cadence®

Design Variance User Guide

Product Version 23.1
September 2023

Last Updated: September 2019

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

in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

Preface

About This Guide

The *Design Variance User Guide* demonstrates the major features of the Design Variance solution, which is part of the front-to-back flow for PCB design. The Design Variance solution lets you create and manage variants. Variants are designs that are different from each other by small differences. For example, consider a design, `USA,` that has18 resistors, four capacitors, and two shunt terminators, and a design, `India`, that has the same configuration as the `USA` design with only a minor difference in the resistance value of one resistor. You can easily create such designs using the Design Variance solution. By the time you complete Chapter 3, "Creating the Variant Database,", you will find that you can create and manage much more complex variations in designs using the Design Variance solution.

In the *Design Variance User Guide*, you will learn about the Variant Editor tool, which is used to create and manage variants. You will also learn about associated features in Design Entry HDL and PCB Editor for managing variants.

Note: To learn about the powerful Bill of Materials (BOM) tool named BOM-HDL, which creates BOM reports and offers extensive customizing, see Generating BOM Reports.

How to Use This Guide

This guide covers both concepts and skills required to use the Design Variance solution. The organization of the guide is based on the logical flow for creating and managing variants. The first chapter introduces the Design Variance solution and each subsequent chapter explains tasks performed using Variant Editor.

The procedures explained in the guide are generic, that is, the procedures are design independent. For most procedures, appropriate examples are also included for further explanation of concepts.

Note: If you are a new user and want to quickly apply your skills and create design variants, see Design Variance Tutorial in `CDSDoc`. This tutorial uses a single-example design for which you create variants and perform all type of customizing available in the Design variance solution.

As opposed to the *Design Variance Tutorial*, the purpose of this guide is to include all procedures (and not the more important ones as covered in the *Design Variance Tutorial*) and serve as a source of reference information. Each procedure in this guide includes detailed conceptual information that lets you understand the need for a feature and essential prerequisites for completing a task.

If you are a new user and have never used the Design Variance solution, it is recommended that you first complete the *Design Variance Tutorial*. However, if you have worked with the Design Variance solution, you can use this guide to find detailed information for specific topics of interest.

The *Design Variance User Guide* assumes that you are familiar with the basic tools in the front-to-back flow for PCB design:

- Project Manager
- Design Entry HDL
- PCB Editor

Brief Outline of Different Chapters

This guide is organized into seven chapters and four appendices. The first chapter introduces the conventional procedure for creating variants and the new procedure for creating variants using the Design Variance solution.

In Chapter 1, “Introduction to the Design Variance Solution”, you will learn about the need for the Design Variance solution. You will understand how tedious the conventional procedure for creating design variants is and how the Design Variance solution simplifies the procedure of creating design variants. You will also learn about Variant Editor and its features.

In Chapter 2, “Getting Started with Variant Editor”, you will learn how Variant Editor fits in the front-to-back flow. You will learn to start Variant Editor, and customize the display of panes and properties.

In Chapter 3, “Creating the Variant Database”,you will learn to create the variant database, which contains information about any customization changes in the base schematic and in variants. You will create alternates, alternate groups, functions, and variants. You will also learn to search for components and synchronize the variant database with the schematic.

In Chapter 4, “Annotating Variant Information”, you will learn to backannotate the changes in the base schematic or variants to the Design Entry HDL schematic.

In Chapter 5, “Managing Variant Information in PCB Editor”, you will learn to create BOM reports for variants in PCB Editor. You will also learn to create assembly drawing layers for variants in PCB Editor.

In Chapter 6, “Variant Editor Command Reference”, you will learn the functions of the menu commands and system commands supported by Variant Editor. You will learn how to effectively use these commands.

In Chapter 7, “Variant Editor Dialog Boxes”, you will find a description of the options in the Variant Editor dialog boxes. This is the same description that you get when you choose *Help* in any dialog box.

In Appendix A, “Variant Editor Glossary”, you will find the definitions of the most common terms associated with Variant Editor.

In Appendix B, “Reference Information”, you will find miscellaneous information about the Design Variance solution.

In Appendix C, “Variant Editor FAQs”, you will find answers to frequently asked questions about Variant Editor.

In Appendix D, “BOM-HDL FAQ”, you will find answers to frequently asked questions about BOM-HDL.

Typographic and Syntax Conventions

The following list describes the syntax conventions used for tools used in the Design Variance solution. Where applicable, exceptions to these conventions are clearly indicated.

`literal (LITERAL)`	Non-italic or (UPPERCASE) words show key words that you must enter literally. These keywords represent command (function, routine) or option names.
argument	Words in italics show user-defined arguments for which you must substitute a value.
|	Vertical bars (OR-bars) separate possible choices for a single argument. They take precedence over any other character. For example, `command` *argument* | *argument*
[]	Brackets denote optional arguments. When used with OR-bars, they enclose a list of choices. You can choose one argument from the list.

{ }	Braces are used with OR-bars and enclose a list of choices. You must choose one argument from the list.
...	Three dots (...) show that you can repeat the previous argument. If they are used with brackets, you can specify zero or more arguments. If they are used without brackets, you must specify at least one argument, but you can specify more. *argument*...: You should specify at least one argument, but more are possible [*argument*]...: You can specify zero or more arguments
,...	A comma and three dots together show that if you specify more than one argument, you must separate those arguments by commas.
Courier font	Indicates command line examples.

1

Introduction to the Design Variance Solution

Overview

Designers often need designs that differ from each other in minor ways. For example, designs may need to be customized for targeted market segments, destination countries, or feature sets. These differences are often small and are made of incremental changes on the base design. For example, you might need two designs, `USA` and `India`, that have only one difference, which is in the resistance value of one resistor. However, creating these differences is often difficult. You can end by repeating the whole design cycle for each design.

The Design Variance solution lets you create and manage designs that are different from each other by minor differences (these designs are also called variants). Using the Design Variance solution is simple. All you have to do is to create the base design in Design Entry HDL and then define the variant component in the Cadence tool called Variant Editor. In the example of the `USA` and `India` designs, first create the base design, say `USA,` and then define the changed resistor in the `India` design.

Variant Editor supports an intuitive user interface (UI). Without resorting to complex editing of text files, you can define variant components, generate Bill of Materials (BOM) reports, annotate special designators to any components, annotate variant data, and merge variant databases using Variant Editor.

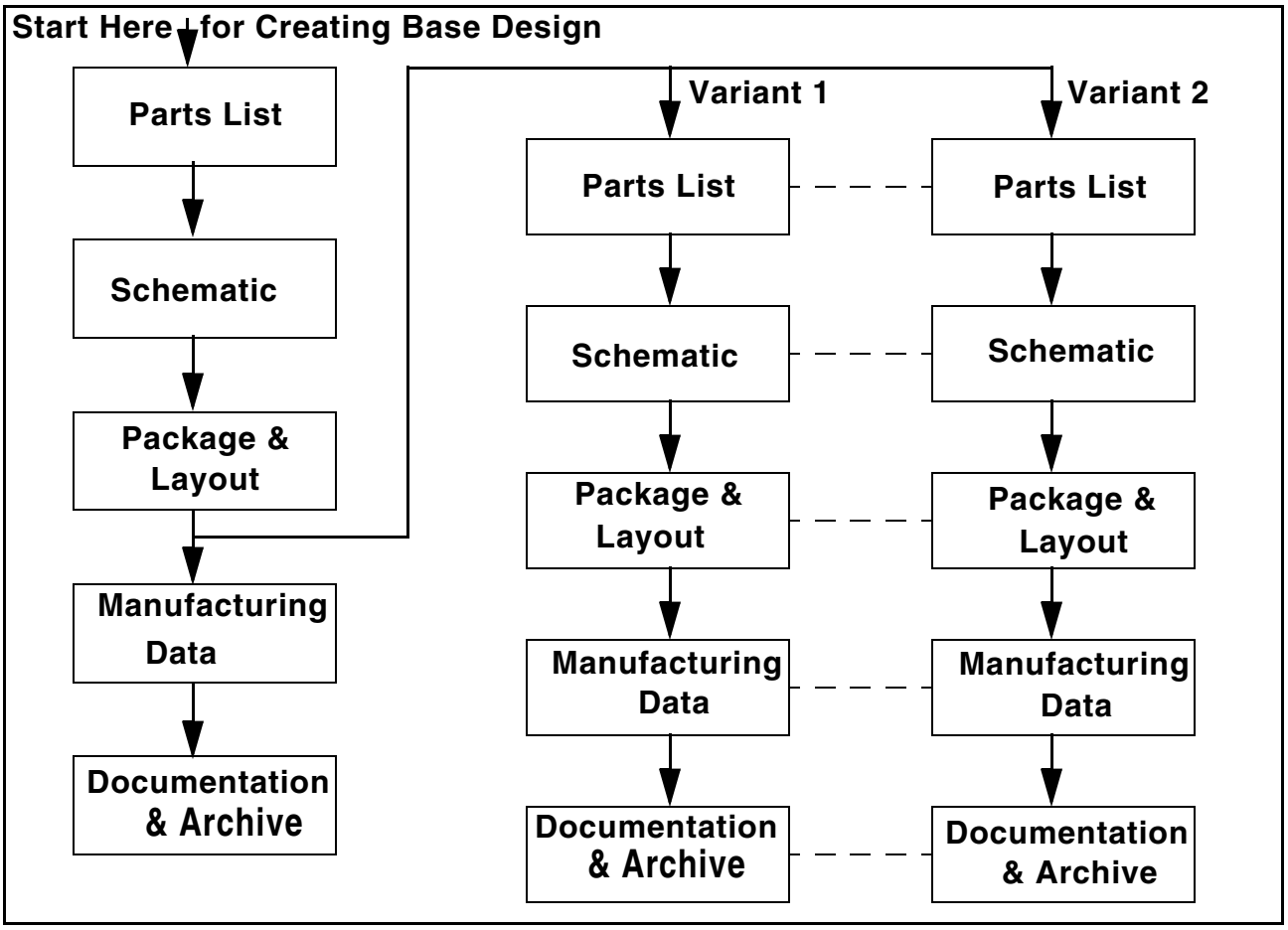
In summary, the Design Variance solution offers the following advantages:

- Reduces the maintenance effort of the design set
- Decreases the development time thus enabling faster time to the market
- Reduces the probability of errors
- Reduces cost through consistent part selection

Conventional Procedure for Variant Design

The conventional variant design procedure is graphically represented, as follows:

Figure 1-1 Conventional Variant Design Procedure



The conventional procedure for creating variations in a design includes the following steps:

- Define functions and parts lists for all the potential product variants.
- Create a schematic that is the union of all foreseen parts and connections, which can be required for any given variant.
- Generate a packaged netlist and parts list for the base design.
- Lay out the base design.
- Generate the board manufacturing data for the base design.

- Generate documentation and archive the base design.
- For each design variation, copy the base design database.
- Make appropriate parts lists and schematic changes for each variant. (This procedure is manual and is prone to errors.)
- Generate the assembly data and a BOM for each variant.

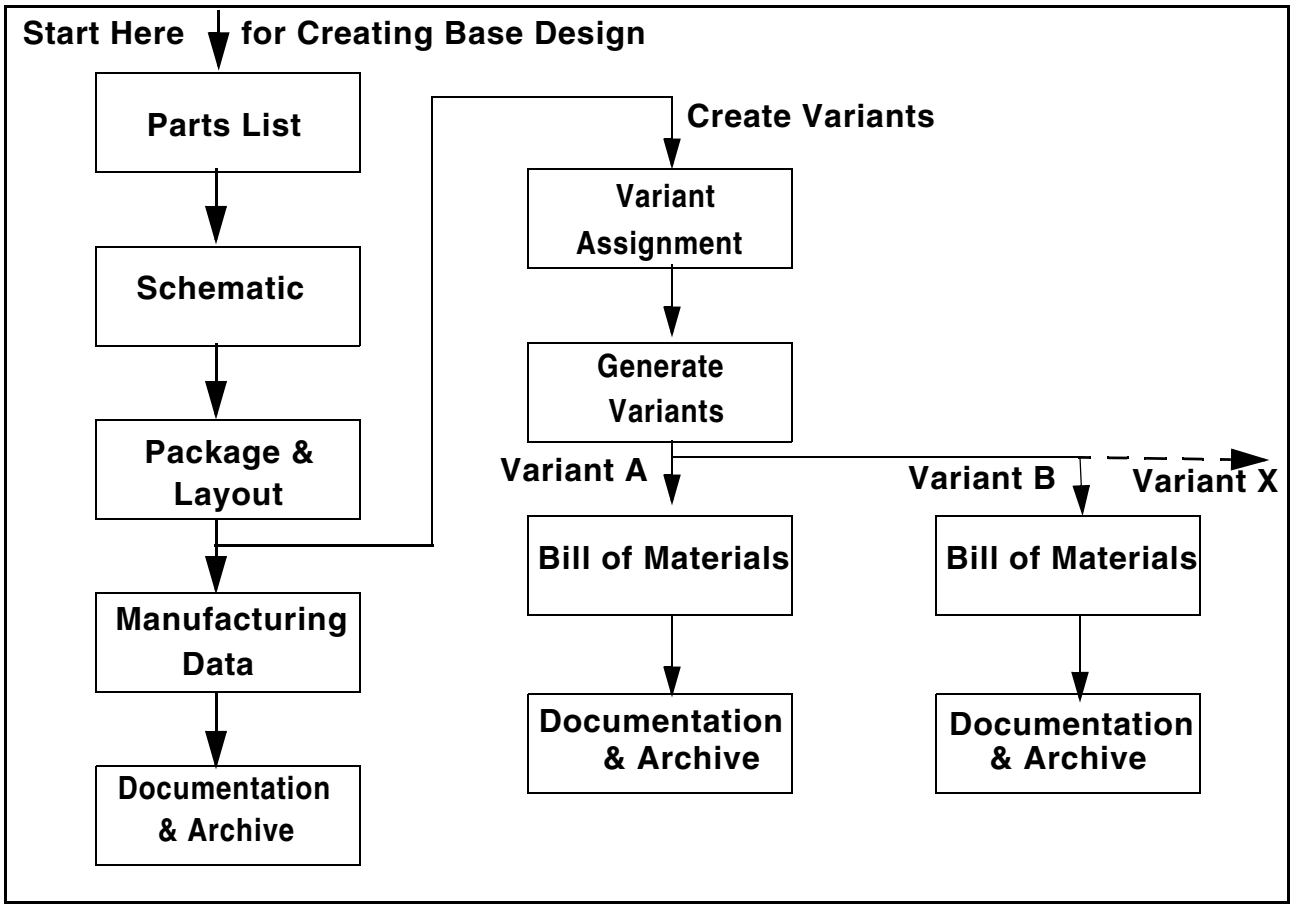
The conventional procedure for variant design has the following limitations:

- There is a significant amount of redundancy, which makes work prone to errors.
- A copy of each variant needs to be maintained, which wastes disk space.
- Future changes are tedious because all variants require individual modification. These changes can cause errors.

Variant Design Using Variant Editor

The new method for variant design is illustrated in the following graphic:

Figure 1-2 Variant Design Using Variant Editor



Whether you create variants using Variant Editor or without using it (that is, using the conventional procedure), the first three steps are common. These steps use Design Entry HDL to make the base schematic, package it, and generate a layout. However, at this point is where the two procedures start to differ. In the conventional procedure for creating design variants, you copy the base design database for each variant, and then make appropriate parts lists and schematic changes for each variant. In the new procedure, Variant Editor is used to define all variations. Variant Editor outputs a file containing the variant information, which is read by PCB Editor to create the variant assembly drawing. Variant information can also be annotated to the schematic.

Detailed steps to create variants using Variant Editor are as follows:

1. Create the base schematic with Design Entry HDL.

   This schematic is the combination of all foreseen parts and connections that are required for any given variant.

2. Package the design.

   Package the design using Packager-XL. Variant Editor reads the Packager-XL output files and creates a spreadsheet view of the design. See Packager-XL Reference for more information about packaging a design.

3. Assign the variant data.

   Assign the variant data to the base design as follows:

   a. Assign alternate values - By default, the value of the component on the base schematic is considered the preferred value. You can change this value. Additionally, you can define up to 99 alternate values. See Adding Alternate Values on page 35 for more information about assigning alternate values to components.

   b. Define alternate groups - You need to define alternate groups when there are multiple, functionally equivalent, components in the base schematic (that have different footprints) out of which only one component is to be used for any variant. See Defining Alternate Groups on page 38 for more information.

   c. Create functions - Create a function if you need a set of components in the base schematic to be included in any variant as a group. When you create a function, you can either include all the components in the function in the variant or include none of them in the variant. Functions can contain alternate groups and components that have been assigned alternate values in step 1. See Creating Functions on page 39 for more information.

   d. Create the required variants - A variant can contain functions, alternate groups, components customized for that variant, and the components that are not customized for that variant. See Managing Variants on page 43 for more information.

4. Create the board layout.

   Generate the board by using PCB Editor. This board must contain the superset of all footprints.

5. Generate the variant assembly drawings.

   Generate the assembly drawings for different variants in PCB Editor. See Creating Variant Assembly Drawings on page 81 for more information.

Variant Editor Features

Variant Editor supports the following features:

- Uses Physical Part Table (PPT) driven data to define variant component values.
- Generates the Bill of Material (BOM) that reflects the electrical *stuff list* for a variant.
- Generates a delta list of components from the base design for a variant.
- Generates a comparative BOM of different variants.
- Generates BOM reports in multiple formats, such as the spreadsheet format and HTML format.
- Annotates variant data from the variant database into the schematic.
- Generates the interface file that is read by PCB Editor to create variant assembly drawings.
- Cross-probes with Design Entry HDL.
- Filters components in the Component List.
- Supports associated mechanical parts and callouts.
- Supports global search of specific components.
- Synchronizes the changes made in the variant database with the changes made in the original schematic.
- Replaces an existing component with another component that has a different name or a non-compatible footprint.

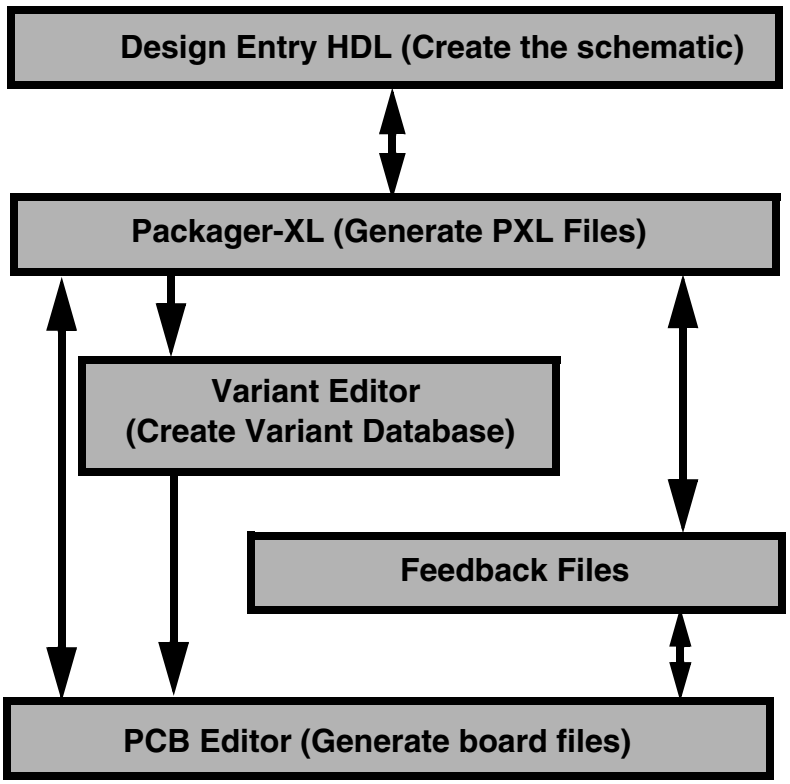
2

Getting Started with Variant Editor

How Variant Editor Fits in the Front-to-back Flow

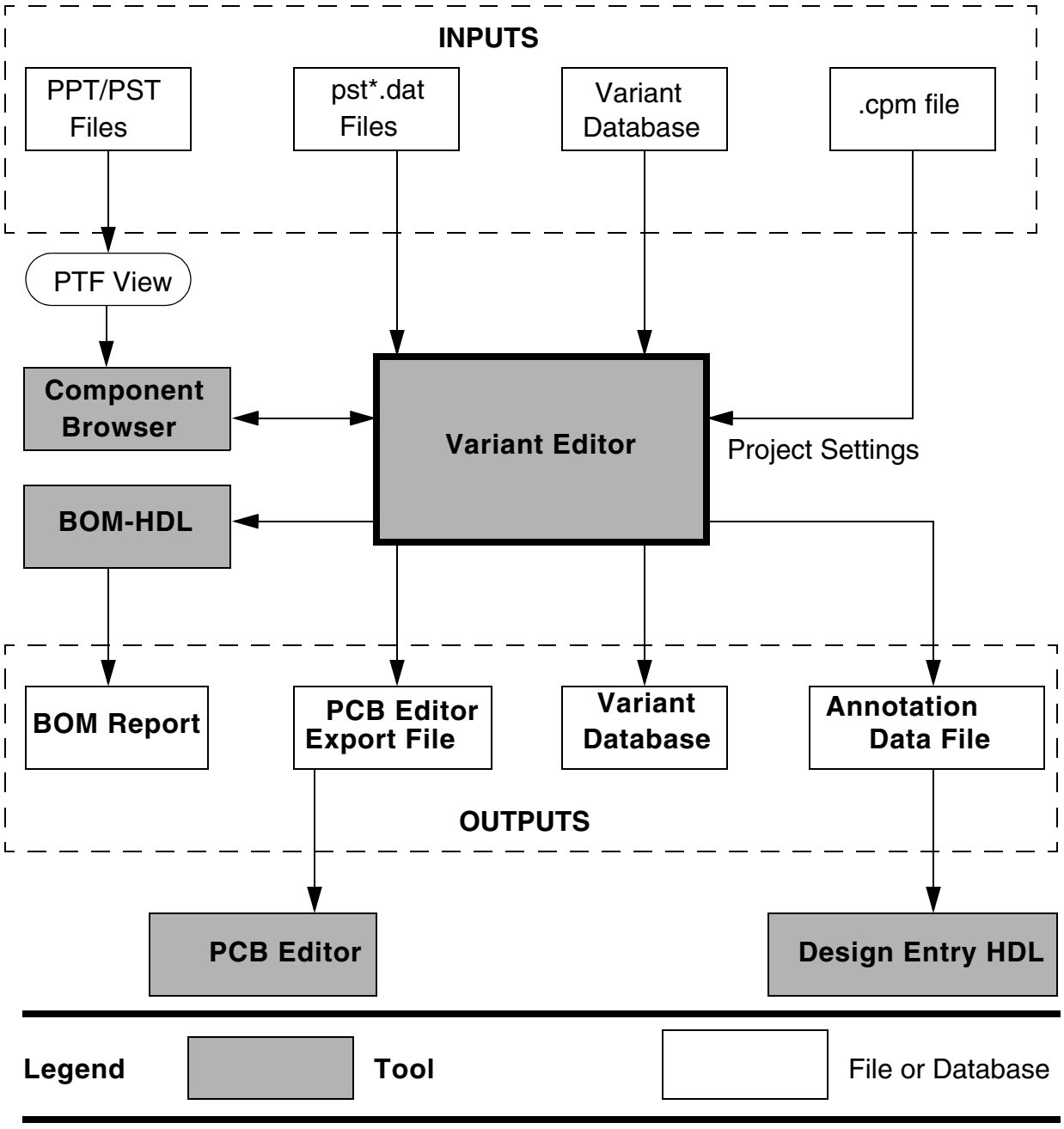
Variant Editor is a GUI-based application that allows you to create variants and generate variant BOMs for a given design. The following figure shows how Variant Editor fits in the front-to-back flow.

Figure 2-1 Front-to-back Flow Using Variant Editor: High-level Flow



The following figure expands all the inputs to Variant Editor and outputs from Variant Editor.

