



Destination Management

Table of Contents

Welcome to the World of Circuit Design

Chapter 1 Performing Design Variation Manager

1. Design Variation Manager	1 - 1
●Design Variation Manager	1 - 1
2. Defining Design Variations	1 - 5
●Variant properties definition file (vmsys.rsc)	1 - 5
●Variant rule file (vmdata.rsc)	1 - 6
●Edit Variant rule file	1 - 8
3. Setting Design Variations	1 - 15
●Design Variations	1 - 15
●Setting Design Variations	1 - 16
●Confirming Design Variation setting	1 - 19
4. Setting Propweties for Destinations	1 - 20
●Displaying a Design Variations	1 - 20
●Setting Different Properties for Each Design Variations(Sheet Editor)	1 - 23
●Setting Different Properties for Each Design Variations (Component Designerr).....	1 - 29

Chapter 2 Outputting Design Variations

1. Printting Design Variations.....	2 - 1
●Printing From the Sheet Editor	2 - 1
●Printing From the Plotout dialog	2 - 2
*Printing No Mount Components.....	2 -3
2. Outputting Design Variations (Netlist)	2 - 4
● Outputting Design Variations (Netlists,Component Lists)	2 - 4
3. Interface with the PCB Design Tool	2 - 9
● To PCB System	2- 9

Welcome to the World of Circuit Design

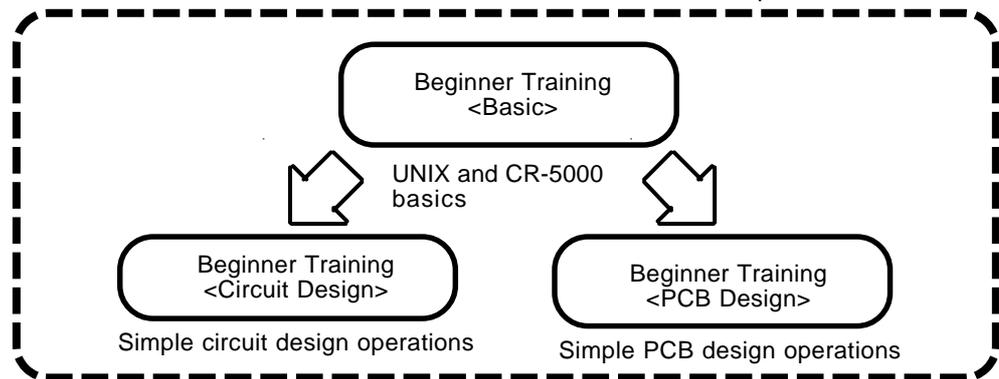
Welcome to the world of CR-5000/Circuit Design.

This manual will give you a further introduction into the world of circuit design using CR-5000.

This CR-5000/System Designer <Destination Management> provides the users with the step-by-step information about the CR-5000/System Designer to generate the destination specific data, allowing smoother operation and management.

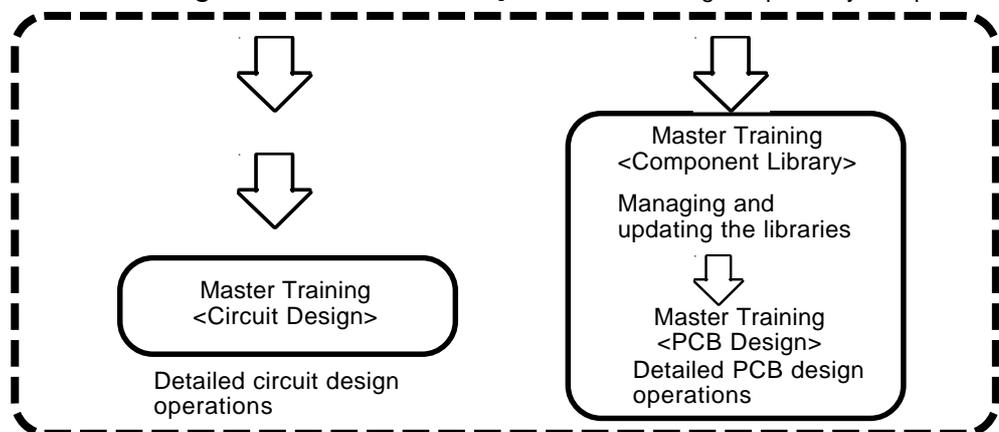
Beginner Training

Aims to get you to the level of an operator's assistant.

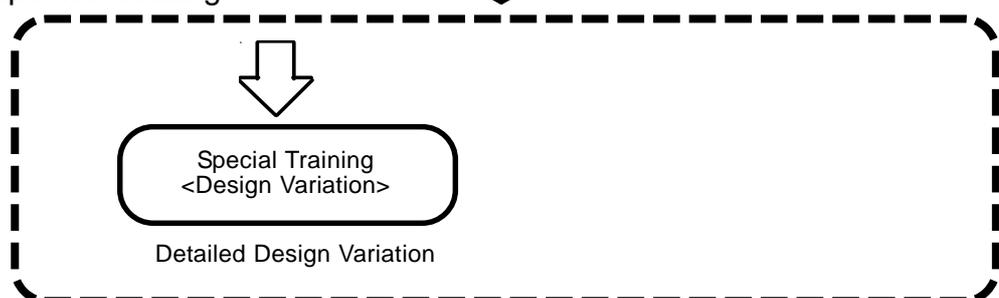


Master Training

Aims to familiarize you with the knowledge required by an operator.



Special Training



The next step is for you to review your intended training course.

●Circuit design only

1. Beginner Training "Basic"
2. Beginner Training "Circuit Design"
3. Master Training "Circuit Design"

●PCB design only

1. Beginner Training "Basic"
2. Beginner Training "PCB Design"
3. Master Training "Component Library"
4. Master Training "PCB Design"
5. Master Training "CAM"

●Circuit design and PCB design (Select either of the following options.)

Starting with circuit design

1. Beginner Training "Basic"
2. Beginner Training "Circuit Design"
3. Master Training "Circuit Design"
4. Beginner Training "PCB Design"
5. Master Training "Component Library"
6. Master Training "PCB Design"
7. Master Training "CAM"

Starting with PCB design

1. Beginner Training "Basic"
2. Beginner Training "PCB Design"
3. Master Training "Component Library"
4. Master Training "PCB Design"
5. Master Training "CAM"
6. Beginner Training "Circuit Design"
7. Master Training "Circuit Design"

1. Design Variation Manager

● Design Variation Manager

In circuit design, even though the circuit itself may not change, circuit components (part names) or properties such as circuit constants may differ depending on the product's functions (grade, for example) or the target market (whether intended for USA or Europe, for example).

To allow for such cases, System Designer enables variant-specific values to be set for the properties of components in a schematic. This enables items such as component lists and schematic output to be produced for specific variants from a single schematic.

Setting variant-specific properties enables the following:

1. Properties can be set and displayed depending on the schematic variant.
2. Components used in the schematic can have different properties set for each variant type.
3. Variant-specific properties can be input or edited from Component Browser.
4. Variant-specific schematics can be output.
5. The netlist processor can produce variant-specific net output.
6. The netlist processor can output variant-specific component lists.

The image shows a circuit schematic with two dialog boxes overlaid. The 'Set Design Variation Type' dialog box is on the left, and the 'Select Destination' dialog box is on the right. Both dialog boxes contain tables for setting properties for different variants.

Set Design Variation Type Dialog:

NTSC/PAL/UL	NTSC	PAL
Chip Set	Chip Set I	Chip
Voltage	100V	120V
Audio	Hifi	Mono

Select Destination Dialog:

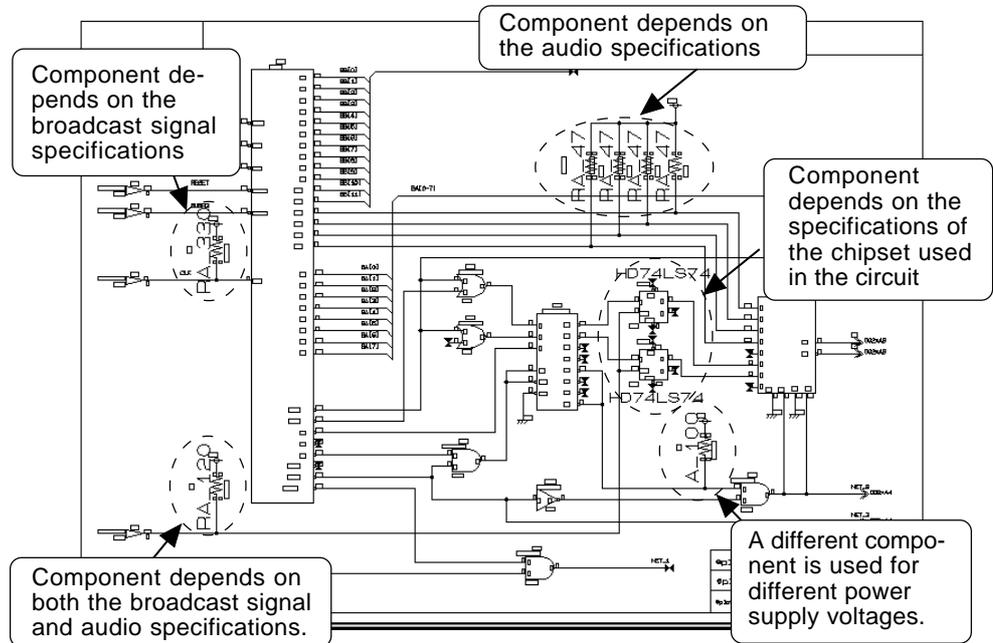
Destination	A Series (Japan)	NTSC	PAL	UL
Chip Set	Chip Set I	Chip Set II	Chip Set III	
Voltage	100V	120V	200V	
Audio	Hifi	Mono		



The design in \$CR5ROOT\data\SDopeguide\sample2.cir use of Design Variation . Use this circuit as a reference.

Design Variation in CR-5000/System Designer can set variant information for each component. This enables variants to be specified based on factors such as product functions and target market.

 **EXAMPLE**
sample2.cir



A variant can be specified as a combination of functions. The above diagram shows an example in which individual components in the circuit depend on parameters such as the broadcast signal and audio specifications.

In CR-5000/System Designer, Design Variation Manager is expressed by the terms "Design variation type" and "Destination of Design variation".

Design variation type

- The various functions of a circuit are called "Design variation type". The above circuit has the following four Design variation types and each Design variation type has a set of available options.

Broadcast signal specifications	(NTSC, PAL, UL specifications)
Audio specifications	(HiFi, Mono)
Chip specifications	(Chipset 1, Chipset 2, Chipset 3)
Power supply voltage	(100V, 120V, 200V)

Destination of Design variation

- Each combination of Design variation type options (functions) is called a "Destination of Design variation". For example, the following combination of Design variation types represents one Destination of Design variation. A different combination would represent a different Destination of Design variation.

