

Design Variance Tutorial

Product Version 23.1
September 2023

© 1991–2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

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

Allegro® Design Entry HDL contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

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

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

All other trademarks are the property of their respective holders.

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

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

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

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

Contents

1

<u>Introduction</u>	9
<u>Objectives</u>	9
<u>Nature of Chapter</u>	9
<u>Estimated Completion Time</u>	9
<u>Purpose</u>	10
<u>Audience</u>	10
<u>Prerequisites</u>	10
<u>How to Use This Tutorial</u>	11
<u>Multimedia Demonstrations</u>	11
<u>Summary</u>	12
<u>What's Next</u>	12
<u>Recommended Reading</u>	12

2

<u>Getting Started</u>	13
<u>Objectives</u>	13
<u>Nature of Chapter</u>	13
<u>Estimated Completion Time</u>	13
<u>Copying the Tutorial Database</u>	14
<u>Understanding the Database</u>	14
<u>Understanding the Design</u>	15
<u>Defining Variations in Base Design</u>	16
<u>Frequently Used Terms in This Tutorial</u>	17
<u>Summary</u>	19
<u>What's Next</u>	19
<u>Recommended Reading</u>	19

3

<u>Creating the Variant Database</u>	21
<u>Objectives</u>	21
<u>Nature of Chapter</u>	21
<u>Estimated Completion Time</u>	21
<u>Overview</u>	22
<u>Becoming Familiar with the Design</u>	23
<u>Opening Variant Editor</u>	23
<u>Customizing the Display</u>	26
<u>Assigning Alternate Values</u>	29
<u>Need for Alternate Values</u>	29
<u>Task Overview</u>	29
<u>Steps</u>	29
<u>Defining Alternate Groups</u>	32
<u>Need for Alternate Groups</u>	32
<u>Task Overview</u>	33
<u>Steps</u>	33
<u>Creating Functions</u>	34
<u>Need for Functions</u>	34
<u>Task Overview</u>	34
<u>Steps</u>	34
<u>Creating a Variant</u>	36
<u>Need for a Variant</u>	36
<u>Task Overview</u>	36
<u>Steps</u>	36
<u>Customizing the Information in a Variant</u>	39
<u>Need for Customizing Information</u>	39
<u>Task Overview</u>	39
<u>Steps</u>	39
<u>Defining a DNI Component in a Variant</u>	41
<u>Need</u>	41
<u>Task Overview</u>	41
<u>Steps</u>	42
<u>Global Searching</u>	43
<u>Need</u>	43

Design Variance Tutorial

<u>Task Overview</u>	44
<u>Steps</u>	44
<u>Synchronizing the Variant Database With the Changes in the Schematic</u>	48
<u>Need</u>	48
<u>Task Overview</u>	48
<u>Steps</u>	49
<u>Using Compatible JEDEC TYPES</u>	53
<u>Need</u>	53
<u>Task Overview</u>	53
<u>Summary</u>	56
<u>What's Next</u>	56
<u>Recommended Reading</u>	56

4

<u>Generating BOM Reports</u>	57
<u>Objectives</u>	57
<u>Nature of Chapter</u>	57
<u>Estimated Completion Time</u>	57
<u>Overview</u>	58
<u>Getting Started</u>	59
<u>Loading BOM-HDL</u>	59
<u>Creating the BOM for the Base Schematic</u>	62
<u>Task Overview</u>	62
<u>Steps</u>	62
<u>Report Explanation</u>	63
<u>Customizing the BOM Template</u>	63
<u>Overview</u>	63
<u>Displaying the Customize Template Dialog</u>	64
<u>Changing the Report Parameters</u>	65
<u>Task Overview</u>	65
<u>Steps</u>	66
<u>Report Explanation</u>	69
<u>Including Associated Mechanical Parts and Mechanical Kits</u>	72
<u>Introduction</u>	72
<u>Task Overview</u>	73

Design Variance Tutorial

<u>Steps</u>	73
<u>Explanation</u>	73
<u>Reading Mechanical Part Information in BOM Reports</u>	75
<u>Task Overview</u>	75
<u>Steps</u>	75
<u>Report Explanation</u>	76
<u>Editing the Callouts Information</u>	77
<u>Introduction</u>	77
<u>Task Overview</u>	77
<u>Steps</u>	77
<u>Setting Filters on Parts</u>	80
<u>Steps</u>	81
<u>Report Explanation</u>	83
<u>Creating a Variant BOM Report</u>	84
<u>Introduction</u>	84
<u>Task Overview</u>	84
<u>Steps</u>	84
<u>Report Explanation</u>	85
<u>Creating the Variant Comparison BOM Report</u>	88
<u>Introduction</u>	88
<u>Task Overview</u>	88
<u>Steps</u>	88
<u>Report Explanation</u>	88
<u>Summary</u>	90
<u>What's Next</u>	90
<u>Recommended Reading</u>	90

5

<u>Backannotating Variant Information</u>	91
<u>Objectives</u>	91
<u>Nature of Chapter</u>	91
<u>Estimated Completion Time</u>	91
<u>Overview</u>	92
<u>Preparing for Backannotation</u>	92
<u>Guideline</u>	92

Design Variance Tutorial

<u>Opening the Backannotation Dialog</u>	93
<u>Backannotating the Base Schematic</u>	94
<u>Task Overview</u>	94
<u>Steps</u>	94
<u>Backannotating a Variant</u>	95
<u>Task Overview</u>	95
<u>Steps</u>	95
<u>Summary</u>	98
<u>What's Next</u>	98
<u>Recommended Reading</u>	98

6

<u>Creating BOMs and Variant Assembly Drawings</u>	99
<u>Objectives</u>	99
<u>Nature of Chapter</u>	99
<u>Estimated Completion Time</u>	99
<u>Overview</u>	100
<u>Exporting the PCB Editor Interface File</u>	100
<u>Overview</u>	100
<u>Creating BOM Reports in PCB Editor</u>	100
<u>Task Overview</u>	100
<u>Steps</u>	101
<u>Report Explanation</u>	101
<u>Creating Variant Assembly Drawings</u>	103
<u>Task Overview</u>	103
<u>Prerequisite for Creating a Variant Assembly Drawing</u>	103
<u>Steps</u>	103
<u>Summary</u>	108
<u>What's Next</u>	108
<u>Recommended Reading</u>	108
<u>Appendix A</u>	109
.....	120

<u>Index</u>	121
---------------------	-----

Design Variance Tutorial

Introduction

Objectives

- Define the purpose of the Design Variance tutorial
- Identify who should use the Design Variance tutorial
- Identify the prerequisites for using the Design Variance tutorial
- Define how to use the Design Variance tutorial

Nature of Chapter

Conceptual

Estimated Completion Time

15 minutes

Purpose

This tutorial demonstrates the major features of the Design Variance solution, which is provided as part of the front-to-back flow for PCB design. The Design Variance solution lets you create and manage similar designs with small differences, such as a difference in the resistance of a resistor. Using the Design Variance solution, you can define the base design and determine the variants on that design. The base design shares a common set of core elements. The variants may differ because of targeted market segments, destination country, or feature set. You will learn to perform all these steps in this tutorial.

Audience

You will find this tutorial useful if you are a printed circuit board (PCB) designer and need to create similar designs from a base design, with only minor changes in each design with respect to the base design. For example, you may need to:

- Change the value of some components.
- Define mutually exclusive group of components with only one component or one set of components winning; that is used in a particular design.
- Remove specific components from the design.
- Add, replace or delete a set of components as associated functions.

Using Variant Editor and other tools in the Cadence PCB Design Variance solution, you can accomplish the above tasks.

Prerequisites

For implementing the Design Variance solution, you should know how to use the following:

- Design Entry HDL (the schematic editor)
- Project Manager (the project creation tool)
- PCB Editor (the layout tool)

How to Use This Tutorial

There are multiple ways in which you can use this tutorial:

1. Use it as a learning aid to create and manage design variants—This is the recommended method. In this mode, you need 8 hours to complete the tutorial. You will learn by trying all steps on a sample design.
2. Learn specific procedures— You can use the tutorial as a quick reference to learn only those procedures that you frequently perform. For example, you might need to create BOM reports. However, in this method, you may not be able to try the steps in the example exercises because the exercises are hardcoded and must be performed in the recommended series.

Multimedia Demonstrations

To quickly understand the main features of the design variance flow for Cadence PCB tools, go to the [SPB Video Library Page](#).

Summary

The Design Variance tutorial is meant for PCB designers who need to create and manage designs that are different from each other in minor ways. The best way to use this tutorial is to complete it from the beginning to the end and try all steps in the software as you proceed.

What's Next

In the next chapter, [Getting Started](#), you will learn to unzip a sample design and identify its characteristics. The sample design would be used for creating design variants. The chapter will also define the frequently-used terms in the tutorial.

Recommended Reading

You can find more information about the following products in Cadence Documentation:

- Design Entry HDL —See the [*Allegro Design Entry HDL User Guide*](#) and the [*Allegro Design Entry HDL Tutorial*](#)
- PCB Editor—See [*Allegro PCB and Package User Guide*](#)
- Project Manager—See the [*Allegro Project Manager User Guide*](#)

Getting Started

Objectives

- Copy the tutorial database to your system
- Identify the contents of the tutorial database
- Identify the different variations that will be made to base design
- Define the important terms in the tutorial

Nature of Chapter

Conceptual

Estimated Completion Time

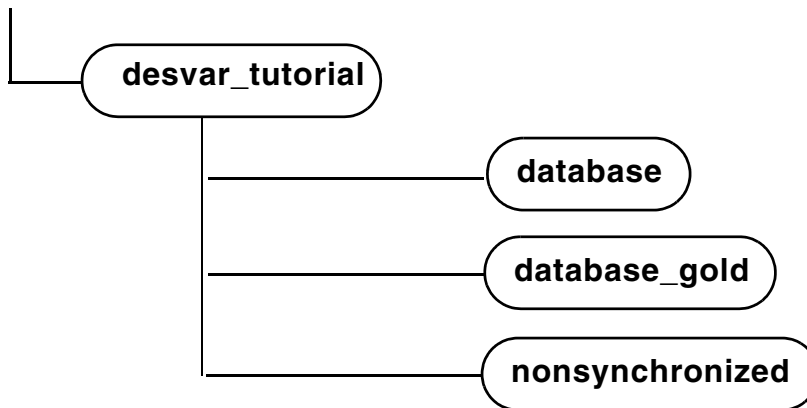
45 minutes

Copying the Tutorial Database

This tutorial provides a hands-on-exercise on managing variants. To gain the most from this tutorial, you should try out all the steps in the flow as documented in the tutorial in the database accompanying the tutorial.

You can unzip the tutorial file available at the `<your_inst_dir>/doc/vedit_tut/tutorial_examples` directory. On the Windows platform, unzip the `vedit_db.zip` file.

You will get the following structure on unzipping the tutorial database.



The `desvar_tutorial` directory consists of three directories:

1. **database**—This directory includes the base schematic and the board. The board contains the components that are used in all the variants. You will use the `dsp` design in the `database` directory to complete most procedures in this tutorial.
2. **database_gold**—This directory is a gold copy of the `database` directory.
3. **nonsynchronized**—This directory includes a design, which has some changes made to the schematic after the creation of the variant database. This directory is used to complete the schematic-variant database synchronization procedure in the tutorial.

Understanding the Database

The tutorial contains a directory called `database` that contains all the required data for creating and managing variants. The `database` directory has the following contents:

- **The top level design `dsp`**—This design serves as the base design and includes all the components needed to create the variants covered in this tutorial. The design includes

the base schematic and the board, which are present in the `sch_1` view and the *physical* views respectively, under the design `dsp` in the design library `worklib`. The design also includes two lower-level blocks: `memory` and `rom`. These blocks are also present in the design library `worklib`.

- **The component library `comp`**—This library contains all the components that are used in the design.
- **The `ptfs` directory**—This directory contains all the required physical part table (`ptf`) files.
- **The `footprints` directory**—This directory contains the Allegro symbols and padstacks.
- **The project file (`dsp.cpm`)**—This file contains the setup information for the project.

Understanding the Design

In this tutorial, you start with a design based on the `ADSP2101` 16-bit fixed-point digital signal processor (DSP) and create variants. This design is an example of the core of a DSP system. It contains only the DSP and its associated memory. All the other peripherals, such as ADCs, additional write and read ports, are not included in this example.

This base design contains the following:

- The `ADSP2101` DSP
- Boot ROM for the `ADSP2101` DSP
- External program memory
- Provision for serial port

Defining Variations in Base Design

You can introduce several variations to the base configuration. For example, you can:

- **Change the size of the Boot ROM**—A larger 32K ROM (27C256) could be used for large boot codes, spanning several boot pages while a smaller 8K ROM (CY7C263) could be used for smaller boot codes limited to one boot page.
- **Include, exclude, or change the size of the external program memory**—You may not have the external memory at all, if you think the system could work properly with the internal (on-chip) program memory itself. You could vary the speed of the external program memory as well. In the design used in this tutorial, a set of three CY7C199 components is used to create a 32K deep external memory 24-bits wide.
- **Include or exclude the serial port**—The serial port introduces another source for a possible variation. Its presence could provide a method to introduce inter-processor communication if several core modules were to be interconnected remotely. The absence of the serial port, of course, would mean that no serial communication is enabled (that is, the module is to be used as a standalone module). The serial port is available through a 9-pin connector (CONN9).
- **Change the processor speed**—You can change the processor speed to interface it with slower peripherals. This brings another source of variation. To change the processor speed, change the value of the crystal module that feeds the clock into the DSP. Two values, 5MHz and 10MHz, have been used in this tutorial.

In this tutorial, you will create the following variations:

1. **Changing components** (27C256 versus CY7C263) - This variation is an example of an Alternate Group. You will connect both components in parallel, with the choice of installing only one component in a variant. You can also customize the values of the components (27C256 or CY7C263) for different variants depending on which component and component value is required in a variant.
2. **Including or excluding a group of components** (External Program Memory (CY7C199)) - This variation is an example of a function. You will define two functions, each of which includes a set of three CY7C199 components. These functions will be included or excluded as a whole from the variant.
3. **Component value change and defining DNI**.
 - a. You will specify an Alternate Value for the 9-pin serial connector (CONN9). The alternate value defined for CONN9 will apply generically to all variants unless the information for CONN9 is customized for a variant.

- b. You will define `CONN9` as not installed in a design by assigning it a DNI status in some of the variants where it is supposed to be absent.

4. Value change of multiple components simultaneously (the crystal modules)

—You will assign one of the two values (5MHz or 10 MHz) for the crystal module in any variant. This variation is also an example of a value change.

Frequently Used Terms in This Tutorial

You will find the following terms frequently used in this tutorial:

Variant

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

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 a 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 of the component in Variant Editor. However, the changed value should map to the same footprint as that of the value in the base schematic.

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 preferred value. You can define up to 99 alternate values for a component.

Alternate Group

An alternate group is a set of functionally equivalent components that have different footprints. Only one component of an alternate group is included in a particular variant.

DNI

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

Summary

You learned how to copy the tutorial database to your system. You can use this database to create and manage variants. You also learned the composition of the tutorial database.

What's Next

In the next chapter, [Creating the Variant Database](#), you will learn to create variants by defining and manipulating alternate values for components, alternate groups, and functions. You will also learn to customize the information in a variant and define the DNI status to components.

Recommended Reading

For more information about different design variations and creating and managing variants, see [Design Variance User Guide](#).

Design Variance Tutorial

Getting Started

Creating the Variant Database

Objectives

- Identify the variations that you can create in a design
- Identify the different components in the design
- Launch Variant Editor and become familiar with its interface
- Customize the display of properties in the right pane of Variant Editor
- Assign alternate values for a component in Variant Editor
- Create an alternate group in Variant Editor
- Create functions in Variant Editor
- Create variants in Variant Editor
- Customize the information in a variant
- Define a DNI component in a variant
- Search for components with variant information using Global Find
- Synchronize the information in the schematic and the variant database
- Replace the value of components using compatible JEDEC TYPES

Nature of Chapter

Skill (includes concepts and practice)

Estimated Completion Time

3 hours

Overview

The variant database is a view called *variant*, which contains information about all the variants. 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 also backannotate the variant information to the base schematic.

Using Variant Editor, you can define variants in the following ways:

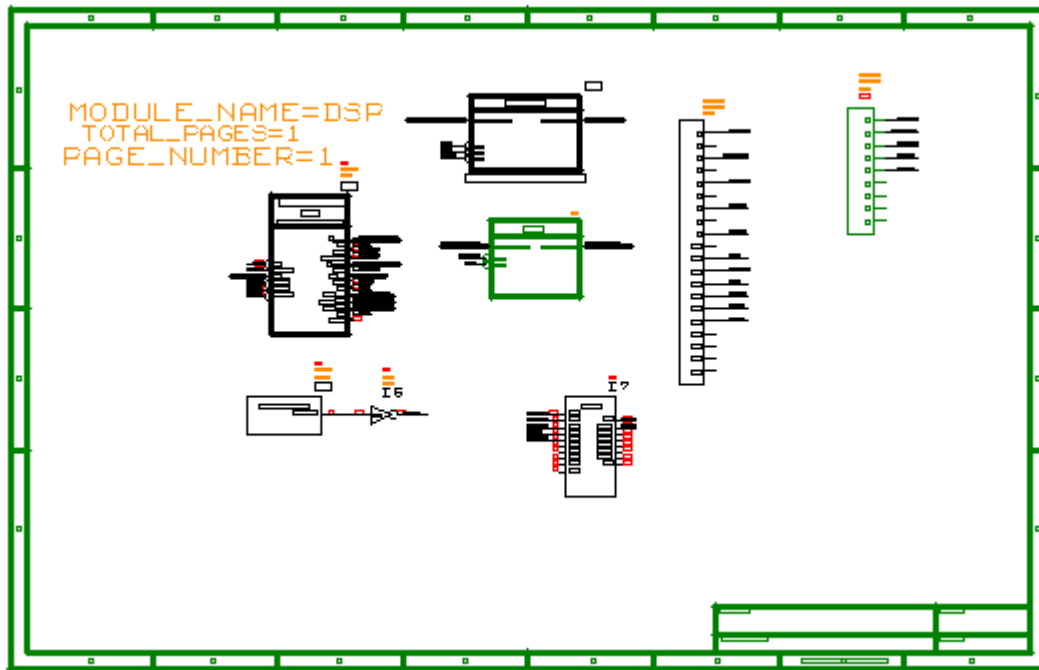
- 1. Assigning alternate values for a component** - An alternate value is a value that is used as a replacement to the preferred value of the component. You can assign a maximum of 99 alternate values for a component. In this chapter, you will assign the alternate values for the J1 (CONN9) component. You will also change the preferred value for the U1 (CONN20) component.
- 2. Assigning a Do Not Install (DNI) status to a component** - If you assign a DNI status to a component in a particular variant, then it is not installed in that variant. In this chapter, you will assign the serial port (CONN9) the DNI status in some of the variants (where its functionality is not required).
- 3. Defining alternate groups** - An alternate group is a set of functionally equivalent components that have different footprints, out of which only one component is included in a particular variant. In this chapter, you will create an alternate group named ROM consisting of the components U9 (27C256) and U10 (CY7C263-35) so that in any variant only one of these two components is installed. When you create the alternate group, one of the components will automatically become preferred and the other component will become alternate 1. You will also customize the ROM alternate group for different variants, depending on which of the two components needs to be installed for a particular variant and the required value for it.
- 4. Creating functions** - A function is a set of components that form a feature or a logical function. This set of components is either included or excluded from the variant as a whole. In this chapter, you will create two functions, MEMORY1 and MEMORY2, which will include the set of three CY7C199 components (U6, U7, and U8) from the MEMORY block. You will also change the preferred values of the components in the MEMORY2 function.

You will perform all the steps described above in the following sections. However, before you make changes to the variant database, you should become familiar with the design and the Variant Editor user interface.

Becoming Familiar with the Design

1. Open the `dsp.cpm` project under the `database` directory in Project Manager.
2. Click *Design Entry*.

The *dsp* design is loaded in Design Entry HDL.



It includes a MEMORY block, a ROM block, a VLSI IC (ADSP2101 - DSP), an oscillator clock (XTALOSC), and connectors (CONN9 and CONN20). The MEMORY block is the external program memory, which consists of three memory chips (CY7C199). The ROM block consists of two chips—27C256 and CY7C263-35.

Opening Variant Editor

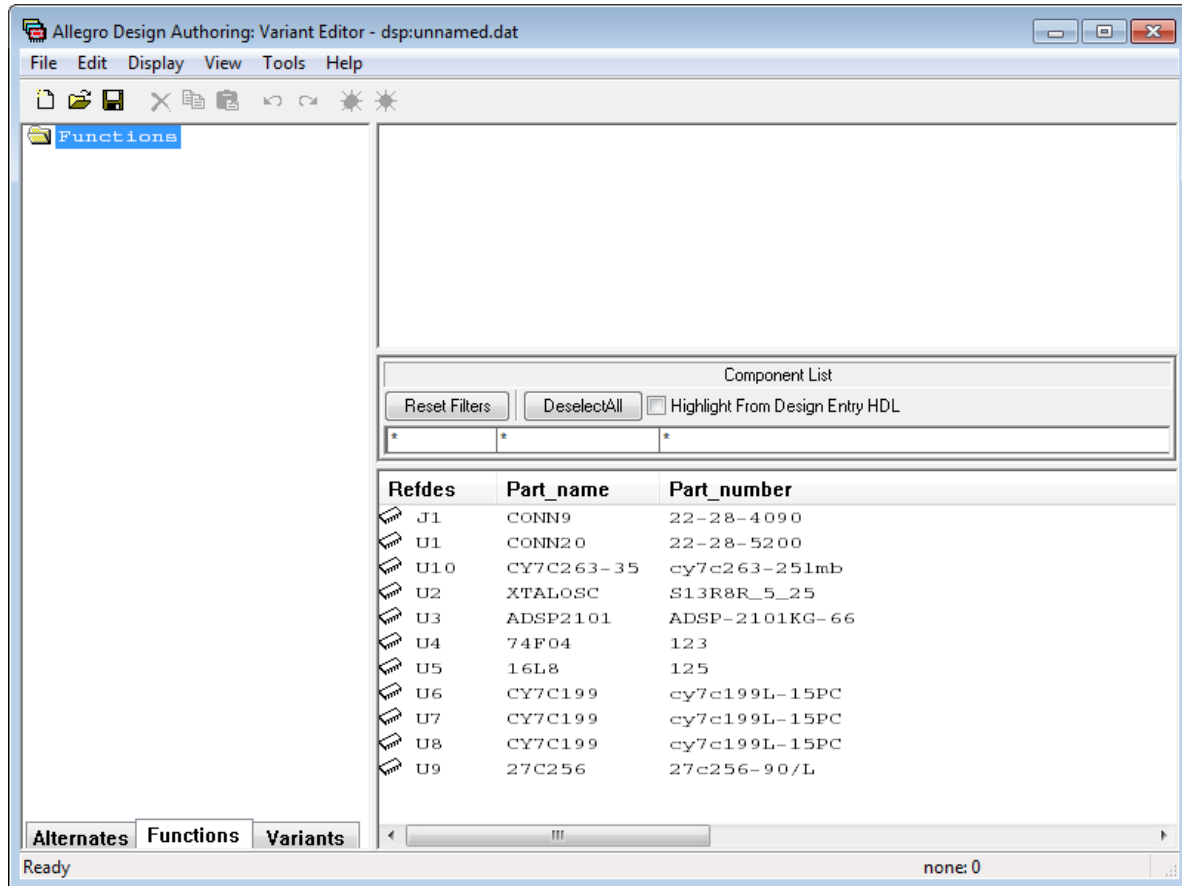
- ➔ In *Project Manager*, choose *Tools – Variant Editor*.