Figure 2-2 Front-to-back Flow Using Variant Editor: Low-level Flow



Variant Editor accepts the following inputs:

1. PPT/PST Files

These files are located in the part table files (`PTF`) view of the design for which the variants are being created. These files contain information about the different parts in the design. Variant Editor accesses these files through the Component Browser.

2. pst*.dat files (`pstxprt.dat` and `pstxref.dat`)

   The `pstxprt.dat` file lists each reference designator and the section assigned to it. This information is also available in a simplified format in the `pstxref.dat` file. The `pstxref.dat` file also contains the logical-to-physical bindings, the schematic instance, and the component information used by Packager-XL for each part in the design.

3. Variant database

   The variant database (`variant.dat`) stores the variant data. This database is an ASCII file, which contains a list of variant value entries for all variant components in a design. Since the schematic can also contain variant data, it is important to sync variant data between the variant database and the schematic through the packaged files.

4. The project (*.cpm*) file

   The project file contains the project settings. The project settings include information about the name of the last variant file used, details about the state of different properties, and the default values used in different dialog boxes. The information in the project file helps Variant Editor to maintain persistence with the last used state.

Variant Editor uses the inputs described in the preceding section to produce one or all of the following four outputs:

1. BOM Report

   Variant Editor can provide detailed BOM reports using the BOM tool, that is, BOM-HDL. You can generate BOM reports for the base schematic or for any variant. You can also generate a variant comparison report, which contains part number-based comparison details between the components of the base schematic and all the variants.

2. PCB Editor Export File (*variants.lst*)

   Variant Editor generates a `variants.lst` file, which PCB Editor uses to extract information about all the variants stored in the physical view of the top-level design.

3. Variant database

   The changes made to the different variants in a Variant Editor run are again saved in the variant database. This ensures that the changes are available for future runs.

4. Annotation data file

   The annotation file contains all the information about the variant properties in the base schematic and in all variants. Design Entry HDL uses the information in this file to

annotate information on the schematic. You can control annotation by assigning different properties to components that have variant information. See Chapter 4, “Annotating Variant Information,” for more details about annotating variants.

Before You Start Using Variant Editor

Variant data can be defined only in the post-packaging phase. Therefore, before you start using Variant Editor, ensure that you have the generic schematic constructed in Design Entry HDL and packaged using Packager-XL.

Starting Variant Editor

You can start Variant Editor in one of the following ways:

- From the *Tools* menu in the Project Manager, choose *Variant Editor*.
- Specify the `vedit -proj` <*`project_name`*> command at the command prompt.

  If <*`project_name`*> does not exist in the current directory, Variant Editor will open the File browser, which allows you to browse to the project file. If <*`project_name`*> does exist in the current directory, Variant Editor opens the project.

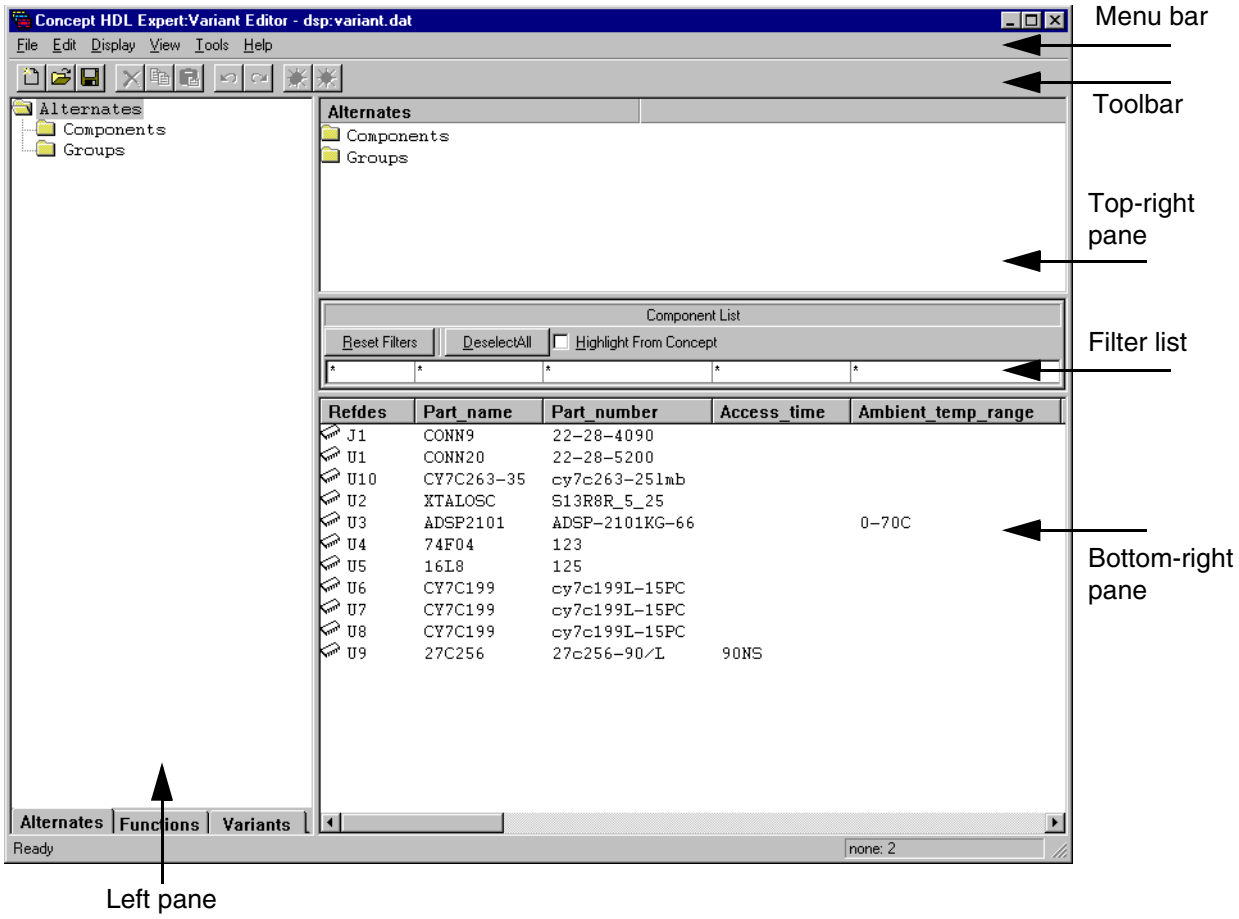
The Variant Editor main window appears. See Variant Editor Main Window figure on page 25.

To ensure that multiple users cannot open Variant Editor at the same time, a `.lck` file is created when you open Variant Editor. The file is saved in the same directory that stores the `variant.dat` file. When you exit Variant Editor, the `.lck` file is automatically deleted.

Variant Editor User Interface

The Variant Editor main window displays the current design in a three-pane view as depicted in the following figure:

Figure 2-3 Variant Editor Main Window



The Variant Editor main window consists of the following parts:

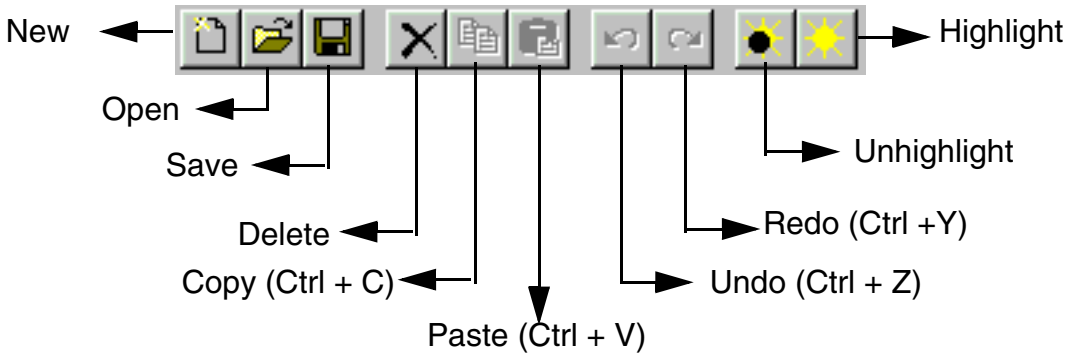
Menu Bar

The menu bar includes six menus: *File*, *Edit*, *Display*, *View*, *Tools*, and *Help*. See Main Menu Commands on page 85 for details about the menu commands.

Toolbar

The Variant Editor toolbar includes eleven toolbuttons that allow you to quickly run the most commonly-used commands. The Variant Editor Toolbar figure on page 26 describes the functions of all toolbuttons.

Figure 2-4 Variant Editor Toolbar



You need not remember the associated command with each toolbutton. If you move the mouse over any toolbutton, a tooltip displays the name of the function. To run the command associated with any toolbutton, click on the button.

Note: All toolbuttons have associated menu commands, and you can use either the menu bar or the toolbar to run the command. Some commands can also be run from the keyboard. For example, you can use the `Ctrl + C` key combination to perform the `Copy` command.

Left pane

The left pane allows you to choose any of these tabs: *Alternates*, *Functions,* or *Variants*. Based on the tab you choose, Variant Editor shows the variant information for components, alternate groups, functions, or variants.

The left pane shows a tree view. Based on the object selected in the tree view, properties for that object appear in the top-right pane.

Top-right pane

The top-right pane displays the properties of the object selected in the left pane tree view.

Bottom-right pane

The bottom-right pane lists all the base components in a design. You can select these components to assign variant information. Note that all the components are displayed along with their `part_numbers`. These `part_numbers` correspond to the values chosen for the components in the base schematic.

Filter List

The Filter list is located between the top-right and bottom-right panes. This is a fixed-size frame that allows you to apply filters on the properties selected. See Setting and Resetting Filters for more information about applying filters.

Customizing Variant Editor User Interface

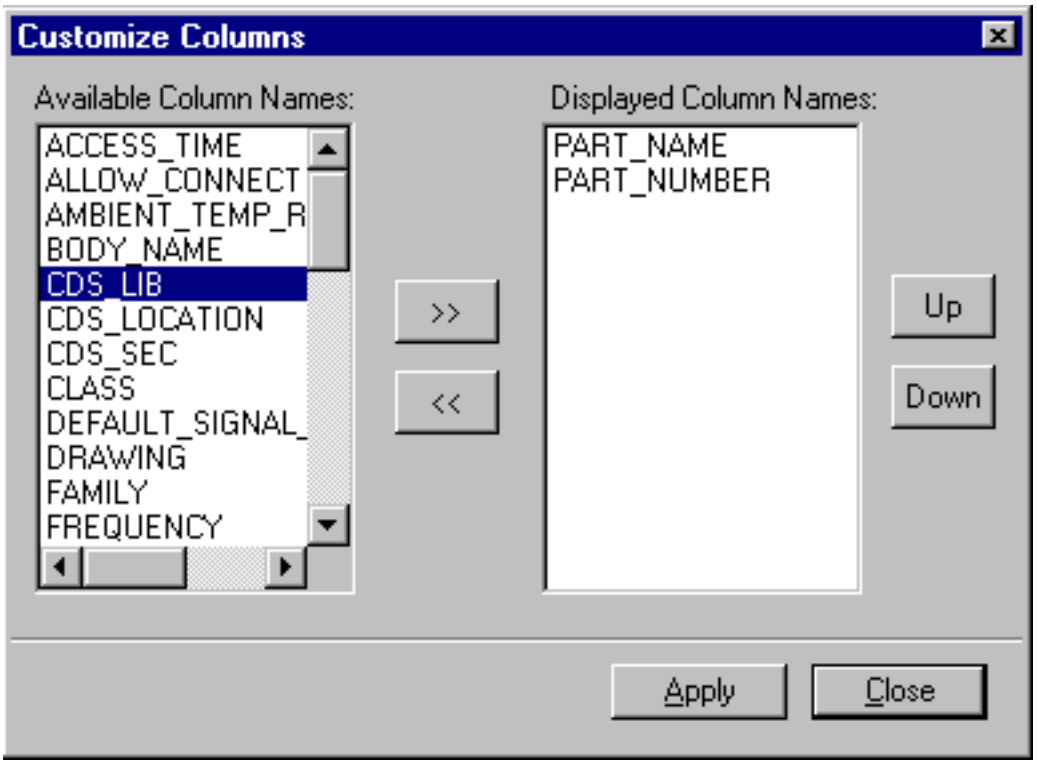
Setting the Properties to View

When you load a variant database, the bottom-right pane displays only the `part_name` and `part_number` properties. To see other properties, you can customize the display by manually editing the column header. To edit the column header, do the following:

1. From the *View* menu, choose *Customize Columns*.

   The Customize Columns dialog appears.

Figure 2-5 Customize Columns Dialog Box



To display a property in the right panes of the Variant Editor main window, you must choose the property from the *Available Column Names* list and place it in the *Displayed Column Names* list.

There are three ways in which you can choose a property:

1. Individual property selection

   You can choose the property by clicking on its name then clicking *>>* to move the property to the *Displayed Column Names* list.

2. Multiple properties selection (random selection)

   To select multiple properties, click on the first property. Hold the `Ctrl` key down and select as many properties as required then click the *>>* button. The selected properties are moved to the *Displayed Column Names* list.

3. Multiple properties selection (serial)

   To choose properties serially, click on the first property you want. Hold the `Shift` key down and choose the last property in the series. This selects all the properties from the first property to the last property. To transfer the selected properties to the *Displayed Column Names* list, click *>>*.

Note: You can move a property from the *Displayed Column Names* list back to the *Available Column Names* list by choosing the property and clicking on the *<<* button.

Note: After you have selected the properties in the Customize Columns dialog box, click the *Apply* button to make Variant Editor reflect the changes. When you click *Apply*, the selected properties are displayed in the right panes. The value for each property is displayed wherever the property is applicable for a component. If a property is not applicable for a component, its corresponding value is not displayed.

Note: VARIANT and FUNCTION are keywords in Variant Editor and are therefore not displayed as property columns. Do not use VARIANT/FUNCTION as property names to capture user properties. The VARIANT property is used for variant data annotation in schematic.

Setting and Resetting Filters

If you have a big design with hundreds of components, searching for specific components can be difficult. To simplify searching, you can use filters to display specific components in the right panes.

To apply a filter for any property, complete the following steps:

1. Choose the box (with the * sign) at the top of the desired property.

   A drop-down button appears.

2. Click on the drop-down button.

3. Choose the value of the property that you want to use for filtering.

   All components with the selected property value appear.

   Note: You can type the value of a property in the filter field for the property and Variant Editor will display all the components with the selected property value. For example, typing `CONN9` in the filter field for the `PART_NAME` column displays the component with the `PART_NAME` `CONN9`.

After filtering the components, you can reset the filters by doing the following:

Click the *Reset Filters* button.

Note: You can also press the `F5` key to reset all filters.

To broaden the scope of your filter to list all components, use * in the *Filter* field.

3

Creating the Variant Database

Overview

The variant database is a new view called variant, which contains files that store information about variants in a design. This database can be used to create BOM reports for each variant and generate the interface file containing the variant information that is read by PCB Editor to create the variant assembly drawing. You can later backannotate the variant information to the base schematic.

Using Variant Editor, you can modify the components in the base schematic for use in a variant. You can define variants in the following ways:

- Assign a Do Not Install (`DNI`) status to a component

  If you assign a `DNI` status to a component in a particular variant, the component is not installed in that variant.

- Change values for components

  This variation is one in which a component value changes from one variant to another. This affects the BOM but not the physical layout. The most common types of design variations are based on value change. See Changing the Preferred Value (Change Values) for details.

- Add alternates to a component

  An alternate value is a value that replaces the preferred value of the component. You can assign a maximum of two alternate values for a component. See Adding Alternate Values for details.

- Change the properties of a component

  You can change the user-defined properties of a component such as PACK_TYPE, PLATING_OPTION, and so on. You cannot change the key properties of a component. See Changing Component Properties for details.

- Define alternate groups

An alternate group is a set of functionally equivalent components that have different footprints. This set of components is called an alternate group. From this alternate group, only one component is included in a particular variant. When you create an alternate group, one of the components will automatically become preferred and the other component will become alternate 1.

The reason for having multiple components on the schematic is that each component has a different board footprint, therefore not allowing basic plug compatibility. The bare printed circuit board has placeholders for all components in the alternate groups. See Defining Alternate Groups for details.

- Create functions

  A function is a set of components that form a feature or a logical function. You cannot include specific components of functions in a variant. You can either include all the components of a function in a variant or exclude all the components of a function from a variant. See Creating Functions for details.

Changing the Preferred Value (Change Values)

1. Choose the row corresponding to the component (whose preferred value is to be changed) in the bottom-right pane and click on the *Add to Alternate Components List* option from the shortcut menu.

   By default, the component is assigned the preferred value.

   Note: You can simultaneously move multiple components from the bottom-right pane to the top-right pane.

2. Choose the component in the left pane so that the row corresponding to it is displayed in the top-right pane.

3. Choose the row corresponding to the component in the top-right pane, and click on the *Change Value* option from the shortcut menu.

   The Part Table Filter dialog box is displayed with the PPT rows that match the footprint of this component.

4. Choose the new preferred value and click *OK*.

   The Part Table Filter dialog box closes. The original row with the status as `Pref` is assigned the status `-', and a new row with the status `Pref` is added.

   Note: You can change the status of the preferred component in the Customize Variant Settings dialog box.

Defining Alternate Values

1. To define the alternate value for a component, choose the row corresponding to the component in the top-right pane and then click on the *Add Alternates* option from the shortcut menu.

   The Part Table Filter dialog box is displayed with all the relevant PPT rows for the selected component.

2. Choose the new preferred value and click *OK*. The Part Table Filter dialog box closes, and a new row with the status `Alt1` is displayed in the top-right pane.

   A new row for the selected component with the status `Alt1` is displayed in the top-right pane.

3. Repeat step 1 and step 2 to assign the second alternate value for the selected component.

   A new row for the `J1` component with the status `Alt2` is displayed in the top-right pane.

Note: You cannot create alternates for a component if the component with the same value already exists.

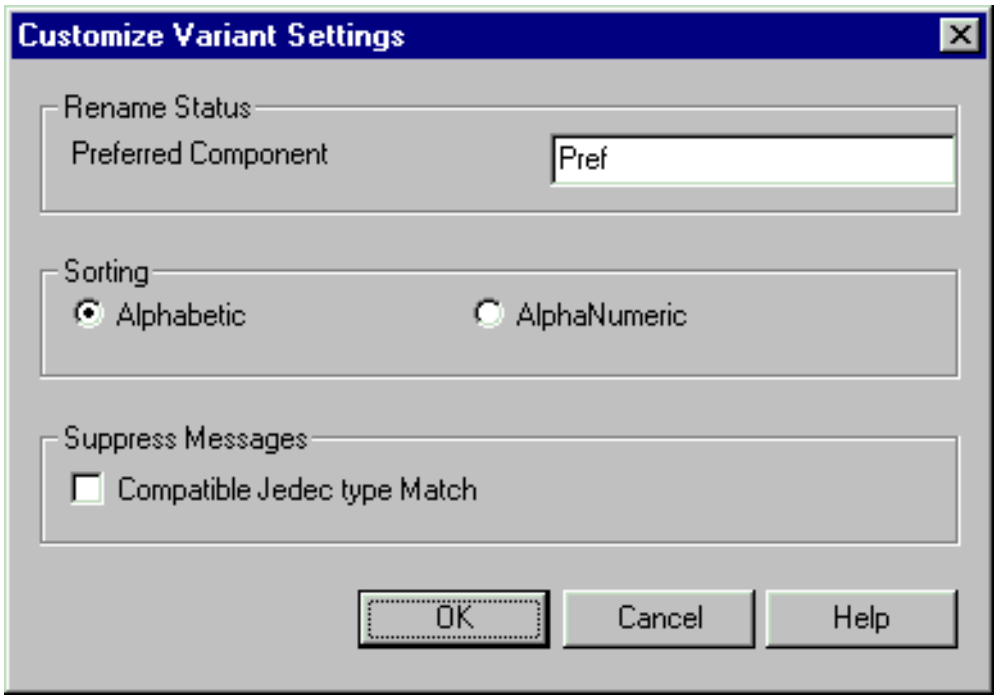
Note: You can define up to 99 alternates for a component.

Changing the Preferred Status Designator for Components

1. To change the status designator for the preferred component, choose *Options in the Tools* menu.

   The Customize Variant Settings dialog box appears.

Figure 3-1 Customize Variant Settings Dialog Box



2. Type the new status for the preferred component in the *Rename Status Preferred Component* field.

   Note: You can use only alphabets or numerals to define the new status designator.

3. Click on the *Ok* button to accept the new status designator.

The status of the preferred component will change to the new value.

Changing the Status of Alternates

➤ Choose the PPT row corresponding to the component (whose status is to be changed) in the top-right pane and click one of the following options:

   a. **Make Pref**—The status of the selected PPT row will change to `Pref` and the PPT row that had the status as `Pref` will show the status of the selected row.

   b. **Make First Alternate**—The status of the selected PPT row will change to `Alt1` and the PPT row that had the status as `Alt1` will show the status of the selected row.

c. **Make Second Alternate**—The status of the selected PPT row will change to `Alt2` and the PPT row that had the status as `Alt2` will show the status of the selected row.

d. **Make Alternate**—This will display the Make Alternate dialog box where you can choose any of the available alternate. The status of the selected PPT row will swap with the status you chose in the Make Alternate dialog box.

Adding Alternate Values

Overview

By default, the value of a component on the base schematic is the preferred value. However, you are not limited to using this value. You can change the preferred value and define up to 99 alternate values for the component. Any of these alternate values can be used to replace the preferred value of the component.

Note: When you define an alternate value for a component, you can select values from only those PPT rows that have the same footprint, that is, `JEDEC_TYPE`. If you want to use alternate values that do not have the same `JEDEC_TYPE`, see Using Compatible JEDEC_TYPEs on page 63.

To define an alternate value, choose the *Alternates* tab and then click on the *Components* folder.

Note: The information assigned in the *Alternates* tab is applicable to all variants unless some customizing is done for a particular variant.

Changing Component Properties

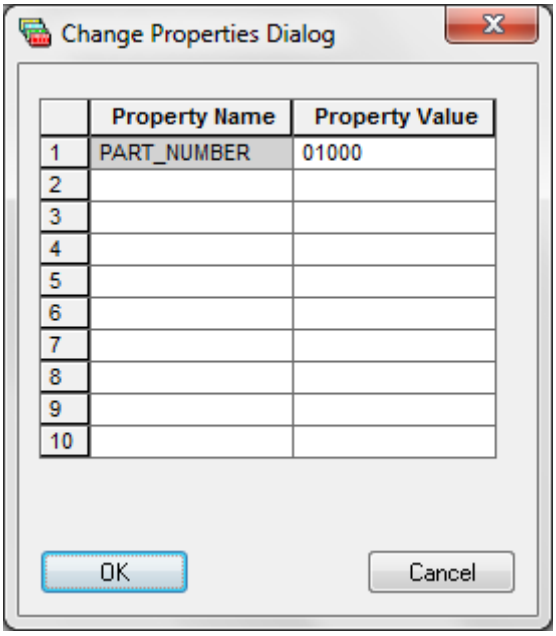
Overview

Variant-specific properties can be specified for instances that have been added to a variant. You can add or modify these user-defined properties of a component for a specific variant, such as PACK_TYPE, PLATING_OPTION, and so on. You cannot change the key properties of a component.