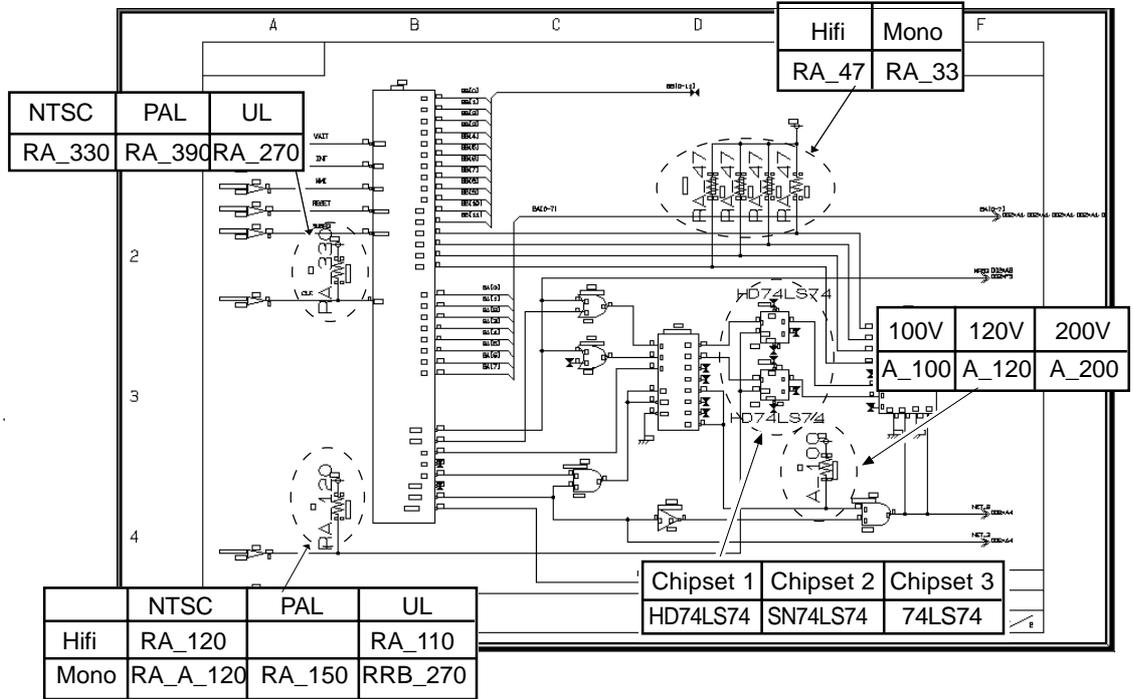
Example variant #1

Broadcast signal = NTSC
Chipset = Chipset 1
Power supply voltage = 100V
Audio = HiFi

Example variant #2 (a different variant)

Broadcast signal = UL
Chipset = Chipset 2
Power supply voltage = 120V
Audio = HiFi

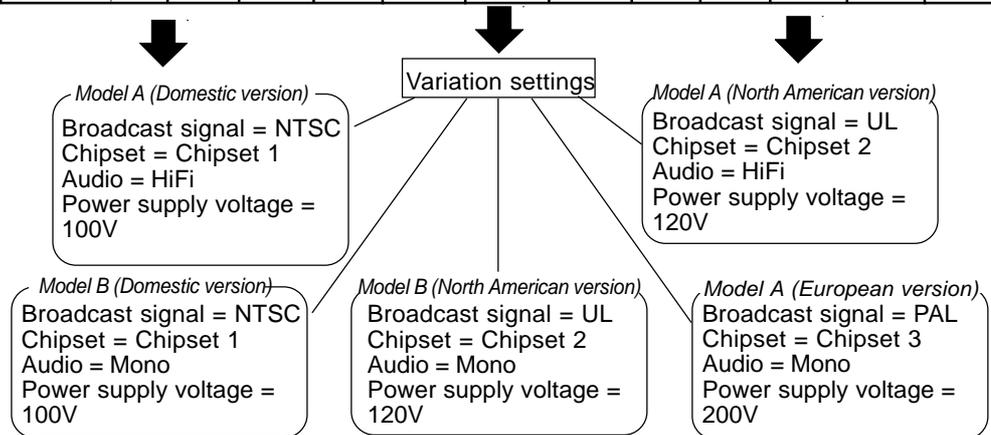
Managing Design variations in this way enables component properties to be set for each Design variation type, as follows.



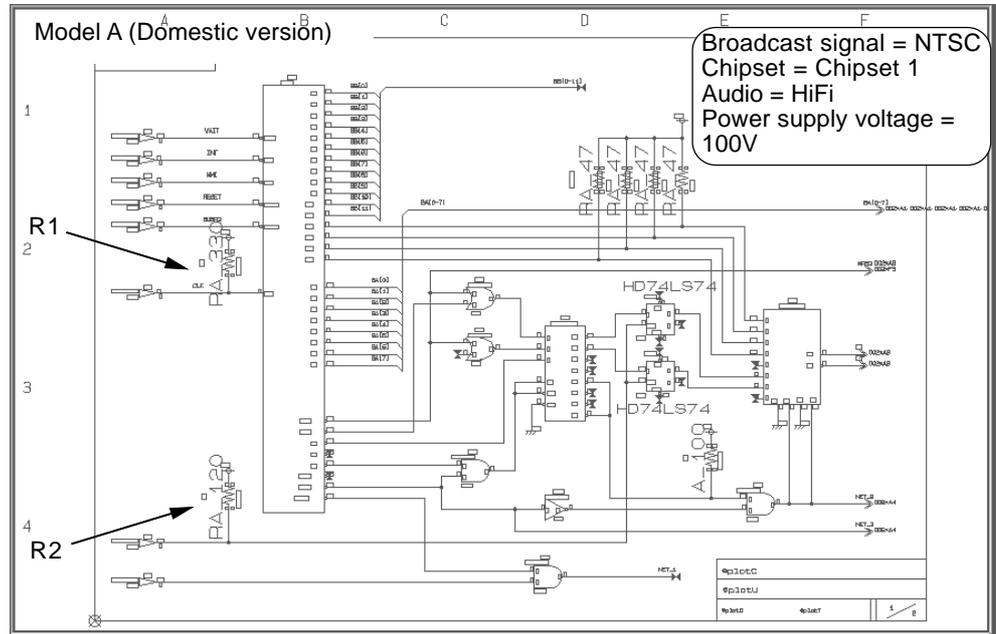
Next, the required combinations of variation types can be specified to define Destination of Design Variations.

In the above example, there are 54 possible combinations of Design variation options. However, if only the five combinations listed below are required in practice, then only these five need be set as Design variations.

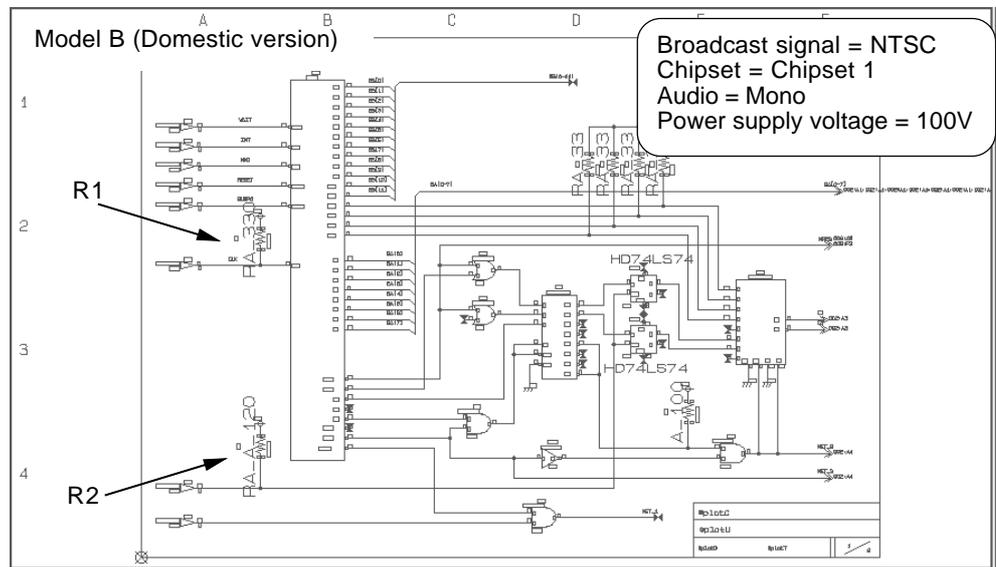
Variation Type	Broadcast Signal		Chipset			Audio		Power Supply		
	NT	UL	Chipset1	Chipset2	Chipset3	Hifi	Mono	100V	120V	200V
ModelA (North American version)	<input type="radio"/>									
ModelB (Domestic version)	<input type="radio"/>									
ModelB (North American version)	<input type="radio"/>									
ModelB (Domestic version)	<input type="radio"/>									



The following shows the schematic for one of the Destination of Design Variations.



The following shows the schematic for a different Design variation. Note that the property values are different.



The property value for R1 only changes when the broadcast signal variation type changes (model B, domestic version). However, the property value for R2 changes when the combination of two variation types, the broadcast signal and the audio specifications, change (model B, domestic version).

Design Variation Manager applies for the following tools.

- Schematic Editor
- Component Browser
- Netlist
- Plotter

2. Defining Design Variations

It is necessary to define the following two files in order to set design variation for a schematic.

Variant properties definition file	vmsys.rsc
Variant rule file	vmdata.rsc

● Variant properties definition file (vmsys.rsc)

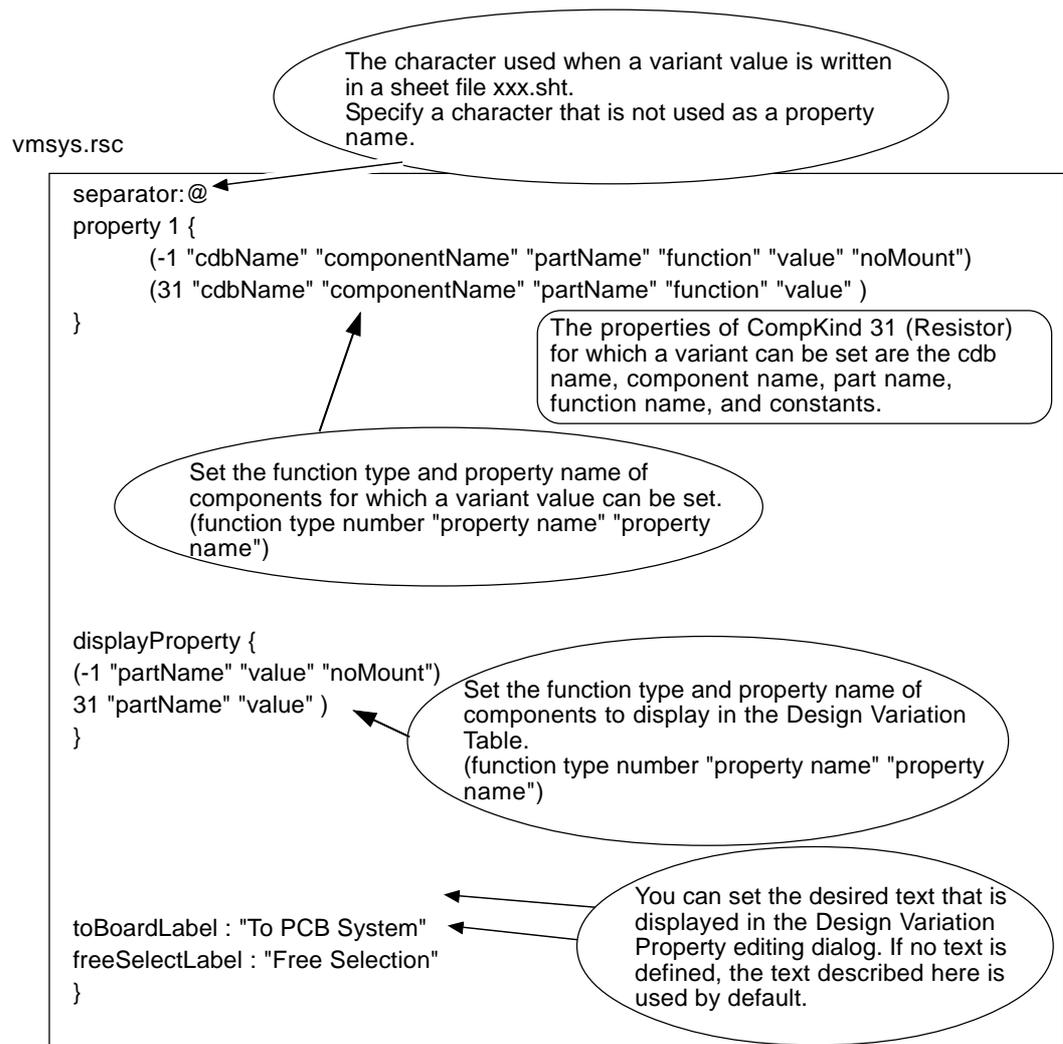
Variant properties definition file vmsys.rsc (\$ZDSROOT/info/vmsys.rsc)

Only one variant properties definition file exists in the system. The file contains common settings for all schematics.

The properties for which variant values can be set are defined for each function type of a component.



NOTE Always define the function type definition file (CompKind) before performing variant management.



● Variant rule file (vmdata.rsc)

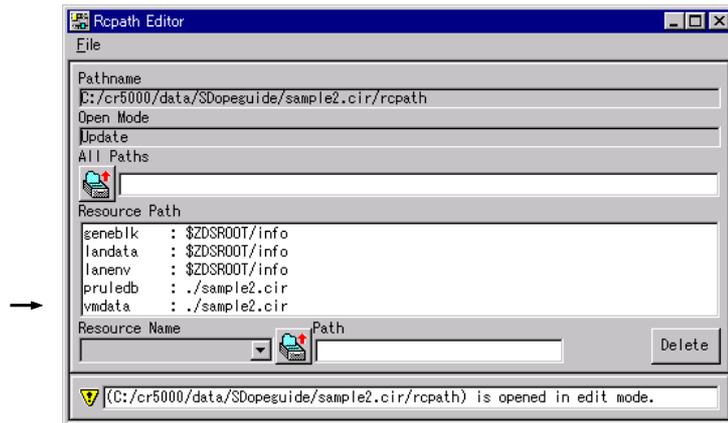
Defines the combination of Design variation type and Destination of Design variation.

