ignores signals that are not connected to flag bodies (offpage or port). If you want CRefer to read and process such signals, select the *Cref Signals Not Connected to Flagbodies* check box. CRefer will attach cross-references to the SIG_NAME property of signals.

For example, you have a signal A<2..0> synonym-ed to another signal B<2..0> at the top level. The A<2..0> signal is connected to a flag body while signal B<2..0> is not. If you do not select the *Cref Signals Not Connected to Flagbodies* check box, neither of the two signals will be cross referenced.

CRefer is optimized to generate cross references for signals with input/output type specified as a flag body. It is strongly recommended that you use flag or port bodies for cross- referencing purposes and not use the Cref Signals Not Connected to Flagbodies check box. Using flag or port bodies will help you avoid placement problems. You should define all flag bodies in the Cref data file.

To cross-reference a design with replicated or read-only blocks, you must place flag or port bodies on all signals that need to be cross-referenced.

If you do not add flag bodies to replicated or read-only blocks and check the *Cref Signals Not Connected to Flagbodies* box, CRefer will add empty placeholders, such as `$XR0=?`, to these signals. CRefer will also not annotate cross references in such cases.

Show Warnings for Unique Signals

Select this check box to show warnings for unique signal names. Unique signals occur only once in the design. By default, CRefer suppresses all warnings.

Skip Schematic Annotations

This option skips the placement of cross-references on a schematic. If cross-references are already present, they are not removed. To remove them, you need to run CRefer with the *Remove All Cross References* option selected in the main CRefer dialog box. This option overrides the *Generate Flattened Schematic* option.

Distinguish Between Ports and Offpages

Select this check box to specify whether you want CRefer to distinguish between ports and offpage symbols when placing cross references.

By default, the *Distinguish Between Ports and Offpages* check box is set to *Off*. In the default selection, CRefer will place both flat and hierarchical cross references in ports and offpage symbols. However, if you select the *Distinguish Between Ports and Offpages* check box, CRefer places hierarchical cross references on ports and flat cross references on offpage symbols.

CRefer uses the `HDL_PORT` property to distinguish between ports and offpage symbols. CRefer places hierarchical cross references ONLY on ports that have the `HDL_PORT` property on a pin. CRefer places flat cross references ONLY on offpage symbols that do not have the `HDL_PORT` property on the pin.

Retain Duplicate Entries

Select this check box to retain the duplicate entries in the signal and part cross references. If the *Retain Duplicate Entries* check box is selected, CRefer produces cross references such as:

```
5A1>2A2<2A2<2A2<
```

Notice that the signal `2A2` is being repeated three times. If the *Retain Duplicate Entries* check box was not selected, CRefer would have recorded only one entry for the signal `2A2`.

Note: By default, CRefer omits the duplicates if there are two or more cross references within the same grid square on a page.

Omit Zone Information

Select this check box to cross reference by page number only and omit the zone (page grid) information.

Note: By default, CRefer includes information about the page border zones in the cross references. For example, when the *Omit Zone Information* check box is not selected a signal may have the following cross reference `1C7^`. However, when the *Omit Zone Information* check box is selected, CRefer will record the cross reference as `1^`.

Sort by Page Number Only	Select this check box to sort the signal cross references only by page number, and not by the input/output type. **Note:** By default, CRefer puts entries in the following order: ❑ Output signals ❑ Input signals ❑ Bidirectional signals If any entries remain, they are sorted by the page number.
Show Signal Names in Hierarchical Cross References	Select this check box to display signal names in hierarchical cross references. For example, consider the scenario where you have the signal `CLK` in `TOP` connected to the pin `A` in `MID`, and then you cross reference the design. Now if the Show Signal Names in Hierarchical Cross References check box is selected then the cross reference will be `CLK@1C7^`. If the check box is not selected, the cross reference will be `1C7^`.
Show Block Names in Hierarchical Cross References	Select this check box to write the name of the block from which the signal originates in the cross reference information for hierarchical designs. The cross reference will appear as: `[Signal_Name@][_Block_Name_@]Page#Ygrid Xgrid[Type]`
Make Page Title Invisible	Select this check box to make the XR page title invisible. By default, the XR page title is displayed above the page border.

Procedures

Configuring Run and Write Options

Cross Referencer Options - Format Tab

Procedures

The Cross Referencer Options – Format tab allows you to configure CRefer formatting options. You can use this tab to specify whether or not the locations of the existing crefs will be reused. You can also specify to ignore input, output, and hierarchical arrows. Further, you can define the text size of annotated CRefer properties and the space between each property.

The Cross Referencer Options - Format tab is organized in three group boxes, *Write Options*, *Formatting Options*, and *Text Spacing Options*.