Variant Editor displays the *dsp* design in a three-pane view as illustrated:

Design Variance Tutorial

Creating the Variant Database

Figure 3-1 Variant Editor Main Window



Note: Now see this multimedia demonstration on [Starting Variant Editor](#).

The Variant Editor main window comprises the following parts:

Menu Bar

The menu bar includes six menus: *File*, *Edit*, *Display*, *View*, *Tools*, and *Help*.

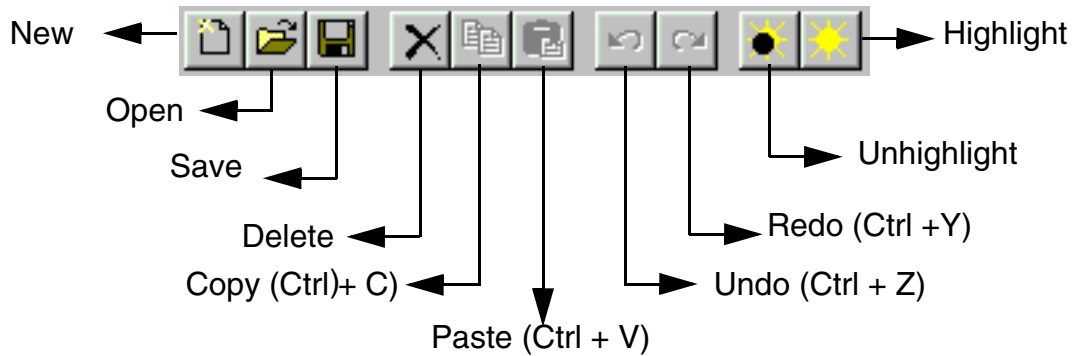
Note: For more information about the menu commands, see [Design Variance User Guide](#).

Design Variance Tutorial

Creating the Variant Database

Toolbar

The Variant Editor toolbar includes ten toolbuttons that allow you to quickly perform the most commonly used commands. The following figure describes the functions of all toolbuttons.



You need not remember the associated command with each toolbutton. If you move the mouse over any toolbutton, a tooltip displaying the name of the function is displayed. To perform the command associated with any toolbutton, click the toolbutton.

Note: All buttons have associated menu commands, and you can use the menu bar or the toolbar to perform the command. For some commands, you may even use 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 select any of these tabs: *Alternates*, *Functions*, or *Variants*. Depending on the selected tab, you can view the variant information for components, alternate groups, functions, and variants. The left pane displays a tree view. Depending 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 valid components that you can select for assigning 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, that is, these `PART_NUMBERS` retain the preferred value of the components.

Filter List

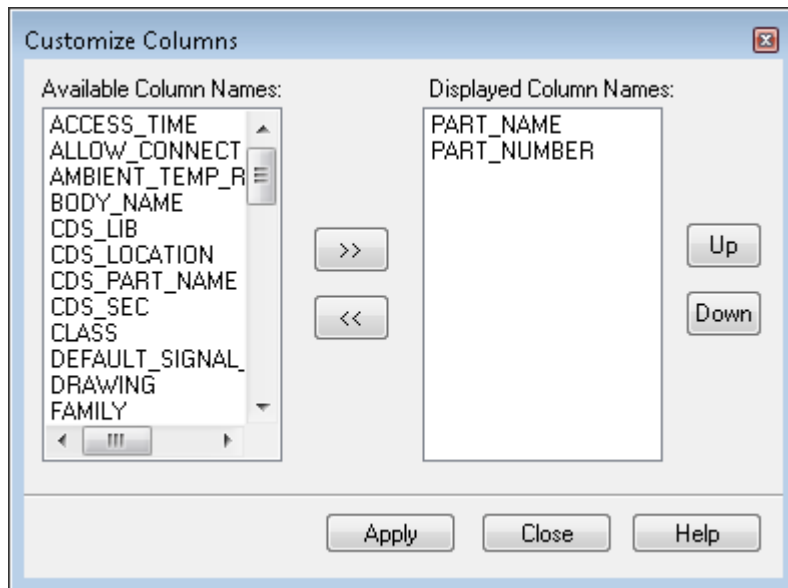
The Filter list located between the top-right and bottom-right panes. This is a fixed size frame that allows you to apply filters on the selected properties.

Customizing the Display

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.

1. Choose *View – Customize Columns*.

The *Customize Columns* dialog is displayed.



You can select the properties from the *Available Column Names* list and move them to the *Displayed Column Names* list.

1. Select the properties to display.

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

- ❑ **Individual property selection**—You can select a property by clicking its name. Next, you need to click `>>` to move the property to the *Displayed Column Names* list.

Design Variance Tutorial

Creating the Variant Database

- ☐ **Multiple properties selection (random)** — To select multiple properties, first select one property by clicking it. Next, select another property by keeping the `Ctrl` key pressed and clicking the property. Repeat this operation to select as many properties as required. After selecting the properties you want to move, click `>>`. This will transfer the selected properties to the *Displayed Column Names* list.
- ☐ **Multiple properties selection (serial)** — This procedure is applicable when you are serially selecting a number of properties. Start by selecting the first property by clicking it. Next, keeping the `Shift` key pressed, select the last property in the series. As a result of your action, all the properties beginning from the first property to the last property are selected. To transfer the selected properties to the *Displayed Column Names* list, click `>>`.

To move a property from the *Displayed Column Names* list back to the *Available Column Names* list, select the property and click `<<`.

After you have selected the properties in the *Customize Columns* dialog, you must click the *Apply* button to ensure that the changes are applied. This displays the selected properties in the right pane. The values for the properties are displayed wherever the property is applicable. If a property is not applicable for a component, no value is displayed for it.

2. Select the following properties in the *Available Column Names* list:

- ☐ ACCESS_TIME
- ☐ CLASS
- ☐ FREQUENCY
- ☐ JEDEC_TYPE
- ☐ MATING_END
- ☐ MECH_KIT1
- ☐ MECH_PART1
- ☐ PCTAIL
- ☐ PLATING_OPTION
- ☐ SPEED

Note: The property values displayed in the bottom-right pane correspond to the values selected in the base schematic.

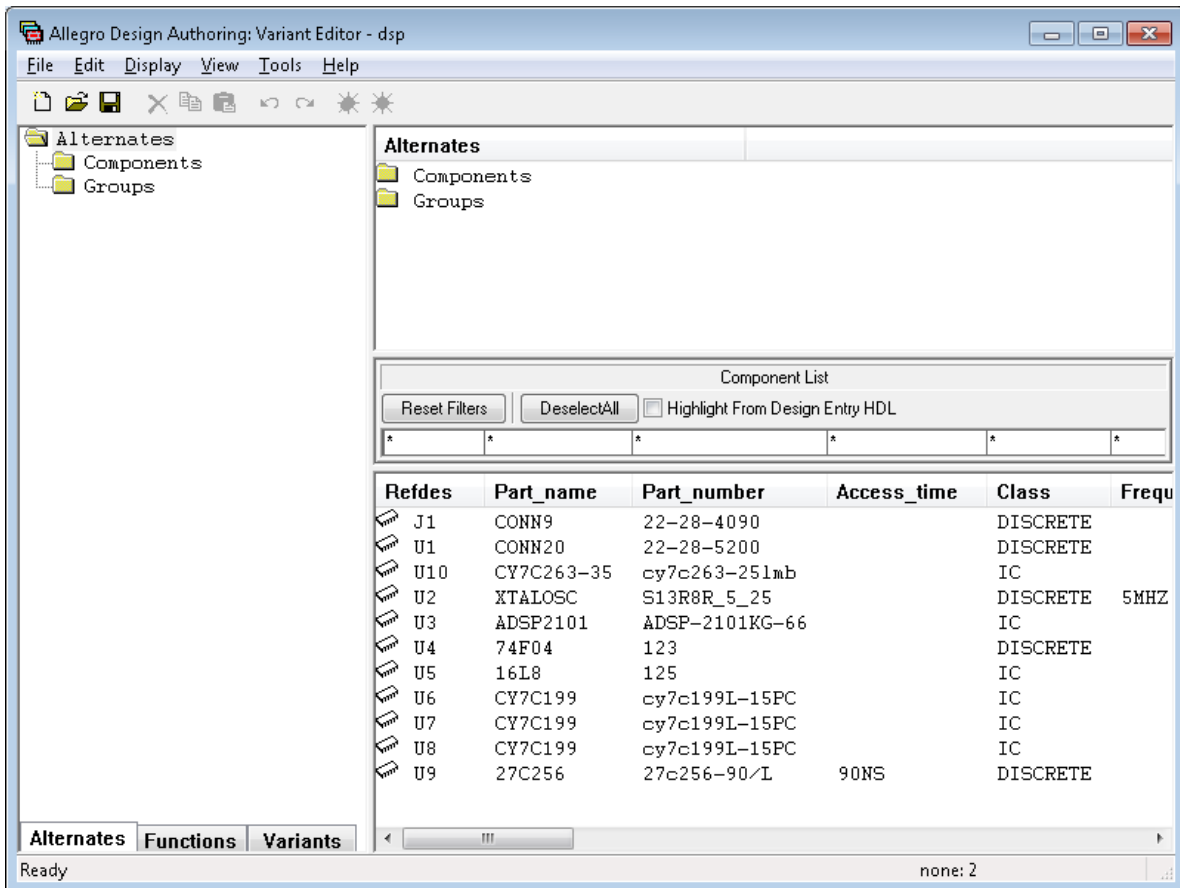
3. Click `>>` to move the selected fields to the *Displayed Column Names* list.

Design Variance Tutorial

Creating the Variant Database

4. Click *Apply*.
5. Click *Close*.

The specified fields are added to the display. Compare the following figure with [Figure 3-1](#) on page 24.



Assigning Alternate Values

Need for Alternate Values

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 as a replacement to the preferred value of the component. To define an alternate value, select the *Alternates* tab.

Note: The information assigned in the *Alternates* tab is applicable to all variants unless you a particular variant has been customized.

Task Overview

You will define alternate values for the J1 (CONN9) connector, and change the preferred value for the U1 (CONN20) connector.

Steps

1. Ensure that the *Alternates* tab and *Alternates* folder are selected in the *Variant Editor*.
2. Select the rows corresponding to the J1 and U1 components in the bottom-right pane.
3. Right-click and choose *Add to Alternate Components List*.

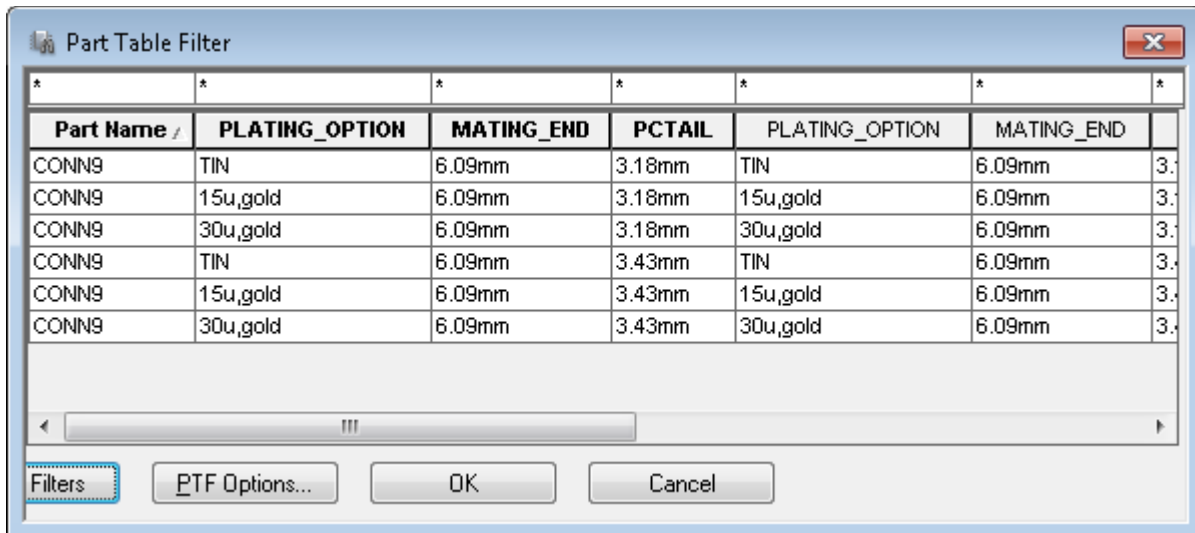
J1 and U1 are added to the left pane, and the properties of the U1 component are displayed in the top-right pane.

4. Select the J1 component in the left pane so that the row corresponding to it is displayed in the top-right pane.
5. Select the row corresponding to the J1 component in the top-right pane.
6. Right-click and choose *Add Alternates*.

The *Part Table Filter* dialog is displayed with all the PPT rows that have the JEDEC_TYPE as CONN9 for the J1 component.

Design Variance Tutorial

Creating the Variant Database



7. Select the row with the 22-28-5090 as the PART_NUMBER.

You might have to move the horizontal and vertical scroll bars to select a row or display the required property.

8. Click *OK* to confirm the row selection.

The *Part Table Filter* dialog closes, and a new row with the status *Alt1* is displayed in the top-right pane.

Refdes	Status	Part_name	Part_number	Access_time	Class	F
J1	Pref	CONN9	22-28-4090		DISCRETE	
	Alt1	CONN9	22-28-5090		DISCRETE	

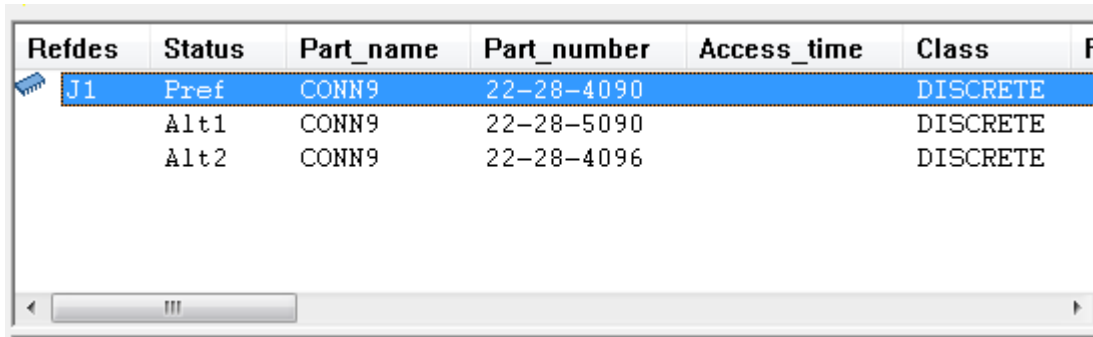
Note: You may move the properties relevant to the connector (J1), such as PCTAIL, PLATING_OPTION, CLASS, JEDEC_TYPE, up in the *Customize Columns* dialog so that they can be viewed without having to move the scroll bars.

9. Repeat steps 6 through 8 to assign the PPT row with the PART_NUMBER 22-28-4096 as the second alternate value for the J1 component.

Design Variance Tutorial

Creating the Variant Database

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



A screenshot of a software interface showing a table with the following data:

Refdes	Status	Part_name	Part_number	Access_time	Class	F
J1	Pref	CONN9	22-28-4090		DISCRETE	
	Alt1	CONN9	22-28-5090		DISCRETE	
	Alt2	CONN9	22-28-4096		DISCRETE	

10. To change the preferred value for the U1 component, select it in the left pane.

The properties for the U1 component are displayed in a row in the top-right pane.

11. Select the row.

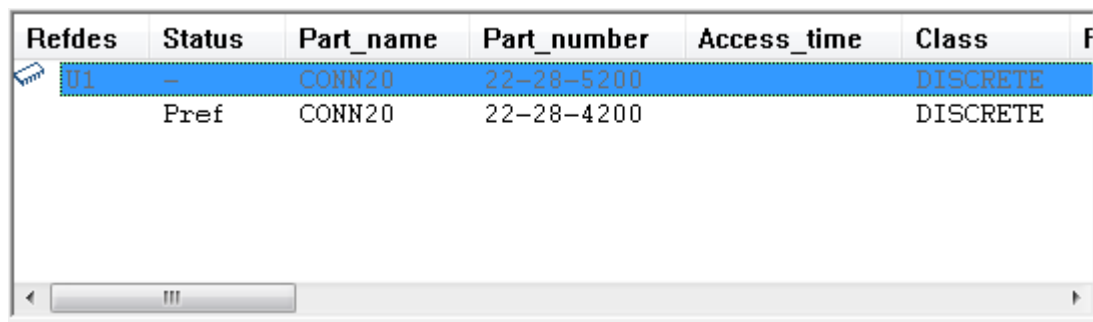
12. Right-click and choose *Change Value*.

The *Part Table Filter* dialog is displayed.

13. Select the row with the Part_Number 22-28-4200.

14. Click *OK* to confirm the row selection.

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



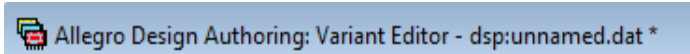
A screenshot of a software interface showing a table with the following data:

Refdes	Status	Part_name	Part_number	Access_time	Class	F
U1	-	CONN20	22-28-5200		DISCRETE	
	Pref	CONN20	22-28-4200		DISCRETE	

Design Variance Tutorial

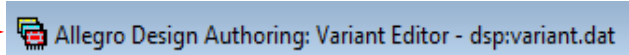
Creating the Variant Database

15. Choose *File – Save As* and save the variant database with the name `variant.dat`.



← Before saving the variant database.

After saving the variant database.



Note: You can define up to 99 alternates for a component. The procedure for creating additional alternates is the same as the procedure for creating the first and second alternate.

Note: Now see this multimedia demonstration on [*Creating Alternates*](#).

Defining Alternate Groups

Need for Alternate Groups

There are many designs that may 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, create alternate groups.

Task Overview

You will define an alternate group for the components in the ROM block (27c256 and cy7c263-35).

Steps

To create an alternate group, do the following:

1. Ensure that the *Alternates* tab is selected.

2. Right-click *Groups* and choose *New Group*.

A new group with the default name *NewGroup* is displayed.

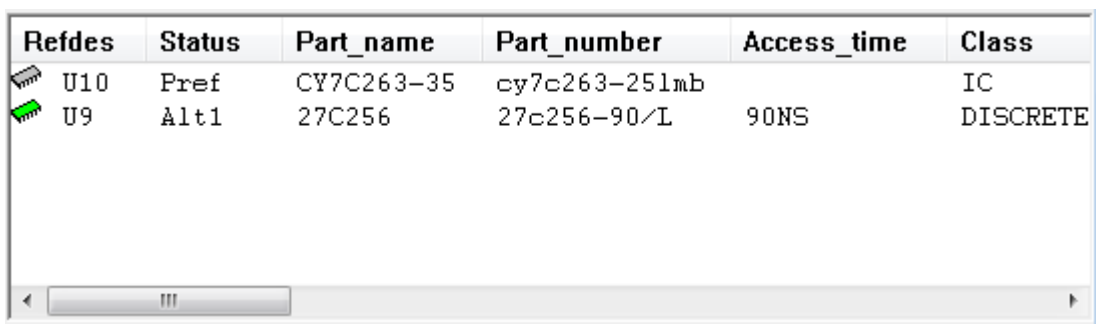
3. Rename this group to ROM.

You can also create a new group by pulling down the *Edit* menu and selecting the *New Group* option.

4. Select the rows corresponding to the U9 and the U10 components in the bottom-right pane.

5. Right-click and choose *Add to Alternate Group*.

The U9 and U10 components are added to the alternate group ROM. Note that one component is assigned the *Pref* status, while the other component is assigned the *Alt1* status.



The screenshot shows a table with the following data:

Refdes	Status	Part_name	Part_number	Access_time	Class
U10	Pref	CY7C263-35	cy7c263-251mb		IC
U9	Alt1	27C256	27c256-90/L	90NS	DISCRETE

The assigning of status (preferred, first alternate, second alternate, or third or subsequent alternates) is sequential. Variant Editor assigns the first component added to an alternate group as preferred. The subsequent components are treated as alternates. All components that are not customized in the top-right pane retain their preferred value.

You may even customize the information in an alternate group. Whatever information you change in the alternate group will be generically applied to all variants. You can, however, override this information by customizing the information for that alternate group in a particular function or variant. To learn about customizing the variant information, refer to [Customizing the Information in a Variant](#) on page 39.

Note: Now see this multimedia demonstration on [Creating Alternate Groups](#).

Creating Functions

Need for Functions

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. This means that you either add all the components defined in the function in the variant or add none of those components in the variant.

Task Overview

You will define two functions: `MEMORY1` and `MEMORY2`. `MEMORY1` contains the memory components: `U6`, `U7`, and `U8` with the values that correspond to the base schematic. `MEMORY2` contains the memory components: `U6`, `U7`, and `U8`, but with changed values.