Files defined in the resource path file (rcpath) and referenced in each schematic can be specified.



ATTENTION

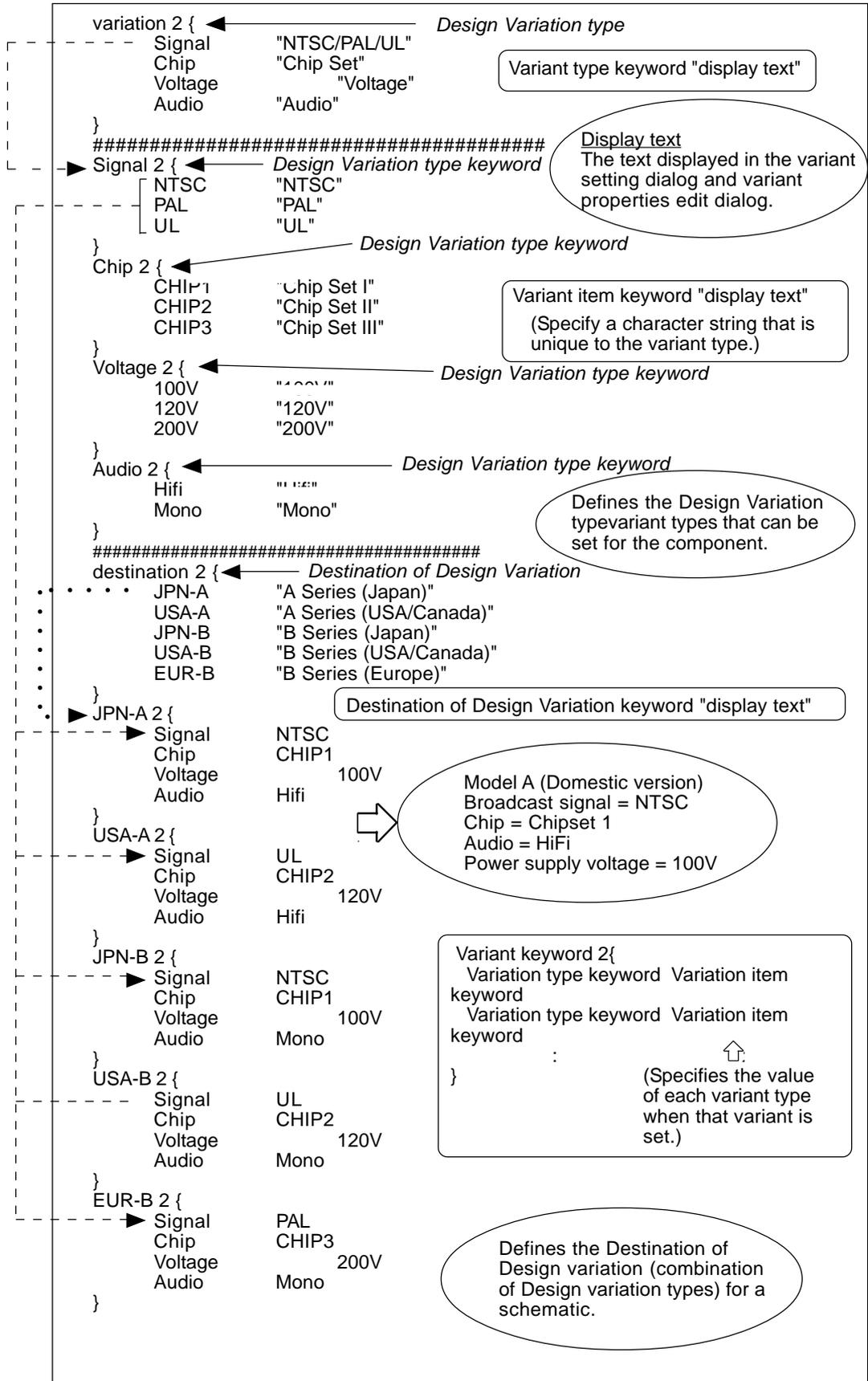
Design Variant management cannot be performed if there is no definition in rcpath. Use the resource path editor to add to rcpath. Double click on rcpath in the CR-5000 Design File Manager to open the resource path editor.



NOTE

It is possible for the system to reference a single variant rule file which defines common variant management, or for each schematic to reference a variant rule file and define variants.

vmdata.rsc



● Edit Variant rule file

Edit of variant rule file is edited by design variation resource editor.

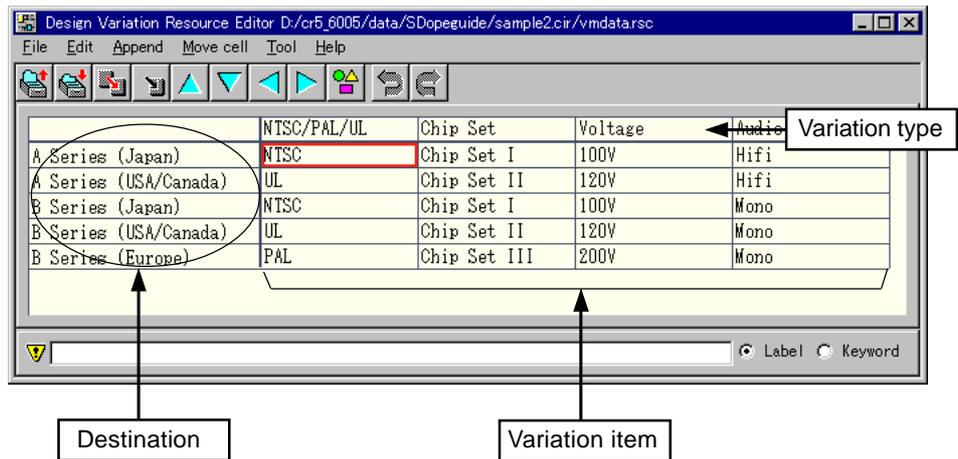
When the new variant rule file is created, perform **file** ⇒ **New Open** ⇒ **Design Variation Property Spec** from CR-5000 Design File Manager.

Design variation resource editor enable the following:

1. Design variation type can be defined.
2. Destination of Design variation can be defined.

From file/directory of CR-5000 Design File Manager, double click [vmdata.rsc(design variation resource)] or select [vmdata.rsc(design variation resource)] then select **Tool** → **Action** → **Design Variation Resource Editor** from the menu bar.

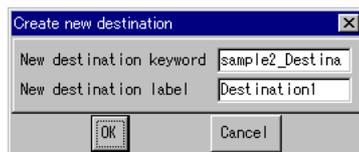
Design variation resource editor appears.



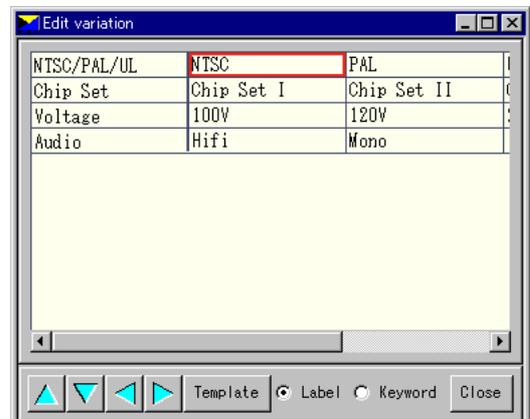
Change destination dialog



Change destination dialog

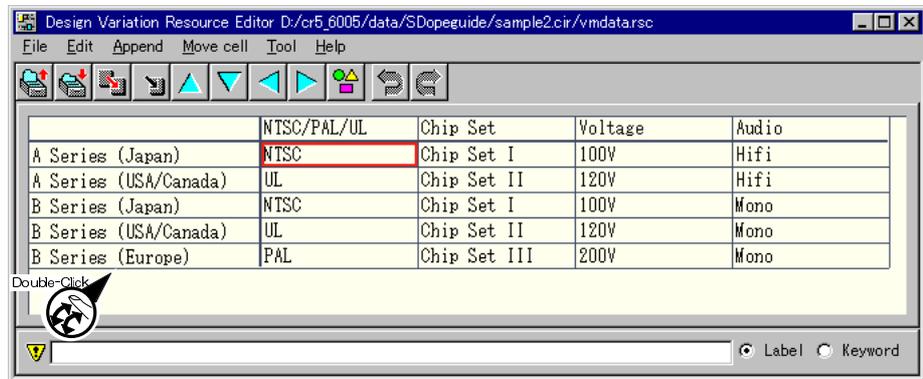


Edit and Add Variation type, Variation item dialog



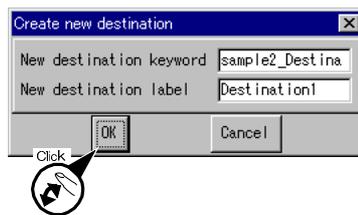
1. Set the destination of design variation.

Set the destination from the Design Variation Resource Editor.



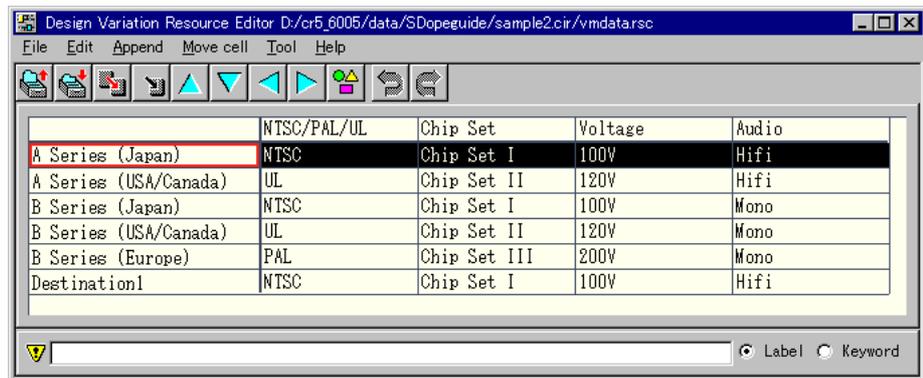
1) Adding the destination of design variation

Add a new design variation to the end of the list displayed by selecting **Append** → **Append destination** from the menu bar.



Specify the adding keyword (destination name) and label of destination of design variation.

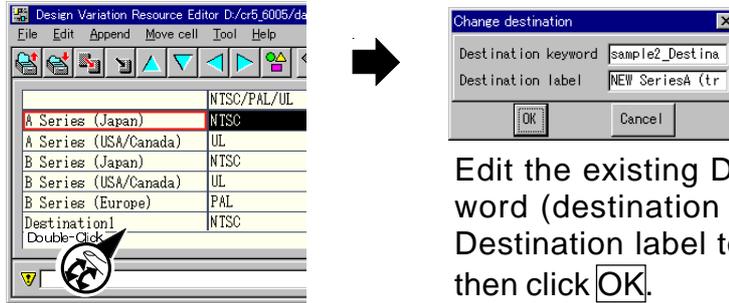
Click **OK**.



ATTENTION It is possible to edit the destinations using the Assist Menu.

2) Editing the destination of design variation

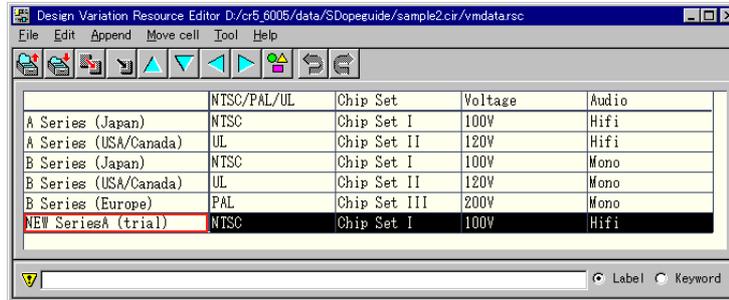
Edit the destinations at the selected cell.



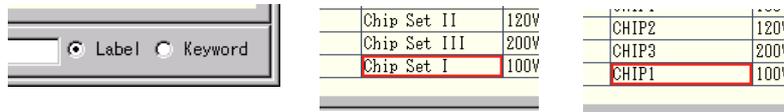
Edit the existing Destination keyword (destination name) and the Destination label to be displayed, then click **OK**.

Double-Click the cell of destination keyword to edit.

The edited result is reflected..