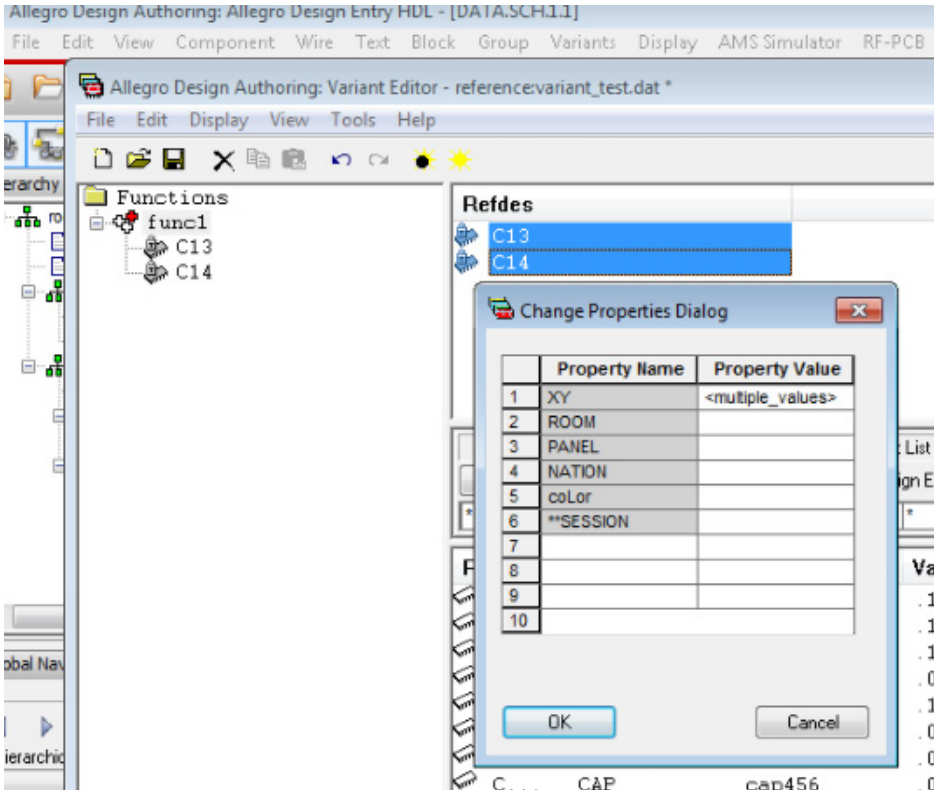
Changing Properties of Components

1. To change the properties of one or more components, select the components whose properties you want to modify in the Variants tab.

   The Change Properties dialog box appears and displays user-defined properties for the selected components. These properties that are configured as custom columns using *View — Customize Columns* in Variant Editor. See Customize Columns for details. Part table (PTF) properties are not displayed in this dialog box as they cannot be modified.



If the components you select have the same properties but with different values, Variant Editor will display the value as *<multiple_values>*.



Note: To change properties for multiple components at a time, the components must have the same logical part names and compatible JEDEC_TYPEs. If you want to use alternate values that do not have the same JEDEC_TYPE, see Using Compatible JEDEC_TYPEs on page 63.

Note: If you want to modify the PTF properties of a component, use the Change Value (in Variant Editor) command or the Modify Component command (in the schematic).

2. Add or modify properties as required and click *OK*.

   These properties are stored for the selected variant and are available for BOM generation.

Defining Alternate Groups

Overview

There are many designs that include a set of parallel components (each with a different footprint), out of which only one component is installed in a particular variant. To create such sets of components, you can create alternate groups.

Creating an Alternate Group

➤ Right-click on the *Groups* folder in the *Alternates* tab and choose the *New Group* option from the shortcut menu.

A new group with the default name *NewGroup* is displayed. Rename this group to specify the name for the alternate group. When renaming, use only alphabets, numerals, or the underscore (_) character.

Note: You can also create a new group by selecting the *New Group* option in the *Edit* menu.

Note: You can use alphabets, numerals, and underscore to name a alternate group, function, or variant. You cannot use space while naming any alternate group, function, or variant.

Adding Components to an Alternate Group

➤ Choose the rows corresponding to the components that you want to include in the alternate group in the bottom-right pane and then choose the *Add to Alternate Group* option from the shortcut menu.

The selected components are added to the alternate group. Notice that one component is assigned the `Pref` status, while the other component is assigned the `Alt1` status. If you had selected three components, then the third component would had been assigned the `Alt2` status.

The assigning of status (`Pref`, `Alt1`, or `Alt2`) is sequential. Variant Editor assigns the first component added to an alternate group as preferred. The subsequent components are treated as alternates.

You can even customize the information in an alternate group. Whatever information you change in the alternate group is generically applied to all variants. You can, however, replace this information by customizing the information for that alternate group in a particular function

or variant. See Customizing the Information in a Variant on page 49 for more information about customizing a variant.

Tip

It is a good practice to keep saving the changes made to the design. To save the design, choose *Save* from the *File* menu.

Creating Functions

Overview

If you need to add a set of components that form a feature or a logical function in multiple variants, you can create a function. Later, you can include or exclude the function from a variant as a whole: that is, you either add all the components defined in the function in the variant or add none of those components in the variant.

Creating a Function

1. Click on the *Functions* tab.
2. Right-click on the *Functions* folder in the *Functions* tab and choose the *New Function* option from the shortcut menu.

   A new function with the default name *NewFunction* is displayed. You can rename this function by selecting *Rename* from the shortcut menu.

   Note: You can also create a new function by doing the following:

   - pulling down the *Edit* menu and selecting the *New Function* option.
   - pressing the `F2` key.

Adding Components to the Function

➤ Choose the components that you want to add to the function in the bottom-right pane and then choose the *Add to Function* option from the shortcut menu.

The selected components are added to the new function.

The components added to a function show a lock sign. This sign means that these components are locked and that any customizing of values for these components will be

applicable only to this function. When you customize the values in a function, the alternate values in the function replace the generic values, if any, defined through the *Alternates* tab.

Customizing Values in a Function

If you have created a function, you can customize its values by doing the following:

1. Choose the function in the left pane.

   The reference designators of the components stored within the function are displayed in the top-right pane.

2. Choose all components in the top-right pane and then click on the *Change Value* option from the shortcut menu.

   The Part Table Filter dialog box is displayed.

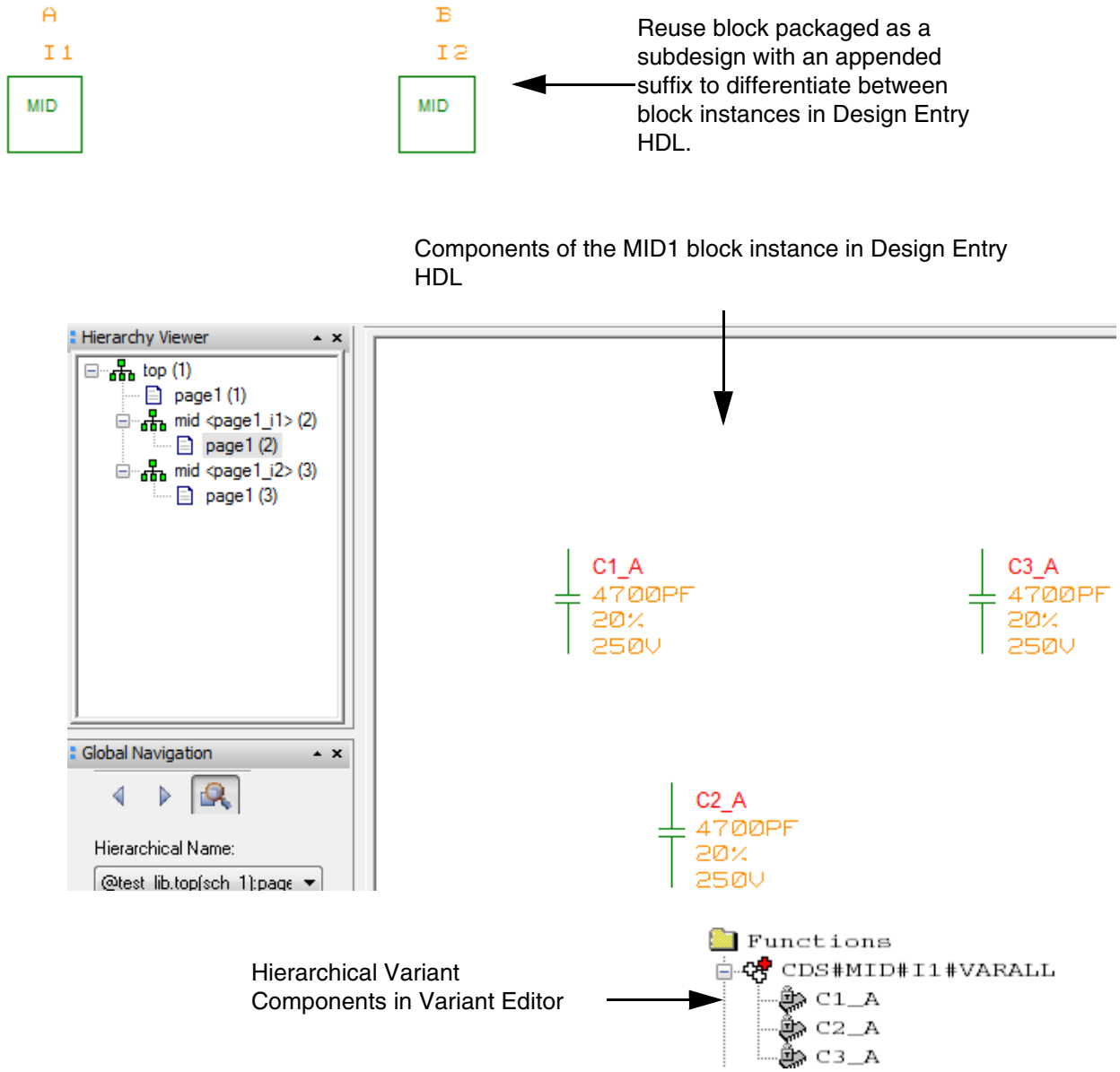
3. Choose the new value and confirm selection by clicking *OK*.

   The values of the components within the selected function have got changed.

Hierarchical Variant Display in Functions

Design Entry HDL supports hierarchical variant tasks in the schematic canvas. Variants defined in lower-level hierarchical blocks can be applied on block instances in higher-level blocks. These variants are referred to as hierarchical variants in DE-HDL documentation.

When you enable hierarchical variants for a block in DE-HDL, the variant is added to the Functions tab in Variant Editor.



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

Locking and Unlocking Components and Groups

The *Lock* command locks the details of the alternates defined on components so that you can change/customize values only for that function/variant. The *Lock* command is used when you want to change the value of alternates or alternate groups in a function or variant. To lock or unlock a component, do the following:

- Choose *Lock* from the *Edit* menu.

  The component is locked, and a lock icon appears at the top of component icon.

- To unlock a locked component, choose a locked component and click on *Unlock* from the *Edit* menu**.**

This will unlock the component. An unlocked component is assigned generic values. Changes made to the unlocked component reflect on all instances of that component (unless some instances are changed after locking within particular functions or variants.)

By default, the *Lock* command is not available for components or alternate groups in the *Alternates* pane. You can unlock components or alternate groups within a variant. If you have included a function in a variant, you cannot unlock components in that function.

Highlighting and Dehighlighting a Component in the Schematic

You might often need to locate a component on a schematic by selecting it in Variant Editor. You can do this by using the *Highlight Source* command. However, before you run the *Highlight Source* command, ensure that the schematic is open in Design Entry HDL.

To highlight a component in the Design Entry HDL schematic, choose *Highlight Source* from the *Display* menu.

The component is selected in Design Entry HDL. After performing the required operations, you can dehighlight the component.

To remove the highlight from the component in the Design Entry HDL schematic, choose the component in Variant Editor and click on *Dehighlight Source* from the *Display* menu.

The component is no longer highlighted and the component appears in its normal colors.

Managing Variants

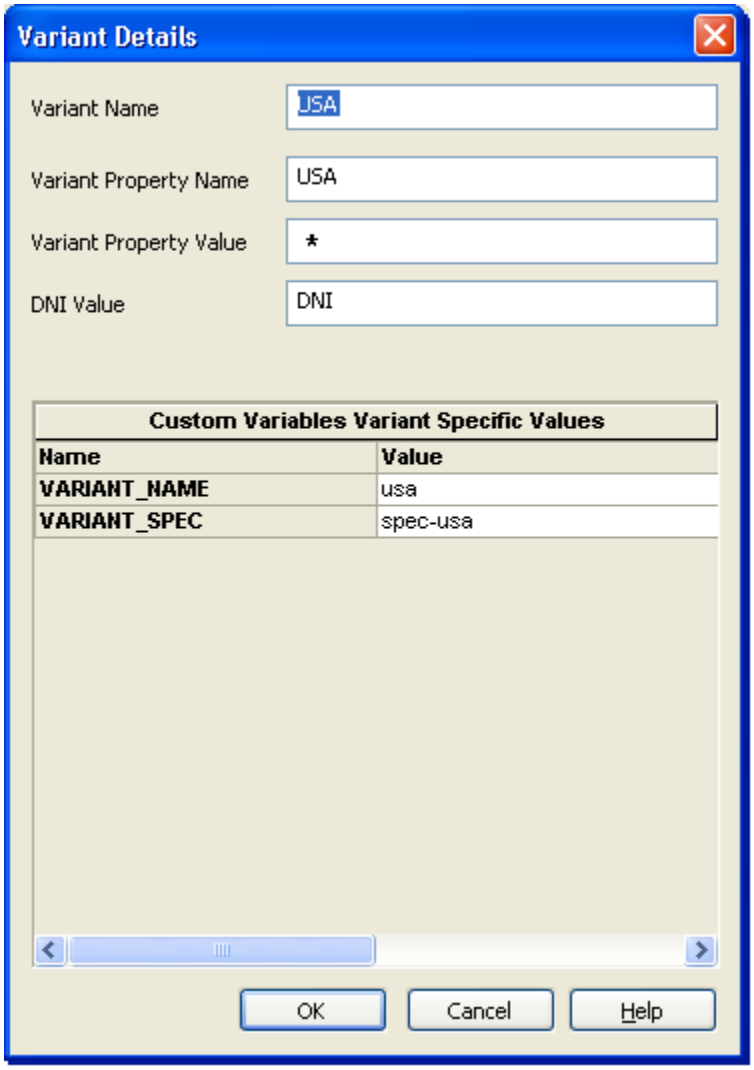
A variant is a variation of the base design created to generate a separate product. To create a variant, you only need to define differences from the base design. You do not require to redefine those components that have the same value as the generic schematic.

The Variant view is a complete hierarchical schematic with occurrence-specific data and Cross Referencer data. Plotting or publishing a PDF of the complete hierarchical schematic with the occurrence data, cross reference data, and variant data is supported in DE-HDL.

Creating or Editing a Variant

1. Click on the *Variants* tab.
2. To create a new variant, do one of the following:
   - ❑ Choose *Edit – New Variant*.
   - ❑ Right-click on the *Variants* folder in the *Variants* tab and choose the *New Variant* command from the pop-up menu.
   - ❑ Press the F2 key.
3. To edit or rename a variant, do one of the following:
   - ❑ Right-click a variant and choose the *Edit/Rename* command from the pop-up menu.
   - ❑ Select a variant and choose *Edit – Edit/Rename*.

The *Variant Details* dialog box is displayed.



4. Specify a name for the variant in the Variant Name dialog.

   The following characters can be used in variant names:

Hyphen (-)

Underscore (_)

Forward slash (/)

The following special characters cannot be used in variant names:

Plus (+)

Backward slash (\)

Round brackets ()

Curly brackets {}

Square brackets []

Asterisk (*)

Ampersand (&)

Circumflex (^)

Percentage (%)

Dollar ($)

Hash (#)

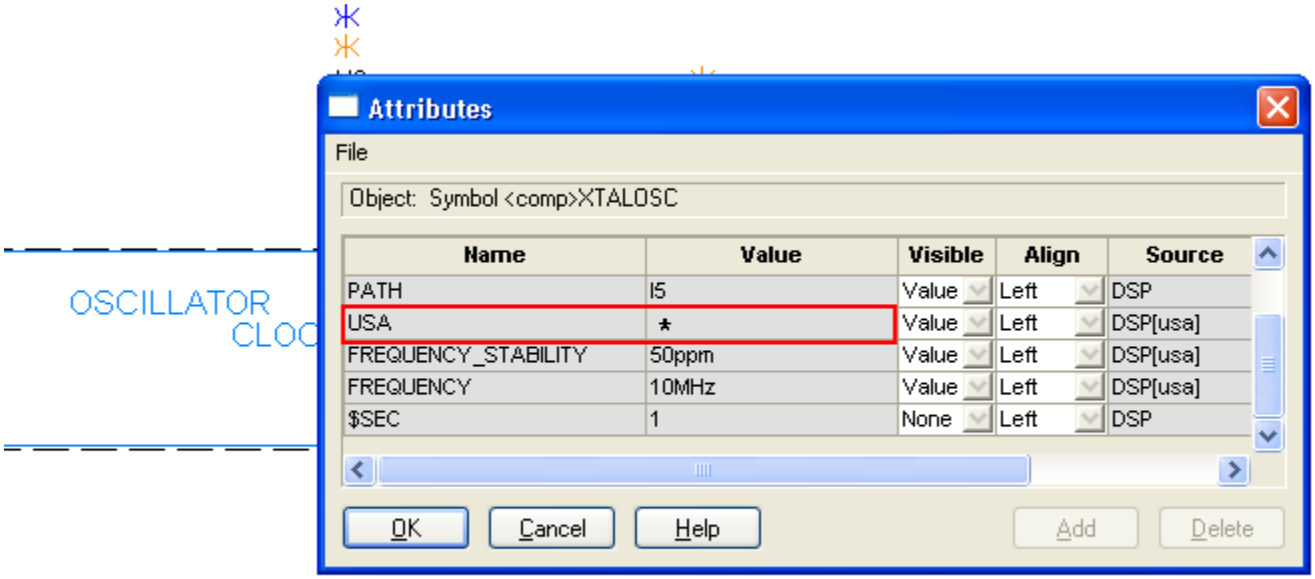
The at symbol (@)

Exclamation mark (!)

Apostrophe (‘)

Tilde (~)

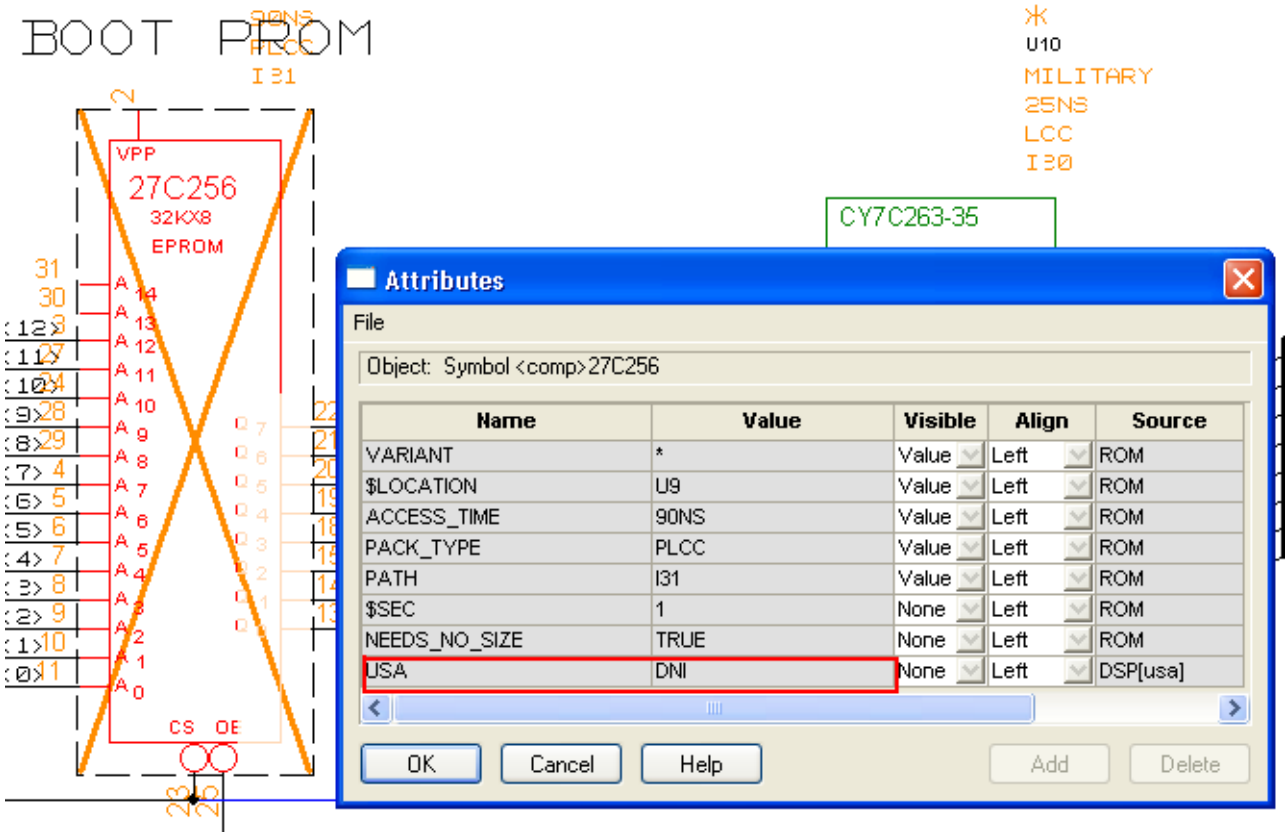
5. Specify the associated Variant Property Name and Variant Property Value, which is annotated to all the components in the base schematic that have variant information.



Note the variant property name and value appearing in the Attributes form of Design Entry HDL. Also note that the *Source* column in the Attributes form displays the name of the property that is specific to the currently enabled variant.

6. Edit the *DNI* value, if required.

Components marked as DNI appear with a *cross mark* in the schematic. The X graphic that runs across the bounding box of the component is the default icon to indicate the DNI status, and cannot be modified.



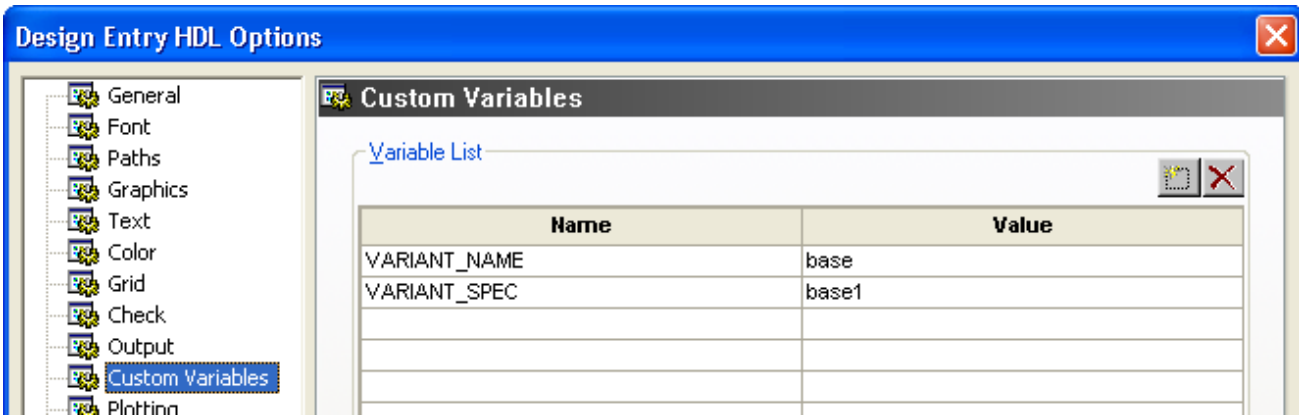
Note: On the UNIX platform, any component marked as DNI is not plotted in HPF plots.

7. Specify a variant-specific value for the custom variable(s).

All custom variables defined in the project are displayed in this dialog with their default values.You can specify a *variant-specific* value to the custom variable in this dialog. The value of the custom variable on the canvas is substituted with the value you specify here for the variant in the variant view.

Note: You can also add the system-defined custom variable, `CON_VARIANT_NAME`, to the base schematic to display the variant name on the canvas for a variant.

You define custom variables in the Design Entry HDL Options dialog:



8. Click *OK*.

Adding Components, Alternate Groups and Functions in a Variant

Before you add any component to the variant, the information in the generic tab (the *Alternates* tab) is applicable to all the variants. It means that the alternate values defined for all the components, and information contained in the alternate groups is applicable to the variant.

You can add components, alternate groups within a variant and change or customize their values. To add a component or alternate group to a variant, perform the following steps:

1. Choose the variant in the left pane.
2. Choose the required component or alternate group in the bottom-right pane and click on the *Add to Variant* from the shortcut menu.

The component or alternate group that you selected is added to the variant. The component is added with the status ‘locked’. Locking allows you to customize the component or alternate group information within the variant. The change that you make is applicable only to that variant.

You can also add functions to a variant.

➤ To add a function to a variant, choose the function in the bottom-right pane and click on the *Add to Variant* from the shortcut menu.