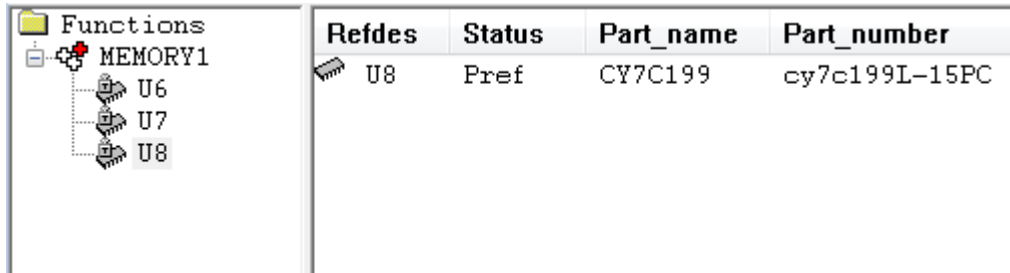
Steps

1. Choose the *Functions* tab.
2. Choose *Edit – New Function*.
A new function with the default name `NewFunction` is displayed.
3. Rename this function to `MEMORY1`.
4. Choose three components, `U6`, `U7`, and `U8`, in the bottom-right pane.
5. Right-click and choose *Add to Function*.

Design Variance Tutorial

Creating the Variant Database

The U6, U7, and U8 components are added to the MEMORY1 function.



Note: 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.

6. Create a new function MEMORY2.
7. Add the U6, U7, and U8 components to MEMORY2 by repeating steps 4 and 5.
8. Choose the MEMORY2 function in the left pane.

The reference designators of the three components are displayed in the top-right pane.

9. Choose the three components: U6, U7, and U8 in the top-right pane.
10. Right-click and choose *Change Value*.

The *Part Table Filter* dialog is displayed.

11. Choose the row with the SPEED 20ns.
12. Click *OK* to confirm the row selection.

The values in the three components are changed.

Exercise 1

1. Verify that the values of the components in the MEMORY2 function are changed.
2. To create a function which is similar to an existing function, use the copy-paste operation. Try this out by creating a function—MEMORY3—by creating a copy of the MEMORY2 function. After you create the MEMORY3 function, remove it.

When you have completed the exercise, refer to the answers to Exercise 1 in Appendix A.

Note: Now see this multimedia demonstration on [Creating a Function](#).

Creating a Variant

Need for a Variant

A variant of the base design is created to generate a separate product. To create a variant, you have to define the differences from the base design.

Task Overview

You will perform the following tasks:

1. Define three variants: INDIA, EUROPE, and USA.
2. Add the MEMORY1 function to the INDIA variant.
3. Add the MEMORY2 function to the EUROPE variant.

Steps

1. Choose the *Variants* tab.
2. Choose *Edit – New Variant*.

Design Variance Tutorial

Creating the Variant Database

The Variant Details dialog is displayed.

Custom Variables Variant Specific Values	
Name	Value

In Variant Name, specify `INDIA`.

Variant Property Name, Variant Property Value and DNI Value are displayed with default values. You can modify them, if required. For this tutorial, leave the default values unchanged.

Variant Property Name displays the associated property name that is annotated in the Design Entry HDL schematic for all the components in the base schematic that have variant information or that have the DNI status assigned to them.

Variant Property Value displays the associated property value that is annotated in the Design Entry HDL schematic for all the components in the base schematic that have

Design Variance Tutorial

Creating the Variant Database

variant information or that have the DNI status assigned to them. The value indicates that the component is part of this particular variant.

If you do not want to include a component in a variant, you can set the status of that component as Do Not Install (DNI). DNI Value displays the associated value that is annotated in the Design Entry HDL schematic for all the components in the base schematic that have variant information and that have the DNI status assigned to them.

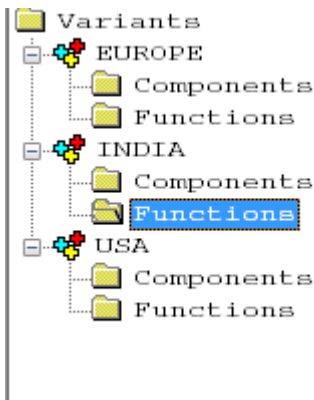
Click *OK* to close the Variant Details dialog.

3. Create two more variants: `EUROPE` and `USA`.

The information in the generic tab (the *Alternates* tab) is applicable to all the variants. It means that the alternate values defined for the `J1` (`CONN9`) component, the change in preferred value for the `U1` (`CONN20`) component, and the change in information in the ROM alternate group is applicable to all the three variants: `INDIA`, `EUROPE` and `USA`.

The `MEMORY1` and `MEMORY2` functions will have to be explicitly added to a variant, if you want the components in that function to be present in a particular variant.

4. Expand the `INDIA` variant by clicking **+** to the left of the `INDIA` folder and choose the *Functions* folder under the `INDIA` variant.



The `MEMORY1` and `MEMORY2` functions are displayed in the bottom-right pane.

5. Drag the row displaying the `MEMORY1` function from the bottom-right pane and drop it to the top-right pane.

The `MEMORY1` function is added to the `INDIA` variant, which means that all the three components (`U6`, `U7`, and `U8`) in the `MEMORY1` function are installed in the `INDIA` variant.

6. Use the procedure described in steps 4 and 5 to add the `MEMORY2` function to the `EUROPE` variant.



Exercise 2

Add both, the `MEMORY1`, and `MEMORY2` functions, to the `USA` variant and save the variant database. Undo this step.

When you have completed the exercise, refer to the answer to [Exercise 2](#) in Appendix A.

Note: Now see this multimedia demonstration on [Creating a Variant](#).

Customizing the Information in a Variant

Need for Customizing Information

Whatever information is defined in the *Alternates* tab is applicable to all variants. If you want to override this information, move the component or alternate group to the top-right pane (in the *Variants* tab) and customize its value for a particular variant. You can include or exclude individual functions from a variant.

Task Overview

You will customize the information for the `ROM` alternate group in the `EUROPE` variant by adding it to the top-right pane and then customizing the component values.

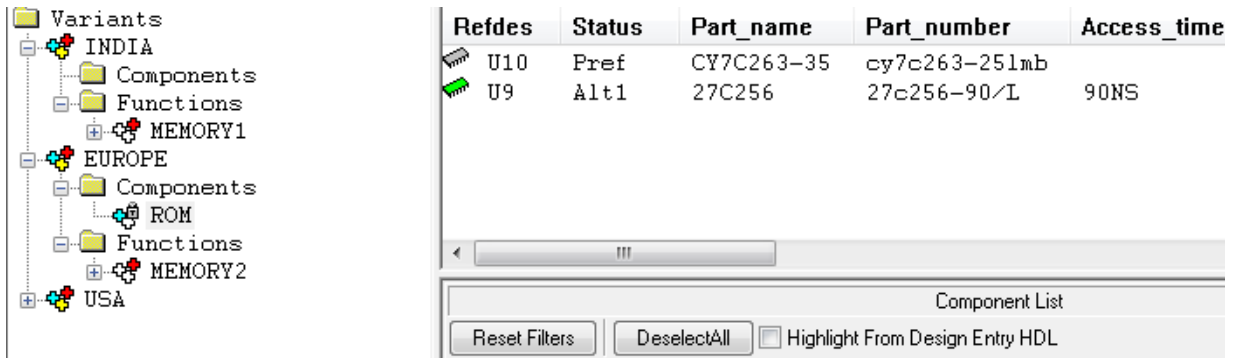
Steps

1. Choose the *Components* folder under the `EUROPE` variant.
2. Drag the row displaying the `ROM` alternate group from the bottom-right pane and drop it in the top-right pane.

Design Variance Tutorial

Creating the Variant Database

The alternate group ROM is added to the EUROPE variant. Note that the U10 component is assigned the *Pref* status and the U9 component is assigned the *Alt1* status.



3. Choose the row with the *Pref* status.
4. Right-click and choose *Make First Alternate*.

A Variant Editor message box prompts you to retain the current status of the row. Click *Yes*. When you choose *Yes*:

- ☐ The current status is retained.
- ☐ You will be allowed to add a new row for the same reference designator with a new value and with the status as *alternate1*.
- ☐ The existing row corresponding to the U9 component, which has the *alternate1* status, will become *DNI*.

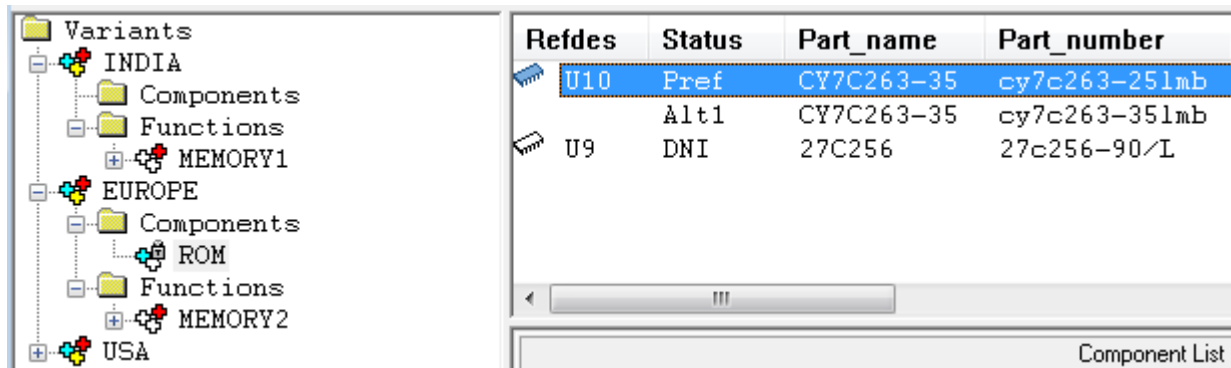
The *Part Table Filter* dialog is displayed. You can select a new value for the component.

5. Select the row whose *SPEED* is *35ns*.
6. Click *OK* to confirm the selection.

Design Variance Tutorial

Creating the Variant Database

Note that a new row is added to the U10 component with the status *Alt1* and the row corresponding to the U9 component is assigned the status *DNI*.



Exercise 3

1. Customize the alternate group ROM for the INDIA variant by specifying the U10 component as *Alt1* and the U9 component as *Pref*. What change is required in the alternate group ROM to achieve this state?
2. There is no alternate group added to the top-right pane for the USA variant. In such a case, which U9 and U10 component information will be applicable for the USA variant?

When you have completed the exercise, refer to the answers to [Exercise 3](#) in Appendix A.

Defining a DNI Component in a Variant

Need

If you do not want to include a component in a variant, you can set the status of that component as Do Not Install (*DNI*).

Task Overview

You will define the status of the J1 component in the INDIA variant as *DNI*.


Design Variance Tutorial

Creating the Variant Database

Steps

1. Choose the *Components* folder under the `INDIA` variant.
2. Drag the `J1` component from the bottom-right pane and drop it in the top-right pane.

The `J1` component is added to the `INDIA` variant with all its alternates.

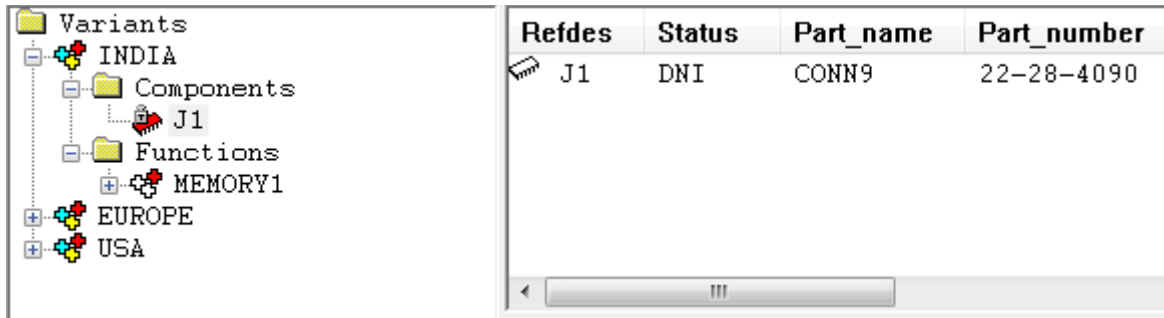
Refdes	Status	Part_name	Part_number	Access_time	Class
 J1	Pref	CONN9	22-28-4090		DISCRETE
	Alt1	CONN9	22-28-5090		DISCRETE
	Alt2	CONN9	22-28-4096		DISCRETE

3. Select any one of the three rows corresponding to the `J1` component. For example, choose the row with the `PART_NUMBER` 22-28-4096.
4. Right-click and choose *Do Not Install Component*.

Design Variance Tutorial

Creating the Variant Database

The Preferred row is displayed with the status **DNI**, while the two alternate rows are removed. As a result, the **J1** component will not be added to the **INDIA** variant. However, the **J1** component will be added to the other two variants.



The screenshot shows a tree view on the left with 'Variants' expanded, containing 'INDIA', 'Components', 'Functions', 'MEMORY1', 'EUROPE', and 'USA'. The 'Components' folder is expanded, showing 'J1'. To the right is a table with the following data:

Refdes	Status	Part_name	Part_number
J1	DNI	CONN9	22-28-4090



Exercise 4



Tip

You can perform this exercise in two ways: by using the *Change Value* command or by using the *Add Alternates* command. Try to complete the exercise both ways.

Change the **FREQUENCY** and the **FREQUENCY_STABILITY** properties for the **U2** component in the **USA** variant.

The base schematic **FREQUENCY** and the **FREQUENCY_STABILITY** property values are **5MHZ** and **25PPM**, respectively. Change these values to **10MHZ** and **50PPM**, respectively. Assign the preferred status to the component with the changed values and assign the **Alt1** status to the component with the base schematic value.

When you have completed the exercise, refer to the answer to [Exercise 4](#) in Appendix A.

Note: Now see this multimedia demonstration on [Assigning the DNI Status to a Component](#).

Global Searching

Need

If you are working on a complex design that has hundreds of components and you have customized the properties of components at multiple places, then the task of managing the design becomes complex. The same component may have different values at different places, and it may be difficult to locate a component with a particular value. To view all the

Design Variance Tutorial

Creating the Variant Database

customized changes on a component or alternate group in all functions and variants, use the Global Find feature.

You can use the Global Find feature for a range of functions, such as the following:

- 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 the values of components - You can choose specific components in the search result and change its value.
- Delete multiple components that have specific customized changes 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.

Task Overview

You will use Global Find to do the following:

- Find all customized changes on the `J1` component.
- Restrict the Global Find results to list customized changes only on components with the status `Preferred`.
- Delete the rows corresponding to the `J1` component that have the status `Alt1` and `Alt2`.
- Navigate to the `J1` component that has the status `DNI`.

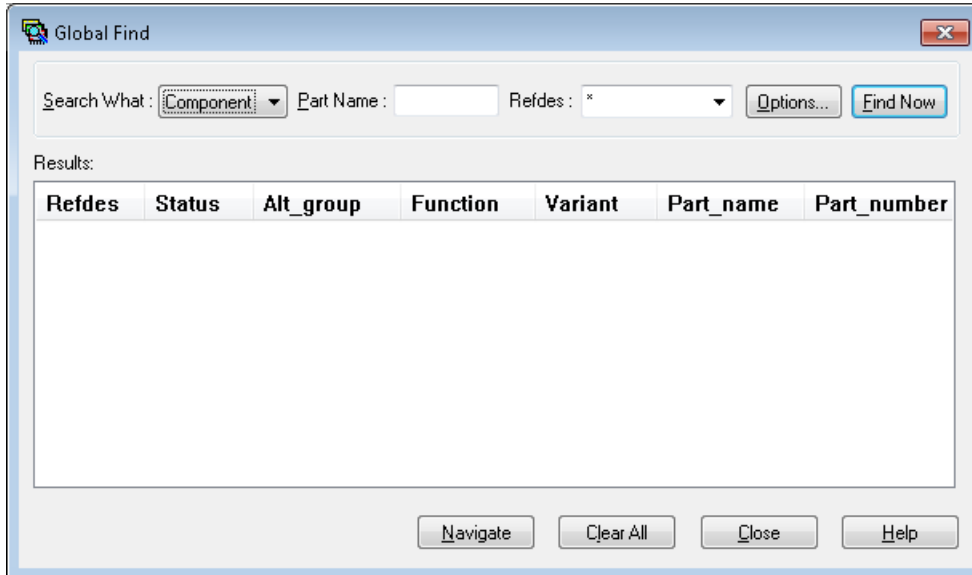
Steps

1. Choose *Tools – Global Find*.

Design Variance Tutorial

Creating the Variant Database

The *Global Find* dialog appears.



Note that the *Search What* field displays *Component*. This signifies that you are searching for components. You can change this selection to *Group* or *Function* depending upon whether you are searching for alternate groups or functions. In the current exercise, you are searching for the J1 component, so leave the selection in the *Search What* field as it is.

You can search for components by specifying a specific part name or reference designator in the *Search What* field.

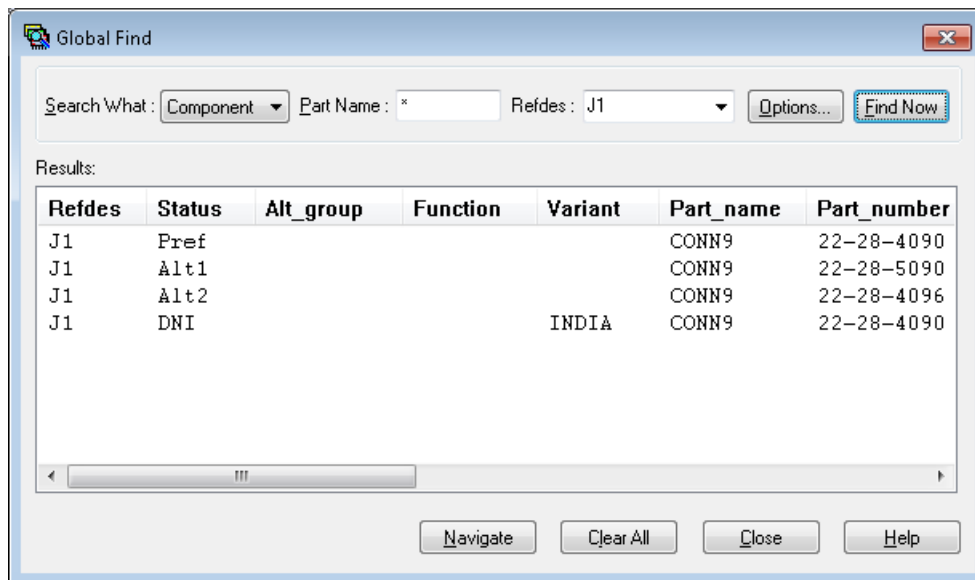
You will now search for the components with the reference designator J1.

2. Type J1 in the *Refdes* field and click *Find Now*.

Design Variance Tutorial

Creating the Variant Database

The *Global Find* dialog returns four rows corresponding to the J1 component. The component has been customized in the *Alternates* tab and for the INDIA variant.



If you have entered a reference designator in the *Refdes* field, Variant Editor remembers that value. You may access a previously-entered reference designator value by using the list button to the right of the *Refdes* field.

The *Global Find* dialog 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 include all properties displayed in the right panes of Variant Editor and any other property that has been customized for any component.

To restrict the scope of Global Find, use the *Options* dialog.

3. Click *Options* to display the *Options* dialog.

Use this dialog to restrict the search results to components in the *Alternates* tab and to components from functions only, variants only, or both functions and variants.

4. Uncheck the *Include Component Alternates* box to restrict the search to only those J1 components that have the preferred status.

5. Click *OK*.

The *Options* dialog closes, and the *Global Find* dialog is displayed.

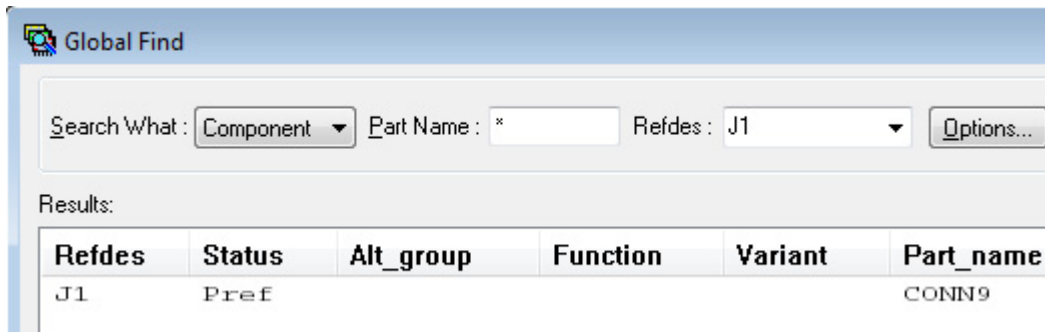
You have set the search options for the new search.

6. Click *Find Now* to conduct a new search.

Design Variance Tutorial

Creating the Variant Database

The search is now limited. Only one row corresponding to the J1 component is displayed.



7. Choose *Clear All* to clear all search results in the *Global Find* dialog.

To understand the operations that you can perform on the results in the *Global Find* dialog, enlarge the scope of Global Find.

8. Choose the *Include Component Alternates* check box again in the *Options* dialog and search again.

This step will display the same results as step 2. You may also revert to standard search options by clicking the *Reset All* button in the *Options* dialog.

There are two rows corresponding to the J1 component, which have the status *Alt1* or *Alt2*. You will now delete these rows simultaneously.

9. Select the row corresponding to the J1 component with the status *Alt1*. To select a row, click the reference designator value. Now keeping the *Shift* key pressed, select the J1 component with the status *Alt2*.

10. Right-click and choose *Delete*.

Click *Yes* in the confirmation message box. The rows corresponding to the J1 component with the status *Alt1* or *Alt2* are deleted.

Note: You cannot delete a row that has *DNI* as the status.

You can quickly navigate to a component in the Global Find results. To navigate to the J1 component that has the status *DNI*, select the row corresponding to the J1 component with the status *DNI*. To select the row, select the *Refdes* value of the J1 component.

11. Click *Navigate*.

The J1 component in the *INDIA* variant is selected in the left pane and all properties corresponding to it are displayed in the top-right pane.

12. Click *Close* to close the Global Find dialog.



Exercise 5

1. Use the Global Find feature to find all customized changes for the components in the alternate group ROM in the design.
2. Find all customized changes for the `CONN20` component.
3. Use Variant Editor to undo all the changes (the deletion of the rows corresponding to the J1 component with the status `Alt1` or `Alt2`) made using the *Global Find* dialog. After undoing the changes, save the variant database because it will be used in later sections.

When you have completed the exercise, refer to the answer to [Exercise 5](#) in Appendix A.

Close Variant Editor. This variant database will be used in the next section. Close Design Entry HDL and Project Manager if these tools are still open.

Synchronizing the Variant Database With the Changes in the Schematic

Need

Variant Editor can detect changes between the variant database and the original schematic. You can synchronize the variant database and the schematic, if needed. You can also retain the differences between the variant database and the changed schematic. In such cases, you must explicitly choose **not** to synchronize the variant database and the schematic because the default option in Variant Editor is to synchronize the variant database and the schematic based on the winning canonical path.