ATTENTION

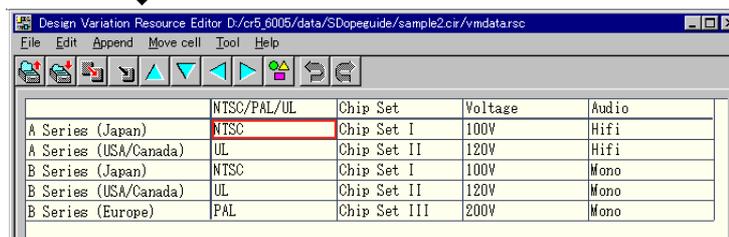
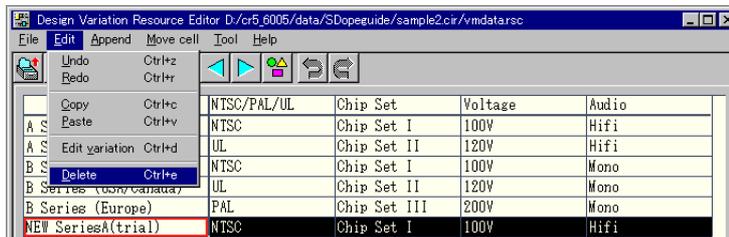


You can directly edit the destination data by changing the display mode to “Label” or “Keyword” and clicking a cell you want to edit.

3) Deleting the destination of design variation

Delete the selected destination cell

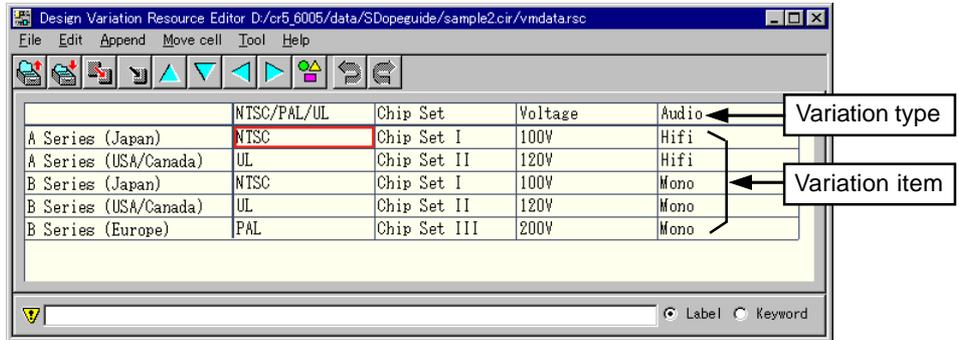
Click **Edit** → **Delete** from the menu bar.



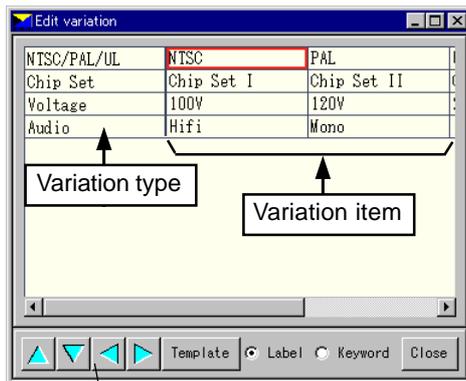
1. Set Design variation type and Destination of Design variation.

Set the design variation types and design variation items in the Resource Editor. Double-click the cell where the design variation type or the design variation item is.

Select **Edit** → **Edit variation** from the menu bar or 

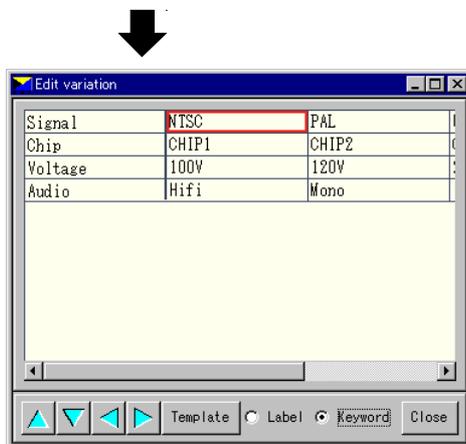


↓ Edit and Add Design Variation Type dialog appears.



Moving the line of design variation type a cursor is on to top and bottom.

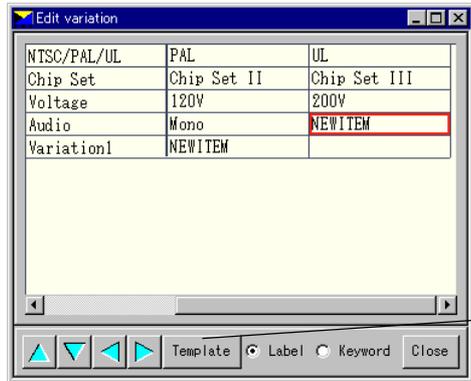
If the [display keyword] is checked, the keywords for the design variation type or design variation item will be changed. They can be edited.



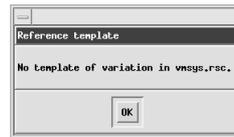
1) Addition of design variation type and design variation item

A new design variation type should be added to the end of the existing design variation type list, and a new design variation item should be added to the end of the displayed item list on the column of the cell selected.

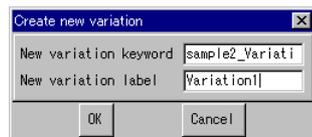
Click **Add variation** or **Add item** from the Assist menu.



This dialog can load design variation type and design variation item that they are described in \$ZDSROOT/info/vmsys.rsc as template.

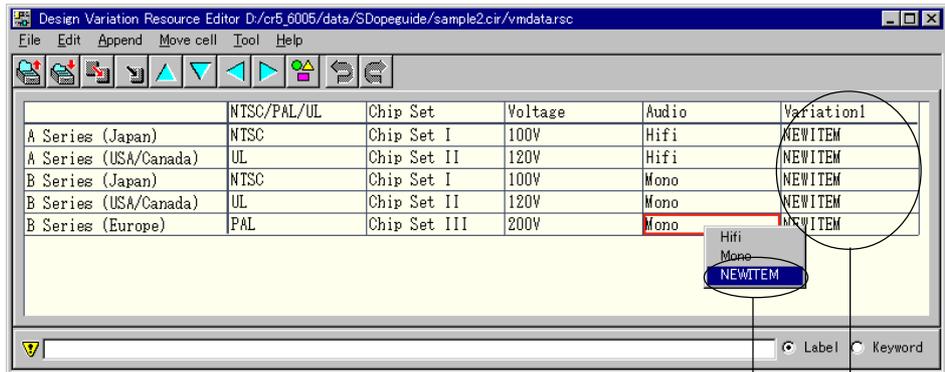


Please add the part of template to the end of line in vmsys.rsc with same format as vmdata.rsc.



Specify keywords and related text for the design variation type and design variation item.

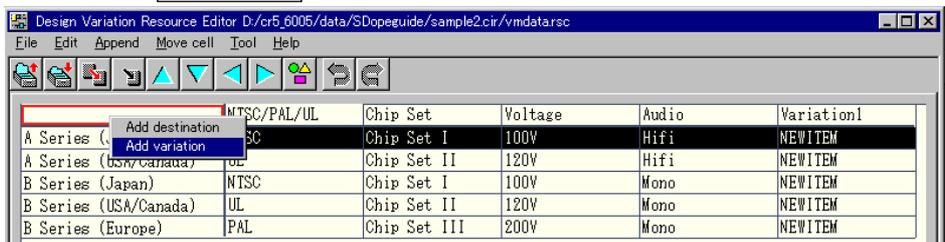
Click **OK**.



Design variation type and design variation item are added.

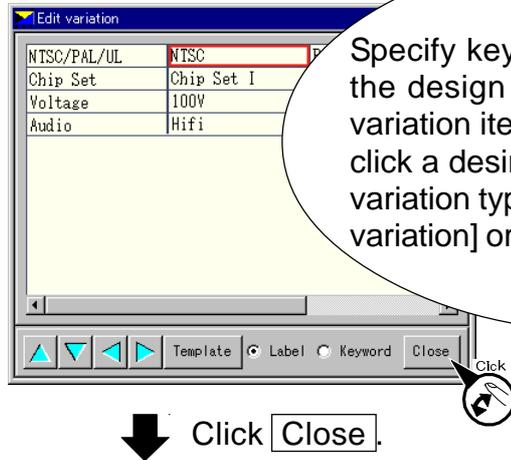


You can use the design variation resource editor to add a design variation type. To do this, click **Add variation** from the Assist menu in the editor.

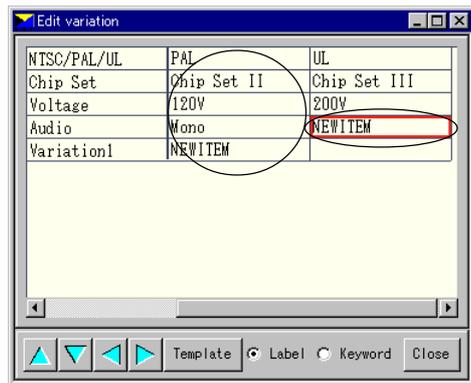


2). Editing design variation type and design variation item

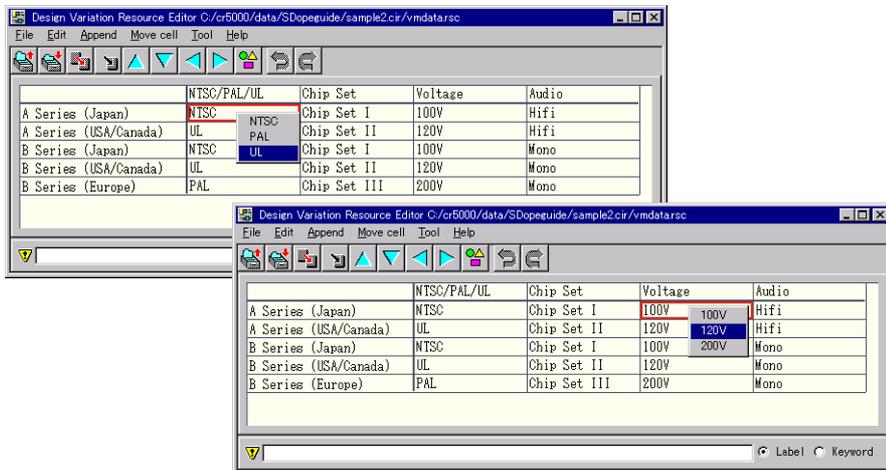
Edit the design variation type and design variation items in the selected cell.



The edited result is reflected..



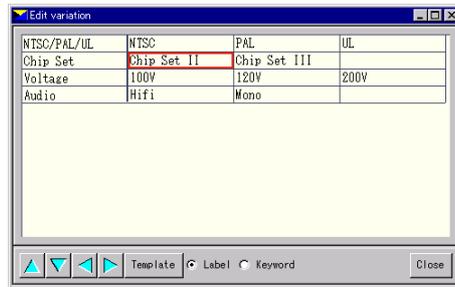
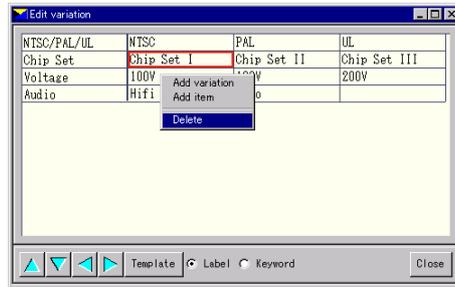
Switch the design variation item in the selected cell. Open the assist menu and change the design variation item.



3) Deleting design variation type and design variation item

Delete the design variation type and the design variation item in the selected cell.

Click **Delete** from the assist menu.

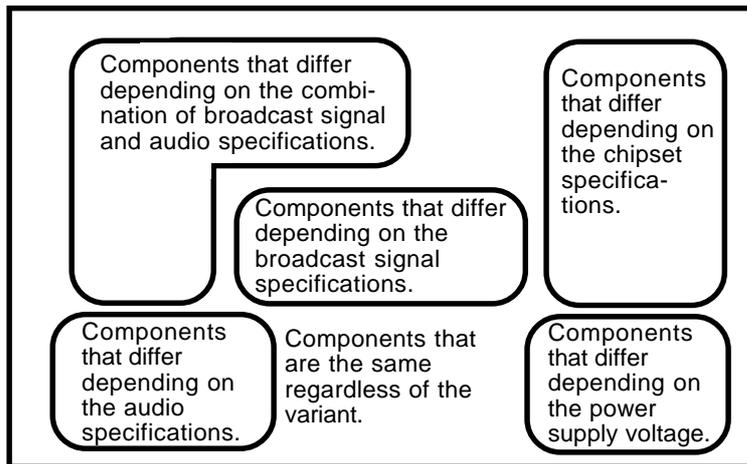


3. Setting Design Variations

● Design Variations

ATTENTION Variants can be set for the schematic directory used in the previous section "2. Defining Design Variations".