Unlike components and alternate groups, you cannot customize a function within a variant.

Customizing the Information in a Variant

Whatever information is defined in the *Alternates* tab is applicable to all variants. If you want to replace this information, move the component, or alternate group in a variant and customize its value for that variant. You can include or exclude individual functions from a variant and thereby customize the information contained within variants.

To customize the properties of a component or alternate group, first choose the component in the top-right pane and click on one of the following options:

1. **Do Not Install Component -** Choose this option to change the status of the selected row, of a component or an alternate group, to `DNI`. All the existing alternates of the component are deleted.
2. **Change Value** - Choose this option to change the value of an existing component. When you choose the *Change Value* option, the status of the existing component changes to '-' and a new row with the status same as that of the existing component appears.
3. **Make Preferred** - Choose this option to change the status of the selected row of a component or an alternate group to `Pref`. If the selected row corresponds to a component, the status change is immediate. If the selected row corresponds to an alternate group, you can retain the existing status.

   For example, consider an alternate group `ROM` that contains two resistors `R1` and `R2` with the status `Pref` and `Alt1`. Now if you choose the *Make Preferred* command on the `R2` component, Variant Editor message box is displayed asking whether the current status of the row be retained or not. If *Yes* is selected:

   - The current status of `R2` is retained.
   - You are allowed to add a new row for the same reference designator with a new value and with the `Pref` status.
   - The status of the existing row corresponding to the `R1` component changes from `Pref` to `DNI`.
4. **Make First Alternate** - Choose this option to change the status of the selected row of a component or an alternate group to `Alt1`. If the selected row corresponds to a component, the status change is immediate. If the selected row corresponds to an alternate group, you have the option to retain the existing status.

   For example, consider an alternate group `ROM` that contains two resistors `R1` and `R2` with the status `Pref` and `Alt1`. Now if you choose the *Make First Alternate* command on the `R1` component, the Variant Editor message box is displayed asking whether the current status of the row be retained or not. If *Yes* is selected, (1) the current status of `R1` will be retained; (2) the user will be allowed to add a new row for the same reference

designator with a new value and with the status as `Alt1`, and (3) the existing row corresponding to the `R2` component, which has the `Alt1` status, will become `DNI`.

5. **Make Second Alternate** - Choose this option to change the status of the selected row of a component or an alternate group to `Alt2`. If the selected row corresponds to a component, the status change is immediate. If the selected row corresponds to an alternate group, you have the option to retain the existing status.

   For example, consider an alternate group `ROM` that contains three resistors `R1`, `R2` and `R3` with the status `Pref`, `Alt1`, and `Alt2`. Now if you choose the *Make Second Alternate* command on the `R1` component, Variant Editor message box is displayed asking whether the current status of the row be retained or not. If *Yes* is selected, (1) the current status of `R1` will be retained; (2) the user will be allowed to add a new row for the same reference designator with a new value and with the status as `Alt2`, and (3) the existing row corresponding to the `R3` component, which has the `Alt2` status, will become `DNI`.

6. **Delete** - Choose this option to change the status of the selected row of a component or an alternate group to `DNI`. When you delete an alternate, the status of all alternates bubbles up.

   For example, consider an alternate group `ROM` that contains three resistors `R1`, `R2`, and `R3` with the status `Pref`, `Alt1`, and `Alt2`. If you delete `R1`, `R2` and `R3` are assigned the `Pref` and `Alt1` status, and R1 is removed from the list in the top-right pane. If you delete `R2`, `R1` remains as it is while `R3` is assigned the `Alt1` status and R2 is removed from the list in the top-right pane. If you delete `R3`, the status of `R1` and `R2` is unaffected while the status of `R3` changes to - signifying dummy `DNI`.

Tip

It is a good practice to keep saving the variant file as you make changes to it.

Note: The default variant file is named as `variant.dat` and is saved in the `variant` view. Variant Editor on Save generates files specific to each Variant containing Variant specific information. The file is created with the name of the variant and a `.ba` extension. You can use a different name for the variant file. While naming the file, you can use alphabets, numbers, and the underscore letter.

Viewing a Variant

To view variant-specific information on the schematic, you need to open the design in Variant Editor and save the data once. On saving the design, Variant Editor generates files specific to each Variant containing Variant-specific information. These files are used for dynamic display of data in DE-HDL schematics. Variant details can be defined at the time of creating

a new variant or editing existing Variant details. Once saved, the Variants Icon and the *View – Variant menu* in Design Entry HDL list the variants present in the design.

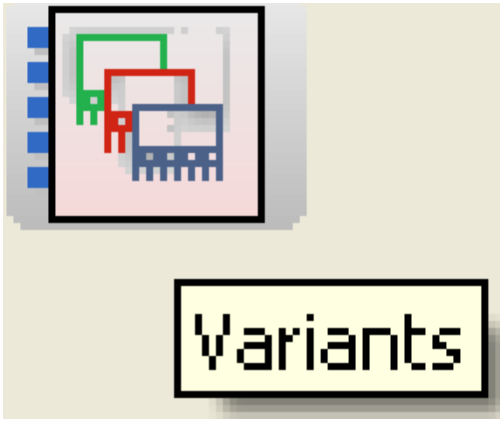
You need to ensure that both Variant Editor and Design Entry HDL are launched from the same instance of Project Manager. Otherwise, the Variant data in these tools will not be in sync and there will be discrepancy in the variant data being displayed.

You can change from base schematic view to any variant view using the following options to view a variant and variant-specific values in a schematic:

- Variants Icon
- View – Variant Menu Command

Variants Icon

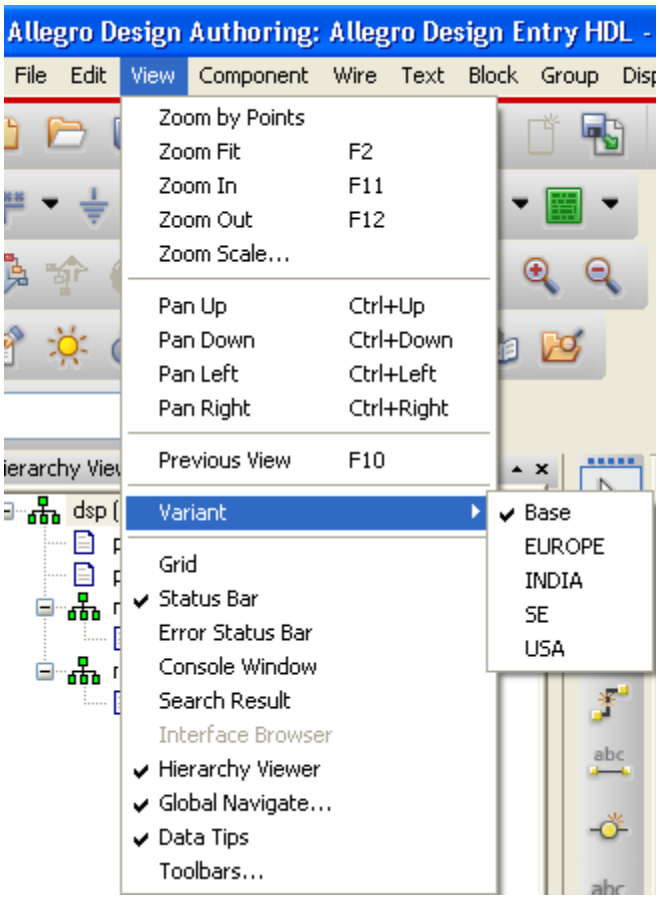
You can select the variant by clicking the *Variants* icon available in the Variant toolbar in Design Entry HDL.



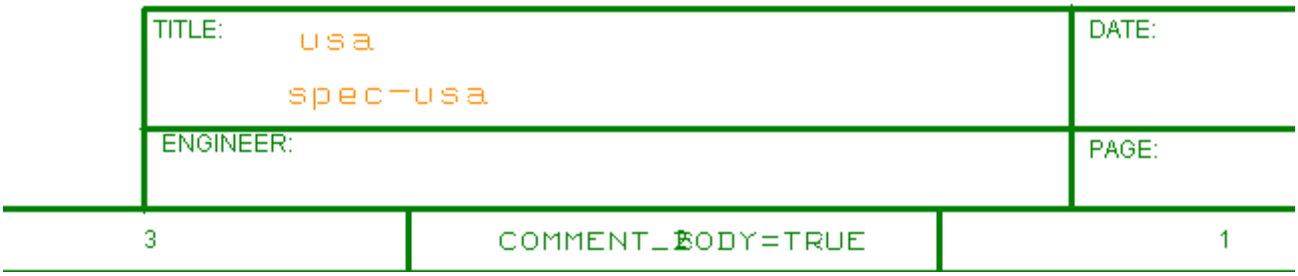
When you click this icon, a list of available variants is displayed. You can choose to view a specific variant from the list.

View – Variant Menu Command

Use the *Variant* menu command under the *View* menu. This command corresponds to the toolbar icon.



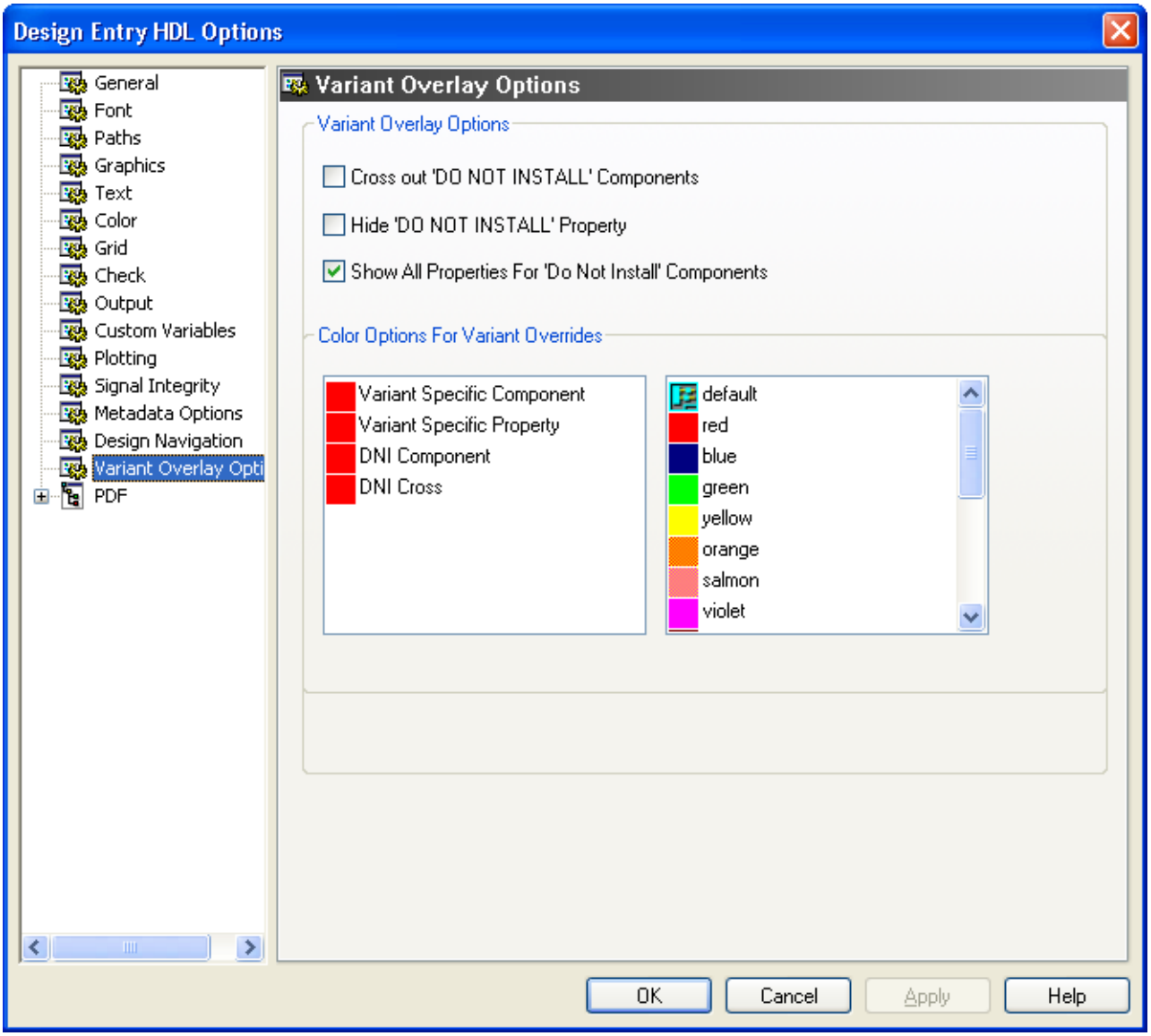
By default, the base schematic is selected. When you select a variant from the list, the design variant appears. You can view the variant-specific information specified in Variant Details dialog in the schematic.



Displaying Specific Information in Variant View

The Design Entry HDL Options dialog includes the *Variant Overlay Options* tab, where you can set options to display selective information in a variant: display or hide the DNI property, cross out the DNI components, or show or hide all properties of DNI components on the

canvas. You can also specify color options for variant-specific components, variant-specific properties, DNI components, and DNI cross.

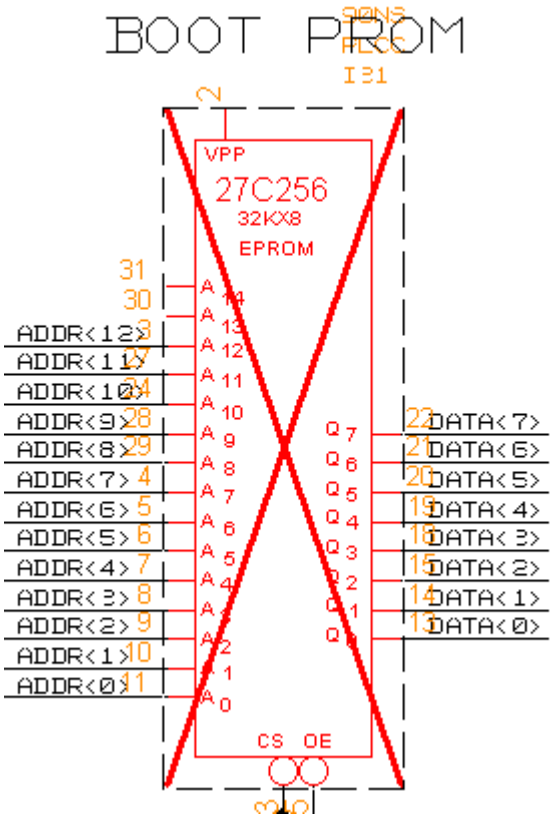


The following table provides a description of the options available in this dialog:

Table 3-1 Design Entry HDL Options — Variant Overlay Options

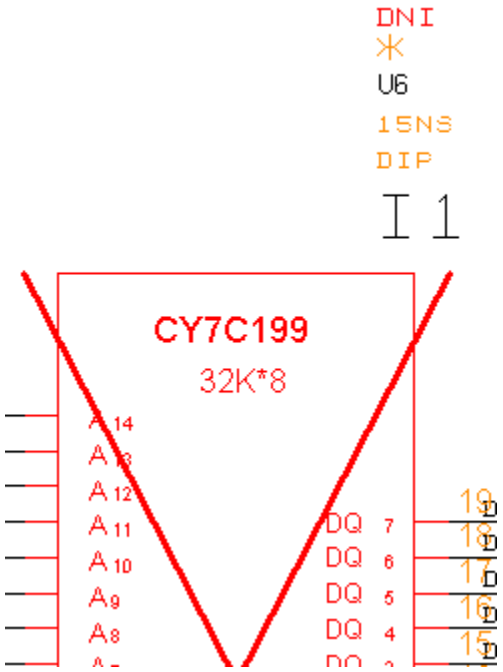
Option	Description

Cross out 'DO NOT INSTALL' Components	Select this option to make a DNI component appear with a cross mark in the variant view.



Hide 'DO NOT INSTALL' Property	Select this option to hide the DNI property in the variant view.

Show All Properties For 'DO NOT INSTALL' Components	Select this option to show all the properties of DNI components in the variant view.



Color Options For Variant Overrides	Specify colors in which the following items appear on the schematic in the variant view: ■ Variant-Specific Components ■ Variant-Specific Property ■ DNI Component ■ DNI Cross

Global Searching

Overview

Variant Editor allows you to customize the properties of a component at multiple places. For example, you can customize the properties in alternates and alternate groups. You can also customize the properties of components in functions or variants. As a result, components can have different values for different customizing. To view all the customizing on a component or alternate group in all functions and variants, you can use the Global Find dialog box.

In addition to viewing all customizing on a component or alternate group in all functions and variants, the Global Find dialog box can also be used to do the following functions:

- View all the variants where a particular function is located.
- Remove a component or remove the alternate value of a component from all variants or selected variants, simultaneously.
- Change value of components - You can choose the desired components in the search result and change its value.
- Delete multiple components that have specific customizing simultaneously. Change the alternate value of a component, for some selected variants, to the same PPT row.
- Navigate to a specific component in Variant Editor.

Note: The behavior of any operation performed by the Global Find dialog box is the same as the behavior of the same operation when performed in Variant Editor. See Performing Operations on Global Find Results on page 60 for more details.

Finding All Customizing for a Component

The steps to find all customizing for a component in Variant Editor are:

1. To invoke the Global Find dialog box, choose *Global Find* in the *Tools* menu.

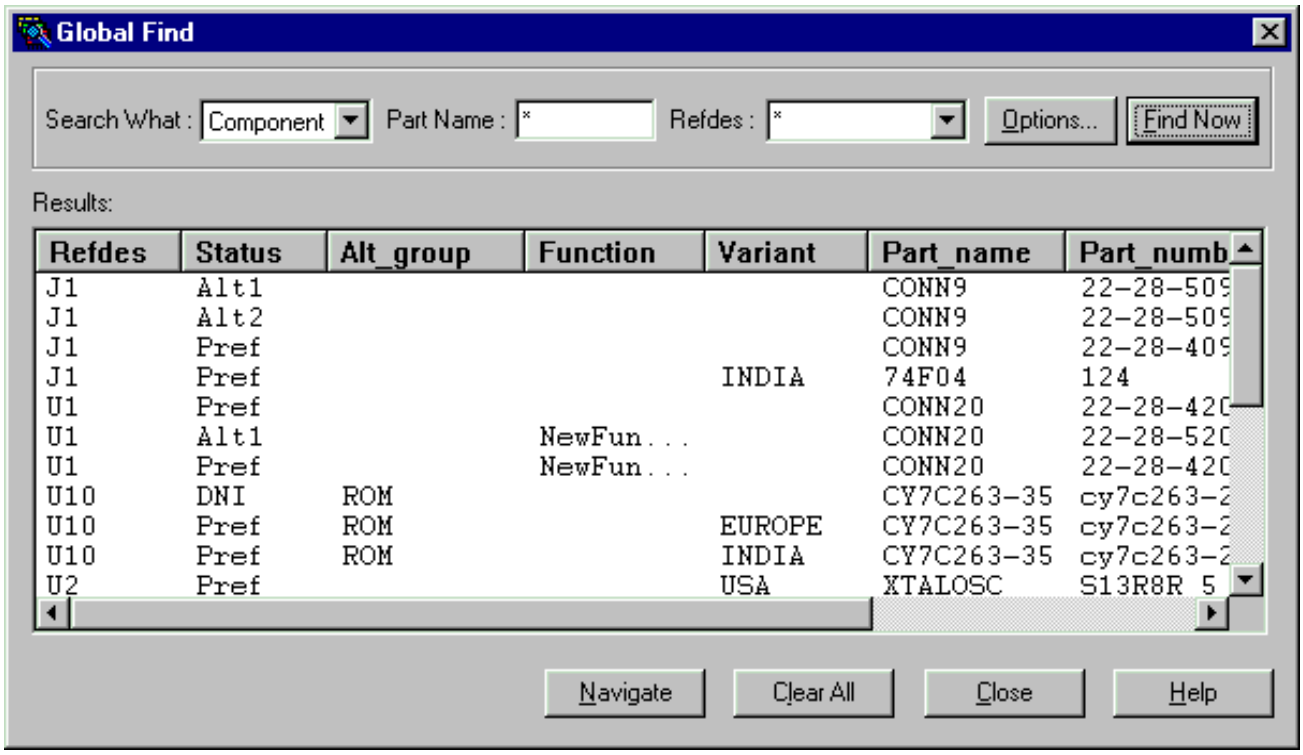
   The Global Find dialog box is displayed. The default option in the *Search What* field is component.

2. To find all customizing for some particular component, enter its part name or reference designator in the *Part Name* or the *Reference Designator* field.

3. Click on the *Find Now* button.

   All customizing for the component is displayed. See Global Find Dialog Box on page 58.

Figure 3-2 Global Find Dialog Box



Note: The Global Find dialog box displays only those components that have been moved to the top-right pane of Variant Editor. Any component that is not moved to the top-right pane is not customized. The properties displayed in the Global Find dialog box include all properties displayed in the right panes of Variant Editor and any other property that has been customized for any component.

Viewing Functions in a Variant

To view a particular function across all variants, complete the following steps:

1. To invoke the Global Find dialog box, choose the *Global Find* option in the *Tools* menu.
2. Choose *Function* in the *Search What* field.

   The *Part Name* field is disabled. You can only enter the name of the function.
3. Type the name of the function in the *Name* field or search the function in the *Name* list.
4. Click on the *Find Now* button.

   All the variants where the specified function is located are displayed. The result displays only two columns, *Function* and *Variant*.

5. If you want to run another search, click on the *Clear All* button to clear the previous search results.

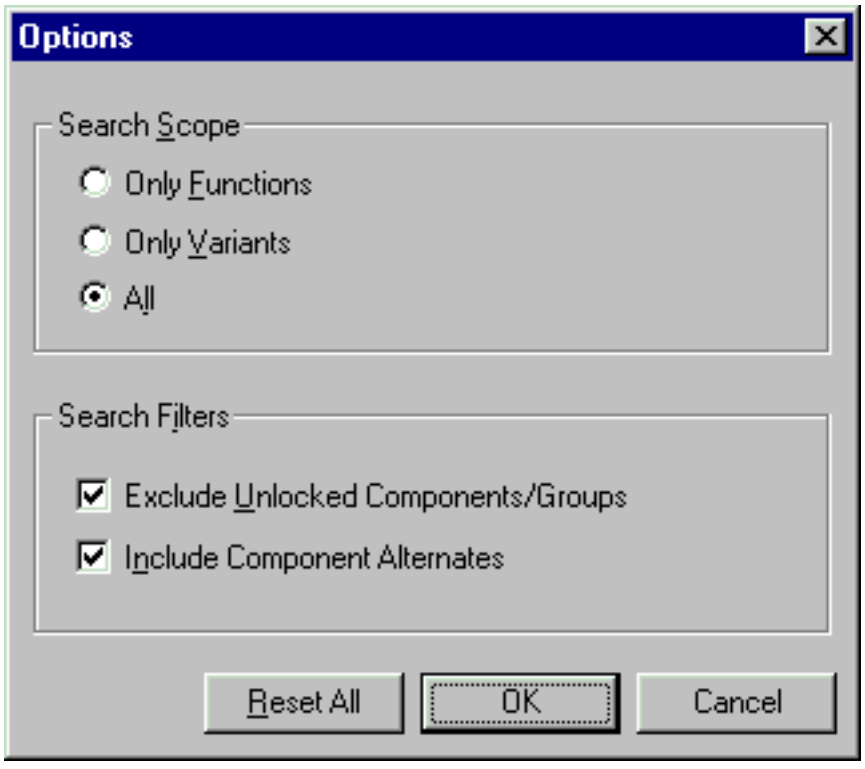
Customizing the Global Search

By default, the Global Find dialog box searches for the customizing in all components, alternate groups, functions, and variants. However, you can limit the scope of global search. You can exclude customizing of components within either functions or variants by selecting the *Only Variants* or *Only Functions* radio button. To customize the scope of search, complete the following steps:

1. Click on the *Options* button in the Global Find dialog box.

   The Options dialog box is displayed. By default, the search is performed across components, alternate groups, functions, and variants.