Task Overview

The `nonsynchronized` database under the `desvar_tutorial` directory contains an example design that has changes that were made in the schematic after the creation of the variant database. These changes caused the schematic and the variant database to be out of sync. In this section, you will learn to synchronize the schematic and the variant database.

Understanding the changes in the schematic

The changes made in the schematic are:

- The instance I10 (CONN9) is swapped with the instance I8 (CONN20).

This change was made by changing the CONN9 component's PATH property from I10 to I8, and the CONN20 component's PATH property from I8 to I10.

- The LOCATION property of the XTALOSC component is changed to U20.

This change will effectively change the reference designator for the component.

After these changes were made, the design was packaged using the *Design Sync – Export Physical* command from Project Manager.



Variant Editor defines variants using the data in the packaged view, which is created or updated only when you package the design. Therefore, if you change the schematic, ensure that you package the schematic.

Steps

1. Open the dsp project from the nonsynchronized folder in Project Manager.
2. Open Design Entry HDL and verify the following:
 - ❑ The changed canonical path of the CONN9 and CONN20 components
 - ❑ The reference designator of the XTALOSC component is U20

To accomplish these tasks, you need to:

- a. Use the search feature to find the occurrences of the CONN9 and CONN20 components.

The full canonical path of the two components will be returned.

- b. To see that this path is different from the dsp design in the database directory, use the same procedure to check the canonical path of the ADSP2101 and XTALOSC components in that design.

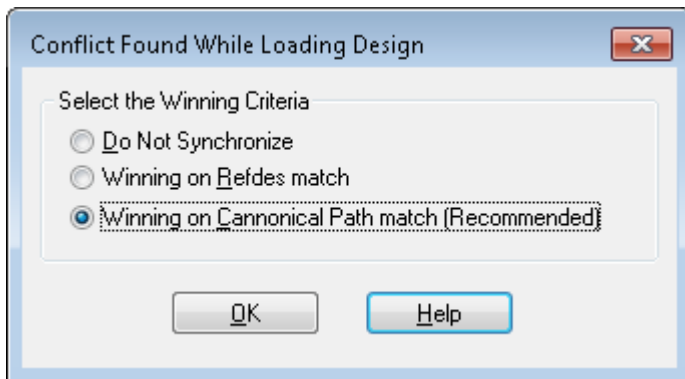
You can also use the Attributes form in Design Entry HDL to verify that the LOCATION property of the XTALOSC component is U20.

Design Variance Tutorial

Creating the Variant Database

3. Open Variant Editor by selecting *Tools – Variant Editor* command from Project Manager.

The *Conflict Found While Loading Design* box displays.



This box prompts you to define the winning criteria for synchronizing the variant database and the schematic. Note that the default winning criteria is *Winning on Canonical Path match (Recommended)*.

4. Accept the default *Winning on Canonical Path match (Recommended)* option.
5. Click *OK* to synchronize the variant database and the schematic.

Variant Editor displays a message about errors/warnings that were detected during importing/loading the variant database.

6. Click *OK* to read the actual error or warning.

The *Error/Warning messages* information box displays two warning messages (Warning 04). To read the warning, expand the size of the dialog. Just move the mouse pointer to any side of the dialog. A double arrow line will display. Click and drag the line to resize the dialog to the required size.

Warning 04 reads: Cannot merge the variant properties of variant instance J1, component with same canonical path not present in design. A similar warning for the variant instance U1 is also displayed.

7. Click *OK* to close the error/warning messages information box.







Understanding the Synchronization Results

The variant instance for U20 is synchronized. Note that in the lower-right pane of the *Alternates* tab, the U2 instance is replaced with U20. Also note the results for the U2 component, which was customized for the USA variant. Variant Editor has retained the

Design Variance Tutorial

Creating the Variant Database

customizing information for this component, although it has changed the reference designator from U2 to U20.

	Refdes	Jedec_type	Part_name	Part_number	A..	Class	Frequency
	J1	CONN9	CONN9	22-28-4090		MECH	
	U1	CONN20	CONN20	22-28-5200		MECH	
	U20	OSC_DIP4	XTALOSC	S13R8R_5_25		IC	5MHZ
	U3	PGA69	ADSP2101	ADSP-2101KG-66		IC	
	U4	DIP14_3	74F04	123		IC	
	U5	SOIC20W	16L8	125		IC	

The U1 and J1 variant instances are not synchronized. This means that all customized changes for the U1 and J1 variant instances are lost. Note the change in the *Alternates* tab. If you think that this change is the change that you wanted, then you may save the variant database. For the purpose of this tutorial, do not save the variant database.

Exploring Other Synchronization Options

Consider the design described in the previous section. If you are not satisfied with the Canonical Path match synchronization, you could decide not to synchronize the variant database.

1. Without saving the variant database, choose *File – Reload Design*.

Click *No* if prompted to save the variant database. The *Conflict Found While Loading Design* dialog is displayed.

2. Choose *Do Not Synchronize*.

3. Click *OK* to confirm your selection.

Variant Editor displays a message about errors/warnings that were detected during importing/loading the variant database.

4. Click *OK* to read the error or warning.

The *Error/Warning messages* information box displays three warning messages (Warning 04). These messages are for the J1, U1, and U2 components. Variant Editor is unable to synchronize any of these instances.

5. Click *OK* to close the information box.

All customized changes for the J1, U1, and U2 components are removed and these components are returned to the lower-right pane in all three tabs: *Alternates*, *Functions*, and *Variants*.

Note: For a detailed description of how Variant Editor synchronizes the variant database and the changed schematic, see Design Variance User Guide.



Exercise 6

Perform Refdes match synchronization on the variant database and check the results.

When you have completed the exercise, refer to the answer to Exercise 6 in Appendix A.

Note: Now see this multimedia demonstration on *Synchronizing the Variant Database and the Schematic*.

Using Compatible JEDEC_TYPES

Need

To replace the values of two components, for example, component A with component B, you need to ensure that both components, A, and B, have the same footprint. The `JEDEC_TYPE` property defines the footprint to be used in the PCB Editor design for the component in the logical netlist, so when replacing values of components, ensure that the components have the same `JEDEC_TYPE`.

This is a limitation if you need to replace the value of one component with the value of another component that does not have the same `JEDEC_TYPE`. Variant Editor overcomes the limitation by allowing support for compatible `JEDEC_TYPES`. You can define compatible `JEDEC_TYPES` by specifying them in a file named `cjedectype.txt` in a directory named `cdssetup`. The `cdssetup` directory is located at the same level as the project file. The use of compatible `JEDEC_TYPES` also helps if you are adding alternates rows for a component by selecting PPT rows of components with compatible `JEDEC_TYPES`.

Task Overview

You will first define the `DIP14_3` and `DIP28_3` `JEDEC_TYPES` as compatible. Next, you will change the values of the `U4` component, which has PPT rows using the `DIP14_3` and `DIP28_3` `JEDEC_TYPES`.

Editing the `cjedectype.txt` File



Tip

To create the `cjedectype.txt` file, copy the `cjedectype.txt` file located at `<your_install_dir>/share/cdssetup` and paste it under the `cdssetup` directory, which is at the same level as the project file.

For this tutorial, a copy of the `cjedectype.txt` file is already copied to the `dsp` design in the `/nonsynchronized/cdssetup` directory.

1. Open this file in a text editor.

The first few lines in the file describe the format for defining compatible `JEDEC_TYPES`. The format is simple. List all compatible `JEDEC_TYPES` in a row, separate each `JEDEC_TYPE` with a space, and end the row with a semi-comma(,).

Design Variance Tutorial

Creating the Variant Database

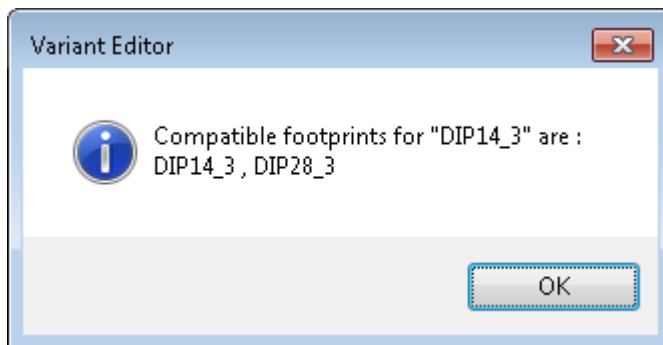
2. To define DIP14_3 and DIP28_3 as compatible JEDEC_TYPES, type DIP14_3 DIP28_3 ; and save the file.
3. To ensure that Variant Editor reads the changed information in the `cjedectype.txt` file:
 - a. Close Variant Editor without saving the design.
 - b. Open Variant Editor again using the *Winning on Canonical Path match (Recommended)* option.

Click *OK* in the *Error/Warning messages* message box then *OK* in the *Error/Warning messages* information box.

Replacing Rows With Compatible JEDEC_TYPES

1. In the *Alternates* tab in Variant Editor, drag the U4 component (the one with the JEDEC_TYPE DIP14_3) from the lower, bottom-right pane to the top-right pane of the and select it.
2. Right-click and choose *Change Value*.

A message box displays the message that the compatible footprints for DIP14_3 are DIP14_3, DIP28_3.



3. Click *OK* to close the message box.

The *Part Table Filter* dialog is displayed. Note that the *Filters* field for the JEDEC_TYPE column displays the value *. This value signifies that all compatible JEDEC_TYPES are displayed. If there were no compatible JEDEC_TYPES for the U4 component then the *Filters* field for the JEDEC_TYPE column would have displayed the value DIP14_3, signifying that you can only select rows with the JEDEC_TYPE value DIP14_3.

4. Select the row with the PART_NUMBER 130 and JEDEC_TYPE DIP28_3 and click *OK*.

Design Variance Tutorial

Creating the Variant Database

A new row with the status as *Pref* and JEDEC_TYPE DIP28_3 is displayed, and the existing row whose JEDEC_TYPE value is DIP14_3, is displayed with *Status* as '-'.

Refdes	Status	Jedec_type	Part_name	Part_number	Access_time	Class	Frequ
U4	-	DIP14_3	74F04	123		IC	
	Pref	DIP28_3	74F04	130		IC	



Exercise 7

1. Try to replace the value of the U7 component which has the JEDEC_TYPE value DIP28_3, with the value DIP14_3 . What happens? Can you explain the results?
2. Add an alternate row for the U4 component by defining a different JEDEC_TYPE, DIP28_3. For the alternate, select the row whose PART_NM property is 131.

When you have completed the exercise, refer to the answer to [Exercise 7](#) in Appendix A.

Suppressing the Compatible Jedec Type Match Messages

You can suppress the display of warning messages for compatible JEDEC type matches. For this, do the following:

1. Choose *Tools – Options* in Variant Editor.
2. Choose the *Compatible Jedec Type Match* check box in the *Suppress Messages* group.
3. Click *OK* to close the box.

If you now try to create an alternate for the U4 component, a warning message will not be displayed.

Close Design Entry HDL, if it is open, and Variant Editor. If prompted to save the changes, click *Yes*. Close Project Manager.

Summary

You learned different ways to create variations in a design. You learned how to create alternate values for components. You created alternate groups, functions and variants, and customized the value of components in variants.

You also learned to find components with variant information in a design and synchronize the variant database with the schematic.

What's Next

In the next chapter, [Generating BOM Reports](#), you will create BOM reports for individual variants and a Variant Comparison BOM report (that provides a part number-based comparison between the components of the base schematic and all the variants). You will also learn to customize the BOM report and include filters to generate listing of specific components.

Recommended Reading

For more information about different design variations and creating and managing variants, see [*Design Variance User Guide*](#).

Generating BOM Reports

Objectives

- Identify the different types of BOM reports that you can create using BOM-HDL
- Launch BOM-HDL and become familiar with its interface
- Create a BOM report for the base schematic
- Customize the BOM template
- Include associated mechanical parts and kits in BOM reports
- Edit the Callouts information in a BOM report
- Apply filters on parts to generate BOM reports for specific parts
- Create a Variant BOM report
- Create a Variant Comparison BOM report

Nature of Chapter

Skill (includes concepts and practice)

Estimated Completion Time

3 hours

Overview

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

There are three types of BOM reports:

- **The BOM report for the base schematic**—This report contains the 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.
- **Variant BOM report**—This report contains the 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.
- **Part-number based comparison BOM report**—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.

In this chapter, you will learn to create all three types of BOM reports and analyze the differences between each BOM report. Besides creating BOM reports, you will also learn to use the BOM-HDL tool to customize the BOM report. You can customize a BOM report in the following ways:

- **Control the format of BOM reports**—You will learn how to control the header in the BOM report, add descriptions, and display the report in the HTML or spreadsheet format.
- **Control the content displayed in BOM reports**—You will learn to control the properties that should be displayed in the BOM report and how to sort the different rows in the BOM report.
- **Filter different parts from the report**—You will learn to set conditions to filter parts based on multiple conditions.
- **Define the callouts file**—You will also learn to create a callouts file, which includes the list of mechanical parts to be displayed in the BOM report.
- **Include mechanical kits and associated mechanical parts in BOM reports**—You will learn to define mechanical kits, which specify a set of mechanical parts that are included together in the BOM report. You can also set that all mechanical parts associated to electrical parts are included in the BOM report.

Getting Started

To create a BOM report, you need to launch the BOM-HDL tool and define the type of BOM report and report format. However, before you launch the BOM-HDL tool, ensure that you have the BOM template file and that you have packaged the design.

Note: For more information about the BOM template file and how BOM-HDL searches the BOM template file, see [Design Variance User Guide](#).

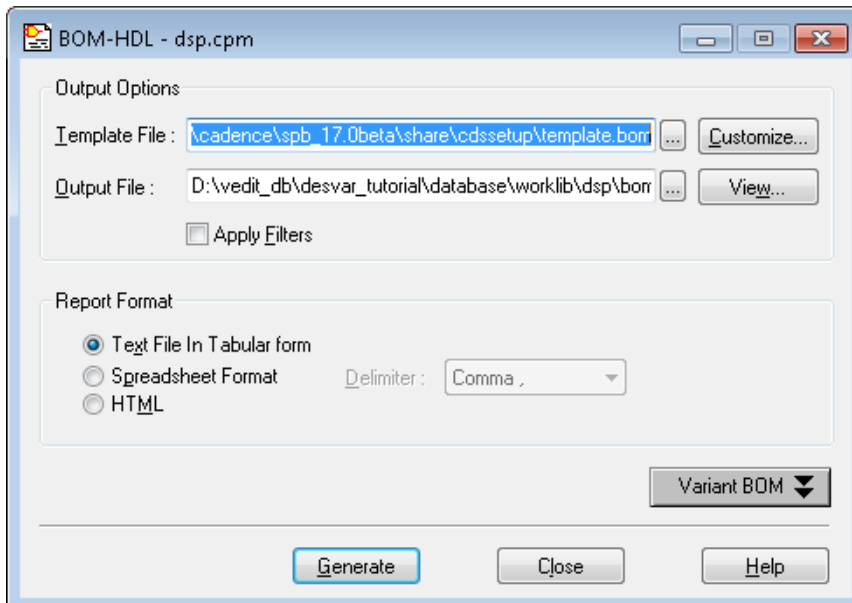
Loading BOM-HDL

To load the BOM-HDL tool, do the following:

1. Load the `dsp` design under the `database` directory in Project Manager.
2. Choose *Tools – Packager Utilities – Bill of Materials*.

A warning message prompts you to package the design since changes were made to the design since it was last packaged. For the purpose of this tutorial, we will not package the design. Click *No*.

The BOM-HDL dialog is displayed.



Design Variance Tutorial

Generating BOM Reports



Tip

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

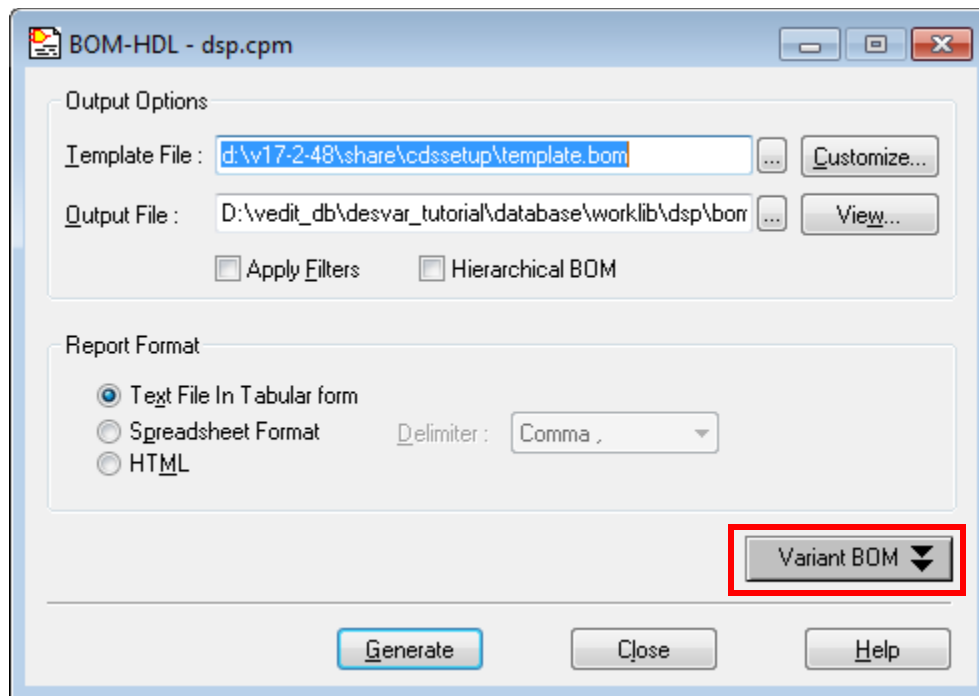
- ☐ From the *Tools* menu of Variant Editor, choose the *Generate Report* command.
- ☐ From the *Tools* menu of Design Entry HDL, choose *Packager Utilities -Bill of Materials*.
- ☐ Specify the `bomhdl -proj ...` command at the command prompt.

A message is displayed prompting you to package the design.

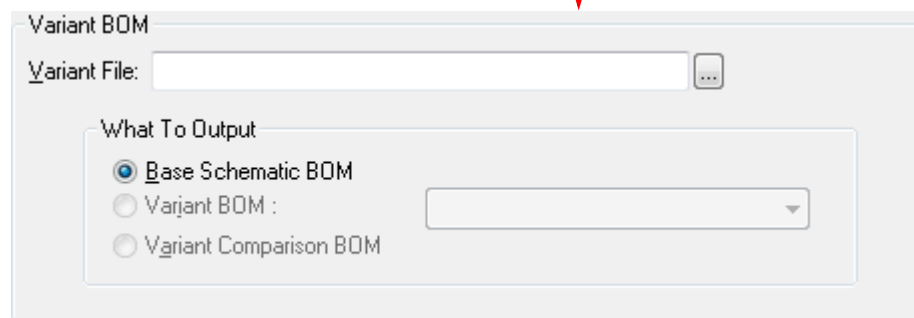
The *BOM-HDL* dialog is organized in three group boxes: *Output Options*, *Report Format*, and *Variant BOM*. You can specify the path to the `bom.template` file and the `<output file>` where the BOM report will be generated. You can also specify the format of BOM report as text file, spreadsheet format, or HTML format.

Design Variance Tutorial

Generating BOM Reports



Click the Variant BOM button to expand the dialog and view the Variant BOM options.



The *Variant BOM* group box allows you to specify the variant file, and select the type of BOM report: *Base Schematic BOM*, *Variant BOM*, and *Variant Comparison BOM*. You will generate each of these reports in the subsequent sections.

Creating the BOM for the Base Schematic

Task Overview

You will create the BOM report for the base schematic using the `template.bom` file in the `bom` directory under the `dsp` design.

Steps

To create a BOM report, do the following in the *BOM-HDL* dialog:

1. Specify the template as `template.bom`.

The `template.bom` file contains the BOM report customized information. Subsequent sections explain how to edit this file.

Note: Note that the *Output File* field displays that the `BOM.rpt` file will be generated in the `bom` view of the top-level design `dsp`. All BOM reports are created in the `bom` directory of the top-level design. You can change the name of the output file where the BOM report is generated.

By default, the *Text File In Tabular form* option button is selected, which signifies that the BOM report will be generated in text format. You can change this option. However, for this tutorial, leave this option unchanged.

Note that the *Base Schematic BOM* radio button is already selected. You need not change it.

Note: If you have generated a variant BOM report or variant comparison BOM report in the current session, the corresponding radio button will be selected. If the *Base Schematic BOM* radio button is not selected, click it. To create variant BOM reports, see [Creating a Variant BOM Report](#) on page 84. To create a variant comparison BOM report, see [Creating the Variant Comparison BOM Report](#) on page 88.

2. Click *Generate* to generate the BOM report.

BOM-HDL displays a message box stating that the BOM report is successfully generated. This report is named `BOM.html` and is stored in the `bom` directory under the root design.

3. Click *Yes* to view the BOM report.

Report Explanation

You will note that the report is divided into two sections: the Header section and the list of components in the design. The Header section lists the title of the BOM report, the date of report creation, the design name, the template file path, and the name of the Callouts file.

For each component included in the design, one row of information is displayed. This row lists the values of various columns, such as `PART_NAME` and `REFDES`, as selected in the `bom.template` file.



Exercise 8



Tip

Click the *Customize* button located to the right of the *Template File* field and explore various customizing options. Do not save any changes to the BOM template file as it might impact the procedures covered later in this chapter.

1. Can you explain why there are only six rows in the header section of the BOM report. How can you add, delete, or change the content in the Header section?
2. How would you control the quantity and cost information for each part in the BOM report?
3. How do you include mechanical parts in the BOM report?

When you have completed the exercise, refer to the answers to [Exercise 8](#) in Appendix A.

Note: Now see this multimedia demonstration on [Generating a BOM Report](#).

Customizing the BOM Template

Overview

The BOM template defines the properties that will be displayed in the BOM report. It also defines the layout of the BOM report. You can customize the BOM template to following:

- **Change the report parameters** - You can change the report header and the format of the report. You can specify the path to the Callouts file and specify a customized header file to be used as the header of the BOM report.

Design Variance Tutorial

Generating BOM Reports

- **Change physical part information** - You can define which properties should be displayed for PPT rows and what should be the alignment for each property column. You can also define the sorting criteria for properties.
- **Change the variant specific settings** - You can define whether you want to include alternates and DNI components in BOM reports. You can also change symbols to determine the status of preferred, alternate, and DNI components.