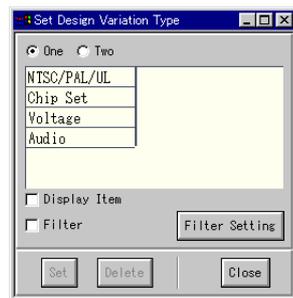
Rather than setting variants for all components in the schematic, variants are set for specific components only.



In this case, the schematic includes components whose property values are the same regardless of the variant and components whose property values differ depending on the broadcast signal specifications and power supply voltage.

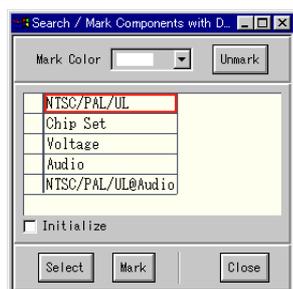
The Schematic Editor is used to set the design variation type and check the parts for the destination for each component.

From the menu bar, select **Attribute** → **Set Design Variation Type**



Set Design Variation Type dialog appears.

From the menu bar, select **Attribute** → **Search/Mark Components with Design Variation**



Search/Mark Components with Design Variation dialog appears.

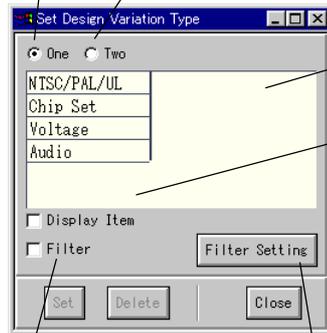
● Setting Design Variations

The following operations can be performed from the variant setting dialog.

1. Set variants for a component.
2. Delete the variants set for a component.

Select one Design variation type only.

Multiple (2) Design variation types can be specified.



Lists the Design variations specified in the variant rule file (vmdata.rsc).

A Design Variation Item List will be displayed for each design variation type.

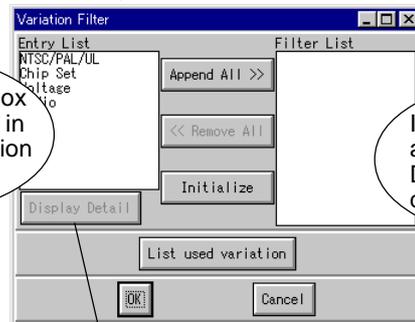
Set Set the variant types for the component.

Delete Delete the variant types set for the component.

Close Close the dialog.

When this is checked, only design variations specified in "Filter Setting" are listed.

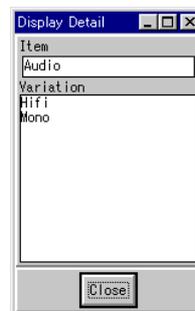
It is possible to set or delete only a specific group of variation types.



Items in this list box are not displayed in Set Design Variation Type dialog.

Items in this list box are displayed in Set Design Variation Type dialog.

It is possible to check the design variation items of the design variation types presently selected from the Entry List.



1.Set variants for a component.

Specify that the component can have variant properties.
Also set the variant types.



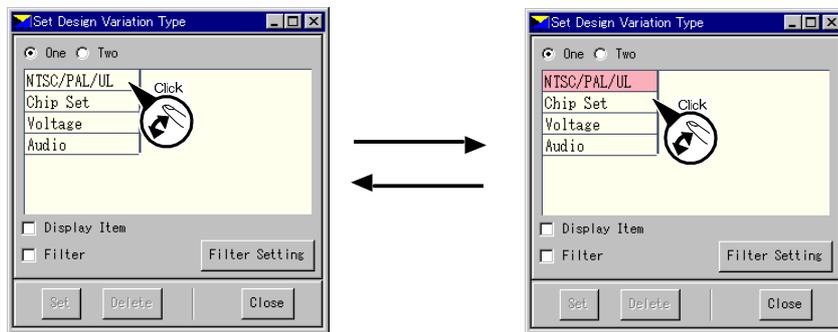
For components for which variant types have been set, the only property values that can be set are those that are variant options for the variant types set for the currently specified variant.

(1)Select the component for which to set a variant.

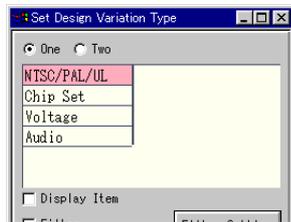
(More than one component can be selected.)

(2)Click on the desired variant types in the variant type list of the Design variation setting dialog.

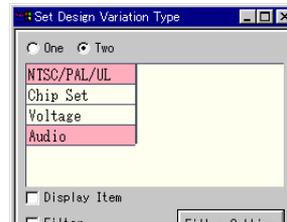
Clicking on a cell that is already selected deselects that cell.



● Single selection
Select when there is only one Design variation type.

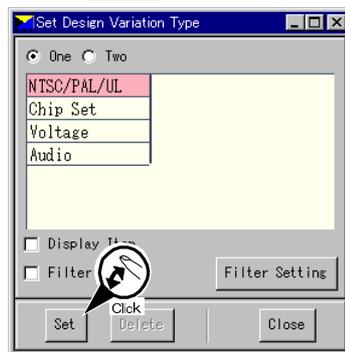


● Multiple selection
Select two Design variation types.

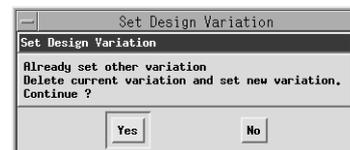


Set multiple selection for components for which property values depend on the combination of the variant items of two different variant types.

(3)Click **Set**.



The following message appears if a different variant type is set for a component that already has a variant setting.
Please check.

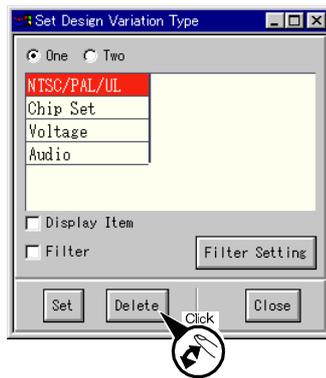


2.Delete the Design variations set for a component.

This deletes the variant setting for a component. This also deletes the property values for the variant items of previously set variant types.

(1)Select a component with a variant setting.
(More than one component can be selected.)

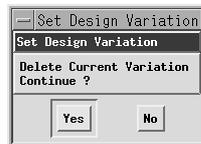
(2)Click **Delete** in the Design variation setting dialog.



The design variation type will be highlighted in red, if the selected component has the design variation type set.



A confirmation dialog appears.



● Confirming Design Variation Setting

This searches the schematic sheet for components that have the specified variant and selects or displays a mark to indicate those components.

The following can be done from the Search/Mark Components with Design Variation dialog.

1. Mark a component for which a design variation type is assigned.
2. Select a component for which a design variation type is assigned.
3. Confirm and check the design variation types used on an open schematic sheet.



Search Components are displayed in a state where they are clicked and selected by using cursor.

Mark Displays the component in the color specified in the dialog. The component remains marked until the mark is cleared.

Specifies the mark display color.

Clear the mark display.

Displays the design variation types used on the presently open schematic sheet.

Select Select a component for which the destination selected from the design variation type list is assigned.

Mark Mark a component for which the destination selected from the design variation type list is assigned.

Close Close the dialog.

In the case you have consecutively specified searches and markings, specify whether or not the status of the components selected or marked is cleared.

- (1) Click on the search item in the Design variation type list of the Design variation setting dialog.

When marking the components, specify the marking color.

- (2) Click **Select** or **Mark** in the Design variation setting dialog.



If **Select** or **Mark** button is clicked when any design variation type is not selected, you can start a search for components for which the destination has not been assigned yet.

If you set a color for a mark, the mark will appear in that color. (If you double click, the mark is deleted.)

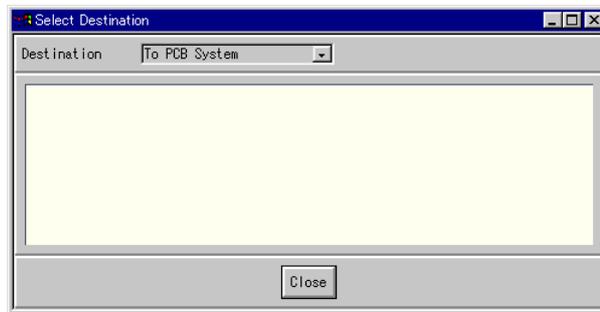
4. Setting Properties for Destinations

● Displaying a Design Variation

The properties displayed on the schematic can switch between the following in accordance with Design variation management.

1. Display a variant defined in the variant rule file.
2. Display a combination (one only) that is not defined as a variant in the variant rule file. ⇒ (variable combination)
3. Display for passing component shape data to the PCB design tool. ⇒ (board values)
(See "●Interface with the PCB Design Tool".)
4. Changing display of no mount components.

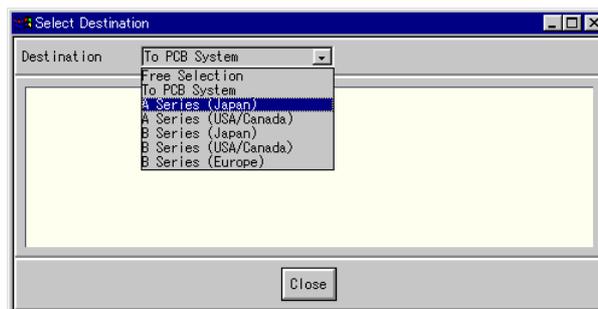
Use the variant properties edit dialog to change the displayed Design variation. From the menu, Click **Attribute** → **Select Destination** .

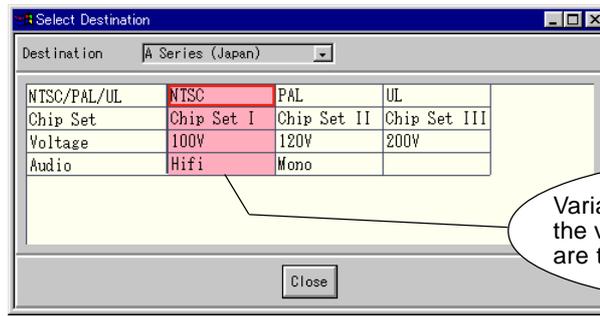


NOTE The variant displayed on opening the file is the "board values".
If not set, the schematic displays the values set when the components were input.

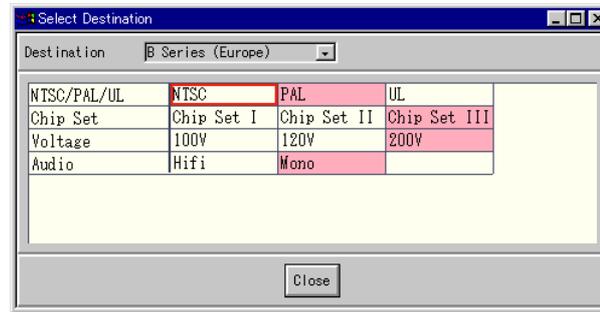
1. Display Destination of Design Variation defined in the variant rule file.

Click the option button in the Design variation list and select one of the available Design variations.





Variant combinations defined in the variant rule file (vmdata.rsc) are the cells displayed in pink.

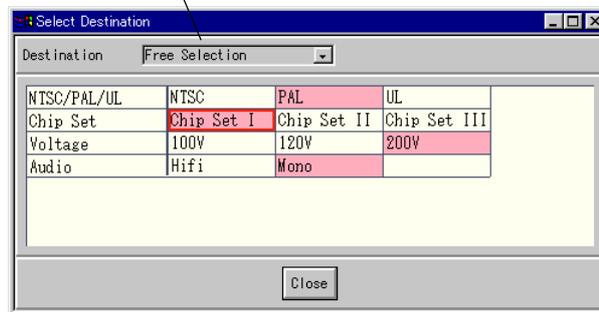


2. Display a combination (one only) that is not defined as a variant in the variant rule file. ⇒ (Free Selection)



The combination of Design variation types displayed for a variable combination is not saved. The purpose of the function is the temporary display of a combination that is not defined in the variant rule file.

Click the option button in the Design variation list and select [Free Selection].



Select by double clicking on item cells from amongst the Design variation types.