Figure 3-3 Options Dialog Box



2. To limit the scope of search within only functions, click on the *Only Functions* radio button.

The search will return the customizing for all generic components and alternate groups, and the customizing for all components in all functions.

3. By default, the unlocked components are not displayed in the search list. To display the unlocked components in the search list, choose the *Exclude Unlocked Components/ Groups* check box.
4. To prevent the display of the component alternates in the search list, clear the *Include Components Alternates* check box.
5. To store the search options, click *OK*.

   You are returned to the Global Find dialog box. You can run another search.

Performing Operations on Global Find Results

You can perform four functions on the components returned in the search list of the Global Find operation.

1. **Navigate**

   If you choose a component in the Global Find dialog box and do `Navigate`, the row corresponding to the component is selected in the left pane of Variant Editor. The alternate value of the selected component is also displayed in the top-right pane of Variant Editor.

   - To run the `Navigate` operation, choose a row in the search list by selecting its *`Refdes`* column, and click on the *Navigate* button.

     Note: The `Navigate` operation is not available if you have selected multiple rows in the search results.

2. **Delete**

   The `Delete` operation deletes all customizing for the rows selected in the search list. For example, assume you are searching for the capacitor `C1` and are returned five rows: two in *Alternates* tab, two for the `USA` variant (with the status `Pref` and `Alt1`) and one for the `China` variant. You can use the `Delete` operation to delete the preferred row in *Alternates* tab, the preferred row in the `China` variant, and the row with the status `Alt1` in the `USA` variant.

   - To delete any row click on it, and choose *Delete* from the shortcut menu.

     Note: You cannot delete a row with `DNI` status. It is already deleted. You can only remove the row from the alternate group, function, or variant.

3. **Change Value**

   The `Change Value` operation invokes the Part Table Filter and allows you to change the alternate or preferred value for the selected component. For example, consider you searched for the resistor `R4` and the search list returned eight PPT rows for `R4`, where each row represents an alternate value. You can choose any of these rows and perform the `Change Value` operation. The selected row is assigned the changed values.

   - To change the value of any PPT row, choose that row and choose *Change Value* from the shortcut menu.

     Note: If you choose two components with different reference designators, you cannot perform the `Delete` or `Change Value` operations.

4. **Remove**

   The `Remove` operation simultaneously removes the selected components, alternate groups, or functions in the search list from the selected variants or functions.

Synchronizing the Variant Database and the Schematic

Overview

If you change the original schematic after creating the variant database, on packaging, the changed schematic and the existing variant database will be out of sync with the newly created netlist database. Therefore, it is important that Variant Editor synchronizes the changes between the variant database and the original schematic.

Synchronization involves merging the changes made in the schematic with the variant database so that the latest schematic information is correctly reflected back in the variant database. Synchronization preserves the sanctity of the data between the schematic and the variant database and minimizes the loss of the earlier variant database.

How Synchronization Works

Synchronization between schematic data and the variant database is based on a match between part names, and `JEDEC_TYPE` compatibility. Variant Editor synchronizes the two based on the following rules:

1. If the canonical path and reference designator (the Location attribute on the schematic) match, information in the schematic is inherited as is.

2. If either the reference designator or the canonical path do not match, Variant Editor displays the Conflict Found While Loading Design dialog box. This dialog box allows you to decide whether or not you want to synchronize data between the variant database and the schematic. If you want to synchronize the data, you can synchronize the data based on the winning reference designator or the winning canonical path.

   - If you specify that Variant Editor should synchronize the variant and schematic information based on the winning reference designator, Variant Editor searches the schematic for a reference designator match and the information is inherited.
   - If you specify that Variant Editor should synchronize the variant and schematic information based on the winning canonical path, Variant Editor checks the schematic for the canonical path match and inherits the information.

   Note: The default synchronization in the Conflict Found While Loading Design dialog box is based on the winning canonical path. Therefore, if you do not change this option, synchronization takes place on the winning canonical path.

   - If you decide not to synchronize the variant and schematic information, the previous variant customizing information on that component is lost.

Tip

To minimize variant information loss during synchronization of the variant database and a schematic where the change involves only repackaging the design, choose synchronization based on the winning canonical path. However, if you have only reordered the instances in a design, specifying synchronization on the winning reference designator match reduces variant information loss.

3. If some new component is introduced into a schematic, based on whether the new component is a new package or an additional section in an existing package, the following takes place:

   - New component is a new package - The new component is always incorporated into the variant database. This component does not have any variant information that needs to be inherited.
   - New component is added as a section in an existing package - The variant information of the other sections is inherited by the new section.

4. If a component is deleted from the schematic:

   - If the deleted component was a section of any multi-sectional component, no changes are made in the variant database.
   - If the deleted component was part of an alternate group, then that group is deleted from the variant database.

Note: To correctly inherit an alternate group information, it is necessary that all the components/sections are in the schematic without any change in the canonical path as in the variant database.

Note: For a multi-sectioned component, the canonical path of the first present section in the variant database for every reference designator is used for matching the canonical path.

Using Compatible JEDEC_TYPEs

It is not always possible to replace a component with the same JEDEC_TYPE. There might be cases where you might need to replace a component with a different JEDEC_TYPE. Variant Editor allows you to define components with compatible JEDEC_TYPEs, that is, components which have different JEDEC_TYPEs but which can be replaced with each other and thereby occupy the same footprint on the board.

You can define compatible JEDEC_TYPEs by creating a file named cjedectype.txt in a directory named cdssetup. The cdssetup directory is located at the same level as the project file. An example of the cjedectype.txt file is enclosed.

```
########################################################################
# File for jedec type compatibility
########################################################################
#
C200901_010 C200901_011 C200901_011 C200901_011 C200901_012 C200901_013;
ADDAMS_CAP ADDAMS_CAPC ADDAMS_CAPD;
CC0603 CC1812 CC1812 CC1825 CC1206 CC1812;
RC1206 RC0603 RC2010;
```

The cjedectype.txt file defines four sets of compatible JEDEC_TYPEs. Each set of compatible JEDEC_TYPEs is defined in a single line ending with a semi colon (;). Compatible JEDEC_TYPEs are separated by a space.

Note: You must place the cjedectype.txt file in a directory named cdssetup. Variant Editor, however, uses CSF Search to find whether the cjedectype.txt file exists in your hierarchy.

How compatible JEDEC_TYPEs Work

Assume that you have a component with the RefDes U1 and the JEDEC_TYPE DO_35_NP. Also assume that this component has two compatible JEDEC_TYPEs: DO_35_NP and DO_35_NP2.

If you try to change the value of U1, Variant Editor searches for its compatible JEDEC_TYPEs in the cjedectype.txt file. If compatible JEDEC_TYPEs for the selected component exist, a dialog box with a list of the available compatible JEDEC_TYPEs is displayed. Since U1 has compatible JEDEC_TYPEs, a dialog box displays the following message:

```
Compatible footprints for "DO_35_NP" are: DO_35_NP, and DO_35_NP2.
```

Note: You can set Variant Editor to suppress the display of the compatible JEDEC_TYPE message. To do so, select the *Suppress Messages Compatible Jedec type Match* check box in the Customize Variant Settings dialog box.

Choose *OK* to continue changing the value of U1. The Part Table Filter dialog box is displayed. The JEDEC_TYPE column displays the symbol * signifying that all available JEDEC_TYPEs for the component are displayed. You can now choose any available part and change the value of U1.

Note: Choosing only those parts that correspond to compatible JEDEC_TYPEs is recommended.

If you choose a component that does not have any compatible JEDEC_TYPE and try to change its value, the Part Table Filter dialog box will display only those components that have the same JEDEC_TYPE.

Replacing Components

After defining functions and variants, Variant Editor allows you to replace a component in a function or a variant with another component, if needed. If a component is part of an alternate group, it cannot be replaced. The replaced component can have a different name or properties, but it should have the same or compatible footprint (JEDEC_TYPE property value).

Components with compatible footprints are those that have different JEDEC_TYPEs but which can be replaced with each other and thereby occupy the same footprint space on the board. See Using Compatible JEDEC_TYPEs on page 63 for more information.

To replace a component, do the following:

1. Choose the component in the *Functions* or *Variants* tab.
2. Select *Replace Component* from the shortcut menu.

   The Replace Component dialog box displays all the libraries and components in your design.

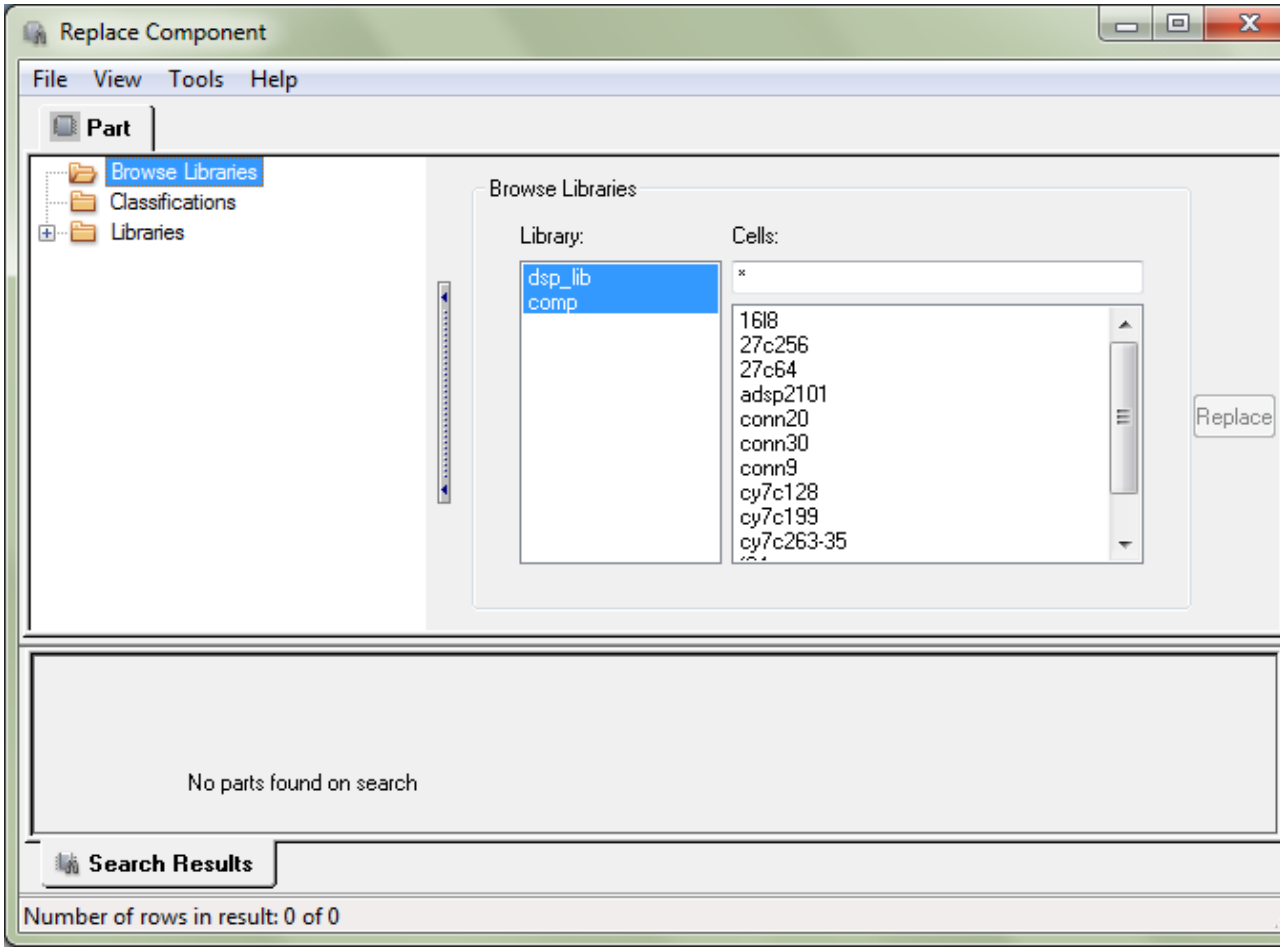
   In earlier versions of Variant Editor, a Replace Variant Component dialog box appeared, which displayed only those components that were part of your packaged design. You could only select from those components.

Starting with this release, you can select any part available in your design library as a replacement. Consequently, the Replace Component dialog box displays all the libraries and components in your design.

The Replace Component dialog box is essentially the Component Browser, which lets you search and select library parts defined in your design. You can search for parts, view details of parts including symbols and footprint, and replace parts.

The Browse Libraries pane displays the Library and the Cells list boxes. The Library list box contains all the Allegro Design Entry HDL libraries listed in the cds.lib file of your schematic. The Cells list box contains the cells available in the library you select. You can select all the libraries under the Library list box by pressing Ctrl+A.

Figure 3-4 Replace Component Dialog Box



3. Select a cell in the Cells list box.

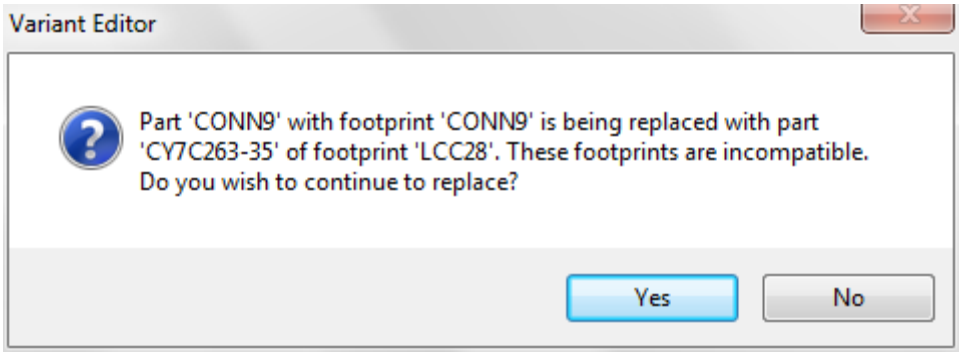
You can filter the cells listed in the Cells list box by entering the initials of the cell name in the text box above the Cells list box. You can also use wildcard characters (*) to filter the cell name.

The physical part table (PPT) rows belonging to the selected cell(s) appear in the Search Results pane.

4. Choose the PPT row with which you want to replace the selected component.
5. Click *Replace*.

When you replace a component in Variant Editor, the tool checks whether the two components—the component that is being replaced and the component that will replace the selected component—have the same or compatible footprints, that is, JEDEC_TYPE properties.

If the footprints are not compatible, a warning message is displayed.



It is recommended that a component not be replaced with another component with a non-compatible footprint. If you continue with the replace operation with a non-compatible footprint, all customized changes on the original component are lost.

If you click *Yes*, the Replace Component dialog box is closed and Variant Editor displays the preferred value of the former component (one that has been replaced) as a grayed out row.

After you replace a component, the new component is displayed in the top-right pane. The `*` symbol in the RefDes field appears signifying that the component is a replaced component. Any further operation, such as `Add Alternates`, on this component will always display the PPT rows corresponding to the new component.

Note: The Replace Component functionality is not backward compatible. If you replace a component in a variant then open the design in an older release, errors may appear.

Synchronizing PTF Information

Overview

In PSD 14.0, Variant Editor would ignore any changes made in the injected properties in the ptf files and retain the properties as contained in the Variant database. In PSD 14.2, Variant Editor can read the changes made in the ptf files in injected properties and use the latest value of these properties as in the ptf files.

When you invoke Variant Editor, it reads the latest ptf files and checks if there is any change in the injected or user-defined properties as compared to the properties stored in the variant database. If a change exists, Variant Editor updates the ppt changes and applies those changes on all variants defined on the design. However, any changes made to the key properties in the ptf files are ignored and for these properties the information in the variant database wins.

Reading PTF Information

1. If you have made any changes in the ptf files, then package the design in the preserve mode.
2. When you invoke Variant Editor, it checks if there is any change in the injected properties in the latest ptf files. If change exists, it displays the following message:

   ```
   Injected properties re-read from ptf files. You need to repackage the design
   (in preserve mode) and re-invoke Variant Editor to update packaged component
   properties.
   ```

   If you have performed step 1 then ignore the above message.
3. Save the variant database. If you exit Variant Editor without saving the variant database, the following message displays:

   ```
   Injected properties of variant components have been updated. Save changes as
   <file_name.dat>.
   ```

   Saving the database allows the latest values to be sued while generating variant

   Note: If you make any changes to properties in the ptf files, you should package the design. If you do not package the design again, you will not be able to get the latest ptf information in the variant database.

   Note: While packaging the design if you choose *Repackage* instead of *Preserve*, then you might get the Conflicts Found While Loading Design dialog box. Select the default option, which is Winning on Canonical Path Match, and chose *OK* to preserve the variant information.

4

Annotating Variant Information

Overview

After creating the variant database, you might want to annotate the variant information onto the schematic to review the design or to plot the schematic.

When you annotate properties to the base schematic, components in the base schematic that have variant information are assigned a property denoting that the components have been assigned variant information.

You can annotate variant information in the following ways:

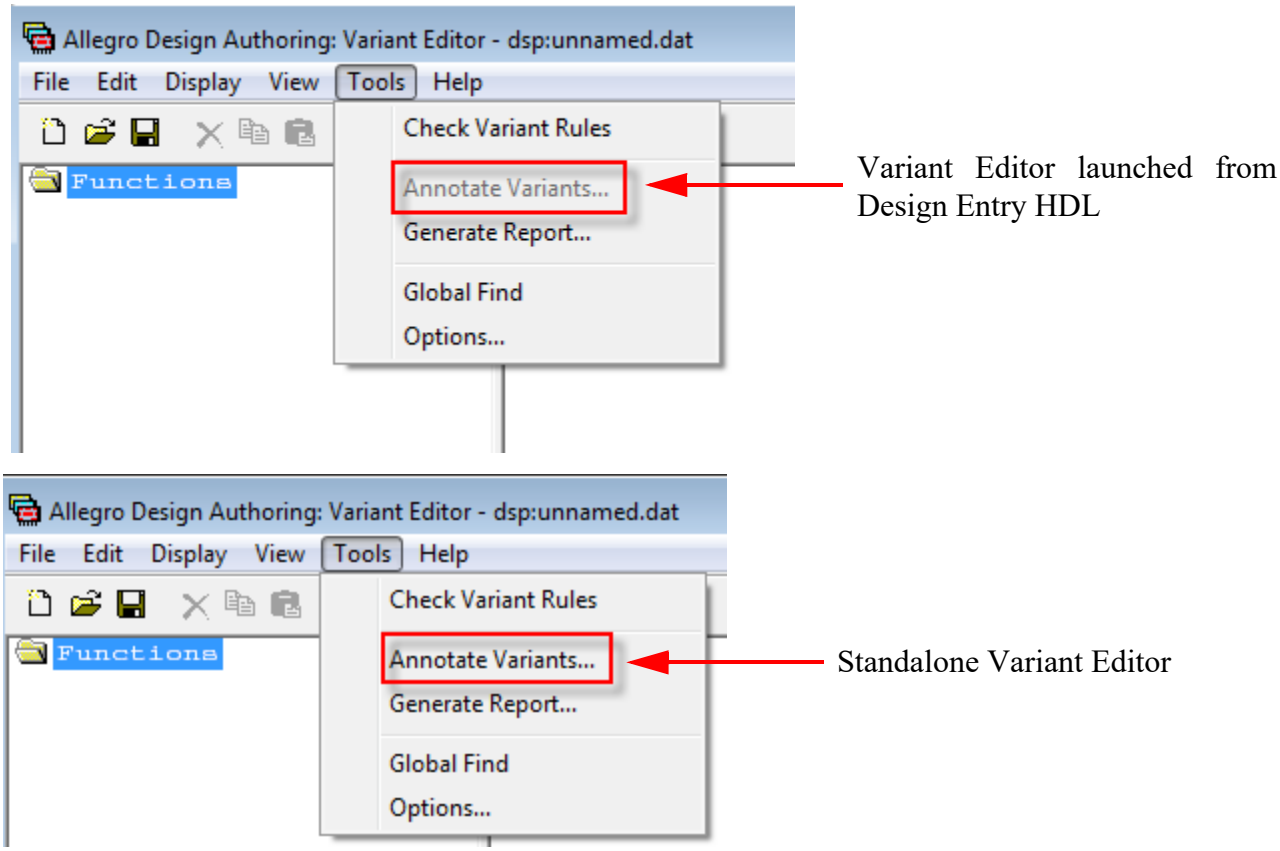
- Annotate variant properties to the base schematic in Variant Editor launched from Design Entry HDL

  When you annotate variant properties to the base schematic in Variant Editor launched from DE-HDL, DE-HDL simply overlays the variant information on the base schematic. With this feature, you can easily view various variant views by dynamically switching between views on the schematic canvas.

  Unlike in releases prior to 16.6-2015, Variant Editor does not create variant-specific files on disk or a flattened view of the schematic. This saves space on disk. Further, because a flattened view is not created, and variant information is simply overlaid on the schematic, you can view occurrence-specific and Cross Referencer data in plots, and publish any variant to a PDF where you view all the variant details.

Important

Since you can annotate properties to the schematic simply by switching variant views, in Variant Editor launched from DE-HDL, the Annotate Variants option is disabled (grayed out). This option is enabled only if you launch Variant Editor in the standalone mode.



See Annotating Variant Properties in Design Entry HDL.

- Using the standalone Variant Editor, you have two options:
  - Annotate properties to a variant

    When you annotate properties to a variant, every component in the variant whose value has changed from the base schematic value, or has the `DNI` status, is assigned a new property. In addition, the changed property values are updated on the components.

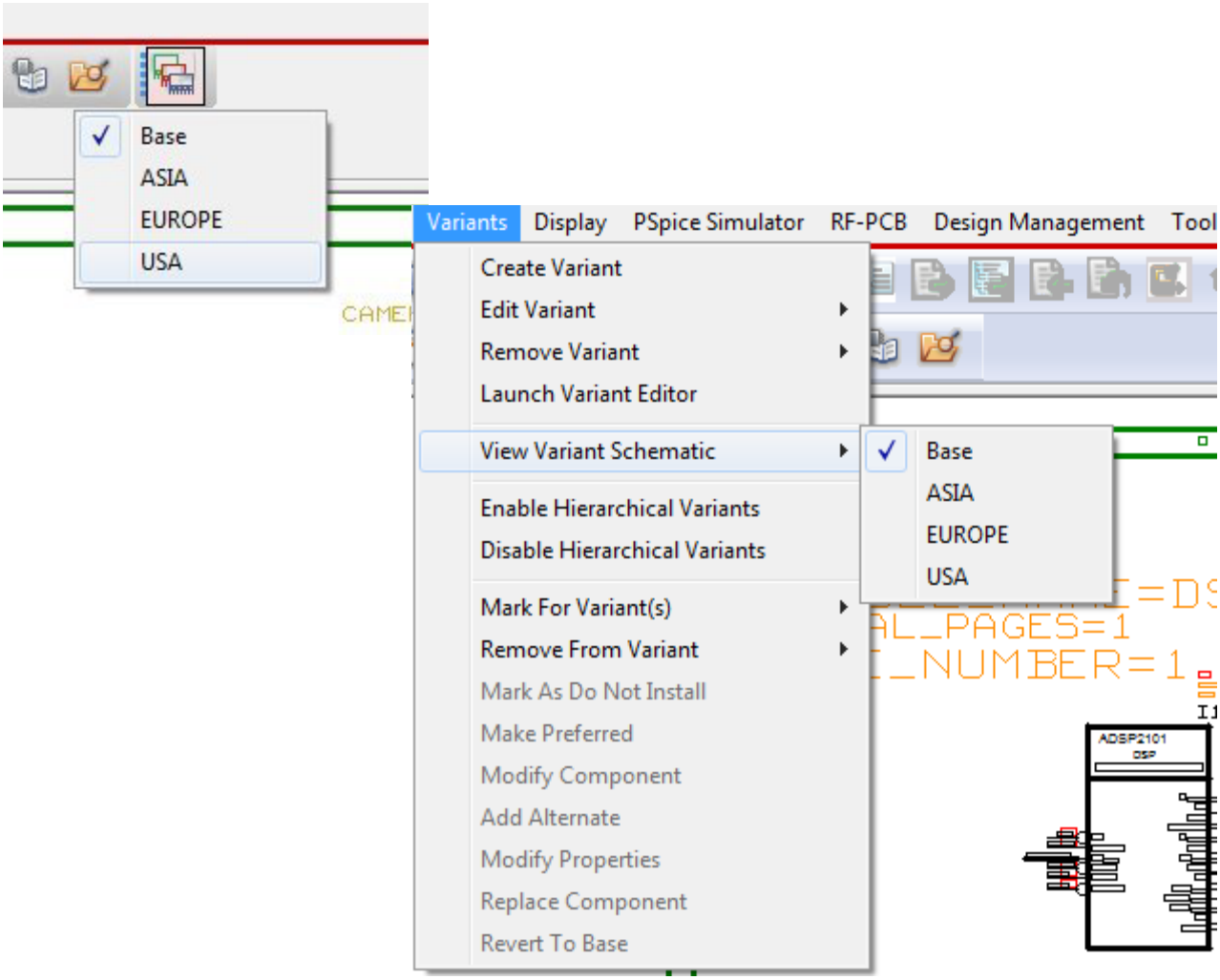
    In this case, a new view for the base schematic is created on disk with the following naming convention: `sch<variant_name>_1`

- Make changes to the original base schematic view, that is `sch_1`. On annotation, Variant Editor creates a new flattened schematic view of the design with the following naming convention: `schbase_1`.