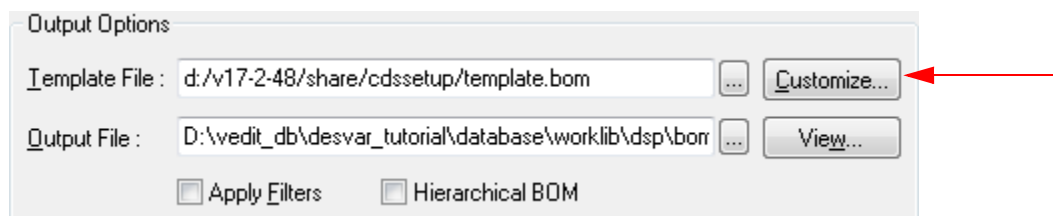
Variant-specific settings can be changed in the BOM template only if you have loaded the variant file in the *BOM-HDL* dialog.

- **Edit the Callouts file** - You can edit the Callouts file by adding or deleting the mechanical parts to be listed in the BOM report. You can also change the quantity of mechanical parts.
- **Filter different parts from the report** - You can set conditions to filter parts based on multiple conditions. For example, you may specify that only those components with *SPEED* less than 15NS are listed in the BOM report.

Displaying the Customize Template Dialog

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

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



Design Variance Tutorial

Generating BOM Reports

The *Customize Template* dialog is displayed with the Report Parameters tab selected.

The screenshot shows the 'Customize Template' dialog box with the 'Report Parameters' tab selected. The dialog has three tabs: 'Report Parameters', 'Physical Part Specifications', and 'Variant Settings'. The 'Report Parameters' tab contains several sections:

- Report Header:** A table with 5 rows and 2 columns: 'HEADER PROPERTY' and 'VALUE'. The rows are: 1. TITLE (checked), Bill of Materials; 2. DATE (checked), 02/02/2016; 3. DESIGN (checked), dsp; 4. TEMPLATE (checked), d:/v17-2-48/share/cdssetup/template.; 5. CALLOUT (checked), D:\vedit_db\desvar_tutorial\database\.... To the right of the table are four icons: up arrow, down arrow, a document with a plus sign, and a document with an X.
- Row Column Separator:** Checkboxes for 'Column Separator', 'Row Separator', and 'Header Separator' (set to '='), and a 'Column Pad' checkbox.
- RefDes:** Radio buttons for 'Each', 'Unique', and 'Ranges' (selected). A 'Range Separator' is set to '-' and 'Minimum Members In Range' is set to 3.
- Callouts:** A 'Title' field with 'Mechanical Parts', a 'File' field with 'D:\vedit_db\desvar_tutorial\database\worklib\dsp\bom\bom.callout', and an 'Edit...' button. There is also an unchecked checkbox for 'Intersperse Associated Mechanical Parts'.
- Miscellaneous:** A 'Header file' field, a 'Page Length' field set to 0, and checkboxes for 'Print Header At Top Of Each Page' (unchecked) and 'Print Column Header' (checked).

At the bottom of the dialog are four buttons: 'Save', 'Save As...', 'Close', and 'Help'.

Changing the Report Parameters

Task Overview

You will change the properties displayed in the Header section of the BOM report by choosing to add the Description property for the Base schematic BOM and changing the date format

Design Variance Tutorial

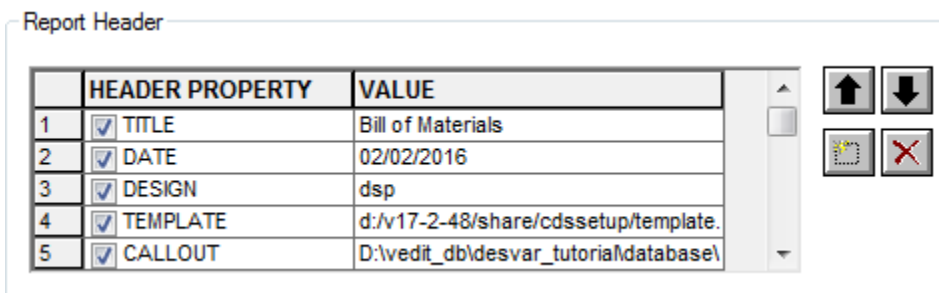
Generating BOM Reports

to include the full month name and time of report generation. You will also include a hyphen (-) as the row separator and specify that Refdes ranges will not be used in the BOM report. You will set BOM-HDL to display mechanical parts along with the electrical parts with which they are associated. You will increase the width of the `PART_NAME` property and include the `JEDEC_TYPE` property in the BOM report. Finally, you will include the serial number for different properties, and set the sorting style to be based in the descending order.

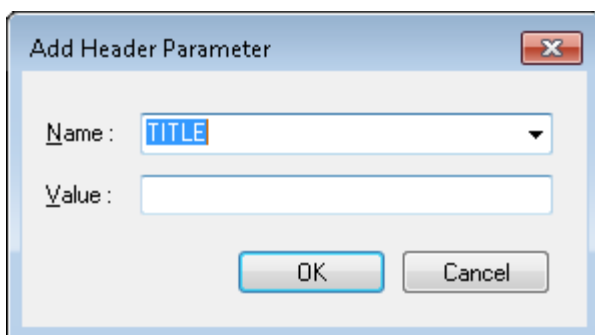
Steps

To change the report parameters, do the following in the *Customize Template* dialog:

1. To add a new property, in the *Report Header* section, click .



The *Add Header Parameter* dialog is displayed.



2. Choose *DESCRIPTION* from the *Name* drop-down list.

In the *Value* field, `Base Schematic BOM` is displayed.

3. Click *OK* to select the property and close the *Add Header Parameter* dialog.

Design Variance Tutorial

Generating BOM Reports

The *DESCRIPTION* property appears in the *HEADER PROPERTY* list.

	HEADER PROPERTY	VALUE
4	<input checked="" type="checkbox"/> TEMPLATE	D:\v17-2-48\share\cdssetup\template.
5	<input checked="" type="checkbox"/> CALLOUT	D:\vedit_db\desvar_tutorial\database\
6	<input checked="" type="checkbox"/> VARIANT	
7	<input checked="" type="checkbox"/> DESCRIPTION	Base schematic BOM

4. To change the *Date* format, click in the *VALUE* field against the *DATE* column.
5. From the *Date* drop-down list, select the month, day, year time format. For example, select a value similar to February 02, 2016 11:20.
6. To specify '-' as the row separator, type '-' in the *Row Separator* field in the *Row Column Separator* group box.

By default, if BOM-HDL finds three or more parts with the same *PART_NAME*, it lists the reference designators corresponding to them as a range.

7. To list each part with the same *PART_NAME* separately, select *Each* in the *RefDes* group box.

By default, all mechanical parts are displayed at the end of the BOM report.

8. To display mechanical parts along with the electrical parts with which they are associated, select the *Intersperse Associated Mechanical Parts* check box in the *Callouts* group box.
9. To change how information related to physical parts is displayed, make changes in the *Physical Part Specifications* tab. Click the *Physical Part Specifications* tab.
10. To increase the width of the *PART_NAME* property so that the name is displayed in one line, type 25 in the *WIDTH* column corresponding to the *BOM_PART* property.

Report Columns

	PROPERTY	MECH	TITLE	WIDTH	JUSTIFICATION	TO
1	<input type="checkbox"/> BOM_ITEM_NUM	<input type="checkbox"/>	S.No.	6	Left	<input type="checkbox"/>
2	<input checked="" type="checkbox"/> BOM_PART	<input type="checkbox"/>	Part Name	25	Left	<input type="checkbox"/>
3	<input checked="" type="checkbox"/> BOM_INST	<input type="checkbox"/>	Ref Des	20	Left	<input type="checkbox"/>
4	<input checked="" type="checkbox"/> BOM_QUANTITY	<input type="checkbox"/>	Qty	4	Right	<input checked="" type="checkbox"/>

11. To add a new property as a column in the BOM report, select the check box corresponding to that property. For example, add the *JEDEC_TYPE* property in the BOM report by selecting the check box corresponding to the *JEDEC_TYPE* property.

Design Variance Tutorial

Generating BOM Reports

12. To ensure that the callouts display desired properties, select the `MECH` check box corresponding to those properties. If the `MECH` check box corresponding to the `BOM_PART` property is not selected, select it.

13. To serialize all properties, choose the *S. No.* check box.

The default sorting style is *Alphabetic*, and the sorting order is *Ascending*. The properties are sorted based on the first property listed in the *Report Columns* grid box. By default, the first property in the *Report Columns* grid box is `BOM_PART`.

This property represents the primitive name used for the part in the `pstchip.dat` file. The BOM report is, therefore, sorted on the `BOM_PART` property. To make another property the key property, move it to the first row in the *Property* column. However, when you change the key property, ensure that all components to be listed in the BOM report must have some non-null value against that property.

14. The BOM report needs to be sorted based on the `BOM_PART` property. However, the properties should appear in descending order. For this, select the *Descending* option in the *Order* list box.

You have made all the necessary changes to the BOM template.

15. Save the template.

16. Click *Close* to close the *Customize Template* dialog.

The *BOM-HDL* dialog is displayed.

17. Select *Text File in Tabular form* under Report Format, if it is not selected.

18. Click the *Generate* button to generate the report.

BOM-HDL displays a message box stating that the BOM report is successfully generated. This report is named `BOM.rpt` and is stored in the `bom` directory under the root design.

19. Click the *Yes* button to view the BOM report.

Design Variance Tutorial

Generating BOM Reports

Report Explanation

The report is divided into three sections: the Header section, the list of components in the design, and the Callouts section. The Header section includes six lines and the last line is Description, with the title Base schematic BOM.

```
TITLE:          Bill of Materials
DATE:          February 02, 2016 14:14
DESIGN:        dsp
TEMPLATE:      d:/v17-2-48/share/cdssetup/template.bom
CALLOUT:       D:\vedit_db\desvar_tutorial\database
               \worklib\dsp\bom\bom.callouts
DESCRIPTION:    Base schematic BOM
```

You will note that the '-' sign is spread all across the report. This is the row separator. There is a serial number column, which begins from 0 and ends in 14, signifying that there are 15 PPT rows. The JEDEC_TYPE column is also included, and the width of the PART_NAME column is increased.

```
-----
S.No.   Part Name           Ref Des           Qty
Unit    Cost
Price
=====
=====
0       XTALOSC-5MHZ,25PPM    U2                1
?       ?
-----
-----
1       CY7C263-35_LCC-25NS,      U10               1
?       ?
        MILITARY
-----
-----
2       CY7C199_DIP-15NS          U6,U7,U8          3
?       ?
-----
-----
3       HS10 (U6) (MECH)         U6                2
2       4.00
-----
```

Design Variance Tutorial

Generating BOM Reports

You will also find that there are six rows that have the word (MECH) in the PART_NAME field. These parts are mechanical parts and they are associated to an electrical part. For example, one PPT row displays the PART_NAME:

WASHER-WA6500 (U3) (MECH)

2	3	HS10 (U6) (MECH)	U6	2
	4.00			
-----	-----	-----	-----	-----
2	4	HS10 (U7) (MECH)	U7	2
	4.00			
-----	-----	-----	-----	-----
2	5	HS10 (U8) (MECH)	U8	2
	4.00			
-----	-----	-----	-----	-----
?	6	CONN9-TIN, 6.09MM, 3.18MM	J1	1
	?			
-----	-----	-----	-----	-----
?	7	CONN20-TIN, 6.09MM, 3.43MM	U1	1
	?			
-----	-----	-----	-----	-----
?	8	ADSP2101_PGA-0-70C	U3	1
	?			
-----	-----	-----	-----	-----
.10	9	WASHER-WA6500 (U3)	U3	5
	0.50			
	(MECH)			
-----	-----	-----	-----	-----
.10	10	BOLT-BO9800, 6.5G (U3)	U3	5
	0.50			
	(MECH)			
-----	-----	-----	-----	-----
2	11	HS10 (U3) (MECH)	U3	2
	4.00			
-----	-----	-----	-----	-----

This PPT row represents a mechanical part named WASHER-WA6500 associated with the electrical part with the REFDES U3. The PART_NAME of U3 can be found by reading the row above the WASHER-WA6500 (U3) (MECH). This PART_NAME is ADSP2101_PGA-0-70C.

Design Variance Tutorial

Generating BOM Reports

The last section in the BOM report is the Callouts section.

```
Callouts File:                      D:\vedit_db\desvar_tutorial
\database\worklib\dsp\bom\bom.callouts
Mechanical Parts
-----
```

S.No.	Part Name	JEDEC_TYPE
=====	=====	=====
0	SCREW-SC8301	?
1	EJECTOR-MP4001	?
2	CONNECTOR-CO6000	?
3	CONN9-15U, GOLD, 6.09MM, 3.43MM	CONN9
4	CO6010	?
5	BOLT-BO9800, 6.5G	?
TOTAL		
=====	=====	=====

This section includes the name of the Callouts file and the list of mechanical parts in the design. Note that the name of the Callouts file is `bom.callouts`. This file is located in the `bom` directory under the root design. BOM-HDL finds callouts information by reading the path to the Callouts file, which is stored in the `bom.template` file. You will learn to customize this path in a later section.

When you create a design, you may define a callouts file to include the list of mechanical parts in your design. A sample Callouts file, `bom.callouts`, is available in the `<your_inst_dir>/share/cdssetup` directory. You may edit this file, or you may simply create a new Callouts file by using the *Callouts Editor* dialog. See [Editing the Callouts Information](#) on page 77 for more information.



Exercise 9

1. Regenerate the BOM report in HTML format.
2. Specify that the row is sorted on the *Ref Des* title in the alphabetic ascending order.



Tip

Make the `BOM_INST` property the key property by shifting it to the top of the Report Columns section in the *Customize Template* dialog.

When you have completed the exercise, refer to the answers to [Exercise 9](#) in Appendix A.

Including Associated Mechanical Parts and Mechanical Kits

Introduction

Some mechanical parts are listed in the BOM reports. This is because BOM-HDL recognizes associated mechanical parts and mechanical kits and includes them in a BOM report. BOM-HDL gets information about the mechanical parts and kits from the ptf files that are specified in the project file.

In the PPT files, an associated mechanical part has two characteristics:

- It has the `CLASS` property defined as `MECH`.
- It is associated to a logical or physical part in the design included in the PPT. To associate a mechanical part to an electrical part, a mechanical property `MECH_PART` is defined in the PPT rows for the electrical part with which the mechanical part is associated.

A mechanical kit includes a pre-defined set of mechanical parts. For example, a connector requires the following mechanical parts: four nuts, five washers, and seven screws. To specify this requirement, you can define a mechanical kit and add it to the schematic. Assume you name the mechanical kit as `KIT`. This mechanical kit has two characteristics:

- It has a part name, `KIT`, and has three `MECH_PART` properties named `MECH_PART1`, `MECH_PART2`, and `MECH_PART3`.

Design Variance Tutorial

Generating BOM Reports

- The mechanical kit, KIT, is associated to an electrical part using the `MECH_KIT` property.

Task Overview

In this procedure, you will analyze the PPT files in the `dsp` design under the `database` directory to find how mechanical parts and mechanical kits are associated in the design.

Steps

1. Find the ptf files used in your design by reading the Project Manager setup.

The `dsp` design uses one PTF file, `myppt.ptf`, which is located in the `ptfs` directory at the following location: `\vedit_db\desvar_tutorial\database\ptfs`.

2. Open the `myppt.ptf` file in a text editor.

Explanation

Example1 shows how the `myppt.ptf` file stores information about associated mechanical parts (the mechanical part in the example is `HEATSINK`).

Example 1

```
PART 'HEATSINK'
CLASS=MECH
{=====
=====}
:PART_NUMBER      = VALUE          | TOL          | POWER        | UNIT_PRICE    ;
{=====
=====}
'HS2001' (~HS10)  = '313'          | '8%'         | '1010'       | '2'
'HS3001' (HS20)   = '222'          | '6%'         | '1030'       | '3'
'HS4001' (!)      = '223'          | '58%'        | '1020'       | '4'
END_PART
```

Note that the `HEATSINK` part has the `CLASS` property defined as `MECH`. You will also find that the `PART_NUMBER` property is followed by three syntax, which help define the row name for the associated mechanical parts. The purpose of each of the three syntax is defined below:

1. `(~any_name)` - The `~any_name` syntax within brackets `()` signifies that the row name will be the same as `any_name`. Therefore, the first PPT row for the `HEATSINK` part will be associated with the electrical part that has `HS10` defined as the row name.

Design Variance Tutorial

Generating BOM Reports

2. (any_name) - The any_name syntax within brackets () signifies that the row name will be the same as PART_NAME-any_name. Therefore, the second PPT row for the HEATSINK part will be associated with the electrical part that has HEATSINK-HS20 defined as the row name.
3. (!) - The ! letter within brackets () signifies that the row name will have the following syntax:

PART_NAME-key property1, key property2, key property3...

Since the HEATSINK part has only one key property, PART_NUMBER, the first ppt row will be associated with the electrical part that has HEATSINK-HS4001 defined as the row name.

Example 2 shows how electrical parts in the myppt.ptf file are associated with mechanical parts (the electrical part in the example is CY7C199 and the mechanical part being associated is HEATSINK).

Example 2

```
PART 'CY7C199'
CLASS=IC
{=====
=====}
:PACK_TYPE          | SPEED      = SPEED      | JEDEC_TYPE          | PART_NUMBER
| MECH_PART1 ;
{=====
=====}
'DIP'                | '15ns' (!) = '15ns'      | 'DIP28_3'           | 'cy7c199L-15PC'
| 'HEATSINK:HS10:2' '
'DIP'                | '20ns' (!) = '20ns'      | 'DIP28_3'           | 'cy7c199L-20PC'
| 'HEATSINK:HEATSINK-HS20:2'
'LCC'                | '15ns' (!) = '15ns'      | 'LCC28'              | 'cy7c199L-15LMB'
| 'HEATSINK:HS10:2'
'LCC'                | '20ns' (!) = '20ns'      | 'LCC28'              | 'cy7c199L-20LMB'
| 'HEATSINK:HEATSINK-HS4001:2'
END_PART
```

The CY7C199 part is associated to the HEATSINK mechanical part using the MECH_PART1 property. In Example 2, the mechanical part property has the following syntax:

'<PART_NAME>:<ROW_NAME>:<Quantity>'

where, ROW_NAME uniquely identifies a PPT row for a mechanical part with the PART_NAME in the mechanical part PPT file.

Design Variance Tutorial

Generating BOM Reports

Note that the `ROW_NAME` for the `MECH_PART1` property uses different syntax as explained in Example 1.

Example 3 shows how the `myppt.ptf` file stores information about mechanical kits (KIT001 and KIT002).

Example 3

```
PART 'KIT'
{
=====
=====}
:PART_NUMBER          = MECH_PART1          | MECH_PART2          |
MECH_PART3            ;
{
=====
=====}
'KIT001' (~KIT001)     = 'WASHER:WASHER-WA6500:5' | 'BOLT:BOLT-BO9800,6.5G:5' |
'HEATSINK:HS10:2'
'KIT002' (~KIT002)     = 'WASHER:WASHER-WA6501:4' | 'BOLT:9801:5'          |
'HEATSINK:HS20:3'
END_PART
```

The KIT001 mechanical kit contains the following parts:

```
5 number WASHER-WA6500
5 number BOLT-BO9800,6.5G
2 number HEATSINK HS10
```

If you associate KIT001 to an electrical part, then all 12 parts in the kit are associated with that electrical part.

Reading Mechanical Part Information in BOM Reports

Task Overview

In this procedure, you will analyze the mechanical part information in a BOM report. The *BOM-HDL* dialog is displayed and the *Intersperse Associated Mechanical Parts* check box is selected in the *Customize Template* dialog.

Steps

1. Choose *HTML* in the *BOM-HDL* dialog.
2. To generate the BOM report, click *Generate*.

Design Variance Tutorial

Generating BOM Reports

3. Select Yes to display the BOM report.

The BOM report is displayed.

Report Explanation

Note that under the PPT row corresponding to the U3 component, there are three rows for the following parts:

WASHER-WA6500 (U3) (MECH) - Quantity = 5

BOLT-BO9800,6.5g (U3) (MECH) - Quantity = 5

HS10 (U3) (MECH) - Quantity = 2

These rows are placed in the BOM report because the KIT001 mechanical kit is associated with the U3 component.

Note that under the PPT row for the U6 component, the following PPT row is present:

HS10 (U6) MECH

Similarly, under the PPT row for the U7 component, the following PPT row is present:

HS10 (U7) MECH

And, under the PPT row for the U8 component, the following PPT row is present:

HS10 (U8) MECH

S.No.	Part Name	Ref Des	Qty	Unit Price	Cost
0	XTALOSC-5MHZ,25PPM	U2	1	?	?
1	CY7C263-35_LCC-25NS, MILITARY	U10	1	?	?
2	CY7C199_DIP-15NS	U6,U7,U8	3	?	?
3	HS10 (U6) (MECH)	U6	2	2	4.00
4	HS10 (U7) (MECH)	U7	2	2	4.00
5	HS10 (U8) (MECH)	U8	2	2	4.00

These rows are placed in the BOM report because the HS10 mechanical part is associated with the CY7C199 electrical part. Example 1 and Example 2 explain how HS10 has been associated with the PART_NAME CY7C199.

The BOM report also lists a Callouts section, which contains six mechanical parts. You will learn to control this section in the next procedure.



Caution

If you edit any ptf file to include information about mechanical parts then package your design to ensure that Variant Editor and BOM-HDL read the latest information.

Editing the Callouts Information

Introduction

Some mechanical parts are listed at the end of BOM reports. BOM-HDL obtains information about these parts using the *Callouts Editor* dialog. You can use the *Callouts Editor* dialog to add or remove mechanical parts. You can also modify the quantity for each mechanical parts.

Task Overview

You will edit the `bom.callouts` file to do the following:

- Include the mechanical part `BOLT` with the `BO9803` as the `PART_NUMBER`
- Remove the mechanical part with the following `BOM_PART` property: `BOLT-9800`

Finally, save the Callouts file with the name `bom1.callouts`.

Steps

To edit the `bom.callouts` file, do the following in the *BOM-HDL* dialog:

1. Click *Customize*.

The *Customize Template* dialog is displayed. Note the name of the Callouts file is `bom.callouts`.