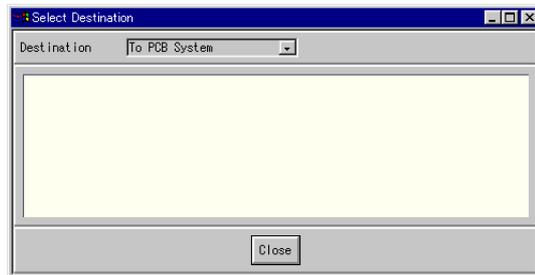


Double clicking on Design variation type items when a variation is selected changes the variant to "Free Selection".



3. Display for passing component shape data to the PCB design tool
 ⇒ [To PCB System]

Click the option button in the Design variation list and select [To PCB System].



Displays the property values defined as "To PCB System" for each component. This is independent of the variants defined in the variant rule file. (For details, see "●Interface with the PCB Design Tool".)

4. Changing the no mount component display

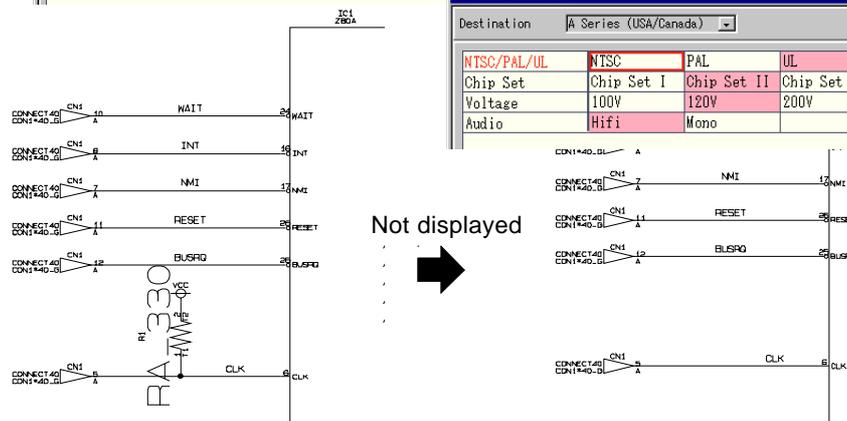
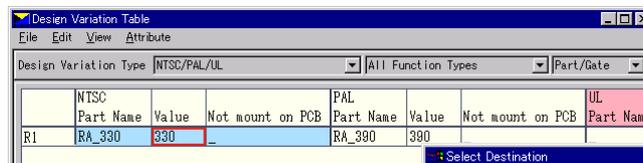
It is possible to change the display style to show the no mount components and the nets connected to them depending on their destinations.



This will apply to only the component property, noMount where the value is set to [YES].

You can specify one of the following 4 display modes by selecting [Data Resource File (landata.rsc)] - [Display Methods] - [Display noMountComponent].

- Normal: The no mount components are displayed.
- Not displayed: The object with this setting is not displayed.
- Dotted line: The object with this setting is displayed with white dotted lines.
- Grid color: The object with this setting id displayed in grid color.



● **Setting Different Properties for Each Design Variation(Sheet Editor)**

After switching the display to a Design variation, you must then set the property values for the variant.

Property values can be edited in the Sheet Editor by the following methods.

1. Design Variation Table dialog
2. Property setting dialog
3. Direct editing in the Sheet Editor

1.Design Variation Table dialog

All of the components with design variation settings that are on schematic sheet are displayed in the Design Variation List" dialog. Simple editing can be done.

- o **Confirm design variation property value.**
Displayed by design variation type or function type.
- o **Cut, copy, paste and delete design variation property.**
- o **Set the value passed on To PCB System and the value for noMount settings.**
- o **Change the destination (Dialog is displayed).**
- o **Access the "Parts Rule Based Search" dialog or the "Change Attribute" dialog.**
- o **Set display property.**
- o **Output the data of all the displayed list.**

From the menu, Click **Attribute** → **Design Variation Table** .

Displays the property of each design variation item one-by-one. The value of the property can be confirmed.

The present destination is displayed in red.
The selected cell is displayed in black.

The design variation type to be displayed can be set.

The function type to be displayed can be set.

Double click the design variation item to change the destination.

Select the component by double clicking on the property value cell and the destination will change.

Select the component by double clicking on the reference.

Specify the action to be applied to the component.

Sets which key is used to trigger highlighting (blue).

Sets the property to be displayed in a list.

● Cut , Copy , paste and Delete variant property values

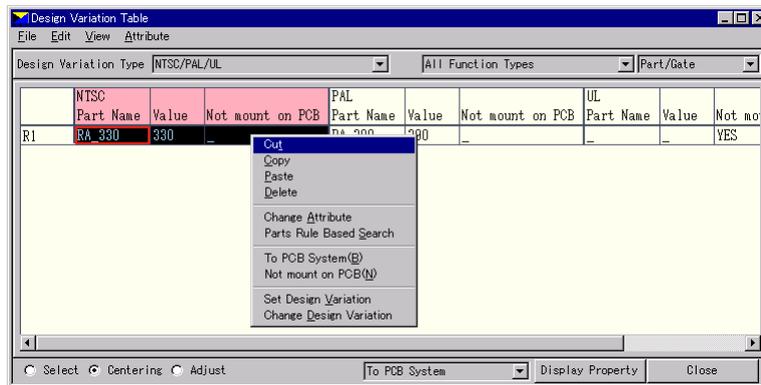
Select the cell in which you want to edit the property value, then from the menu bar select [Edit] or from the Assist Menu execute a command.

⚠ ATTENTION

Property values can be cut, copied, pasted, and deleted in the Edit Design Variation dialog. However, property values cannot be input or modified. Use the property setting dialog to input or modify property values or edit the values directly in the Sheet Editor.

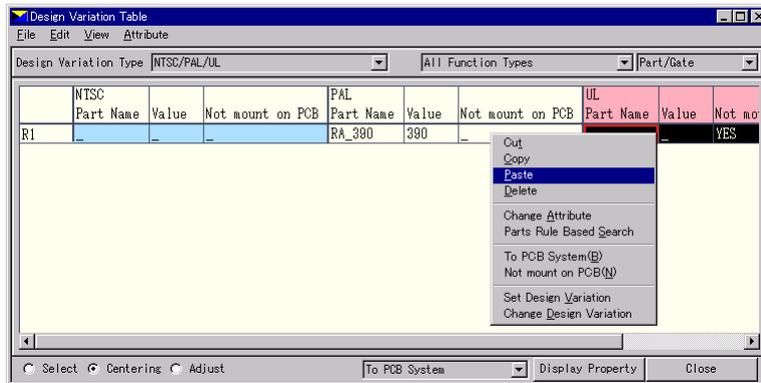
✍ EXAMPLE

Select the cell you wish to cut and click **Cut**.



Select the cell in which to paste and click **Paste**.

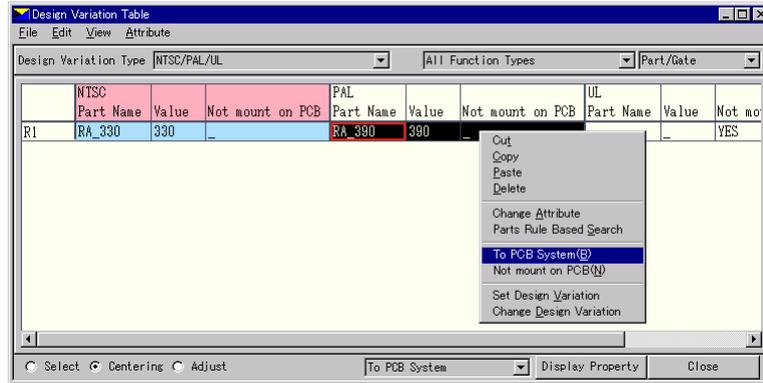
The property value is cut from its original cell and pasted into the specified cell.



● To PCB System

Set the design variation type where the cursor is to To PCB System for that component (Please refer to 3. Interface for PCB Settings).

Select the cell in which you want to edit the property value, then select **Attribute** → **To PCB System** from the menu bar or **To PCB System** from the Assist menu.

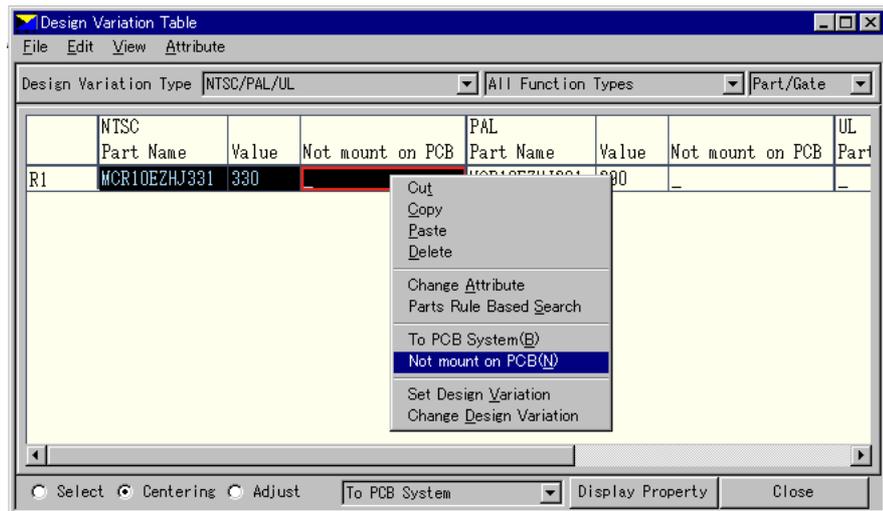


This changes the value passed to the board and changes the display color of the cell.

● Not Mount On PCB

Set the design variation destination component where the cursor is to no mount setting active (YES).

Select the cell in which you want to edit the property value, then select **Attribute** → **Not mount on PCB** from the menu bar or **Not mount on PCB** from the Assist menu.



The property of Not mount is set to "YES".
Specifying "Not mount on PCB" command again while in the "Not mount (YES)" setting changes the setting to "Not mount (No)".

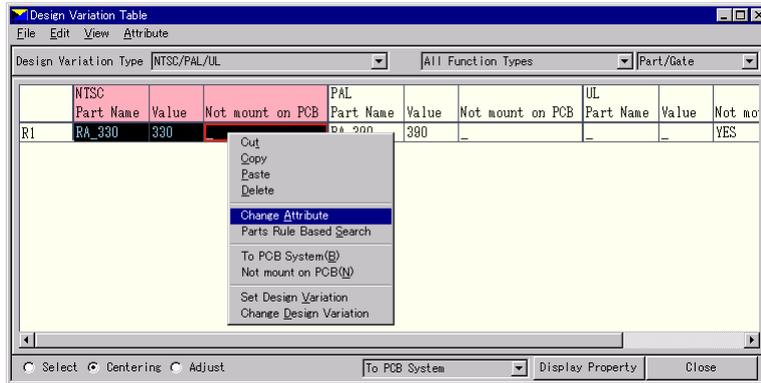


● Calling a dialog

The following dialog boxes can be accessed.

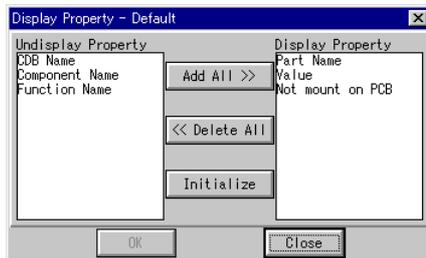
Change Design Variation
 Set Design Variation
 Change Attribute
 Parts Rule Based Search

Select **Attribute** or specify command from the assist menu.



● Setting display Property

The property you want to display in a list can be set.

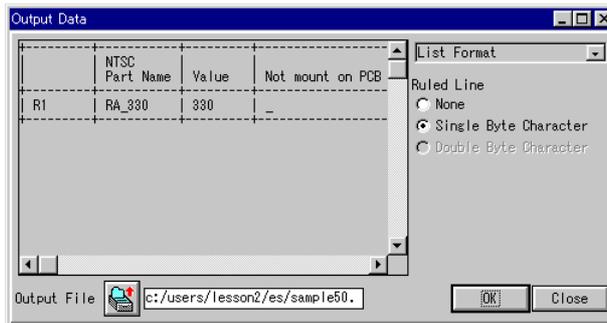


The default display property is set in [displayProperty] within \$ZDSROOT/info/vmsys.rsc. The display property, after editing, is saved in schematic name.cir/logxxxvmtable.ini.

● Outputting the data in the list currently displayed

The list currently displayed can be output in a text-with-line format or a CSV format.