Write Options

The Write Options group box allows you to make soft properties hard and if some properties conflict then make them invisible.

Add Crefs as Hard Properties

By default, while cross referencing a design, CRefer adds all cref (`XR`) properties as soft properties (`$XR`). Select the *Add Crefs as Hard Properties* check box to add cref properties (`XR`) as hard (`XR`).

The use of the *Add Cref as Hard properties* check box is a rare case and is not recommended. However, if you have a specific requirement for creating hard `XR` properties (such as using hard properties in custom scripts to select or filter results from the `.csa` file), you can select the *Add Crefs as Hard Properties* check box to generate crefs as hard properties.

Note: Hard and soft `XR` property values are treated identically by CRefer. When you delete cross references, all hard and soft `XR` properties are deleted. When you clear the *Redo Placement of Crefs* check box, all hard and soft `XR` properties remain intact, that is, these properties are “not moved” during successive runs of CRefer.

Make Overlapping Cref Properties Visible

By default, CRefer tries to place the cross references near the signals. However, there may be cases where due to lack of space cross references may overlap with other properties and cause clutter. In such cases, CRefer will annotate properties but make them invisible. Select the *Make Overlapping Cref Properties Visible* check box to ensure that you can see all Cref properties on the schematic.

Formatting Options

The *Formatting Options* group box allows you to control the writing of the characters. You can also control whether or not the cross references should be recreated every time you run CRefer.

Redo Placement of Crefs

Select this check box to override the previous cross reference placement in a design. This information is available if the design has been cross referenced in the past. You should select this check box if you have changed any of the formatting or text spacing options since the last time you ran CRefer. However, if you want to retain the existing cross references, clear the selection from the *Redo Placement of Crefs* check box.

By default, CRefer reuses the existing cross reference information. This ensures that all soft and hard properties generated during the previous run of CRefer are not moved.

Note: The *Redo Placement of Crefs* option only retains the location of Cref properties. The values of Cref properties are substituted each time you cross reference the design.

Omit Input/ Output Arrows

Select this check box to disable the writing of the signal I/O type in the cross references.

Omit Hierarchical Arrows

Select this check box if you want to omit the hierarchical arrows. By default, CRefer displays the hierarchical arrows (`^` or `v`).

By default, CRefer writes the characters indicating the I/O type of the signals.

Omit Xrefs Down Hierarchy

Select this check box if you want to omit cross-references down the hierarchy.

Text Spacing Options

This group box allows you to make the cross references more readable by specifying the display options such as text size and spacing.

Scale Text

You use this field to increase or decrease the text size of cross references. Type a number to scale the text size in relation to the default display.

Example - To scale the text size to half of the original size, enter `0.5` in the *Scale Text* field. To scale the text size to twice the original size, enter `2` in the *Scale Text* field. This setting (`2`) can be used when the schematic is densely packed.

When you specify the scale text, keep in mind the smallest pages in the design.

Note: Whenever you change the cross reference text size from a previous value, use the `–r` option to redo all cross reference placements of cross references. This will ensure that the cross references are placed according to the new size.

Text Spacing

You use this field to set the space between the cross reference text notes in the schematic. CRefer sets the text space using Design Entry HDL coordinates.

Flag Body and Text Spacing

You use this field to set the space between the flag or port body and the cross reference text. CRefer sets the flag or port body and text space using Design Entry HDL coordinates.

Note: Take precaution in changing the text spacing or the flag body and text spacing. By default, CRefer places optimal space between cross reference text notes. You may not need to set the space. However, if you need to set the text spacing, keep in mind the following parameters:

- To increase the space between the cross reference text notes in the schematic, use a positive integer. To decrease the space between the cross reference text notes in the schematic, use a negative integer.
- The number that you enter for text spacing is in Design Entry HDL coordinates. A very small increase in this number such as `1` or `2` points will not show any effect. The number that you must enter to increase text spacing should be approximately the same size as the minimum grid size.
- CRefer first tries to place the cross reference text at the point that you have specified using the *Text Spacing* and *Flag Body and Text Spacing* fields. However, if it does not find available space to place the cross references, it uses its own algorithm to place those cross references.

Procedures

Configuring Formatting Options

Cross Referencer Options - Reports Tab

Procedures

The Cross Referencer Options - Reports tab allows you to define the types of reports to be generated as outputs. CRefer generates two types of reports: text and schematic.

Text reports are of five types: NetsByPage, BaseNet, Synonym, CrefParts, and PinXrefs. You can generate any of these reports by selecting the corresponding radio buttons in the Cross Referencer Options - Reports tab.