Design Variance Tutorial

Generating BOM Reports

2. To edit the `bom.callouts` file, click the *Edit* button.

Report Parameters Physical Part Specifications

Report Header

	HEADER PROPERTY	VALUE
1	<input checked="" type="checkbox"/> TITLE	Bill of Materials
2	<input checked="" type="checkbox"/> DATE	February 03, 2016 15:53
3	<input checked="" type="checkbox"/> DESIGN	dsp
4	<input checked="" type="checkbox"/> TEMPLATE	D:\v17-2-48\share\cdssetup\template.
5	<input checked="" type="checkbox"/> CALLOUT	D:\vedit_db\desvar_tutorial\database\

Row Column Separator

Column Separator ☐ Row Separator ☐ Header Separator ☐

Column Pad ☐

RefDes

☒ Each ☐ Unique ☐ Ranges

Range Separator ☐ Minimum Members In Range

Callouts

Title

File

☒ Intersperse Associated Mechanical Parts

Miscellaneous

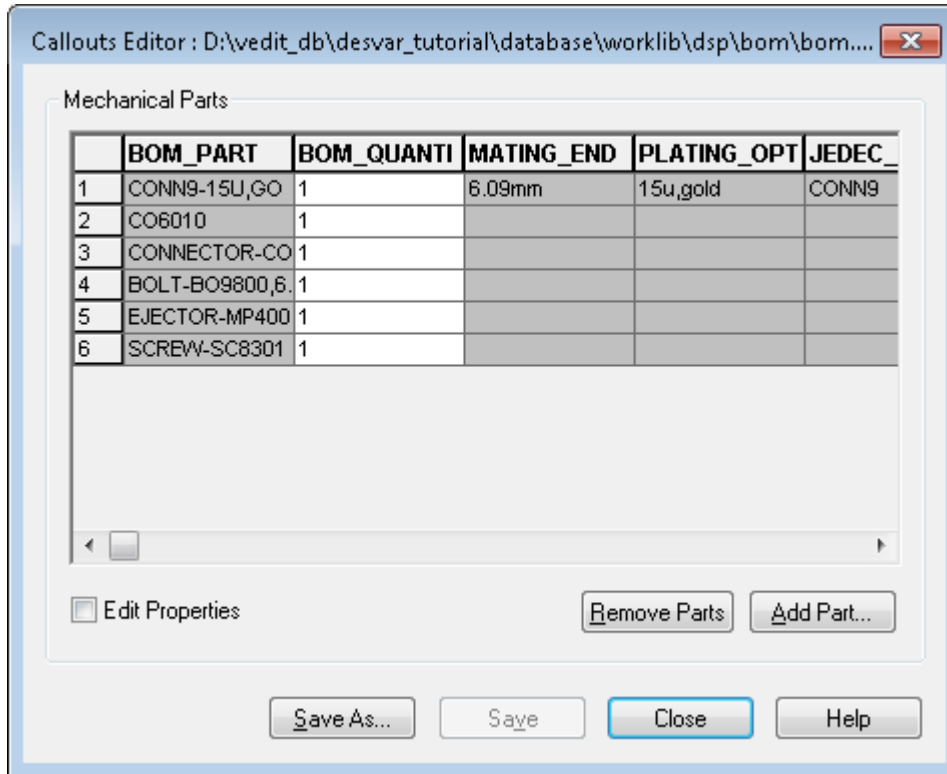
Header file

Page Length ☐ Print Header At Top Of Each Page ☒ Print Column Header

Design Variance Tutorial

Generating BOM Reports

The *Callouts Editor* dialog is displayed.



3. To remove the mechanical part with the `BOM_PART` property `BOLT-9800`, select the row corresponding to serial number 4 by clicking in the `BOM_PART` field.
4. Click the *Remove Parts* button.
The row corresponding to the `BOM_PART` property with value `BOLT-9800` is deleted.
5. To add a new part in the Callouts file, click the *Add Part* button.
The *Physical Part Filter* dialog is displayed. The list of parts with the `CLASS` property defined as `MECH` in the ptf files is displayed in the *Part Names* list.
6. To display the list of PPT rows corresponding to the Part Name `BOLT`, click `BOLT` in the *Part Names* list.
7. Select the PPT row with the `PART_NUMBER` `BO9803` to include the mechanical part `BOLT` with the `PART_NUMBER` `BO9803` in the Callouts file, and click *OK*.
8. To save the Callouts file with a different name, click the *Save As* button.

The *Save Callouts Filer As* file browser is displayed.

Design Variance Tutorial

Generating BOM Reports

9. Type the name of the new Callouts file as `bom1.callouts` and select *Save*.
10. Click the *Close* button to close the *Callouts Editor* dialog.
11. Click the *Save* button in the Customize Template dialog then click *Close* to close the dialog.
12. In the BOM-HDL dialog, click the Generate button to generate the BOM report.

You will find that the path to the Callouts file has changed and that there are six mechanical parts in the file with the part `BOLT-BO9803 , 9 . 2G` replacing the part `BOLT-9800`.



Exercise 10

1. Name the different types of mechanical parts supported by BOM-HDL.
2. What are the two different modes in which BOM-HDL displays mechanical parts in the BOM report?
3. Which files do you need to edit to ensure that you have the necessary mechanical part information in the BOM reports?

When you have completed the exercise, refer to the answer to [Exercise 10](#) in Appendix A.

Setting Filters on Parts

Introduction

If your design includes hundreds or, maybe, thousands of parts, then it may be difficult to locate specific parts in a BOM report. You may then need to list only the parts in the BOM report that meet a particular condition. Using filters, you can define conditions for selecting parts. For example, you may define a condition to include only resistors in the BOM report. To define such a filter, you need to use the following condition:

```
ref des LIKE R*
```

In this condition:

- `ref des` means reference designators
- `LIKE` denotes that the search will be on wildcard entries
- `R*` denotes that the search will return all parts that have their reference designators starting with the letter `R`

Design Variance Tutorial

Generating BOM Reports

Task Overview

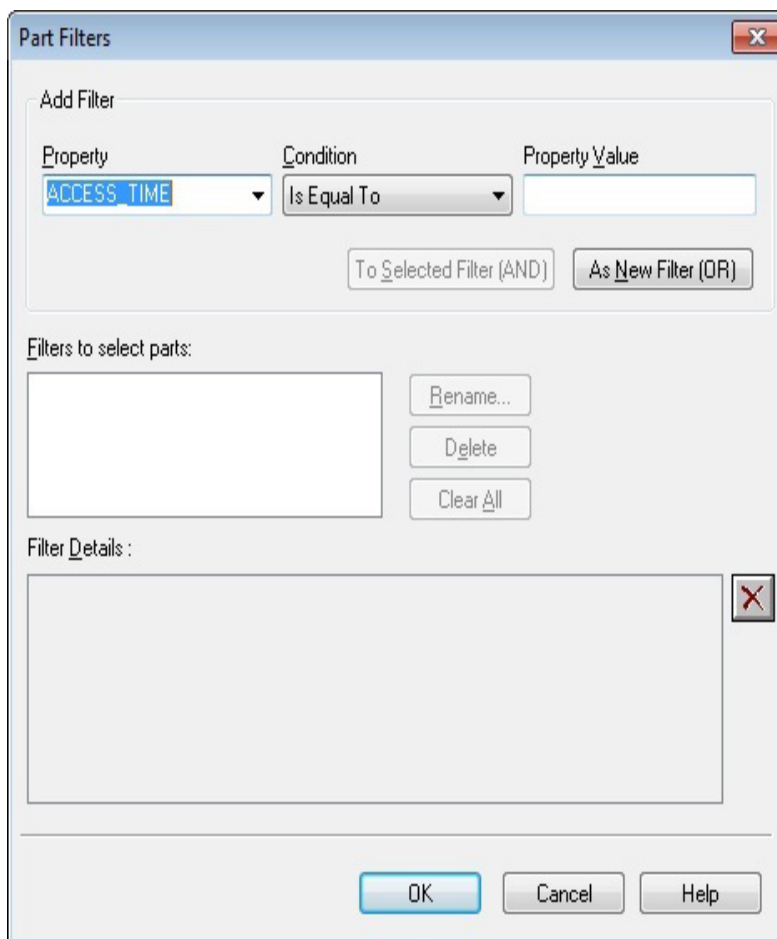
You will define a filter which lists only those rows that have `SPEED` greater than 20NS or `PART_NAME` beginning with the letter X.

Steps

To set up a filter, do the following in the *Customize Template* dialog:

1. If you had closed the *Customize Template* dialog, open it by clicking the *Customize* button in the BOM-HDL dialog:
2. In the *Customize Template* dialog, click the *Physical Part Specifications* tab.
3. Click the *Filters* button.

The *Part Filters* dialog is displayed.



Design Variance Tutorial

Generating BOM Reports

4. Define a filter that lists only those rows that have **SPEED** greater than 20NS. Set the following:
 - a. Select the **SPEED** property in the *Property* list.
 - b. Specify the condition as **Is Greater Than** in the *Condition* list.
 - c. Type 20NS in the *Property Value* field.

After you have entered values in the *Add Filter* group box, you need to specify whether the condition you defined is a new filter or an addition to an existing filter. To define a new filter, select the *As New Filter (OR)* button. To add to an existing filter, select the existing filter in the *Filters to select parts* list and click the *To Selected Filter (AND)* button.

5. For this tutorial, to define a new filter for the selection you made in the last step, click the *As New Filter (OR)* button.

A new filter, named **NEW FILTER 1**, is created in the *Filters to select parts* list. You may rename or delete it. The filter is selected, and the details of this filter are displayed in the *Filter Details* list.

Part Filters

Add Filter

Property: SPEED Condition: Is Greater Than Property Value: 20NS

To Selected Filter (AND) As New Filter (OR)

Filters to select parts:

NEW FILTER 1

Rename... Delete Clear All

Filter Details :

Property Name	Condition	Value
SPEED	Is Greater Than	20NS

OK Cancel Help

Design Variance Tutorial

Generating BOM Reports

6. To add another condition that the `PART_NAME` should begin with the letter `X` as a new filter, select the `BOM_PART` property in the *Property* list, the condition `LIKE` in the *Condition* list, and the value `X*` in the *Property Value* field.

7. To add the new condition as a new filter, click *As New Filter (OR)* button.

The new condition is displayed as *NEW FILTER 2*. The details of the filter in the *Filter Details* list are also changed.

8. Click *OK* to save the details of the part filter.

The *Part Filters* dialog closes and the *Customize Template* dialog is displayed.

9. Save the changes in the BOM template by clicking the *Save* button then the *Close* button to close the dialog.

The *BOM-HDL* dialog is displayed. The filters that you have defined are now stored in the BOM template file. However, these filters are not applied to all the BOM reports by default.

10. To apply filters to the BOM report, select the *Apply Filters* check box in the *Output Options* group box.

11. To generate the BOM report, click the *Generate* button.

12. Click *Yes* to view the BOM report.

Report Explanation

The BOM report lists two rows corresponding to the parts. `Ref Des U2` is included in the BOM report because its `PART_NAME` starts with the letter `X`, and the `U10` row is included in the BOM report because its `SPEED` property has the value `25NS`, which is greater than `20NS`.

Note: BOM-HDL may not be able to filter the following:

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

Creating a Variant BOM Report

Introduction

A variant BOM report contains the 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. For more information about variants and how to manage them, see [Chapter 3, “Creating the Variant Database.”](#)

Task Overview

You will create the Variant BOM report for the `INDIA` variant. This report will include `DNI` components.

Steps

To create the Variant BOM report, do the following in the *BOM-HDL* dialog:

1. Uncheck the *Apply Filters* box in the Output Options group box.
2. Click the *Variant BOM* button to display the variant options if the *BOM-HDL* dialog does not display the variant options.
3. Seed the path to the variant file in the *Variant File* field. For this, click the Browse button next to the *Variant File* field and select `variant.dat`.

Note: If you have not covered [Chapter 3, “Creating the Variant Database.”](#), the *variant.dat* file will not be visible. Select the `variant_orig.dat` file in that case.

4. Click the *Variant BOM* radio button, and select the `INDIA` variant from the drop-down list next to the radio button.

By default, Variant BOM reports do not contain `DNI` components. To set that `DNI` components be listed in variant BOM reports, you need to make changes in the *Variant Settings* tab of the Customize Template dialog.

5. Display the Customize Template dialog by clicking the *Customize* button in the Output Options group box.
6. Click the *Variant Settings* tab.
7. Select the *Include DNI Components list* check box.

Design Variance Tutorial

Generating BOM Reports

8. The *Show Values For DNI Components* check box becomes active. Select this check box to include the values of `DNI` components in the BOM report.
9. Click *Save* to save the changes in the BOM template file, and click *Close* to close the Customize Template dialog.
10. To generate the BOM report, click the *Generate* button.
11. Click *Yes* to view the BOM report.

Report Explanation

The variant BOM report is divided into four sections:

1. Header list - This list has seven PPT rows and is the same as the Base schematic BOM report.
2. Properties list - This list displays PPT rows for all the properties that are included in the `INDIA` variant. Note that no PPT row is included for the J1 or U10 components. PPT rows for these components are included in the *DNI Components List*.

The `Var Status` column is a new column in the property list. This column displays whether a component has `Pref` or `Alt1` or `Alt2` as its status. All properties displayed in the property list have `Pref` as their status. The `Var Status` for the U1 component has the status `Pref*`. The letter `*` in the `Var Status` signifies that the preferred value for the

Design Variance Tutorial

Generating BOM Reports

U1 component is different from the base component; that is, there has been some change in the preferred value for the U1 component.

TITLE Bill of Materials
DATE February 02, 2016 14:54
DESIGN dsp
TEMPLATE d:/v17-2-48/share/cdssetup/template.bom
CALLOUT D:\vedit_db\desvar_tutorial\database\worklib\dsp\bom

VARIANT \bom1.callouts
DESCRIPTION INDIA

S.No.	Part Name	Ref Des	Var Status	Qty	Unit Price	Cost
0	XTALOSC-5MHZ,25PPM	U2	Pref	1	?	?
1	CY7C263-35_LCC-25NS, MILITARY	U10	Pref	1	?	?
2	CY7C199_DIP-15NS	U6,U7,U8	Pref	3	?	?
3	HS10 (U6) (MECH)	U6	Pref	2	2	4.00
4	HS10 (U7) (MECH)	U7	Pref	2	2	4.00
5	HS10 (U8) (MECH)	U8	Pref	2	2	4.00
6	CONN20- TIN,6.09mm,3.18mm	U1	Pref*	1	?	?
7	ADSP2101_PGA-0-70C	U3	Pref	1	?	?
8	WASHER-WA6500 (U3) (MECH)	U3	Pref	5	.10	0.50
9	BOLT-BO9800,6.5G (U3) (MECH)	U3	Pref	5	.10	0.50
10	HS10 (U3) (MECH)	U3	Pref	2	2	4.00
11	74F04_DIP-123	U4	Pref	1	?	?
12	16L8	U5	Pref	1	?	?
TOTAL	-	-	-	27	-	17.000

Design Variance Tutorial

Generating BOM Reports

3. DNI Components list - This list displays PPT rows for all DNI components, that is, J1 and U9. The J1 component was specifically assigned the DNI status for the INDIA variant, therefore, it is included in the DNI component list.

DNI Components List :

S.No.	Part Name	Ref Des	Var Status	Qty	Unit Price	Cost
0	CONN9-TIN,6.09MM,3.18MM	J1	DNI	1	?	?
1	27C256_PLCC-90NS	U9	DNI	1	?	?
TOTAL	-	-	-	2	-	0.000

The ROM alternate group has been customized in the INDIA variant. The U9 component was assigned the *Pref* status and the U10 component was assigned the *Alt1* status. Since, in a variant, only the preferred component is included, the U9 component is included in the INDIA variant while the U10 component is not included in the INDIA variant. As a result, the U10 component is assigned the DNI status.

4. Mechanical Parts list - The PPT rows listed in this category are the same as the Base schematic BOM report.

Mechanical Parts

S.No.	Part Name	JEDEC_TYPE
0	SCREW-SC8301	?
1	EJECTOR-MP4001	?
2	CONNECTOR-CO6000	?
3	CONN9-15U,GOLD,6.09MM,3.43MM	CONN9
4	CO6010	?
5	BOLT-BO9803,9.2G	?
TOTAL	-	-

Note: If you have customized the preferred PPT row for any component in a variant, that row is listed in the BOM report.



Exercise 11

1. Generate a variant BOM report for the `EUROPE` variant and include both alternate and `DNI` components in it. Check the results.

When you have completed the exercise, refer to the answer to [Exercise 11](#) in Appendix A.

Creating the Variant Comparison BOM Report

Introduction

The Variant Comparison 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.

Task Overview

You will create the Variant Comparison BOM report.

Steps

1. In the BOM-HDL dialog, choose a variant file, for example, *variant_orig*.
2. Click the *Variant Comparison BOM* radio button.
3. Click the *Generate* button.

Click *Yes* when prompted to view the report.

Report Explanation

The variant comparison report is generated. The description of the report is `Part Number based comparison BOM`. Note that the components that have not been customized in Variant Editor appear in the `Common Components List` with their `PART_NUMBERS`. For these components, all the part numbers are the same as those in the base schematic. The other components appear with their base `PART_NUMBER` values and the `PART_NUMBER` values in each of the variants. Dash (-) in the report specifies that the base `PART_NUMBER`

Design Variance Tutorial

Generating BOM Reports

value for the base schematic and the base PART_NUMBER value for the given variant is the same.

TITLE Bill of Materials
DATE February 02, 2016 15:01
DESIGN dsp
TEMPLATE d:/v17-2-48/share/cdssetup/template.bom
CALLOUT D:\vedit_db\desvar_tutorial\database\worklib\dsp\bom
 \bom1.callouts
DESCRIPTION Part Number based comparison BOM

S.No.	Ref	Part Number	EUROPE	INDIA	USA
	Des				
0	U9	27c256-90/L	DNI	DNI	DNI
1	U8	cy7c199L-15P	cy7c199L-20	-	DNI
		C	PC		
2	U7	cy7c199L-15P	cy7c199L-20	-	DNI
		C	PC		
3	U6	cy7c199L-15P	cy7c199L-20	-	DNI
		C	PC		
4	U1	22-28-5200	22-28-4200	22-28-4200	22-28-4200
5	J1	22-28-4090	-	DNI	-

Common Components List :

S.No.	Ref	Part Number
	Des	
0	U5	125
1	U4	123

Close the BOM-HDL dialog and Project Manager.

Summary

You learned to generate the three types of BOM reports—base schematic BOM, variant BOM, and variant comparison BOM. You also learned to customize the BOM template.

Next, you learned to add information about mechanical parts and callouts in BOM reports. Finally, you learned to set filters to generate BOM reports for specific parts.

What's Next

In the next chapter, [Backannotating Variant Information](#), you will create a new view for the base schematic in which every component will have a property that will designate whether or not the component has variant information defined for it. You will also create schematic views for each of the variants with the variant properties backannotated for plotting.

Recommended Reading

- For more information about backannotating variant information, see [Design Variance User Guide](#).
- For more information about the functions of each option in BOM-HDL dialog, see [Allegro Design Entry HDL Utilities User Guide](#).

Backannotating Variant Information

Objectives

- Identify the need for backannotation of variant information in the schematic
- Identify the basic requirement for backannotation of variant information and the recommended guidelines
- Identify the functions of the Backannotation dialog
- Create a new flattened schematic view for the base schematic with variant properties
- Create a new flattened schematic view for a variant

Nature of Chapter

Skill (includes concepts and practice)

Estimated Completion Time

45 minutes

Overview

After creating the variant database, you may want to create plots for the schematic with the variant information backannotated on it. Using Variant Editor, you can backannotate the changes to a schematic view. On backannotation, Variant Editor creates a new flattened schematic view of the design, which is very useful for plotting.

Using Variant Editor, you can perform two types of backannotation.

- **Annotate properties to the base schematic**—In this type of backannotation, any component in the base schematic that has variant information is assigned a property denoting that the component has been assigned variant information.
- **Annotate properties to a variant**—In this type of backannotation, every component in the variant whose value has changed from the base schematic value or has the `DNI` status is assigned a property denoting this. In addition, the changed property values will be updated on the components.

In this chapter, you will learn to backannotate the variant information for the base schematic and variants.

Preparing for Backannotation



Before you backannotate the variant information to a design, ensure that Design Entry HDL is running. Variant Editor requires Design Entry HDL for backannotation.

Guideline

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

- `TOTAL_PAGES`
- `PAGE_NUMBER`

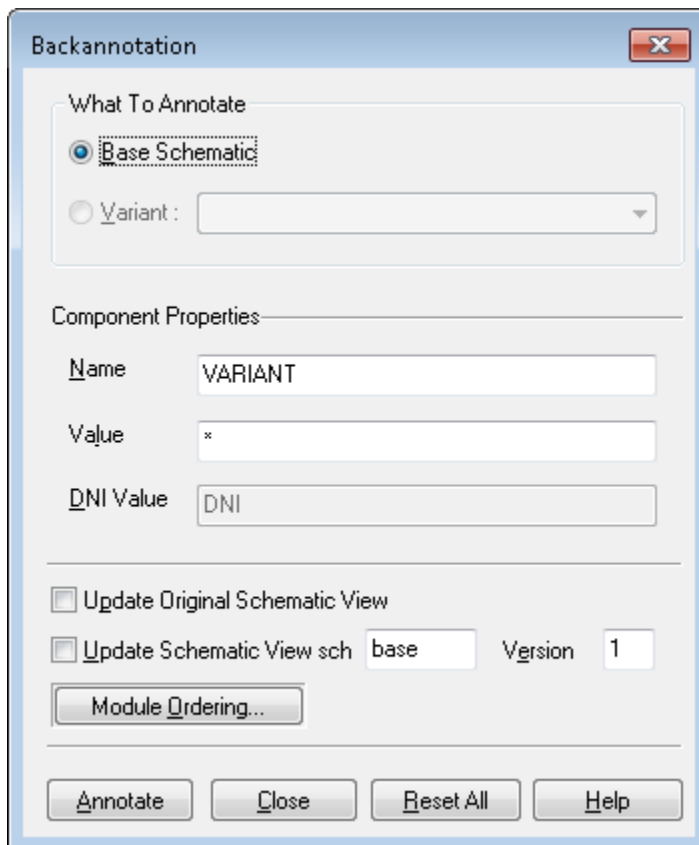
When Variant Editor creates a new flattened view of the design, it updates these properties to match the new view.

Note: In the following steps, when you open Design Entry HDL, you will note that the two properties, `TOTAL_PAGES` and `PAGE_NUMBER`, have already been added to each of the three module pages.