From the menu, Select **File** → **Output Data**.



2. Set Properties dialog

The Set properties dialog can be used to input, edit, or delete property values.

Select a component on the schematic sheet and open the set properties dialog. To change the displayed variant, open the variant property edit dialog.

Use the variant property edit dialog to change the variant.

The 'Select Destination' dialog box shows a dropdown menu for 'Destination' set to 'B Series (Europe)'. Below it is a table with the following data:

NTSC/PAL/UL	NTSC	PAL	UL
Chip Set	Chip Set I	Chip Set II	Chip Set III
Voltage	100V	120V	200V
Audio	Hifi	Mono	

Below the table is a 'Close' button. An arrow points from the text above to the 'Destination' dropdown menu.

The 'Set Properties' dialog box shows a list of properties for a component. The 'Part Name' is 'MCR10EZHJ121'. The 'Value' is '120'. The 'Tolerance' is '+-5%'. The 'MAX Voltage' is 'MAX Voltage'. The 'Part Name' field at the bottom contains 'MCR10EZHJ121'. An arrow points from the text below to the 'Value' field.

The property values in the properties dialog change when you change the variant.

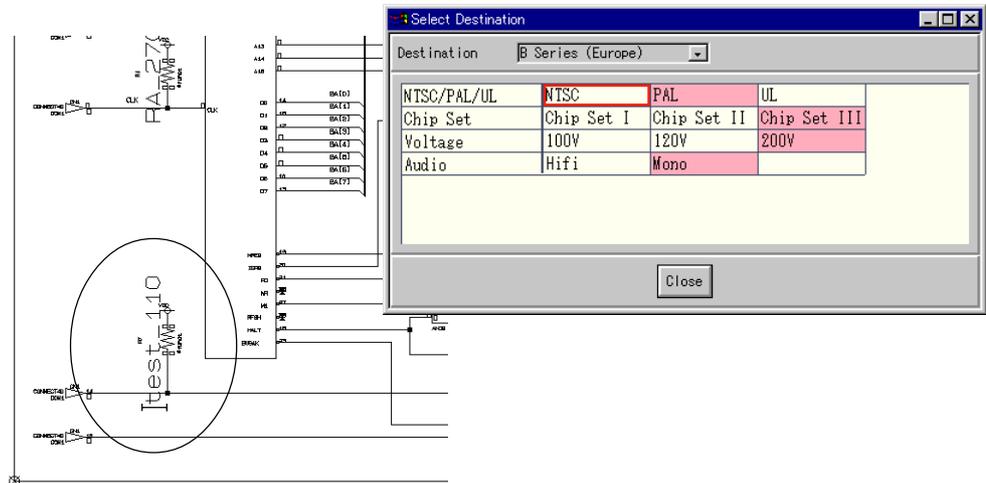
The 'Set Properties' dialog box shows the same component as the previous screenshot, but the 'Part Name' is now 'test_110'. The 'Value' is '120'. The 'Tolerance' is '+-5%'. The 'MAX Voltage' is 'MAX Voltage'. The 'Part Name' field at the bottom contains 'test_110'. An arrow points from the text below to the 'Value' field.

Modify the property values in the Set properties dialog.



3. Direct editing in the Sheet Editor

Select the schematic sheet display in the variant properties edit dialog, then edit values directly.



● **Setting Different Properties for Each Design Variation (Component Browser)**

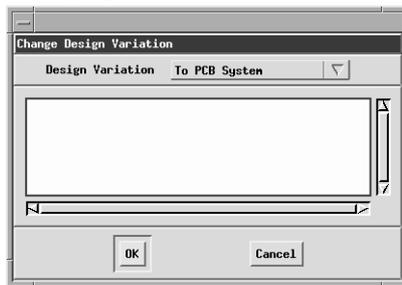
Setting variant properties (input, edit, and delete) can also be performed from the Component Browser.

The following operations can be performed in Component Browser in conjunction with variant management.

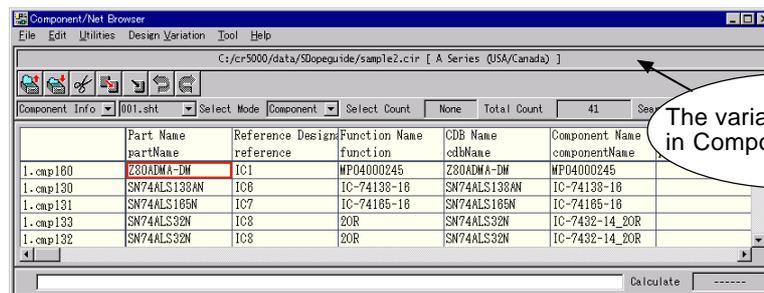
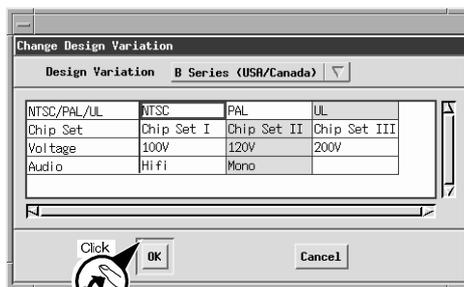
1. Change the variant.
2. Search for objects with variant settings.
3. Mark objects with variant settings.
4. Edit property values.
5. Display a list of variants and edit specific property values.
6. Edit specific property values from a list of variants.

1. Change the variant.

Select **Design Variation** → **Change Design Variation** from the menu .



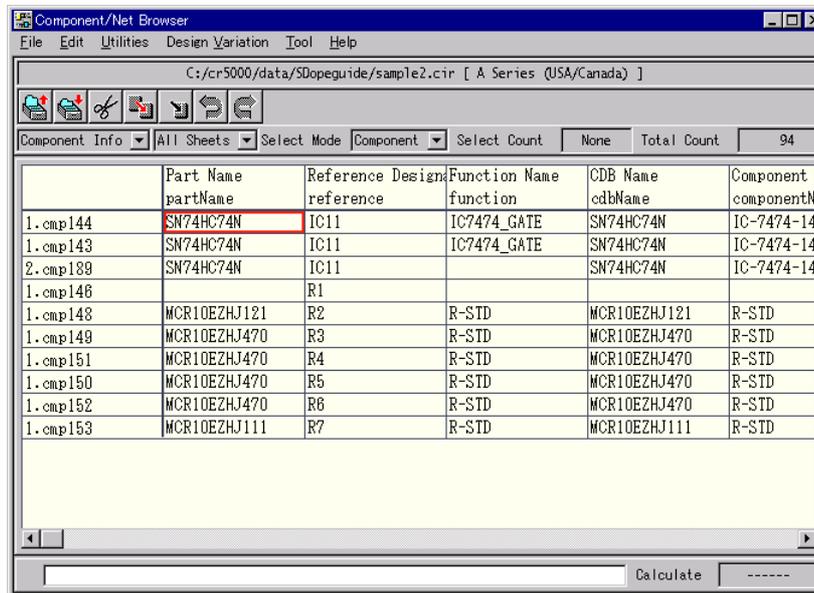
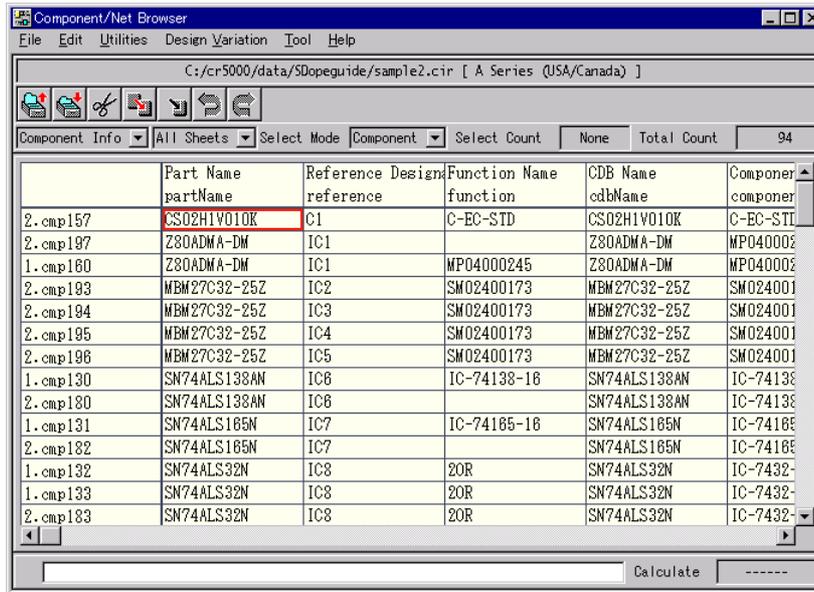
Select the variant you wish to display from the variant list, then click **OK**.



2. Search for objects with variant settings.

This searches for components with a variant setting and displays those components only in the browser.

Select **Design Variation** → **Design Variation Object** → **Search** from the menu.

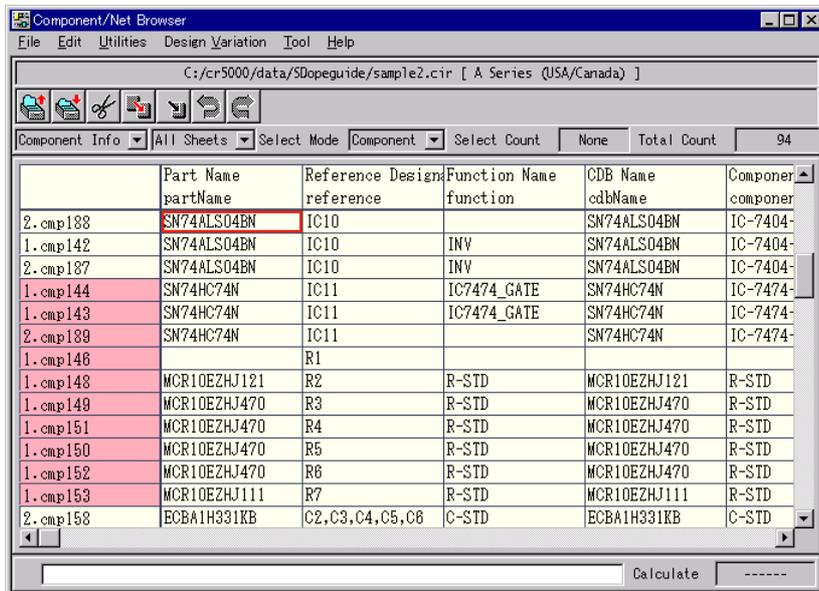
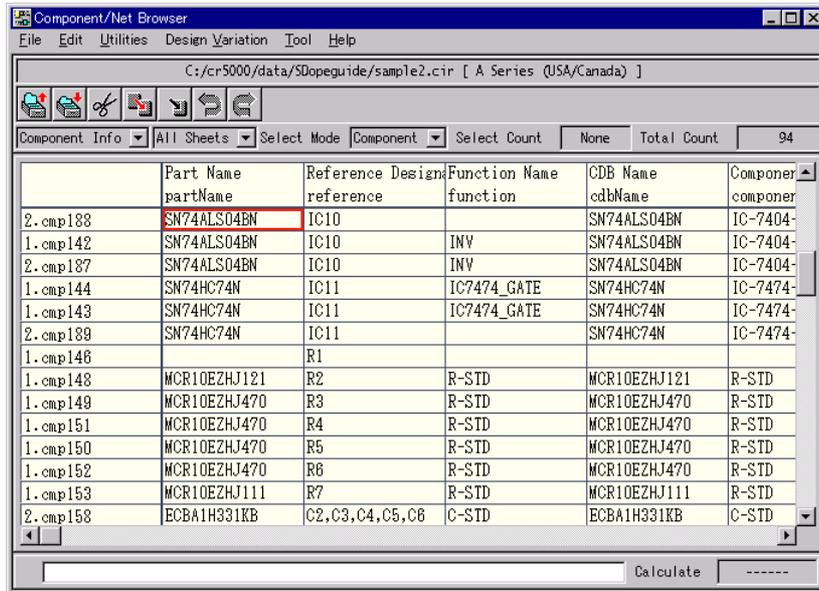


To clear the search, select **Utilities** → **Reset Search** from the menu bar or select **Reset Search** from the Assist menu.

3. Mark objects with variant settings.

This displays a red mark in the component ID display cell (left side) of objects with a variant setting.