See Annotating Variant Properties to the Base Schematic Using Standalone Variant Editor.

Annotating Variant Properties in Design Entry HDL

To annotate variant information to the base schematic in Design Entry HDL, you can simply switch between variant views. For example, if you have three variants—ASIA, EUROPE, USA, you can switch between these views.



Information (properties) specific to the selected variant, say ASIA, is then overlaid on the base schematic. For instance, if component A has been marked as Do Not Install (DNI) for ASIA, that component is displayed with the DNI property in the base schematic.

Annotating Variant Properties to the Base Schematic Using Standalone Variant Editor

To launch Variant Editor as a standalone application, do one of the following:

- Open Project Manager, and choose *Tools — Variant Editor*.
- Enter `vedit -proj <project_name>` at the command prompt.

  Variant Editor opens with the specified project. If `<project_name>` does not exist in the current directory, Variant Editor opens the file browser, which allows you to navigate and select the project file.

Preparing for Annotation

To annotate variant information to a design using the standalone Variant Editor, you must ensure that both, Variant Editor and Design Entry HDL are running. To launch Design Entry HDL, do one of the following:

- Open the schematic design in Project Manager then click *Design Entry*.
- Enter `concepthdl -proj <project_name>` at the command prompt.

For annotation to work in Variant Editor, the concepthdl.scr file must be at the following location: `<your_inst_dir>/share/cdssetup/concept/concepthdl.scr`. If you have set the `CDS_SITE` environment variable to, say /hm/common, ensure that the `concepthdl.scr` file is installed at /hm/common/`cdssetup/concept/` directory.

Note: If the existing design has a variant module order defined, the module ordering is ignored and a confirmation message is displayed after annotation. In addition, you may choose to manually delete the `variant_modules.dat` file from the variant view in the design to avoid the message. Variant Editor annotation uses module ordering defined in Design Entry HDL.

Before you annotate the variant information to a design, add the following properties on the page border of each page of each module in the design:

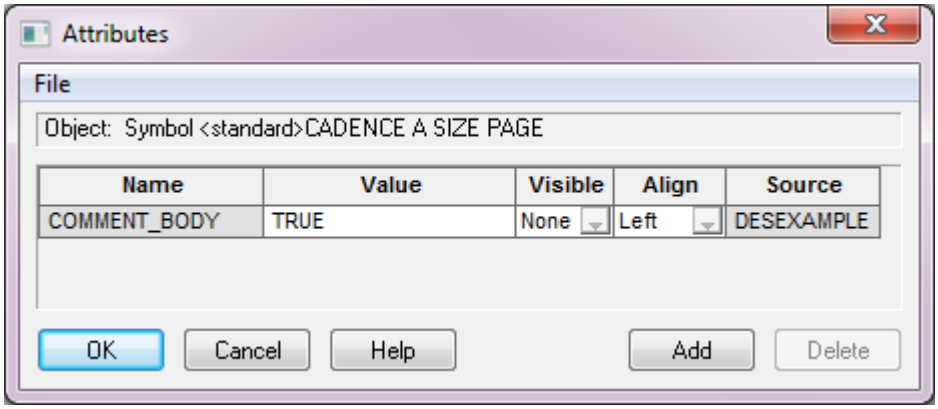
- TOTAL_PAGES
- PAGE_NUMBER

These properties are specific to Variant Editor. When Variant Editor creates a new flattened view of the design, it will update these properties to match the new view.

Adding properties to the definition of a page border

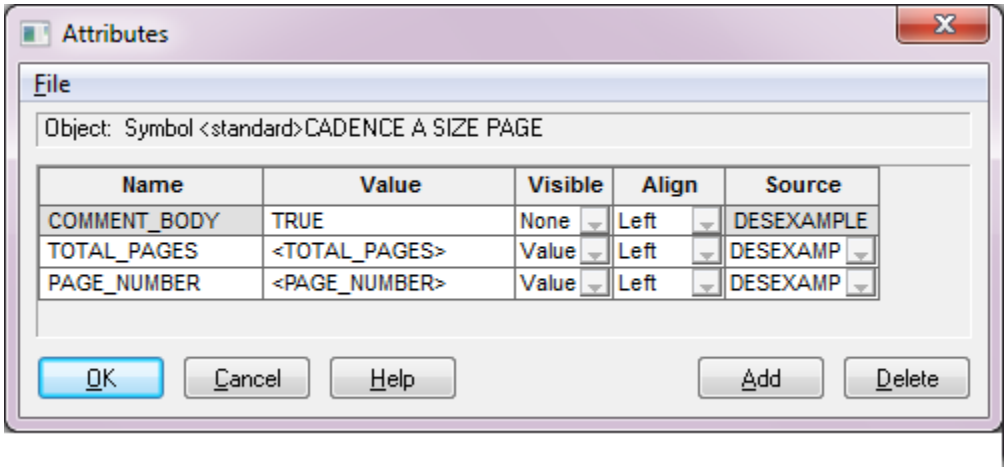
1. Open the page border symbol.
2. Choose *Display > Origin*.
3. Click the *Display Attributes* toolbar button () then click the origin of the symbol.

   The Attributes dialog box is displayed.



4. Click the *Add* button to add a row.

5. Enter the names and values of the properties as illustrated:



6. Click *OK* to save the changes and close the dialog box.

   The page border symbol displays the format string for the custom text. When the page border is instantiated on a schematic page and the variant is annotated, the values of custom variables are substituted.

   For example, when the page border is instantiated on a schematic page and the variant is annotated, the custom variable PAGE_NUMBER will take its actual value on each page. For example, Page 1 or Page 2. For details on annotation, see Annotating a Variant on page 77.

Controlling the Display of Variant Properties in the Schematic Annotated by Variant Editor

When you annotate the variant information on the schematic, Design Entry HDL checks if you have used any placeholder for variant properties. If you have used placeholders for variant properties, Design Entry HDL displays the value of variant properties on the schematic. You can control this display to either name of property, both name and property, or make the display of variant properties invisible.

To change the display of variant properties, use the following environment variable:

```
CDS_VARIANT_PROP_VIS = {BOTH | NAME | VALUE | INVISIBLE}
```

Where,

`BOTH` specifies that both name and value of property appears on the schematic

`NAME` specifies that name of property but not its value appears on the schematic

`VALUE` specifies that value of property but not its name appears on the schematic

`INVISIBLE` specifies that neither name nor value of property appears on the schematic

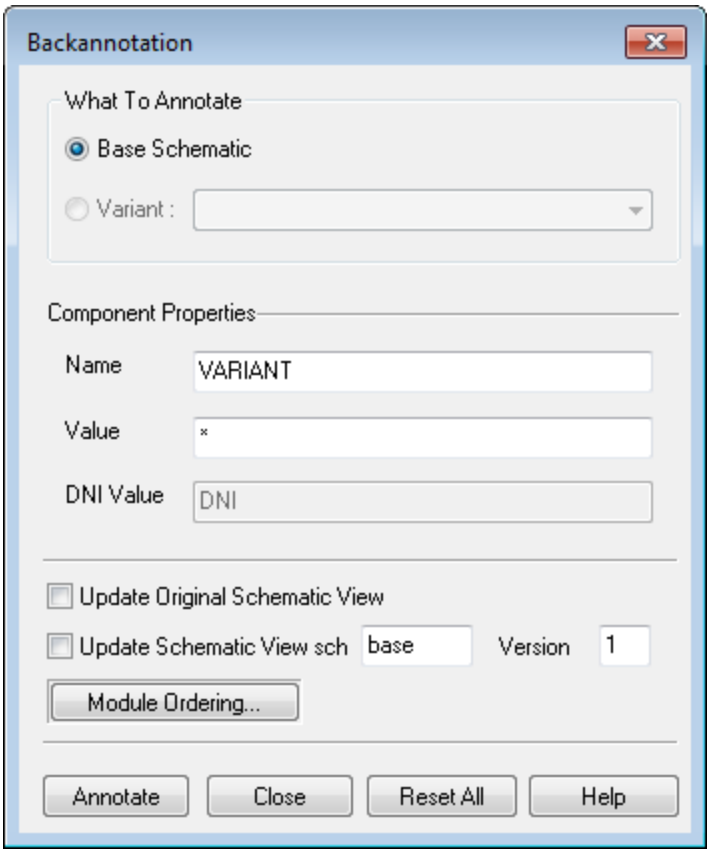
Note: The `CDS_VARIANT_PROP_VIS` environment variable value is case insensitive for value on both UNIX and Windows NT. On UNIX, the name of the environment variable is case sensitive.

Note: If you define an invalid environment variable value, both name and value appear on the schematic.

Backannotation Dialog Box

To open the Backannotation dialog box, choose *Annotate Variants* from the *Tools* menu.

Figure 4-1 Backannotation Dialog Box



The Backannotation dialog box is divided into three main sections:

1. The *What To Annotate* group allows you to choose the base schematic or a variant to be annotated.
2. The *Component Properties* group box allows you to customize the property name, property value, and the DNI value that will be annotated to designate that the component has some variant information.
3. The lower section, which allows you to define whether you want to annotate to the original schematic view or you want to create a new flattened schematic view. However, for a variant, only the latter option applies.

Annotating the Base Schematic

Overview

If you have made any changes to the base schematic by defining variant information in Variant Editor, you can annotate these changes to the Design Entry HDL schematic from Variant Editor. You can create a new view for the base schematic or make changes to the original view, that is `sch_1`.

You can also define a new property and assign it a value. This property will be annotated to all the components in the base schematic that have variant information or that have the DNI status assigned to them.

To annotate variant information to the base schematic, do the following:

1. In the *What to Annotate* group box, choose the *Base Schematic* radio button.
2. By default, the `VARIANT` property with the value `*` is annotated to the Design Entry HDL schematic for all components that have any variant information. You can change this value. Type the property name that will be annotated in the Design Entry HDL schematic in the *Name* field and its value in the *Value* field.
3. To update the original schematic, click on the *Update Schematic View* check box.

   The default name of the new view is `base` and the version number is `1`. You can, if you want, change the name and the version number.
4. Click on the *Annotate* button.

A new flattened schematic view named `schbase_1` is created in Design Entry HDL. The `TOTAL_PAGES` and `PAGE_NUMBER` properties are also updated on each page of the design.

All components that have any variant information or the DNI status are assigned the new property that you defined in step 2.

Annotating a Variant

Overview

You can use Variant Editor to create a new flattened schematic view for any variant. This variant view will have all components and properties that are stored in the base schematic. The variant view also contains a property that you can customize for all components that have any property change from the base schematic or have the status as DNI. The advantage of the variant view is that you can create a schematic for the variant using all the information in the variant database. This saves you time and effort if you were to design the variant schematic from scratch in Design Entry HDL.

To annotate variant information to a variant, do the following in the Backannotation dialog box:

1. Choose the *Variant* radio button in the *What to Annotate* group box.
2. Choose the name of the variant that is to be annotated by selecting it in the *Variant* list

   The *Update Original Schematic View* check box is cleared and grayed out. Whenever you choose a variant for annotation, you cannot update the original schematic view. However, you can create a new flattened view for that variant.
3. By default, the VARIANT property with the value * is annotated to the Design Entry HDL schematic for all components that have any variant information. You can change this value. Type the property name that will be annotated in the Design Entry HDL schematic in the *Name* field and its value in the *Value* field.
4. By default, the VARIANT property with the value DNI is annotated to the Design Entry HDL schematic for all components that have any variant information. You can change the DNI value by assigning a new value in the *DNI Value* field.
5. Choose the *Update Schematic View* check box.

   The default name of the new view is the same as the variant name you selected in step 2, and the version number is *1*. You can, if you want, change the name and the version number.
6. Click on the *Annotate* button.

A new flattened schematic view named *<variant_name>*_1 is created in Design Entry HDL. All components that have any change in property values are assigned a new property

(that you selected in the *Component Properties* box). Any change in property values is also updated in the new view.

5

Managing Variant Information in PCB Editor

Overview

Variant Editor allows you to export the variant information to an interface file that PCB Editor can use to:

1. Generate BOM reports for individual variants. These BOM reports are created in the standard PCB Editor format.
2. Create assembly drawings for individual variants.

Exporting the PCB Editor Interface File

Overview

Before you use PCB Editor to create variant assembly drawings or BOM reports, you need to create the PCB Editor interface file.

Steps

1. From the *File* menu in Variant Editor, choose the *Export* option.

   A message box appears asking if you want to view the interface file (`variants.lst`). This file is created in the physical view of the top-level design.

2. Choose *Yes* to view the file.

The PCB Editor interface file displays all the properties that have change in values from the base schematic for all the variants.

Creating BOM Reports in PCB Editor

Overview

You can create BOM reports for variants from PCB Editor. Include the DNI components in the BOM report.

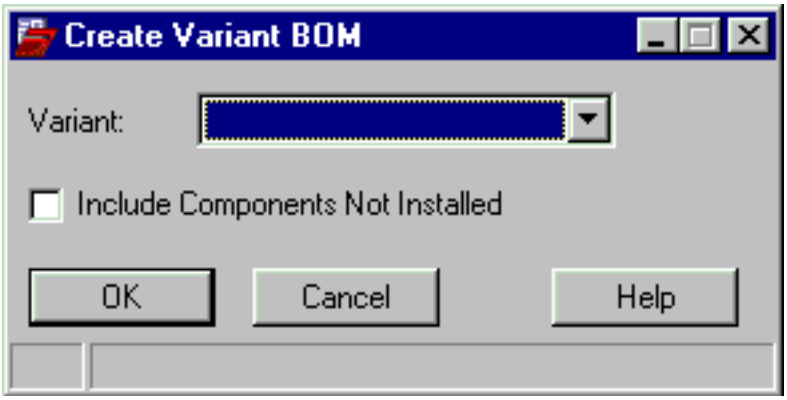
Steps For Creating BOM Reports in PCB Editor

1. Invoke PCB Editor.
2. Choose the *Create Bill of Materials* option in the *Manufacture – Variants* menu.

   Note: You can also run the `variant bom` command from the skill prompt in PCB Editor to generate BOM reports for variants.

   The Create Variant BOM dialog box is displayed.

Figure 5-1 Create Variant BOM Dialog Box



3. Choose the variant for creating BOM report.
4. To include all components with the DNI status in the BOM report, choose the *Include Components Not Installed* check box.
5. Click on the *OK* button.

The variant BOM report named `var-<variant_name>.rpt` is created in the physical view under the top-level design.

Report Explanation

The `var-<variant_name>.rpt` report has two sections of text. The first section displays the package name, component type, component value and tolerance, component class, and reference designator for each component installed in the selected variant. The second section lists the reference designators of all the DNI components.

If you do not to select the *Include Components Not Installed* check box, then the second section in the report that lists the DNI components would not have been present.

Creating Variant Assembly Drawings

Overview

You can create an assembly drawing layer for components belonging to a specific variant of the current design. You can define whether the components on the top or the bottom side of the board will be displayed.

Prerequisite for Creating a Variant Assembly Drawing

Before you create a variant assembly drawing, display the *Color/Visibility* dialog box and set the *All Invisible* option in the *Global Visibility* combo box. Next, choose the subclasses that you want to include in the variant assembly drawing that you are generating.

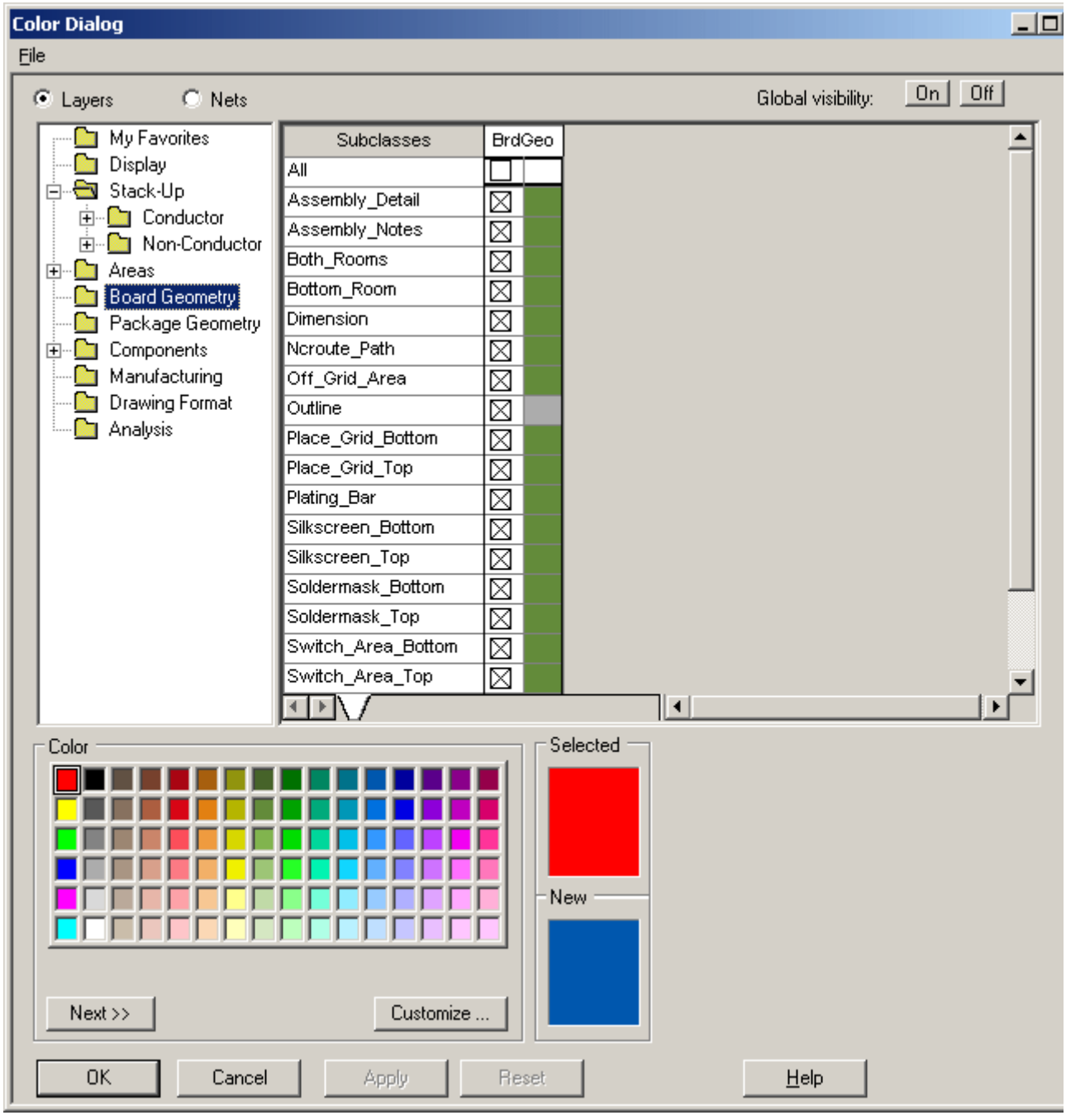
Steps

Selecting Subclasses for Variant Assembly Drawing

1. Choose the *Display – Color/Visibility* option.

   The Color and Visibility Dialog Box figure on page 82 is displayed. You can use this dialog box to control both the colors and the visibility of the various classes and subclasses in Allegro PCB and Package Designer.

Figure 5-2 Color and Visibility Dialog Box



2. Choose the *All Invisible* option in the *Global Visibility* combo box.
3. A message box is displayed. Click on the *Yes* button.

4. Choose *Components* in the *Group* list box and choose the properties that you want to be visible for the selected assembly. For example, if you want to display the `COMP_VALUE` and `REF_DES` properties for the `ASSEMBLY_TOP` group, choose the *COMP_VALUE* and *REF_DES* check boxes corresponding to the `ASSEMBLY_TOP` group.
5. Click *OK*.

You have defined the subclasses that will be visible in the assembly drawing.

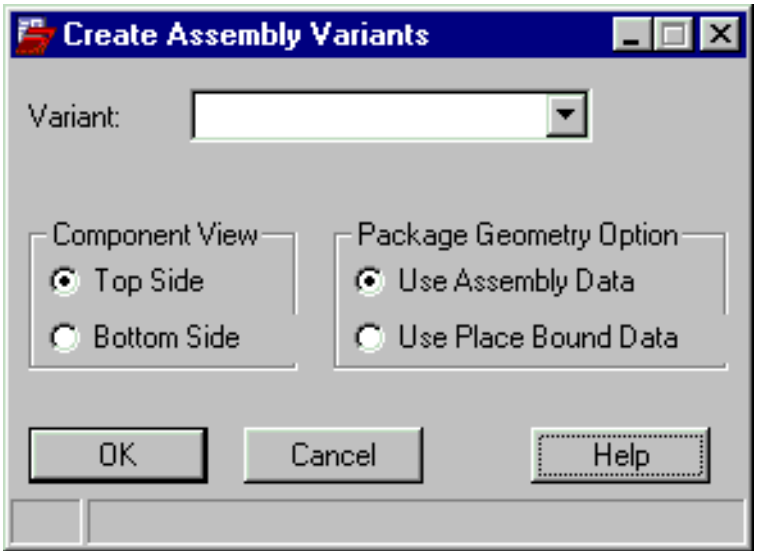
Creating an Assembly Drawing

1. Choose the *Create Assembly Drawing* option in the *Manufacture – Variants* menu.

   The Create Assembly Variant Dialog Box figure on page 83 is displayed. You can use the Create Assembly Drawing dialog box to choose the variant for which the assembly drawing is to be created. You can define whether you are creating an assembly drawing for the top-side components or the bottom-side components. You can also define whether you are using the place bound outlines or the assembly outlines.

   Note: You can also run the `variant assembly` command from the skill prompt in PCB Editor to create variant assembly drawings.

Figure 5-3 Create Assembly Variant Dialog Box



2. Choose the variant for creating the assembly drawing in the *Variant* list box.
3. Depending on whether you want to create the top-side or bottom-side component view, or use the assembly or place bound data for creating the assembly drawing layer, choose

the appropriate radio buttons. The default options are, *Top Side* and *Use Assembly Data*.

4. Click *OK*.

A message ‘Generating assembly drawing `MANUFACTURING/<variant_name>_<component_view>` for design variant `<variant_name>` using assembly data' is displayed. This message signifies that an assembly drawing is created as a subclass under the `MANUFACTURING` class.

Note: Information that is copied to a variant assembly drawing includes the appropriate assembly or place bound outlines (which is the ONE exception that does NOT need to be visible at the time of report generation), and the information from any other subclasses (shapes, lines, and text) that are visible at the time of assembly drawing generation.

Note: You can generate the variant assembly drawing for the components, which are either on the top-side or the bottom-side of the board. If you want to generate the assembly drawings for a variant for both the top- and bottom-side components, generate both the top and bottom sides separately, and display or plot the two sides as superimposed.