Opening the Backannotation Dialog

1. Launch Project Manager and load the `dsp.cpm` file from the `database` folder.
2. Click the *Design Entry* button.
3. In Project Manager, choose *Tools – Variant Editor*.
4. Choose *Tools – Annotate Variants*.

The Backannotation dialog is displayed.



The Backannotation dialog is divided into three main sections:

- ❑ The top section—This section includes the *What To Annotate* group that allows you to select the base schematic or a variant to be backannotated.
- ❑ The middle section—This section includes the *Component Properties* group box that 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.

- ❑ The lower section—This section allows you to define whether you want to backannotate to the original schematic view or you want to create a new flattened schematic view. However, for a variant, only the latter option applies.

Backannotating the Base Schematic

Task Overview

You will create a new flattened view for the base schematic with the property `MY_VARIANT = TRUE` assigned to all the components that have any variant information.

Steps

1. In the Backannotation dialog, select *Base Schematic* in the *What to Annotate* group box.
2. Type `MY_VARIANT` in the *Name* field.
3. Type `TRUE` in the *Value* field.
4. Select the *Update Schematic View sch* check box.

Note: 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.

5. Click the *Annotate* button.

Variant Editor reports successful backannotation.

6. Click *OK*.

A new flattened schematic view named `schbase_1.1.1` is created in Design Entry HDL. This view has three pages, one each for each module: `DSP`, `MEMORY`, and `ROM`. The `TOTAL_PAGES` and `PAGE_NUMBER` properties are also updated on each page of the design.

Note that the components `U3`, `U4`, and `U5` (`ADSP2101`, `74F04`, and `16L8`) do not have the property `MY_VARIANT = TRUE` annotated. This is because no variant information for these components is specified in Variant Editor. All components other than `U3`, `U4`, and `U5` are assigned the property `MY_VARIANT = TRUE`.

7. Close the Backannotation dialog.

Note: Now see this multimedia demonstration on [*Backannotating Variant Information*](#).

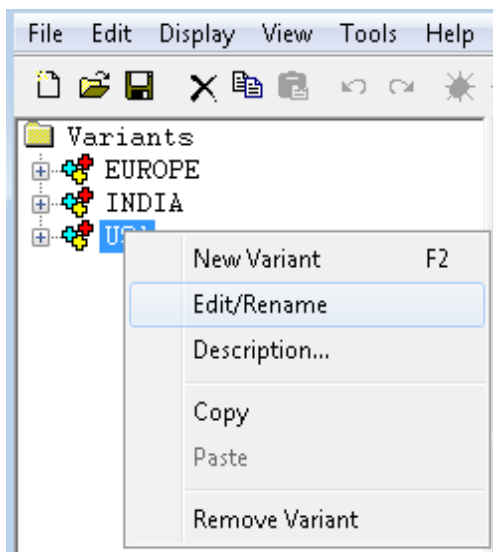
Backannotating a Variant

Task Overview

You will create a new flattened schematic view for the USA variant and assign the property `VAR_USA = TRUE` for all components that have any property change from the base schematic. You will assign the property `VAR_USA = DO_NOT_INSTALL` to all components that have the DNI status.

Steps

1. In *Variant Editor*, make sure that the Variants tab is selected.
2. Right-click on the USA variant and select *Edit/Rename*.



The Variant Details dialog appears.

3. In the *Variant Property Name* field, type `VAR_USA`.
4. Type `TRUE` in the *Variant Property Value* field.
5. Type `DO_NOT_INSTALL` in the *DNI Value* field.
6. Click *OK* to save the changes and close the Variant Details dialog.

Design Variance Tutorial

Backannotating Variant Information

7. Choose *Tools – Annotate Variants*.

The Backannotation dialog is displayed.

8. Select the *Variant* radio button in the *What to Annotate* group box.

9. Select the *VAR_USA* variant.

Note: The fields in the *Component Properties* group box are grayed out. The *Update Original Schematic View* check box is deselected and grayed out. Whenever you select a variant for backannotation, you cannot update the original schematic view. However, you can create a new flattened view for that variant.

10. Select the *Update Schematic View sch* check box.

Note: The default name of the new view is *usa* and the version number is *1*. You can, if you want, change the name and the version number.

11. Click the *Annotate* button.

Variant Editor reports successful backannotation.

12. Click *OK*.

A new flattened schematic view named *schusa_1* is created in Design Entry HDL. This view has three pages, one for each module: *DSP*, *ROM*, and *MEMORY*. The components *U3*, *U4*, and *U5* (*ADSP2101*, *74F04*, and *16L8*) do not have any change in property values because none of these components were customized in Variant Editor. The *U2* component (clock - *XTALOSC*) is displayed with the changed values for the *FREQUENCY* and *FREQUENCY_STABILITY* properties (*10MHz* and *50PPM* respectively). The *U1* component (*CONN20*) is also updated with the new values. Both, the *U1*, and *U2* components (*CONN20* and *XTALOSC*), have the new property *VAR_USA* = *TRUE*.

13. Close the Backannotation dialog.



Exercise 12

1. Which property values have changed for the *U1* component (*CONN20*)?
2. An alternate for the *J1* component (*CONN9*) has been defined in the Variant database. However, the properties for the *J1* component have not changed in the schematic view for the *USA* variant. Can you provide a reason for this?

When you have completed the exercise, refer to the answer to [Exercise 12](#) in Appendix A.

Design Variance Tutorial

Backannotating Variant Information

In the MEMORY page, all the three components (U6, U7, and U8) are assigned the property VAR_USA = DO_NOT_INSTALL. This is because none of the two functions, MEMORY1 or MEMORY2, was included in the USA variant.

In the ROM page, the U9 component (27C256) is assigned the property VAR_USA = DO_NOT_INSTALL while the U10 component (CYC263-5) is not assigned any new property.



Exercise 13

Explain why the U9 component (27C256) is assigned the VAR_USA = DO_NOT_INSTALL property while the U10 component (CYC263-5) has no change in the existing property.

When you have completed the exercise, refer to the answer to Exercise 13 in Appendix A.

Summary

You learned to use the Backannotation dialog to generate a flattened schematic view for the base schematic that includes variant information. You also learned to generate a new schematic view for variants.

What's Next

In the next chapter, [Creating BOMs and Variant Assembly Drawings](#), you will create the PCB Editor interface file using Variant Editor. This file will be used to create assembly drawings and BOM reports for each variant in PCB Editor.

Recommended Reading

For more information about backannotating to the base schematic, see [*Design Variance User Guide*](#).

Creating BOMs and Variant Assembly Drawings

Objectives

- Generate the PCB Editor interface file from Variant Editor
- Create BOM reports in PCB Editor
- Create a variant assembly drawing in PCB Editor

Nature of Chapter

Skill (includes concepts and practice)

Estimated Completion Time

45 minutes

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.
2. Create assembly drawings for individual variants.

In this chapter, you will learn to create the interface file using Variant Editor, and create assembly drawings and BOM reports from PCB Editor. These BOM reports are created in the standard PCB Editor format.

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. In Variant Editor, choose *File – Export*.

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

2. Click *Yes* to view the file.

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

Creating BOM Reports in PCB Editor

Task Overview

You will create a BOM report for the `INDIA` variant in PCB Editor. You will specify that the `DNI` components should also be included in the BOM report.

Design Variance Tutorial

Creating BOMs and Variant Assembly Drawings

Steps

1. Launch PCB Editor from Project Manager by clicking the *Layout* button.

An error message is displayed saying that the license does not include the ability to run the program.

2. Click *OK*.

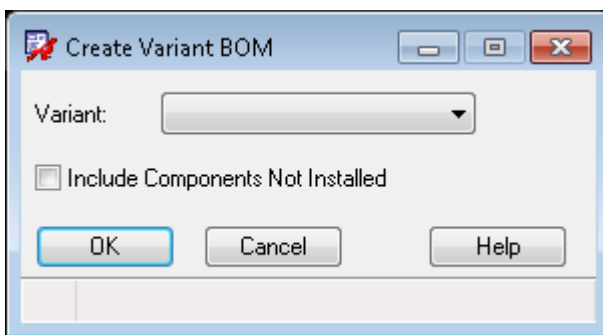
Cadence 23.10 ALLEGRO Product Choices appears.

3. Select *Allegro PCB Designer* and click *OK*.

Allegro PCB Designer appears.

4. Choose *Manufacture – Variants – Create Bill of Materials*.

The *Create Variant BOM* dialog is displayed.



5. Select *India* from the *Variant* drop-down list.
6. Select the *Include Components Not Installed* check box.
7. Click *OK*.

A variant BOM report named *var-india.rpt* is created in the

`<your_work_dir>\vedit_db\desvar_tutorial\nonsynchronized\worklib
\dsp\physical.`

Report Explanation

The `var-india.rpt` report has two sections of text. The first section displays the package name, device type, component value and tolerance, component class and reference designator for each component installed in the `INDIA` variant. The second section lists the reference designators of all the `DNI` components. Note that the `J1` and `U10` components

Design Variance Tutorial

Creating BOMs and Variant Assembly Drawings

figure in this section. The J1 component was explicitly defined as DNI in the INDIA variant. The U10 component is part of the alternate group ROM from which only the preferred component U9 is included in the INDIA variant. Therefore, U10 is assigned the DNI status.

If you had not selected the *Include Components Not Installed* check box, there would not have been a section 2 in the report, which lists DNI components.

```
|                                     Components Not Installed
|
|
Page      2  |
|-----|
|-----|
| 'INDIA' variant of
D:/vedit_db/desvar_tutorial/database/worklib/dsp/physical/test.br
d |
|                                     Feb 02
16:05:42 2016 |
|-----|
|-----|
|                                     |
|      |                                     |
|      |                                     |
refdes |      |
|-----|
|-----|

J1

U9
|-----|
|-----|
| total not installed count      2
|
|-----|
|-----|
```



Exercise 14

Create variant BOM reports for the USA and EUROPE variants and verify whether all the components have been correctly created.

When you have completed the exercise, refer to the answer to Exercise 14 in Appendix A.

Creating Variant Assembly Drawings

Task Overview

You will create an assembly drawing for the `USA` variant in PCB Editor. Since the `dsp` design does not have any bottom-sided component, create the assembly drawing for only the components on the top-side of the board. You will select the following properties: `DEV TYPE`, `REF DES` and `USER PART` to be included in the assembly drawing so that they are visible.

Prerequisite for Creating a Variant Assembly Drawing

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.

In PCB Editor, before you create a variant assembly drawing, you should display the Color dialog (*Display – Color/Visiblity*) and set the *Off* option in the *Global Visibility* combo box. Next, you can explicitly select the subclasses that you want to include in the variant assembly drawing that you are generating.

Steps

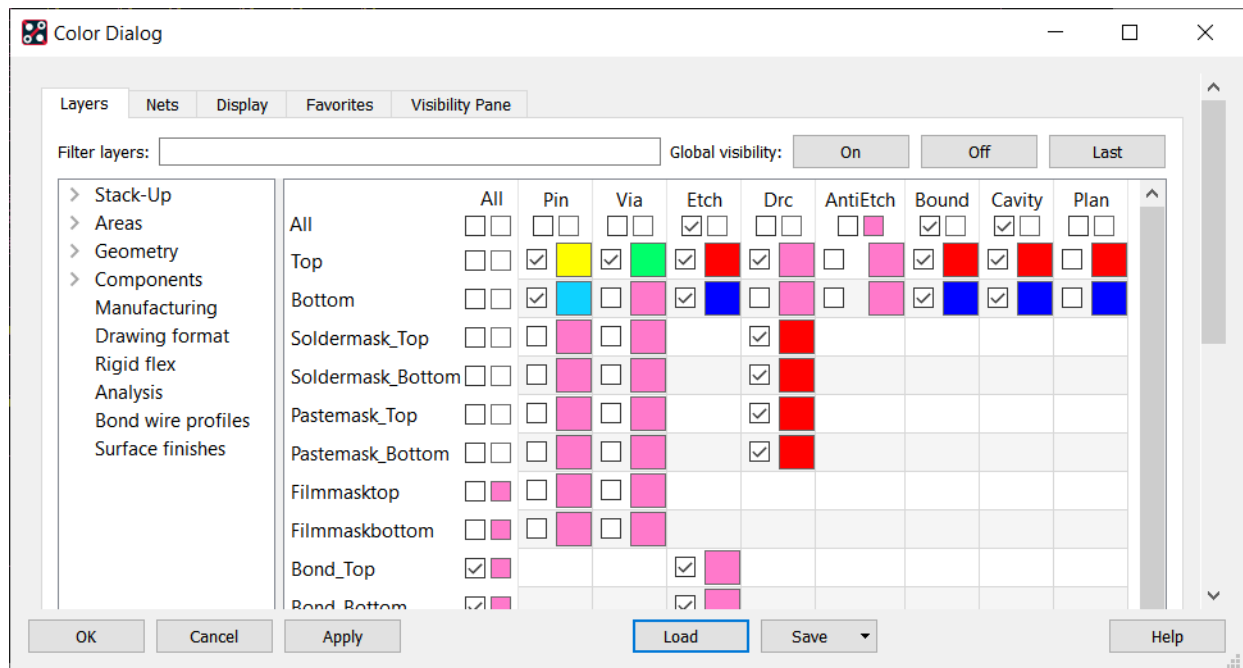
Selecting Subclasses for Variant Assembly Drawing

1. In the PCB Designer window, choose *Display – Color/Visibility*.

The Color dialog is displayed.

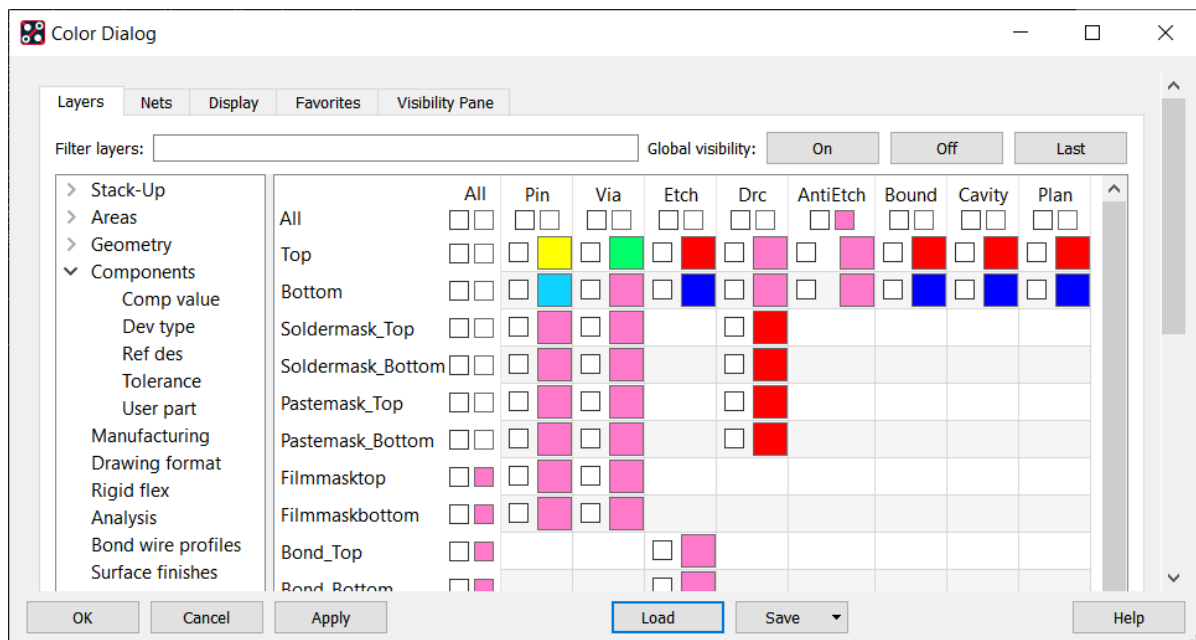
Design Variance Tutorial

Creating BOMs and Variant Assembly Drawings



You can use this dialog to control both the colors and the visibility of the various classes and subclasses in Allegro PCB and Package Designer.

2. Set the *Global Visibility* to *Off*.
3. Click *Components* in the left pane of the Color dialog.



Design Variance Tutorial

Creating BOMs and Variant Assembly Drawings

4. Select the ASSEMBLY_TOP check box for DEV TYPE, REF DES, and USER PART.

	All	CmpVal	RefDes	Tol	UserPart	DevType
All	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>
Assembly_Bottom	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>
Assembly_Embedded	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>
Assembly_Top	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> <input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> <input checked="" type="checkbox"/>
Display_Bottom	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>
Display_Embedded	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>
Display_Top	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>
Silkscreen_Bottom	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>
Silkscreen_Top	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>	<input type="checkbox"/> <input checked="" type="checkbox"/>

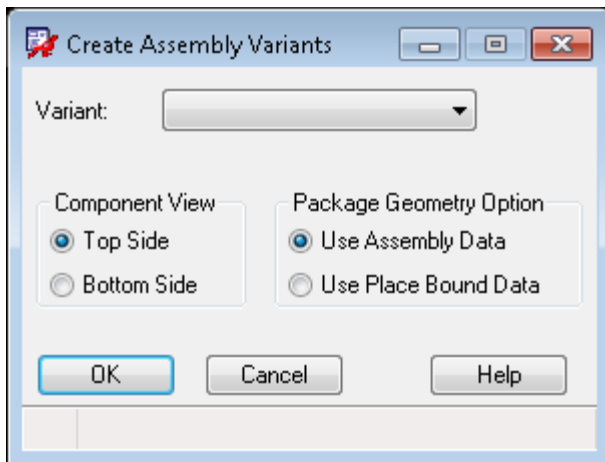
5. Click OK.

You have defined the subclasses that will be visible in the assembly drawing. The properties, DEV TYPE, REF DES, and USER PART, will be visible in the assembly drawing.

Creating the USA Assembly Drawing

1. Choose *Manufacture – Variants – Create Assembly Drawing*.

The Create Assembly Variants dialog is displayed.



You can use the Create Assembly Variants dialog to select the variant for which the assembly drawing is to be created. You can define whether you are creating an assembly drawing for the top-sided components or the bottom-sided components. You can also define whether you are using the place bound outlines or the assembly outlines.

2. Select *VAR_USA* in the *Variant* drop-down list.
3. Retain the default options: *Top Side* and *Use Assembly Data*.
4. Click *OK*.

Note the message “Generating assembly drawing MANUFACTURING/USA_TOP for design variant ‘USA’ using assembly data” in the command window. The assembly drawing is created as a subclass (named USA_TOP where USA represents the variant name and TOP represents the component view) under the MANUFACTURING class.

Viewing the Assembly Drawing

1. Display the Color dialog by selecting the *Display – Color/Visibility* option.
2. Set *Global Visibility* to *Off*.
3. Click *Manufacturing* in the left pane of the Color dialog.

Design Variance Tutorial

Creating BOMs and Variant Assembly Drawings

4. Select the *Var_Usa_Top* check box.

Analysis	Freeform_Legend	<input type="checkbox"/>	
Bond wire profiles	No_Gloss_All	<input type="checkbox"/>	
Surface finishes	No_Gloss_Bottom	<input type="checkbox"/>	
	No_Gloss_Internal	<input type="checkbox"/>	
	No_Gloss_Top	<input type="checkbox"/>	
	No_Probe_Bottom	<input type="checkbox"/>	
	No_Probe_Top	<input type="checkbox"/>	
	Photoplot_Outline	<input type="checkbox"/>	
	Probe_Bottom	<input type="checkbox"/>	
	Probe_Top	<input type="checkbox"/>	
	Var_Usa_Top	<input checked="" type="checkbox"/>	
	Xsection_Chart	<input type="checkbox"/>	

5. Click *OK*.

The *USA_TOP* assembly drawing is displayed. This assembly drawing includes the J1, U1, U2, U3, U4, U5 and U10 components with the following three properties: *DEV TYPE*, *REF DES*, and *USER PART*. The components U6, U7, U8, and U9, being *DNI* components, are not displayed.

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

You can create and verify the assembly drawings for the other variants, *INDIA* and *EUROPE*.

Note: Now see this multimedia demonstration about [Creating a Variant Assembly Drawing](#).

Summary

You learned to create the PCB Editor interface file using Variant Editor. This file is used to create assembly drawings and BOM reports for each variant in PCB Editor. You also learned to create assembly drawings and BOM reports for each variant in PCB Editor.

What's Next

This is the last chapter in the Design Variance Tutorial. In [Appendix A](#) on page 109, you will find answers to all the exercises in the tutorial.

Recommended Reading


For more information about creating assembly drawings and BOM reports in PCB Editor, see the [Allegro PCB and Package User Guide](#).

Appendix A

This appendix contains solutions to the exercises in this tutorial.

Exercise 1

1. Select a component (U6, U7, or U8) under the MEMORY2 function in the left pane. Note that two rows are displayed in the top-right pane. The original row with the SPEED value of 15ns is assigned the status '-' while a new row with the SPEED value of 20ns is assigned the status Pref.

Refdes	Status	Part_name	Part_number	Access_time	Class
 U6	-	CY7C199	cy7c199L-15PC		IC
	Pref	CY7C199	cy7c199L-20PC		IC

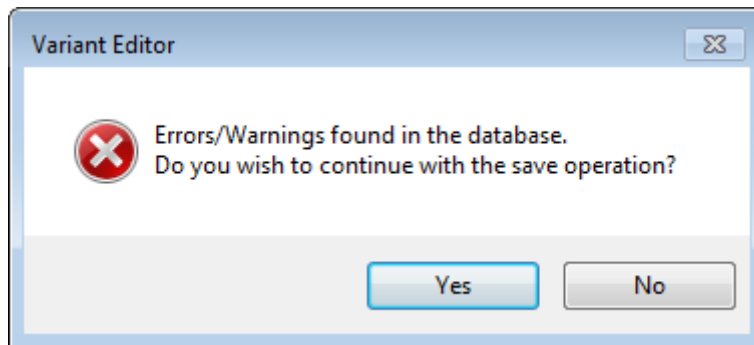
2. To create a new function named MEMORY3 from MEMORY2, do the following:
 - a. Right-click on the MEMORY2 function in the left pane and choose *Copy*.
 - b. Right-click and choose *Paste*.
A new function with the name CopyOfMEMORY2 is displayed.
 - c. Right-click and choose *Rename*.
 - d. Rename the function to MEMORY3.

This procedure was covered to demonstrate a capability of Variant Editor. Right-click MEMORY3 and choose *Remove Function*.

Return to [Creating Functions](#) on page 34.