Schematic reports are of two types: signal and part cross reference reports. By default, these reports are added to the end of the schematic. However, you may store these reports in a separate view by selecting the *Create Separate View for Schematic Reports* radio button.

The Cross Referencer Options - Reports tab includes two group boxes for cref text reports and schematic reports. They are described below:

Cref Text Reports	This group box allows you to generate cref text reports.
NetsByPage Report	Select this check box to generate nets by page reports. This report contains the following columns: `Page`, `Net Name`, `Scope`, `Base Net`, and `Location`. If you append this to the schematic, the maximum numbers of characters these columns can have are 7, 15, 20, 15, and 21, respectively.
BaseNet Report	Select this check box to generate the basenet reports. This report contains the following columns: `Base Signal`, `Synonyms`, and `Location`. If you append this to the schematic, the maximum numbers of characters these columns can have are 20, 35, and 25, respectively.
Synonym Report	Select this check box to create a report that traces those nets whose names change across the hierarchical design. This report contains the following columns: `Page.Scope`, `Net Name`, `Synonyms`, and `Location`. If you append this to the schematic, the maximum numbers of characters these columns can have are 15, 20, 25, and 21, respectively.

CrefParts Report Select this check box to generate the crefparts report. This report contains the following columns: `Reference Designator`, `Physical Part Name`, and `Location`. If you append this to the schematic, the maximum numbers of characters these columns can have are 10, 20, and 50, respectively.

PinXrefs Report Select this check box to generate a pin cross-references report. This report contains the following columns: `Part`, `Bodyname`, `Pin`, `Zone`, and `Net Name`. If you append this to the schematic, the maximum numbers of characters these columns can have are 10, 20, 5, 10, and 35, respectively.

Schematic Report Select this check box to append a selected report to the schematic.

Schematic Reports This group box is enabled when you select a *Schematic Report* check box for any kind of report. It helps you create a separate view for schematic reports.

Add at the end of Root Schematic Select this radio button to add signal and part cross reference reports at the end of the root schematic. This radio button is selected by default, signifying that CRefer appends all Cref reports at the end of the root schematic.

Note: While adding extra pages to the schematic, CRefer uses the default page border specified in the project file. If the default page border is not specified in Design Entry HDL, CRefer searches for the page border in the `cref.dat` file. If information is not available about any custom page borders, CRefer uses the page border from the last page of the schematic to create new pages.

Note: This option is grayed out for hierarchical designs.

Add as a Separate View Select this radio button to create a separate view for the schematic reports. The crefout view is created. This view contains two reports, one each for signal and part cross reference.

Note: If you select the *Create Separate View for Schematic Reports* radio button, the reports added by CRefer at the end of the schematic during any previous run are automatically deleted.

Add as a Separate Cell Select this to create a separate cell structure, into the `sch_1` view of which the CRefer report pages are added. The remaining fields appear grayed until this option button is selected. The `generate_separate_cell` directive in the `cpm` file corresponds to this option.

Note: This is recommended for hierarchical designs.

Place at the End of Root Schematic	Select this to specify that CRefer should append a page to the root schematic and place the new symbol on it.
Cell Name	Specify a name by which the report page cell will be added to the schematic. The default name is `CrefRpt`.
Symbol Information	Use the browse button to open the View Open dialog box to get information about the library, the cell, and the version of the symbol that you want to use.

Note: The *PinXrefs Report* and *Add as a Separate Cell* options take considerably more time than the other options do.

Procedures

Defining Output Reports

CRefer Progress Window

The CRefer Progress window displays the status of the cross-referencing task. The fields in this box are described below:

Details

Click this button to view detailed system messages. CRefer expands the window at the bottom and displays the system messages. You can click on this button again to close the window.

Note: The `creferhdl.lst` file is created in the `temp` directory of the root design. This file records the status of the CRefer run. If CRefer quits on encountering errors, check the `creferhdl.lst` file in the directory specified by the `TEMP` directive in the project file to find details about the errors. The `creferhdl_pinxref.lst file` is also created when the PinXrefs Report is generated. To read the complete log of the CRefer run, see the `cref.log` file in the `temp` directory of the root design.

Cancel

Click this button to cancel the task in progress and exit.

Note: The cross references inserted before you chose to cancel the cross-referencing process will remain in the design.

D

IFF2HDL Dialog Boxes

IFF Import

IFF (Intermediate File Format) is used to transfer a design in machine and application independent format between Electrical Engineering design and Printed Circuit Board (PCB) design environments.

Use this dialog box to import a schematic in IFF format into Design Entry HDL.

Index

Symbols

A

B

C

D

E

F

H

I

L

M

N

O

P

R

S

T

U

V

X