Viewing the Assembly Drawing

1. Display the *Color/Visibility* dialog box by selecting the *Display – Color/Visibility* option.
2. Choose the *All Invisible* option in the *Global Visibility* list box.
3. A message box is displayed. Click on the *Yes* button.
4. Choose *Manufacturing* in the *Group* list box.
5. Select the check box corresponding to the assembly layer that you have created. See Creating Variant Assembly Drawings.
6. Click *OK*.

The selected assembly drawing layer is displayed.

6

Variant Editor Command Reference

Main Menu Commands

Variant Editor contains six menus: File, Edit, Display, View, Tools, and help. Each menu contains a set of related commands as described in the following sections:

File Menu

File – New

Creates a new variant database.

File – Open

Opens the variant database file that you have selected.

File – Save

Saves the current variant information into the variant database.

File > Save As

Allows you to save a copy of the variant file, with a different name, into the variant database.

You cannot modify the default save location of the variant file.

File > Export

Exports the design into an interface file. This file can be read by PCB Editor.

The exported file lists information about all variants. The components are grouped in the following order: base, alternates, and alternate groups. All available components that are not part of alternates or alternate groups are listed as part of the base.

An example of an exported file is listed below:

```
(
    (Variant US
        (base
            (C10 C12 C2 C3 C5 C7 C9)
        )
        (alternate C1
            (C1 VALUE="1UF" VOLTAGE="25V" PART_NUMBER="200500-122")
            (C1 VALUE="10UF" VOLTAGE="6V" PART_NUMBER="200500-287")
        )
        (alt_group cpack
            (C13)
            (C6)
            (C11)
```

File > Reload Design

Loads the current design again.

If you have not saved the changes, the `Reload Design` command allows you to save the changes. However, you can use this opportunity to revert the original unsaved design by clicking `No` in the dialog box.

File > Change Suite

Allows you to choose the product suite for launching Variant Editor. Changing product suites allows you to access components, which are not available in the current product suite.

File > Exit

Exits the tool. If you have not saved the changes, a warning box displays providing opportunity to save the variant database file and quit.

Edit Menu

Edit > Undo

Ignores the changes performed by the last operation.

Edit > Redo

Repeats the last operation.

Edit > New

Adds a new group, function, or variant.

Edit > Rename

Allows the user to rename the selected alternate group, variant, or function.

Note: You can use only alphabets, numbers, and the underscore (_) letter to name the selected alternate group, variant, or function. If you use an invalid letter such as & or $, Variant Editor will display the following message:

```
Renaming failed ...
Invalid characters '&' found.
```

Edit > Description

Allows you to define a description for the alternate group, function, or variant.

Edit > Copy

Copies the information contained in a function or variant into a buffer.

Edit > Paste

Pastes the current information of the variant or the function that has been selected for copy. The pasted information is saved in the file named *copy of <source>.*

Note: The `paste` command copies the current state of the selected variant/function and not the state that was copied using the `copy` command.

Edit > Lock

Locks the details of the alternates defined on components so that you can customize the values only for that function or variant. This command is used when you want to change the value of alternates or alternate groups in a function or variant. To choose the `lock` command, right-click on a component in the Object browser and then click on *Lock*.

Edit > Unlock

Unlocks the component that you have already locked. You cannot customize an unlocked component.

Edit > Restore

Restores the values of alternates on components to those that are generically defined from the *Alternates* tab.

Edit > Replace Component

Replaces a component (which is not a part of an alternate group) added to a function or a variant with another component.

Edit > Remove Component

Removes the components from the alternate list.

Note: All alternates defined on the removed component will be lost.

Edit > Remove Group

Removes the selected alternate group from the alternate list.

Edit > Remove Function

Removes the selected function from the function list.

Edit > Remove Variant

Removes the selected variant from the variant list.

Edit > Delete

Deletes groups, functions, and variants, or removes components from groups, functions, and variants.

Display Menu

Display > Highlight Source

Highlights the component in the Design Entry HDL schematic.

Display > De-highlight Source

Removes the highlight from the component in the Design Entry HDL schematic.

View Menu

View > Customize Columns

Allows you to choose the property names whose values you want to see in the Object Properties window and in BOM reports. For example, you can choose the `PART_NAME`, `PART_NUMBER`, and `CLASS` properties for display.

View > Reset Filters

Resets any filters that might have been applied to the selected components. This command displays all available components in the bottom-right pane.

Tools Menu

Tools > Check Variant Rules

Checks the design for rules, such as PPT value mismatch and ambiguous customizing, and displays errors, if any.

Tools > Annotate Variants

Takes a given variant and annotates it to the schematic.

Tools > Generate Report

Lets you to generate BOM reports for the base schematic, any variant, or variant comparison BOM.

Tools > Global Find

Searches specific components, alternate groups, or functions in the variant database. You can customize the search to display the components that you want.

Tools > Options

Lets you to rename the status designator for the preferred component, define the sorting style, and suppress the display of messages for compatible Jedec types.

Help Menu

Help > Contents

Displays Online information for Variant Editor.

Help > About Variant Editor

Displays a splash screen that contains information about the copyright notice and version details.

System Menu Commands

The following are the available System Menu commands in Variant Editor:

Add Alternates

Adds alternates to the selected components. You need to choose an alternate value for the selected component. If only the preferred value exists for a component, the new value is added with the `Alt1` status. If an alternate with `Alt1` status exists, the new value is added with the `Alt2` status.

Note: This command is available in the *Alternates* tab when you right-click on a component in the Object Properties window.

Note: This menu is not available when both alternates exist or the preferred value of the selected component is DNI.

Add to Alternate Components List

Moves the selected component to the top panel and adds the `Pref` status to it.

Note: This command is available in the *Alternates* tab when you right-click on a component in the bottom-right pane.

Add Function to Variant

Adds the selected function to the selected variant.

Note: This command is available in the *Variants* tab when you right-click on a function in the bottom-right pane.

Add to Alternate Group

Adds the selected components to the selected alternate group.

Note: This command is available in the *Variants* tab when you right-click on a function in the bottom-right pane.

Add to Function

Adds the selected component or alternate group to the selected function and removes it from the bottom-right pane. If the selected component belongs to an alternate group, then all components of the alternate group are added to the function.

Note: Variant Editor adds components to functions in lock mode. However, alternate groups are always added in unlock mode. To make changes to an alternate group added to a function, first lock it and then make changes.

Copy

Copies the information into a buffer. This command is generally followed by the `paste` command.

The `copy` command can be used to copy only a variant or function.

Note: The `copy` command marks a pointer to the variant or the function that needs to be copied. When the `paste` command is selected, the current state of the variant or function is copied. This is in contrast to copying the state of the variant or the group at the time when the `copy` command was executed.

Change Value

Brings up the Design Entry HDL component browser with PPT (physical part table) rows listing all values that you can change. You can choose a different PPT row, thereby selecting a different value for the component. However, the value of the `JEDEC_TYPE` of the component on the base schematic must match that of the selected row. If the selected row is the base row, the base row is assigned a `DNI` (do not install) status and a new row is added with the selected value. The `DNI` status is shown by the `-` sign.

Description

Allows you to add comments about the group, function, or variant.

Delete

Based on which component you choose the `delete` command, the following operations are performed:

- Selecting the `delete` command on a component that has only *Pref* as its status removes the component. However, if the component has alternates and you delete one of the alternates, the status of other alternates changes in a bubbling-up manner. See Design Entry - Bubbling Up on page 115 for details.
- Selecting the `delete` command on component(s) in alternate groups changes the status of the component(s) to `DNI` (do not install). The status of other components in the

alternate group changes in a bubbling-up manner. However, if the component you are deleting is preferred and there are no other alternates, all components are deleted from the alternate group.

Note: The `delete` command is available in the top-right pane.

Do Not Install

Changes the status of the selected row, of a component or an alternate group, to `DNI` (Do Not Install). The existing alternates for a component, if any, are deleted.

Explore

Displays the details of the properties of the selected items.

Lock

Locks the details of the alternates defined on components so that you can customize values only for that function or variant. The `lock` command is used when you want to change the value of alternates or alternate groups in a function/variant. To select the `lock` command, right-click on a component in the top-right pane and choose `Lock`.

Make Alternate

Displays the Make Alternate dialog box where you select an alternate as the new status of the selected row (of a component or an alternate group).

- In case of components, you can swap the status of the selected row with the status of the alternate that you select in the Make Alternate dialog box.

  You cannot retain the status of existing components and create new alternates with the `Make Alternate` command. For example, consider you have a component `R1` with five alternates - `Pref`, `Alt1`, `Alt2`, `Alt3`, and `Alt4`. If you select the PPT row with the `Alt3` status and run the `Make Alternate` command, you can select only `Pref`, `Alt1`, `Alt2`, and `Alt4` as new status. You cannot retain the existing status. Therefore, if you choose the new status as `Alt1`, Variant Editor will swap the status of the PPT rows `Alt1` and `Alt3`.

- In case of alternate groups, you can either swap or retain the status of the selected row with the status of the alternate that you select in the Make Alternate dialog box.

To retain the status of the existing PPT row, choose the *Retain Current Status* check box in the Make Alternate dialog box. If you do not choose the *Retain Current Status* check box, Variant Editor swaps the status of the existing PPT row with the status of the new PPT row.

For example, consider you have an alternate group `ROM` with the following components and values:

```
R1  4.4K Pref
    2.0K Alt1
    3.0K is Alt2
R2  2.4K Alt3
    1.0K Alt4
    4.0K is Alt5
R3  5.0K Alt6
```

Assume you choose the PPT row corresponding to the `R1 (3.0K)` component and run the `Make Alternate` command. Variant Editor displays the Make Alternate dialog box. If you select the PPT row corresponding to the `R2 (1.0K)` component and do not choose the *Retain Current Status* check box, Variant Editor swaps the status of the two rows. If you had chosen the *Retain Current Status* check box, Variant Editor displays the Part Table Filter dialog box, where you can choose a new part (PPT row) that would be replaced with the `R1 (3.0K)`. Assume you select a new part `R1 (4.8K)`. Variant Editor will then retain the existing PPT row for the `R1 (3.0K)` component, create a new PPT row with values `R1 (4.8K)` and status `Alt4` and change the status of the `R2 (1.0K)` component to `DNI`.

Make First Alternate

Changes the status of the selected row, of a component or an alternate group, to `Alt1`.

- In case of components, the swapping of status occurs whenever the `make first alternate` command is used.

  For example, consider you have a component `R1` with the following values:

  ```
  R1  4.4K Preferred
      2.0K Alternate 1
      3.0K is Alternate 2
  ```

  If you choose 3.0K (alternate 2) and use the `make first alternate` command, the original alternate 2 value (3.0K) becomes the alternate 1 value and the original alternate 1 value (2.0K) becomes the alternate 2 value. In simple words, the status of the two alternate values is swapped.

- In case of alternate groups, you have the option of retaining the status of the selected component. If you do not want to retain the status, swapping of status takes place the way it occurs for components.

  To retain the current status, choose a new alternate 1 value. This value is added to the alternate group and is assigned the `Alt1` status. The status of the existing alternate 1 component changes to `DNI`.

Make Preferred

Changes the status of the selected row, of a component or an alternate group, to `Pref`.

- In case of components, swapping of status occurs whenever the `make preferred` command is used.

  For example, consider you have a component `R1` with the following values:

  ```
  R1  4.4K Preferred
      2.0K Alternate 1
      3.0K is Alternate 2
  ```

  If you choose 3.0K (alternate 2) and use the `make preferred` command, the original alternate 2 value (3.0K) becomes the preferred value and the preferred value (4.4K) becomes the alternate 2 value. In simple words, the status of the two values is swapped.

- In case of alternate groups, you have the option of retaining the status of the selected component. If you do not want to retain the status, swapping of status takes place the way it occurs for components.

  To retain the current status, choose a new preferred value. This value is added to the alternate group and is assigned the `Pref` status. The status of the existing preferred component changes to `DNI`.

Make Second Alternate

Changes the status of the selected row, of a component or an alternate group, to `Alt2`.

- In case of components, the swapping of status occurs whenever the `make second alternate` command is used.

  For example, consider you have a component `R1` with the following values:

  ```
  R1  4.4K Preferred
      2.0K Alternate 1
      3.0K is Alternate 2
  ```

If you select 4.4.0K (preferred) and use the `make second alternate` command, the original preferred value (4.4K) becomes the alternate 2 value and the original alternate 2 value (3.0K) becomes preferred. In simple words, the status of two values is swapped.

- In case of alternate groups, you have the option of retaining the status of the selected component. If you do not want to retain the status, swapping of status takes place the way it occurs for components.

  To retain the current status, select a new alternate 2 value. This value is added to the alternate group and is assigned the `Alt1` status. The status of the existing alternate 2 component changes to `DNI`.

New Function

Creates a new function under *Functions*, with the default name as *New_Function*.

New Group

Creates a new group under *Alternate Groups*, with the default name as *New_Group*.

New Variant

Creates a new entry under *Variants*, with the default name as *New_Variant*.

Paste

Pastes the buffered information. The `paste` command is active only when you have used the `copy` command.

Remove Component

Removes the selected component from the list.

Remove Function

Removes the selected function from the top-right pane and places all components in the bottom-right pane.

Replace Component

Replaces a component (which is not a part of an alternate group) added to a function or a variant with another component.

For example, assume you have a component `R1` (`10K`) and this component is included in the function `RES`. If you choose the `Replace Component` command on `R1`, Variant Editor displays the Replace Variant Component dialog box. You use this dialog box to choose a component to replace `R1`. If you choose `C1` to replace `R1`, a new PPT row appears with the Refdes as `*`, `and` status as `Pref`. The row corresponding to the component `R1` will display - as its status, signifying its a dummy DNI.

Restore

Restores the values of alternates on components to those that are generically defined from the Alternates tab.

Unlock

If a component or alternate group is locked within a function, then choosing `Unlock` removes the lock. An unlocked component or alternate group cannot be customized. It always carries the generic alternates defined for it.

7

Variant Editor Dialog Boxes

Variant Details

You can display the Variant Details dialog box by using the *Edit – New Variant* command in the Variant tabbed window. This dialog box also appears when you right-click an existing variant and choose *Edit/Rename* from the pop-up menu. Use this dialog to create a new variant, or edit or rename an existing variant. The properties you specify or modify include the associated property, property value, and the Do Not Install (DNI) status.

The Variant Details dialog box consists of the following fields:

Variant Name Specify a name for the variant that needs to be created or renamed. Note that version 16.6 QIR 6 onwards, variant names can support the forward slash character.

Variant Property Name Displays the name of the property that is annotated to all the modified components in the variant. This helps you to easily identify the modified components by searching for the property name. By default, this field displays the name of the property as `VARIANT`. You can change the default name in the `site.cpm` file.

Variant Property Value Displays the value of the property that is annotated to all the modified components in the variant. This helps you to easily locate the modified components on schematic, by searching for the property value. By default, this field displays the value of the property as `*`. You can change the default value in the `site.cpm` file.

DNI Value Displays the DNI status assigned to the variant. A component is said to have the DNI status when it is not installed in a particular variant. The default value is `DO_NOT_INSTALL`.

Note: If you assign a `DNI` status to a component in a specific variant, the component is not installed in that variant.

Custom Variable Variant-Specific Value A user-defined custom variable in the schematic canvas appears in this list. You can specify a variant-specific value to the custom variable. The value of the custom variable on the canvas is substituted with the value you specify here.

For more information, see Managing Variants on page 43.

Global Find

You can display the Global Find dialog box by using the *Tools – Global Find* command. The Global Find dialog box allows you to find components, alternate groups, or functions in the variant database (`variant.dat` file). You can customize the search to locate specific results. For example, you can find all components which begin with the letter `C` and which are located within functions only.

The Global Find dialog box is divided in the following group boxes:

Search What box - You can define the search parameters in this box. The box contains the following fields:

Search What - Choose whether you want to search component, alternate group, or function from the `Search What` list box. If you are searching components, then you can search by logical part names or by reference designators. For example, you can search for all capacitors with the `refdes C1`. The search result will return all alternates of the capacitor `C1`.

If you search for an alternate group or a function, you can only search by reference designators.

Part Name - Enter the logical part name here. This field is available only when you have selected `Component` in the *Search What* field. Since alternate groups or functions do not have any part names, the *Part Name* field is not displayed for alternate groups or functions.

By default, the *Part Name* field has the value `*` indicating that all components are to be searched.

Refdes - Enter the name of the component, alternate group, or function. You can specify full name or use wildcard operators `?` or `*` to search for components. For example, `U*` will return all components starting with the letter `U`. Similarly, `J?2` will return components such as `J12`, `J22`, `J32` and so on.

Options - Invokes the Options dialog box, which helps you to scope the search within functions or variants. You can also exclude component alternates, or unlocked components or groups from the search.

Find Now - Searches for the result and displays the search result in the *Results* list.

Results list - This list displays the search results. It lists the following information: `Refdes`, `Status`, `Alt_group`, `Function`, `Variant`, `Part_name`, `Part_number`, `Drawing`, and `Cds_location`.

At the bottom of the Global Find dialog box are four buttons:

Navigate - Choose a row in the *Results* list and click on the *Navigate* button to select the corresponding component, alternate group, or function in the top-right pane of Variant Editor.

Clear All - Choose this button to clear the previous search results.

Close - Choose this button to close the Global Find dialog box.

Help - Choose this button to display help about the options in the Global Find dialog box.

Global Find

You can display the Global Find dialog box by using the *Tools – Global Find* command. The Global Find dialog box allows you to find components, alternate groups, or functions in the variant database (`variant.dat` file). You can customize the search to locate specific results. For example, you can find all components which begin with the letter `C` and which are located within functions only.

The Global Find dialog box is divided in the following group boxes:

Search What box - You can define the search parameters in this box. The box contains the following fields:

Search What - Choose whether you want to search component, alternate group, or function from the `Search What` list box. If you are searching components, then you can search by logical part names or by reference designators. For example, you can search for all capacitors with the `refdes C1`. The search result will return all alternates of the capacitor `C1`.

If you search for an alternate group or a function, you can only search by reference designators.

Part Name - Enter the logical part name here. This field is available only when you have selected `Component` in the *Search What* field. Since alternate groups or functions do not have any part names, the *Part Name* field is not displayed for alternate groups or functions.

By default, the *Part Name* field has the value `*` indicating that all components are to be searched.

Refdes - Enter the name of the component, alternate group, or function. You can specify full name or use wildcard operators `?` or `*` to search for components. For example, `U*` will return all components starting with the letter `U`. Similarly, `J?2` will return components such as `J12`, `J22`, `J32` and so on.

Options - Invokes the Options dialog box, which helps you to scope the search within functions or variants. You can also exclude component alternates, or unlocked components or groups from the search.

Find Now - Searches for the result and displays the search result in the *Results* list.

Results list - This list displays the search results. It lists the following information: `Refdes`, `Status`, `Alt_group`, `Function`, `Variant`, `Part_name`, `Part_number`, `Drawing`, and `Cds_location`.

At the bottom of the Global Find dialog box are four buttons:

Navigate - Choose a row in the *Results* list and click on the *Navigate* button to select the corresponding component, alternate group, or function in the top-right pane of Variant Editor.

Clear All - Choose this button to clear the previous search results.

Close - Choose this button to close the Global Find dialog box.

Help - Choose this button to display help about the options in the Global Find dialog box.

Options

When you click the *Options* button in the Global Find dialog box, the Options dialog box appears. The Options dialog box is used to scope the search within functions or variants. You can also exclude component alternates, or unlocked components or groups from the search.

The Options dialog box is divided in the following group boxes:

Search Scope group box - You can define whether you want to search components and alternate groups only in functions or only in variants, or search them in both functions and variants.The group box contains the following options:

Only Functions - Searches for components or alternate groups only in functions and not in variants. The search also returns all customizing for generic components and alternate groups. This option is not valid if you are searching functions in the variant database.

Only Variants - Searches for components or alternate groups, or functions only in variants. The search also returns all customizing for generic components and alternate groups.

`All` - Searches for components or alternate groups, or functions throughout the variant database. The search results returned include results from alternates, alternate groups, functions, and variants.

Note: If you have changed the status of any component or alternate group in any function or variant to unlocked, then that component or alternate group will not appear in the search results of the Global Find dialog box. Only locked components that have any customizing are returned as search result in the Global Find dialog box.

Search Filters group box - You can define whether you want to exclude component alternates, or unlocked components or groups from the search. The group box contains the following options

Exclude Unlocked Components/Groups - By default, this check box is selected indicating that the unlocked components and alternate groups will not be searched. Since you cannot delete unlocked components or alternate groups, you may use this check box to decide whether you want to view those components. If you want to view or remove unlocked components or alternate groups, clear the *Exclude Unlocked Components/Groups* check box.

Include Component Alternates - By default, this check box is selected indicating that the search result will return alternates of components. If you do not want to display alternates of components, you can clear the *Include Component Alternates* check box.

At the bottom of the Options dialog box are three buttons:

Reset All - Choose this button to reset all options to default value.

OK - Choose this button to save the changed values. When you click *OK*, the Options dialog box closes and you are returned to the Global Find dialog box.

Cancel - Choose this button to close the Options dialog box and return to the Global Find dialog box.

Part Table Filter

When you choose the `Change Value` command for a component in Variant Editor, the Part Table Filter dialog box displays. This dialog box is used to include or change physical information of a component from a Physical Part Table (PPT) on the schematic. The Part Table Filter dialog box lets you choose a physical component based on the logical component you selected in the Variant Editor.

The Part Table Filter dialog box contains the following fields:

Part Names - The *Part Name* box displays the part name of the selected component.

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 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 the 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.

OK - Choose this button to confirm the changes and close the Physical Part Filter dialog box.

Cancel - Choose this button to cancel the changes and close the Physical Part Filter dialog box.

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 the settings for annotating the physical properties on the schematic.

The Property Options dialog box contains the following fields:

Property Order box - 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 that has a VOLTAGE value of 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. If you choose Invisible, physical properties are not added on the schematic. If you choose Name, only the names of the physical properties are annotated. If you choose Value, only the values of physical properties are annotated. If you choose Invisible, the physical properties are added on the schematic and are read by all tools, but they 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, then150 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.

OK - Choose this button to apply the changes on the Physical Part Filter dialog box and close the Physical Options dialog box.

Apply - Choose this button to apply the changes on the Part Filter dialog box.

Cancel - Choose this button to close the Physical Options dialog box without saving the changes.

Option Sets>> - Choose the *Option 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 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.

The *Option Sets* part of the Property Options dialog box has the following options:

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.

Backannotation

You can display the Backannotation dialog box by using the *Tools – Annotate Variants* command. The Backannotation dialog box is used to backannotate the changes to the base schematic or variants.

Caution

For backannotation to work in Variant Editor, the concepthdl.scr file must be present at the following location: <your_inst_dir>/`share/cdssetup/concept/concepthdl.scr`***. If you have set the*** `CDS_SITE` ***environment variable to, say /hm/common, ensure that the*** `concepthdl.scr` ***file is installed at /hm/common/***`cdssetup/concept/ directory.`

The Backannotation dialog box contains the following options:

What To Annotate group box - This group allows you to choose the base schematic or a variant to be backannotated. The group box contains the following options:

Base Schematic radio button - Choose the *Base Schematic* radio button to annotate the *Property Name/Value* (as specified in the *Component Properties* group box) on all the components that have any variant property on it.

Variant radio button - Choose the *Variant* radio button to annotate the changes made for a particular variant. If there are more than one variant, you can choose a variant from the *Variant* drop-down list. Once this radio button is selected, the name, value, and DNI value of the variant cannot be edited. Therefore, these options are greyed out in the dialog box.

Note: Backannotation of a variant causes (1) annotation of the *Property Name/Value* (as specified in the *Component Properties* group box) on all the components that have a preferred value different from the base value or are `DNI` in that variant, and (2) updating of the key properties on all components that have a preferred value different from the base value.

Component Properties group box - This group box allows you to customize the property name, property value, and `DNI` value that are to be annotated to the annotation view. The Component Properties group box includes the following fields:

Name - Define the property name in the *Name* field. The default property name is `VARIANT`.