Exercise 2

An error message is displayed about errors in the database. These errors are about the three components, U6, U7, and U8, which are customized in both the functions and are included in the same variant.



If you have a common component or alternate group in two functions then only one of the two functions can be added to a variant. In the present exercise, you added two functions MEMORY1 and MEMORY2. Both of these functions include the three components U6, U7, and U8. Therefore, an error occurred.

Return to [Creating a Variant](#) on page 36.

Exercise 3

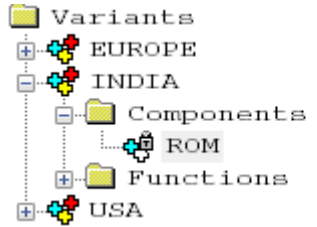
1. To customize the alternate group ROM in the INDIA variant, do the following:
 - a. Select the INDIA variant in the top-left pane.
 - b. Drag the row displaying the ROM alternate group from the bottom-right pane and drop it in the top-right pane.
 - c. Right-click the row corresponding to the U10 component and choose *Make First Alternate*.

A dialog appears prompting you to retain the existing status (Pref) of the U10 component. Click the No button to ensure that the existing status of the U10 component is not retained and that it is assigned the status Alt1. The status of the U9 component changes to Pref.

Design Variance Tutorial

Appendix A

Note that the status of the U9 and U10 components has changed.



Refdes	Status	Part_name	Part_number
U9	Pref	27C256	27c256-90/L
U10	Alt1	CY7C263-35	cy7c263-251mb

2. If an alternate group is not added to a variant for customization, its information as defined in the *Alternates* tab applies for that variant. Therefore, the USA variant will have information as defined in the *Alternates* tab.

Return to [Customizing the Information in a Variant](#) on page 39.

Exercise 4

To customize the value of the U2 (XTALOSC) component in the USA variant, first add the U2 component to the USA variant. Next, you can change the value of the U2 component in the USA variant by using two methods.

1. Change Value

- a. Right-click the U2 component in the top-right pane and choose *Change Value*.

The Physical Part Filter dialog is displayed.

- b. Select the row that has the FREQUENCY value 10MHz and the FREQUENCY_STABILITY value 50ppm.
- c. Click *OK*.

The existing row corresponding to the U2 component is assigned the '-' status while a new row with *Pref* status is added. This row has the FREQUENCY value 10MHz and the FREQUENCY_STABILITY value 50ppm.

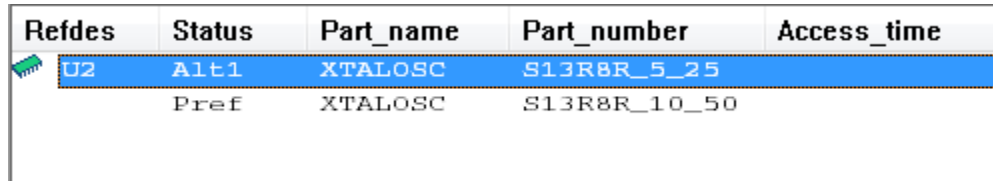
Refdes	Status	Part_name	Part_number	Access_time
U2	-	XTALOSC	S13R8R_5_25	
	Pref	XTALOSC	S13R8R_10_50	

- d. Right-click the row with the '-' status and choose *Make First Alternate*.

Design Variance Tutorial

Appendix A

The status of the row changes to *Alt1*.



Refdes	Status	Part_name	Part_number	Access_time
U2	Alt1	XTALOSC	S13R8R_5_25	
	Pref	XTALOSC	S13R8R_10_50	

2. Add Alternates

- a. Right-click the U2 component in the top-right pane and choose *Add Alternates*.

The Physical Part Filter dialog is displayed.

- b. Select the row that has the FREQUENCY value 10MHz and the FREQUENCY_STABILITY value 50ppm.

- c. Click *OK*.

A new row with *Alt1* as the status is added with the values you selected.

- d. Change the status of the new row to preferred by right-clicking the row and selecting *Make Pref*.

The status of the new row changes to *Pref*, and the status of the earlier preferred row changes to *Alt1*.

Return to [Defining a DNI Component in a Variant](#) on page 41.

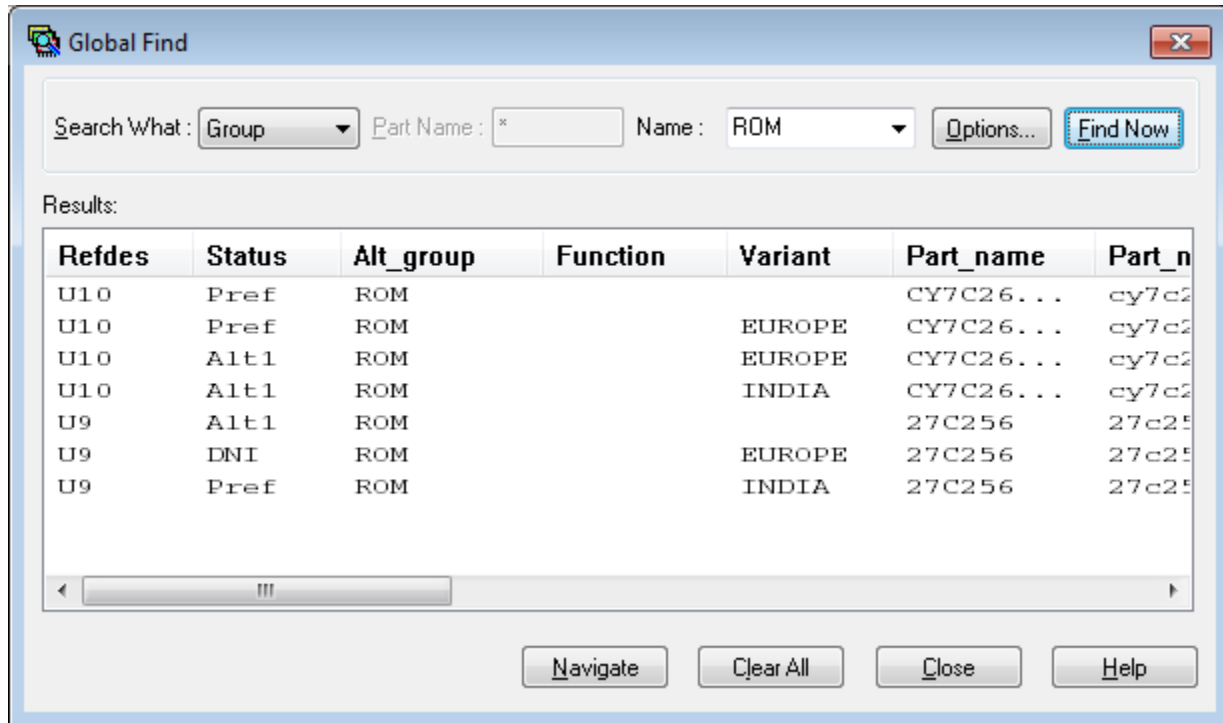
Exercise 5

1. To display the customized changes for all components in the ROM alternate group, do the following:
 - a. Choose *Tools – Global Find*.
 - b. Select the *Group* option in the *Search What* field.
 - c. Select ROM from the *Name* drop-down list and click *Find Now*.

Design Variance Tutorial

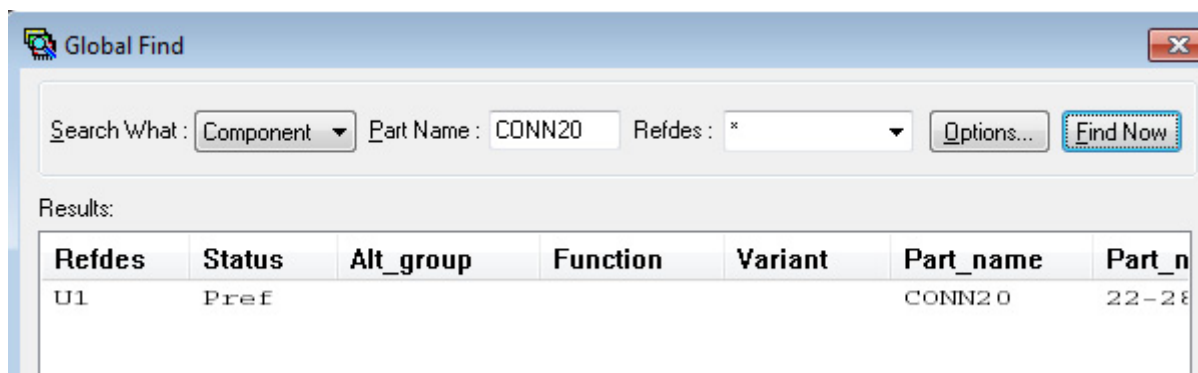
Appendix A

All the components in the ROM alternate group in the *Alternates* tab and in all the variants are displayed.



2. To display the customized changes for the CONN20 component, do the following in the Global Find dialog:
 - a. Select the *Component* option in the *Search What* field.
 - b. Type CONN20 in the *Part Name* field and click the *Find Now* button.

One PPT row corresponding to the CONN20 component in the *Alternates* tab and in all the variants is displayed.



3. Close the Global Find dialog.
4. In Variant Editor, to undo the changes made using the Global Find dialog, or in any pane in Variant Editor, click the `Undo` button.

To save the variant database, choose *File – Save*.

Return to [Global Searching](#).

Exercise 6

To perform `Refdes match` synchronization, do the following:

1. Without saving the variant database, choose *File – Reload Design*.
The *Conflict Found While Loading Design* dialog is displayed.
2. Select the *Winning on Refdes match (Recommended)* radio button.
3. Click *OK* to confirm your selection.
4. Variant Editor displays a message that states that errors/warnings were detected during importing/loading the variant database. To read the actual error/warning, click *OK*.

On checking the results, you will discover that Variant Editor is unable to synchronize the U20 component because its `Refdes` does not match the `Refdes` of the U2 component. However, Variant Editor is able to synchronize the changes to the U1 and J1 components. All customized changes on these components is retained.

Return to [Synchronizing the Variant Database With the Changes in the Schematic](#) on page 48.

Exercise 7

1. When you try to replace the value of the U7 component, you first get the following message:

Compatible footprints for "DIP28_3" are : DIP14_3, DIP28_3


When you click *OK* to close the message box, the Physical Part Filter dialog is displayed. This dialog does not include any PPT row which has the `JEDEC_TYPE` value `DIP14_3`. The reason for this originates from the Physical Part Filter dialog, which lists only those PPT rows that have been defined for the U7 component (that is, the component with the `PART_NAME` `CY7C199`). Since no PPT row with the `JEDEC_TYPE` value `DIP14_3` is defined for the U7 component, no such row is listed in the Physical Part Filter dialog.

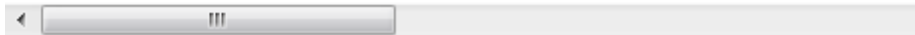
Design Variance Tutorial

Appendix A

2. To create an alternate for the U4 component with the JEDEC_TYPE DIP28_3 and PART_NM 131, do the following:
 - a. Right-click the PPT row corresponding to the U4 component in the top-right pane and choose *Add Alternates*.
 - b. Select *OK* to after reading the message about compatible footprints.
 - c. Select the PPT row with the JEDEC_TYPE DIP28_3 and PART_NM 131.
 - d. Click *OK*.

A PPT row with the status `Alt1` is displayed. Note that the `JEDEC_TYPE` property has the value `DIP28_3` and the `PART_NM` property has the value `131`.

Refdes	Status	Jedec_type	Part_name	Part_number
 U4	-	DIP14_3	74F04	123
	Pref	DIP28_3	74F04	130
	Alt1	DIP28_3	74F04	131



Return to [Using Compatible JEDEC TYPES](#) on page 53.

Exercise 8

You can perform all these tasks using the `bom.template` file and the Callouts file. You will learn to customize the BOM template file in subsequent sections in the tutorial. By customizing the BOM template and Callouts files, you can address not only the questions asked in the exercise but can also add many more subtle and useful customizing features to the BOM report.

Return to [Creating the BOM for the Base Schematic](#) on page 62.

Exercise 9

1. To generate the BOM report in HTML format, do the following:
 - a. Choose the *HTML* radio button in the *Report Format* section of the *BOM-HDL* dialog.
 - b. Click *Yes* to view the BOM report.

Note: If no HTML viewer is selected for your tool, the *Select HTML Viewer* dialog is displayed. Select the HTML viewer that is the executable file of your browser. For example, you can select `chrome.exe` as your browser.

You will note that the file is displayed in a neat format with a table segregating all the properties. The - sign does not appear as the row separator because the HTML format overrides any row and column separators. Therefore, you need not specify row and column separators for the HTML format.

Note: Try creating a BOM report in spreadsheet format, and you will find that row and column separators are not required in that format.

2. To regenerate the BOM report in HTML format and to ensure that the row is sorted on the *Ref Des* title in an alphabetic ascending order, do the following:
 - a. Display the Physical Part Specifications tab in the Customize Template dialog.
 - b. Select the row corresponding to the `BOM_INST` property in the *Report Columns* section and click the *Up* arrow button.
 - c. Select *Ascending* from the *Order* list in the *Sorting* group box.
 - d. Click the *Save* button to save the template file, and click *Close* to close the Customize Template dialog.
 - e. Select the *HTML* radio button and click *Generate*.

The BOM report is generated as a HTML file, and is sorted on the *Ref Des* title in an alphabetic ascending order. Note that the serial number is not listed for rows where the key property, `BOM_INST`, is not unique.

Return to [Changing the Report Parameters](#) on page 65.

Exercise 10

1. There are three different types of mechanical parts that are handled by BOM-HDL. These mechanical parts are:
 - a. **Associated Mechanical Parts**—These parts have their `CLASS` property defined as `MECH`. These parts are associated to an electrical part through the `MECH_PART` property.
 - b. **Mechanical Kits**—A mechanical kit includes a set of mechanical parts, which are defined using the `MECH_PART` property. You can associate a mechanical kit to an electrical part using the `MECH_KIT` property.

- c. Callouts**—These are mechanical parts that are defined using the Callouts Editor dialog. The Callouts Editor dialog displays only those parts that have their `CLASS` property defined as `MECH`.
- 2. The two different modes in which BOM-HDL displays mechanical parts in the BOM report are:
 - a. Intersperse Mode**—In the Intersperse mode, mechanical parts are listed along with the electrical parts to which they are associated. Callouts, however, are separately listed at the bottom of the BOM report. Select the Intersperse mode by checking the *Intersperse Associated Mechanical Parts* box.
 - a. Non-intersperse Mode**—In the Non-intersperse mode, mechanical parts are listed along with the callouts in the Callouts section of the BOM report. The Non-intersperse mode is selected by unchecking the *Intersperse Associated Mechanical Parts* box.
- 3. To ensure that you have the necessary mechanical part information in the BOM reports, you need to edit the following files:
 - a. PTF files**—These files need to be edited to ensure that you have included all information about the associated mechanical parts or mechanical kits. This should be your first step and should be completed before you use the BOM-HDL tool. You must package your design to ensure that all ptf changes are accepted by the variant database.
 - b. Callouts file**—You need to edit the Callouts file to add or delete callouts in the BOM report. The callouts appear in the Callouts section of the BOM report.
 - c. BOM Template file**—You need to edit the BOM template file to specify whether or not mechanical parts will be interspersed along with electrical parts. You also specify the path to the Callouts file and title of the Callouts section in the BOM report.

Return to [Editing the Callouts Information](#) on page 77.

Exercise 11

The steps for generating a variant BOM report for the `EUROPE` variant are:

1. In the *Variant Settings* tab of the Customize Template dialog, select the *Include* radio button in the *Alternates* group box.
2. Ensure that the *Include DNI Component List* check box is selected.
3. Click *Save* to save the changes in the BOM template file, and click *Close* to close the Customize Template dialog.

4. Click the *Variant BOM* radio button, and then select the `EUROPE` variant from the drop-down list next to the radio button.
5. To generate the BOM report, click the *Generate* button; and, accept *Yes* to view the BOM report.

Report Explanation

The variant BOM report is divided into four sections:

1. Header list—This list has seven PPT rows and is the same as the variant BOM report for the `INDIA` variant.
2. Properties list—This list displays PPT rows for all the properties that are included in the `EUROPE` variant, which do not have the `DNI` status. The `Var Status` column displays all three values: `Pref` or `Alt1` or `Alt2`. The character `*` in the `Var Status` column signifies that the value for the concerned components is different from the base component. This character appears for the following PPT rows:
 - ❑ PPT rows for the `J1` component that have `Alt1` and `Alt2` as the status—These rows were customized in the *Alternates* tab. BOM-HDL therefore assigns the character `*` in the `Var Status` column against the PPT rows.
 - ❑ The `Pref` row for the `U1` component—This row was also customized in the *Alternates* tab.
 - ❑ The `Pref` row for the `U6`, `U7`, and `U8` components—These rows are part of the `MEMORY2` function, which was customized in the *Functions* tab. Since `MEMORY2` is included in the `EUROPE` variant, the `Pref` row corresponding to the `U6`, `U7`, and `U8` components display the character `*` in the `Var Status` column.
3. DNI Component list—This list displays PPT rows for the `DNI` components `U9` and `U10`. The `U9` component was explicitly assigned the `DNI` status in the `ROM` group. The PPT row for the `U10` component with the `Alt1` status is assigned the `DNI` status because, from an alternate group, only a component with the Preferred status is included in a variant. The `EUROPE` variant therefore includes only the PPT row for the `U10` component with the `Pref` status.
4. Callouts list—This list is the same as the variant BOM report for the `INDIA` variant.

Return to [Creating a Variant BOM Report](#) on page 84.

Exercise 12

- a. The property `VAR_USA = True` is added to the U1 component (CONN20) because the preferred value of the component was changed in the *Alternates* tab. Its part number has changed to 22-28-4200, which is different from that in the base schematic (22-28-5200).
- b. The J1 component (CONN9) does not have any change in value as the preferred value for that component is the same as the base schematic value. Backannotation of a variant causes the annotation of the *Property Name* and *Property Value* (as specified in the *Component Properties* group) on all the components that have a preferred value different from the base value or that are assigned the DNI status in that variant. Since the J1 component does not have any change in the preferred value or is not DNI, the properties for the J1 component are not changed in the USA variant.

Return to [Backannotating a Variant](#) on page 95.

Exercise 13

In the USA variant, the U9 component (27C256) is assigned the `VAR_USA = DO_NOT_INSTALL` property while the U10 component (CYC263-5) is not assigned any new property. The reason for this stems from the fact that both U9 and U10 are part of the ROM alternate group. Only the preferred component of an alternate group is added to a variant. The preferred component in ROM is U10 and this component has the same value as that of the base schematic because it was not customized in the USA variant. Therefore, U10 is not assigned any property.

The other component in the ROM alternate group U9 has the status Alt1. This component is not included in the USA variant. Therefore, during backannotation, the `VAR_USA = DO_NOT_INSTALL` property is assigned to U9.

Return to [Backannotating a Variant](#) on page 95.

Exercise 14

You can create BOM reports for the USA and EUROPE variant by doing the following:

1. Select *USA* in the *Create Variant BOM* dialog.
2. Select the *Include Components Not Installed* check box to specify that the DNI components are to be included in the BOM report.
3. To create the Variant BOM report, click *OK*.

Design Variance Tutorial

Appendix A

The variant BOM report for the *USA* variant is created. You can repeat steps 1 through 3 to create the variant BOM report for the *EUROPE* variant.

Note that the variant BOM report for the *USA* variant lists U6, U7, U8, and U9 as DNI components. Similarly, the variant BOM report for the *EUROPE* variant lists the U9 component as DNI.

Return to [Creating BOM Reports in PCB Editor](#) on page 100.

Index

A

- alternate groups
 - defining [32](#)
- alternate value [17](#)
- alternate values
 - assigning [22](#), [29](#)
- assembly drawing
 - creating [106](#)
- assigning
 - DNI status to a component DNI
 - assigning status [22](#)
- assigning alternate values [22](#)
- associated mechanical parts
 - including in BOM report [72](#)

B

- backannotating
 - base schematic [94](#)
 - variants [95](#)
- BOM report
 - customizing
 - changing report parameters [65](#)
 - for base schematic [58](#)
 - creating [62](#)
 - for variant [58](#)
 - Part_number based Comparison BOM report [58](#)
 - reading mechanical part information [73](#), [75](#)
- BOM reports
 - creating in PCB Editor [100](#)
- bom template
 - customizing [63](#)
- BOMHDL
 - launching [60](#)

C

- compatible JEDEC_TYPES [53](#)
- creating
 - assembly drawing [106](#)

- BOM report
 - for base schematic [62](#)
 - for variant [84](#)
 - functions [34](#)
 - variants [36](#)
- customizing
 - bom template [63](#)
 - information in variants [39](#)
- customizing BOM report
 - changing report parameters [65](#)
 - editing Callouts information [77](#)
 - including associated mechanical parts [72](#)
 - including mechanical kits [72](#)
- customizing display
 - of properties in Variant Editor [26](#)

D

- defining
 - alternate groups [32](#)
- defining alternate groups
 - alternate groups
 - defining [22](#)
- Design Variance tutorial
 - glossary [17](#)
 - how to use [14](#)
 - overview [10](#)
- dialog boxes
 - Conflict Found While Loading Design [50](#)
 - Customize Template [64](#)
 - Global Find [44](#)
 - Options [46](#)
 - Physical Part Filter [31](#)
- dialogs
 - Callouts Editor [77](#)

E

- editing Callouts information [77](#)

F

- files
 - base schematic BOM report [63](#)
 - Callouts [77](#)
 - cjedectype.txt [53](#)
 - PCB Editor interface file [100](#)
 - ptf [73](#)
 - variant BOM report [85](#)
 - variant comparison BOM report [88](#)
- Filter list [26](#)
- filters [80](#)
 - setting on parts [80](#)
- function [17](#)
- functions
 - creating [22](#), [34](#)

G

- global search [43](#)

L

- launching
 - BOM-HDL [59](#)
 - BOMHDL [60](#)

M

- mechanical kits
 - including in BOM report [72](#)

P

- panes
 - left [25](#)
 - top-right [25](#)
- parts
 - filtering [80](#)
- preferred value [17](#)
- properties
 - selecting [26](#)

S

- setting [80](#)

V

- variant database [22](#)
- Variant Editor
 - Launching [23](#)
 - Main Window [24](#)
 - Menu Bar [24](#)
 - Toolbar [25](#)
- variants
 - creating [36](#)
 - customizing information [39](#)
- viewing
 - assembly drawings [106](#)