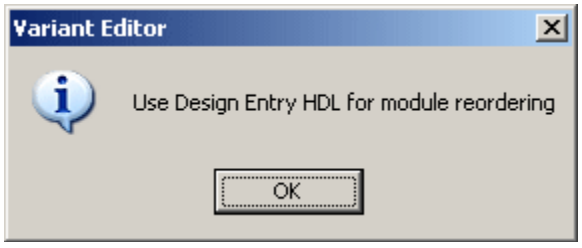
Value - Define the property value in the *Value* field. The default property value is `*`.

DNI Value - Define the value for a component with the `DNI` status in the *DNI Value* field. The default DNI value is `DNI`.

Update Original Schematic View - Choose this check box to update the variant properties on the original schematic. The *Update Original Schematic View* check box is deactivated when you choose any variant for backannotation.

Update Schematic View - Choose this check box to create the view where the variant properties should be annotated. The name of the new view has two parts: name and version. If you are annotating the base schematic, `base` is assigned as the default name of the new view. If you are annotating a variant, the name of the variant is assigned as the default name of the new view. By default, the version assigned to the new view is `1`.

Module Ordering - When you click this button, you are prompted to reorder modules in Design Entry HDL.



Annotate - Choose this button to generate the *<variant name>*.ba file. If the *Update Original Schematic View* check box is selected, the properties (as specified in the Component Properties group) are annotated to the annotation view.

Close - Choose this button to close the Backannotation dialog box.

Reset All - Choose this button to reset all options in the Backannotation dialog box to the original default selection.

Customize Columns

When you choose the *Customize Columns* command in the *View* menu of Variant Editor, the Customize Columns dialog box displays. The Customize Columns dialog box is used to customize the display of properties in the right panes of Variant Editor.

The Customize Columns dialog box is organized in two list boxes:

Available Column Names - This list box includes the list of all properties that can be displayed in the right panes of Variant Editor. However to display any of these properties, you need to move it in the *Displayed Column Names* list box.

Displayed Column Names - This list box includes the list of properties that are displayed in the right panes of Variant Editor. You can exchange properties between the two list boxes. For example, the ACCESS_TIME property is included in the *Available Column Names* list box. To display this property in the right panes of Variant Editor, you can move it to the *Displayed Column Names* list box.

To move any property from the *Available Column Names* list box to the *Displayed Column Names* list box, choose that property and click on the *>>* button.

To move back any property from the *Displayed Column Names* list box to the *Available Column Names* list box, choose that property and click on the *<<* button.

Note: You can choose multiple properties and perform simultaneous operations on them.

To move up or down the display of any property, choose that property and choose *Up* or *Down* buttons. The property that you move up in display is listed first.

Apply - After you have made changes in the display of properties, choose *Apply* to save those changes.

Close - Choose this button to close the dialog box.

Conflict Found While Loading Design

The Conflict Found While Loading Design dialog box displays when Variant Editor finds that there exists differences between the schematic and the variant database. These differences can occur if you have made any changes to the schematic after creating the variant database.

The Conflict Found While Loading Design dialog box lets you use three different ways in which you can resolve conflicts between the schematic and the variant database.

Do Not Synchronize - Choose this radio button to ignore the conflicts between the schematic and the variant database. Variant Editor will load the variant database without resolving the conflicts and the previous variant customizing information on that component is lost.

Winning on Refdes match - Choose this radio button to inherit all schematic data for a component if its `Refdes` matches with any component with similar `Refdes` in the variant database.

Winning on Canonical Path match (Recommended) - This radio button is the recommended selection and is selected by default. Variant Editor matches the canonical paths for all components in the variant database and in the schematic. If the canonical path matches, the information in the schematic is inherited.

After making your selection, choose the *OK* button to close the dialog box.

Customize Variant Settings

The Customize Variant Settings dialog box appears when you select *Tools > Options* in Variant Editor. You can use this dialog box to rename the status designators for the preferred component, set the sorting style, and suppress messages for compatible Jedec type matches.

The Customize Variant Settings dialog box includes the following options:

Rename Status Preferred Component - This field displays the default status designator of the preferred component as pref. To rename the status designator to some other value, enter the new value in the Rename Status Preferred Component field. For example, you can set the status designator of the preferred component to Current by typing Current in the field.

When you change the status designator for the preferred component, the top-right pane of Variant Editor displays the changed status.

Sorting - You can change the sorting style from the default style, which is alphanumeric, to alphabetic.

To change the sorting style to alphabetic, choose the Alphabetic radio button. In alphabetic sorting, Variant Editor sorts the terms A1, A12, and A2 as A1, A12, and A2.

To change the sorting style to alphanumeric, choose the Alphanumeric radio button. In alphabetic sorting, Variant Editor sorts the terms A1, A12, and A2 as A1, A2, and A12.

Suppress Messages Compatible Jedec Type Match - Choose this check box to suppress the display of warning messages for compatible Jedec type matches.

Variant Editor Product Choices

The Variant Editor Product Choices dialog box appears when you select *File – Change Suite* in Variant Editor. The Variant Editor Product Choices dialog box helps you specify the product suite you want to use in a session. A product suites allows you to access components that are not available in the current product suite. The product suites available for use are displayed in the list.

1. Click the product suite you want to use.
2. Select the *Use As Default* check box if you want to invoke the selected product suite every time you invoke Variant Editor.
3. Click *OK*.

The product suites and their components are listed below:

Allegro Design Entry HDL XL

The components of this product suite are

- Allegro Design Entry HDL
- Allegro Design Entry HDL Rules Checker
- Allegro Design Entry HDL Reuse
- Design Variants
- Allegro Physical Viewer
- Design Access Bundle

- Base Verilog Model Library
- Extended Verilog Model Library
- PE Librarian

A

Variant Editor Glossary

Alternate 1

The alternate 1 value is the first alternate value of the component that can be used in place of the preferred value. By default, the alternate 1 value has priority over the alternate 2 value.

Alternate 2

An alternate 2 value is the second alternate value of the component that can be used in place of the preferred value and the alternate 1 value.

Alternate Group

An alternate group is a set of functionally equivalent components that have different footprints, out of which only one is included in a particular variant. The reason for having multiple components on the schematic is that each component has a different board footprint; therefore, not allowing basic plug compatibility. The bare printed circuit board has placeholders for all components in the alternate group.

Alternate Value

An alternate value is a value that is used as a replacement to the preferred value. An alternate value maps to the same footprint as that of the component on the base schematic.

Associated Mechanical Parts

Associated mechanical parts are mechanical parts such as screws, nuts, and bolts that are associated with another physical device such as resistor or microprocessor.

Base Design

The base design is the union of all foreseen parts and connections that are required for any given variant.

Callouts

Callouts are parts included in the bill of materials that have no graphical representation in the schematic. Generally callouts are mechanical parts that are to be added to the BOM.

DNI

DNI is an acronym for Do Not Install. A component is said to have the status as DNI when it is not installed in a particular variant.

Function

A function is a set of components that form a feature or a logical function. This set of components can be included in a variant or excluded from the variant as a whole.

Preferred Value

The preferred value of a component is the value that has priority over its alternate values. By default, the value of the component placed on the base schematic is the `preferred` value. You can change the preferred value to alternate 1 or alternate 2.

Variant

A variant is a variation of the base design created to generate a separate product. All variants share a common base printed circuit board.

B

Reference Information

This appendix lists general miscellaneous information.

Design Entry - Bubbling Up

In Variant Editor when you delete alternate values or components in an alternate group, their status changes or is *bubbled up*. Simply put, if you delete a preferred or an alternate value of a component, the next alternate value takes its place.

For example, consider you have a component with the following values:

```
R1      10K Pref
        12K Alt1
        16K Alt2
        24K Alt3
```

Deleting the `Pref` row results in:

```
R1      10K  -
        12K Pref
        16K Alt1
        24K Alt2
```

Deleting the `Alt2` row in the original example results in:

```
R1      10K Pref
        12K Alt1
        24K Alt2
```

Deleting the `Alt3` row in the original example results in:

```
R1      10K Pref
        12K Alt1
        16K Alt2
```

genVariant

The `genvariant` command is used to backannotate the variant information to the base schematic or create a new flattened view for the design on which the variant information is annotated.

You enter the `genvariant` command from the Design Entry HDL console.

Syntax

```
GenVariant $VariantName
```

Or

```
GenVariant FILE $FileName
```

Where:

$*VariantName* = The name of a variant (.ba) file located in the `variant` view of the top design.

$*FileName* = Complete path to the file containing variant information. To use a relative path, specify a path relative to the `.cpm` file location.

setup.loc File

The setup.loc file contains the default search locations for a file. This file is found in the following locations:

- *<cds_site>*/cdssetup/setup.loc
- *<your_inst_dir>*/share/cdssetup/setup.loc

If the `setup.loc` file is defined in the *<cds_site>* directory, then that file is used else the `setup.loc` file in the *<your_inst_dir>* directory is used.

The search order defined in the *<your_inst_dir>*/share/cdssetup/setup.loc file is based on CSF Search.

CSF Search

The CSF search is used to load a file based on the following logic:

.	Current working directory
$CDS_WORKAREA	User workarea, if defined
$CDS_SEARCHDIR	Area set up by various tools during tool startup
$HOME	
$CDS_PROJRCT	Project storage area, ignored if not defined
$CDS_SITE	Site setup information (Typically this is <$*your_inst_dir*>/share/local)
$your_inst_dir/share	Cadence default setup Information

Note: The CSF search returns the first instance of the file it finds.

Variant Editor Shortcut Keys

You can perform many operations in Variant Editor using shortcut keys.

The following table lists the shortcut keys and the operations they complete.

Shortcut Key	Description
Ctrl+N	Creates a new variant database, named `unnamed.dat`.
Ctrl+O	Opens the variant database that you have selected.
Ctrl+S	Saves the current variant information into the variant database.
Ctrl+E	Exports the variant database into an interface file, which can be read by PCB Editor.
Ctrl+R	Loads the current design again.
Ctrl+Z	Ignores the changes performed in the last operation.
Ctrl+Y	Repeats the last operation.
Ctrl+C	Copies the information contained in a function, or variant into a buffer.

Shortcut Key	Description
Ctrl+V	Pastes the current information of the variant or function that has been selected for copy.
Ctrl+G	Highlights the component corresponding to the selected row in the Design Entry HDL schematic.
Del	Deletes groups, functions, and variants, or removes components from groups, functions, and variants.
F1	Displays the Help file.
F2	Adds a new group, function, or variant.
F5	Resets any filters that might be applied to the selected components.
F12	Saves the variant information into the specified file in the variant database. The operation is same as the *File > Save As* command.

C

Variant Editor FAQs

This appendix lists answers to frequently asked questions (FAQ) about Variant Editor.

- When should I use an alternate, an alternate group, and a function in a variant?
- How is the Change Value command different from the Add Alternates command?
- What is the difference between deleting a component and removing a component?
- How do I display the properties of a modified component in a variant on the schematic canvas?
- How can I update Design Entry HDL with the variant information added to a component in Variant Editor?
- Can I make changes to the base schematic in Design Entry HDL after defining variant information in Variant Editor?
- Can I search a component for the variant information attached to it?
- Can I generate a BOM report from Variant Editor?
- How do I modify the default comparison BOM report to include alternate rows and user-defined properties?
- How do I take variant information to PCB Editor?
- Can I support the logical Part Information Manager in Variant Editor?
- How do I control the sorting of reference designators in a BOM report?
- How do I make all projects in a site have a customized preferred status name?
- How can I view a list of all the variants in a design?

When should I use an alternate, an alternate group, and a function in a variant?

Based on the type of variation you need in a variant, you can use alternates, alternate groups, or functions. The three most common types of variations along with information on how to create those variations is listed as follows:

1. **Value change**—This type of variation involves a value change in a component's properties. For example, you can have the same component with different properties in different designs. This affects the BOM but not the physical layout. To specify a change of value in your design, use alternates.
2. **Footprint variation**—This type of variation involves the use of a mutually exclusive component set. For example, you can have a group of functionally equivalent components that have different footprints, out of which only one component is included in a particular design. There can be multiple components on the schematic so that is that each component has a different board footprint. As a result, there can be basic plug incompatibility. The bare printed circuit board has placeholders for all components in the alternate groups. To create footprint variations, use alternate groups.
3. **Creating a logical feature set**—This type of variation involves the use of functions. For example, you can have a set of components that form a feature or a logical function. Later, you can include or exclude this set from a variant as a whole: that is, you either add all the components defined in the set in the variant or add none of those components in the variant. To create such a logical component set, use functions.

How is the Change Value command different from the Add Alternates command?

By default, the value of the component on the base schematic is considered as the preferred value. You can perform two operations for this value:

1. **Change Value**—The `Change Value` command launches Physical Part Browser and allows you to change the alternate or the preferred value for the selected component. For example, assume that you are searching for the resistor `R4` and the search list returned eight PPT rows for `R4`, where each row represents an alternate value. You can choose any of these rows and perform the change value operation. The selected row will be assigned the changed values.
2. **Add Alternates**—The `Add Alternates` command adds alternates to the selected components. You need to choose an alternate value for the selected component. If only the preferred value exists for a component, the new value is added with the `Alt1` status. If an alternate with the `Alt1` status exists, the new value is added with the `Alt2` status. You can define up to 99 alternates for a component.

The primary difference between the `Change Value` and `Add Alternates` commands is that the `Change Value` command changes the value of the selected component while the `Add Alternates` command creates a new PPT row for the selected component. When you include a component in a variant, the component with the status `Pref` is included. Therefore, if you have multiple alternates and you want to use one of them in a design, you need to make that alternate the preferred one using the `Make Preferred` command.

Another difference between the `Change Value` and `Add Alternates` commands is that you can use the `Add Alternates` command on components in alternates, alternate groups, functions, or variants while you can only use the `Add Alternates` command only on components in the Alternates tab.

What is the difference between deleting a component and removing a component?

When you delete a component, you remove one instance of 'customization' on that component while when you remove a component, you remove all customization from that component. To understand this concept in detail, assume that you have the following customization for the component `R1`:

```
R1  4.4K Pref
    2.0K Alt 1
    3.0K is Alt 2
```

By default, Variant Editor displays only the `Refdes` of all available components in the top-right pane. For the component example here, you will see only `R1` in the top-right pane. If you now select the component and right-click on it to display the available commands, you will see the `Explore`, `Change Value`, and `Remove` commands. If you choose `Remove`, `R1` will be removed from the top-right pane and all customization on it will be lost.

Now assume that you choose `Explore`. This displays the properties for all the alternates of `R1`. If you now right-click on any PPT row, say `R1 4.4K Pref`, to display all available commands, the `Delete` command is available but the `Remove` command is not available. If you now select `Delete`, the PPT row corresponding to the `R1` component would be deleted and the status of the other row would change in a bubble up manner, that is, the row with the `Alt1` status will become `Pref` and the row with the `Alt2` status will become `Alt1`. For more information about bubbling up, see Design Entry - Bubbling Up on page 115.

How do I display the properties of a modified component in a variant on the schematic canvas?

When you modify a component in a variant, additional properties might be added to the schematic because of changes in the selected part. Because these properties might not have placeholders on the symbol, they are not displayed on the schematic canvas. If you want these properties to be displayed on the canvas, you can use the `VAR_OVERLAY_PROPS_VISIBLE` directive in the `START_CONCEPTHDL` section of the `.cpm` file to specify which properties should be displayed on the canvas.

You can define three values for this directive:

- NONE - only variant properties, that is, `VARIANT = *`, will be visible on the schematic
- ALL - all the properties of the modified component that have changed compared to the base instance will be overlaid on the schematic in the variant view. These properties will be displayed in the color defined for changed properties. You can configure the changed property color using the Variant Specific Property option in the Design Entry HDL Options dialog.
- In the third case, you can define the properties (for example, Part Number) that you want displayed on the schematic as follows:

  ```
  VAR_OVERLAY_PROPS_VISIBLE 'Property1' 'Property2' 'Property3'
  ```

  In this case, the schematic will display Property1, Property2, Property3 in the changed property color in the variant view for the variant component.

  Note that the variant properties, that is, `VARIANT=*` will always be visible on the variant component in the schematic in the variant view.

How can I update Design Entry HDL with the variant information added to a component in Variant Editor?

You can annotate the changes you make in a component in Variant Editor to Design Entry HDL. When you annotate properties to the base schematic, components in the base schematic that have variant information are assigned a property denoting that the components have been assigned variant information.

You can annotate variant information in the following ways:

- Annotate variant properties to the base schematic in Variant Editor launched from Design Entry HDL

When you annotate variant properties to the base schematic in Variant Editor launched from DE-HDL, DE-HDL simply overlays the variant information on the base schematic. With this feature, you can easily view various variant views by dynamically switching between views on the schematic canvas.

Unlike in releases prior to 16.6-2015, Variant Editor does not create variant-specific files on disk or a flattened view of the schematic. This saves space on disk. Further, because a flattened view is not created, and variant information is simply overlaid on the schematic, you can view occurrence-specific and Cross Referencer data in plots, and publish any variant to a PDF where you view all the variant details.

Important

Since you can annotate properties to the schematic simply by switching variant views, in Variant Editor launched from DE-HDL, the Annotate Variants option is disabled (grayed out). This option is enabled only if you launch Variant Editor in the standalone mode.

See Annotating Variant Properties in Design Entry HDL.

- Using the standalone Variant Editor, you have two options:
  - Annotate properties to a variant

    When you annotate properties to a variant, every component in the variant whose value has changed from the base schematic value, or has the `DNI` status, is assigned a new property. In addition, the changed property values are updated on the components.

    In this case, a new view for the base schematic is created on disk with the following naming convention: `sch<variant_name>_1`

    Make changes to the original base schematic view, that is `sch_1`. On annotation, Variant Editor creates a new flattened schematic view of the design with the following naming convention: `schbase_1`.

See Annotating Variant Properties to the Base Schematic Using Standalone Variant Editor.

Annotation will update properties to all components in Design Entry HDL that have the following:

- Change in property value from the base schematic
- Do not install status

See Annotating Variant Information for details on annotating variant information.

Can I make changes to the base schematic in Design Entry HDL after defining variant information in Variant Editor?

Variant Editor is a post-packaging tool—that is, it is best used when you have made changes to the schematic and packaged it. However, in real-life situations, you often need to make changes to the base schematic after you have created variant information. You can use the synchronization feature in Variant Editor to synchronize the base schematic and the variant database. Synchronization preserves the sanctity of the data between the schematic and the variant database and minimizes the disordering or loss of the earlier variant database.

Note: For more information on how Variant Editor synchronizes the variant database and the schematic, see Synchronizing the Variant Database and the Schematic on page 61.

Can I search a component for the variant information attached to it?

You can use the Global Find dialog box in Variant Editor to view all the variant information on a component or alternate group in all functions and variants. To access the Global Find dialog box, choose Tools > Global Find.

Note: For more information on how to use Variant Editor to search all customizing on a component, see Global Searching on page 56.

Can I generate a BOM report from Variant Editor?

You can generate a BOM report from Variant Editor in three formats:

1. **Base schematic BOM**—This report contains a list 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.

   To generate the base schematic BOM report, do the following:

   a. Choose *Tools > Generate Reports* in Variant Editor.

   b. Specify the template file path, the output file path, and the report format in BOM-HDL dialog box.You can retain the default selection.

   c. Click the *Generate* button.

2. **Variant BOM**—This report contains a list of all the components used in a particular variant. All the property values, including the part number, correspond to the values chosen in the particular variant.

   To generate the Variant BOM report, do the following:

a. Choose *Tools > Generate Reports* in Variant Editor.

b. Specify the template file path, the output file path, and the report format in BOM-HDL dialog box.You can retain the default selection.

c. Click the *Variant BOM* button to expand variant options.

d. Enter the path to the variant field in the *Variant File* field.

e. Click the *Variant BOM* radio button and select the name of the variant.

f. Click the *Generate* button.

3. **Part-number based comparison BOM**—This 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.

   To generate the part-number based comparison BOM report, do the following:

   a. Choose *Tools > Generate Reports* in Variant Editor.

   b. Specify the template file path, the output file path, and the report format in BOM-HDL dialog box.You can retain the default selection.

   c. Click the *Variant BOM* button to expand the variant options.

   d. Enter the path to the variant field in the *Variant File* field.

   e. Click the *Variant Comparison BOM* radio button and select the name of the variant.

   f. Click the *Generate* button.

How do I modify the default comparison BOM report to include alternate rows and user-defined properties?

By default, 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.

You can modify the comparison BOM report format according to your requirements. For example, you can include user-defined properties for each changed component in each variant of a design comparison BOM report.

To modify the default comparison BOM report, do the following:

1. In the `START_BOMHDL...END_BOMHDL` section of the `.cpm` file, specify the following directive:

   `VAR_COMP_BOM_PROPS`

   Note: When this directive is `ON`, the report generated also automatically includes alternate rows, as well as rows for the preferred values of components.

2. Specify the names of the properties that you want displayed in the report. In this example, we have `'NATION' 'COLOR'`.
3. Choose *Tools — Generate Reports* in Variant Editor.
4. Specify the template file path, the output file path, and the report format in BOM-HDL dialog box. You can retain the default selection.
5. Click the *Variant BOM* button to expand the variant options.
6. Enter the path to the variant field in the Variant File field.
7. Click the *Variant Comparison BOM* radio button and select the name of the variant.
8. Click the *Generate* button.

   The comparison BOM report is displayed.

```
TITLE:                  Bill of Materials
DATE:                   10/30/2017
DESIGN:                 super_design
TEMPLATE:               //v17-2-250/share/cdssetup/template.bom
CALLOUT:                bom.callouts

 Ref      Base           DEMO_VARIANT
 Des
          Part Number    Part Number  Var Status  NATION  COLOR
======    ============   ========================================
  R1      CDN0005-01     CDN0005-01   Pref*       ?       ?
  U1      ?              ?            DNI         ?       ?

Common Components List :
 Ref      Part Number
 Des
======    ============
  R2      CDN0005-01
  R3      CDN0005-05
```

How do I take variant information to PCB Editor?

Variant Editor allows you to export the variant information to an interface file that PCB Editor can use to generate BOM reports or assembly drawings for individual variants. To create the PCB Editor interface file, choose File — Export.

Can I support the logical Part Information Manager in Variant Editor?

Yes, 16.6 QIR 7 onwards, the logical Component Browser is supported in Variant Editor. When you call the PPT browser from Variant Editor, it now opens the Replace Component dialog box, which displays all the libraries and components in your design.

The Replace Component dialog box is essentially the Component Browser, which lets you search and select library parts defined in your design. You can search for parts, view details of parts including symbols and footprint, and replace parts.

How do I control the sorting of reference designators in a BOM report?

When you generate a BOM report using BOM-HDL, the report is sorted on the property that is listed first in the *Reports Column* section of the Customize Template — Physical Part Specifications tab.

By default, the first property in the *Report Columns* section 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 such as `BOM_INST` (which displays reference designators) as the key property, move it to the first row in the *Property* column.

Note: 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.

To move the `BOM_INST` property to the first row, do the following:

1. Select the property by clicking on the row corresponding to it.
2. Click on the *Up* button to move the property to the top.
3. Save the template file.
4. Generate the BOM report.

The BOM report is sorted by reference designators.

How do I make all projects in a site have a customized preferred status name?

To ensure that all projects in a site have a customized preferred status name, choose *Tools — Options* in Variant Editor.

The Options dialog box appears. Enter the new status designator for the preferred component. For example, if you want the status of the preferred component to be `Stat`, enter `Stat` in the *Rename Status Preferred Component* field and click *OK*.

How can I view a list of all the variants in a design?

You can either view the `<variant name>.ba` files in your variant folder, or check the `variant.dat` file for all the variant names. The names following this format: `VAR_DEF_<variant name>`

D

BOM-HDL FAQ

This appendix 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?
- Is there any mechanism by which the report viewing dialog box can be suppressed in the nographic mode?
- How can I import BOM report in MS Excel?
- 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.

Is there any mechanism by which the report viewing dialog box can be suppressed in the nographic mode?

In PSD 14.0, 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
```

Note: In PSD 14.2, BOM-HDL has enhanced the `nographic` mode support and it does not display any dialog box while completing its operation.

How can I import BOM report in MS Excel?

To import a BOM report into MS Excel, perform the following steps:

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 `BOM.rpt` report is generated, open MS Excel 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 MS Excel.

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.

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

Index

Symbols

A

B

C

D

F