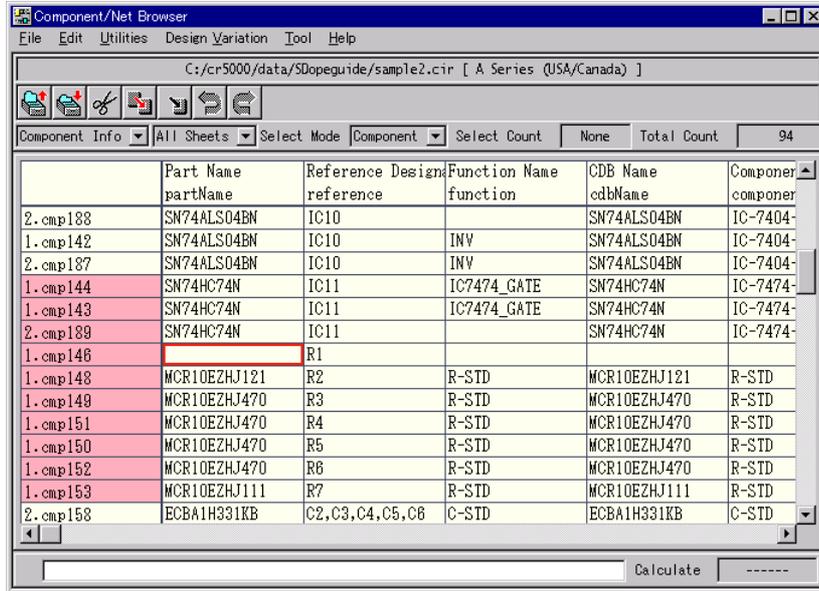
Select **Design Variation** → **Design Variation Object** → **Mark** from the menu bar.



To clear the marks, select **Design Variation** → **Reset Mark** from the menu bar.

4. Edit property values.

Changing the variant also changes the property values displayed in the browser.



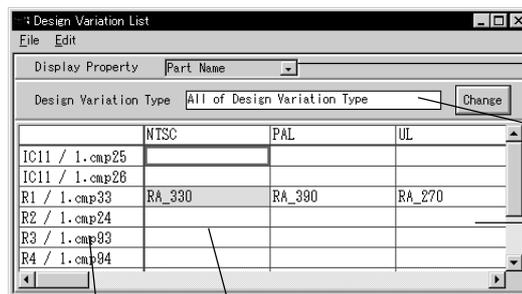
Saving the Component Browser data updates the values in the sheet.

5. Display a list of Design Variations.

The variant list displays a list of components with variant settings and displays the property values set for each variant.

Select **Design Variation** → **Design Variation List** from the menu bar

On opening, the variant list displays all the variant types defined in the variant rule file.



Changes the displayed property name.

Indicates the displayed variant type.

Displays components with variant settings only.

The default value in data output is displayed on the most left part of property value.

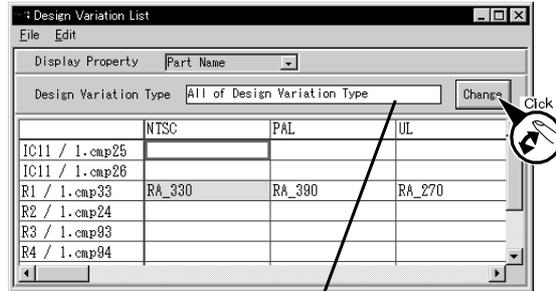
Displays reference/componentID

The cell of "To PCB System" is displayed with blue color.

You can specify specific variant types to change the components displayed in the list.



Click **Change** to open a dialog.

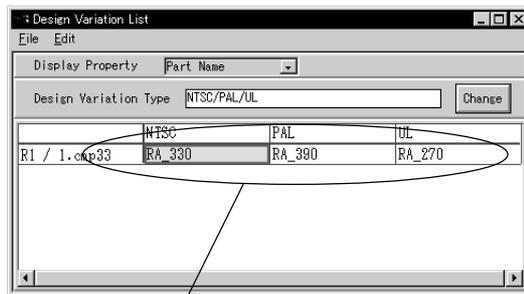


Select one Design variation type.

Click **OK**.



Displays only components with the specified variant setting.



The property values set for each variant

The cell of "To PCB System" is displayed with blue color.

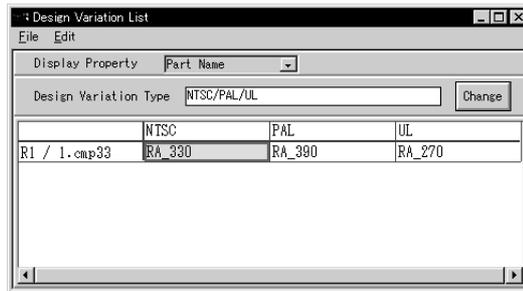


6. Edit specific property values from the Design variation list.

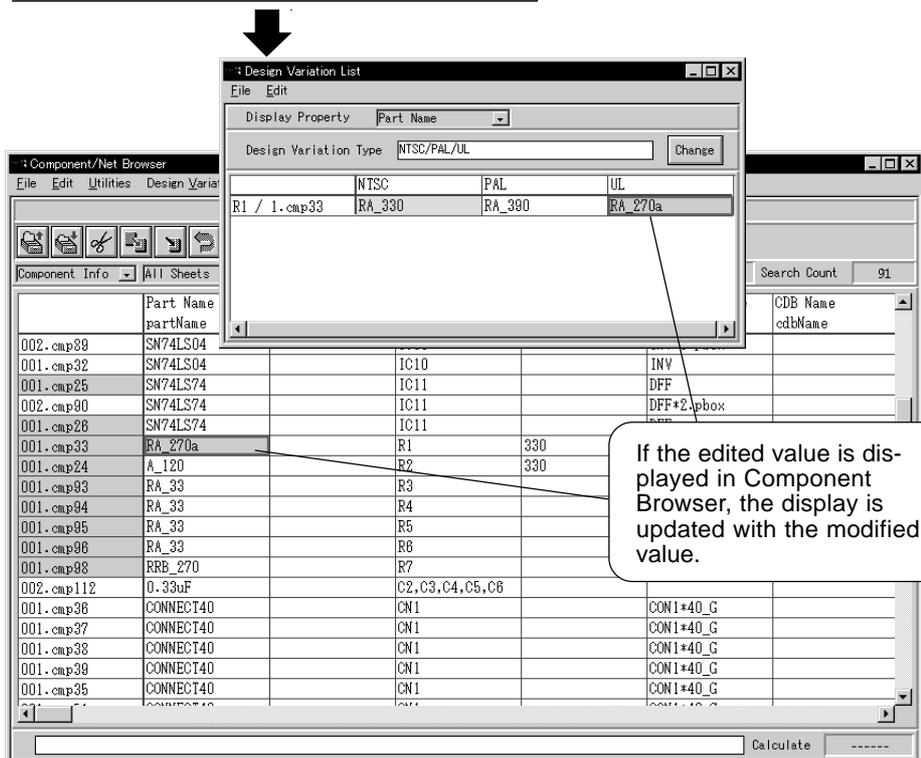
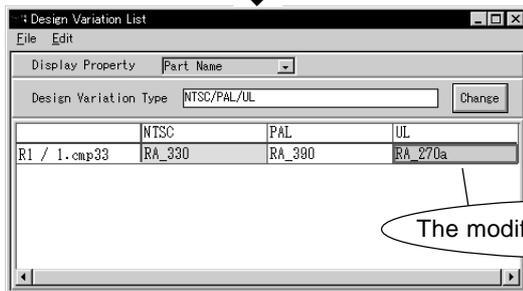
Two methods are available to edit property values from the variant list: "edit specific values" or "edit by cutting, copying, and pasting property values for individual components". Select **Design Variation** → **Design Variation List** from the menu bar.

 **EXAMPLE**

[Edit specific property values.]

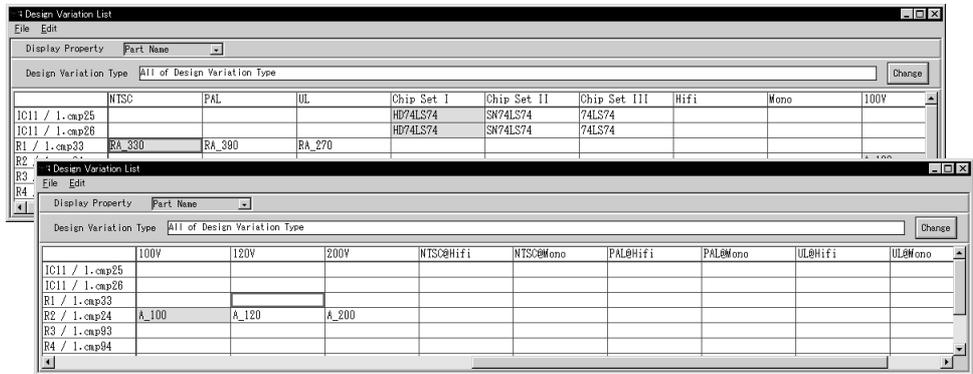


Select variation type and click the cell of the item, then edit the value by using keyboard.



[Edit by component.]

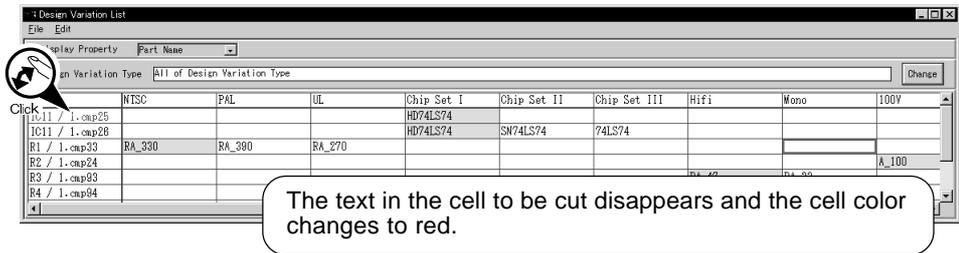
You can cut, copy, and paste between any of the variant properties of the specified component.



EXAMPLE

Cut

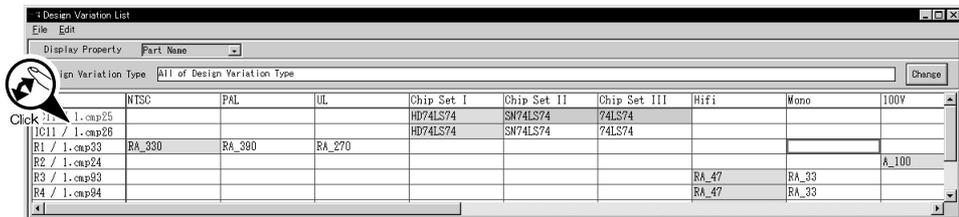
Click on a component cell ID displayed on the left.



Select **Edit** → **Cut** from the menu bar.

Paste

Click on the component cell in which to paste.



Select **Edit** → **Paste** from the menu.

ATTENTION

You can only paste into components with the same variant type. You cannot paste into components with a different variant type.

Copy

Click on the component cell from which to copy.

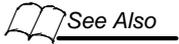
Select **Edit** → **Copy** from the menu bar.

The list display does not change.



NOTE

Data can be exported from the list.
Select **Output Data** from the **File** menu.



See Also

Refer to "CR-5000 System Designer Operation guide -MASTER- Chapter3 Component Browser" for details.

1. Printing Design Variations

You can print a schematic sheet either by "Printing from the Sheet Editor" or "Printing from the plot out dialog".



See Also

Refer to "CR-5000 System Designer Operation Guide -MASTER- Chapter6 Outputs 6-3 Plot Out" for details.

● Printing from the Sheet Editor

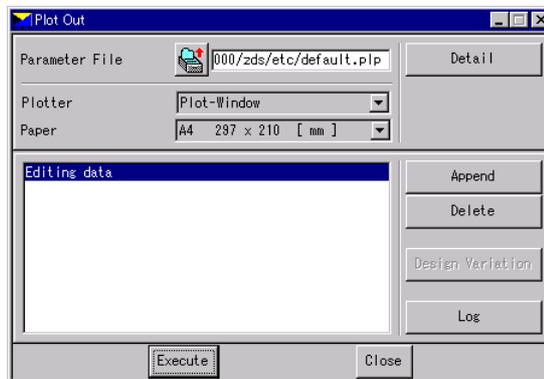
Printing from the Sheet Editor prints the currently displayed variant. Therefore, select display of the variant-specific properties before printing. Thus, before printing the schematic sheet, have the destination you want to print displayed on the screen.



NOTE

The plotter output of the no mount components depends on the settings in the "Plot Out" dialog.

Select **File** → **Print** from the menu bar.

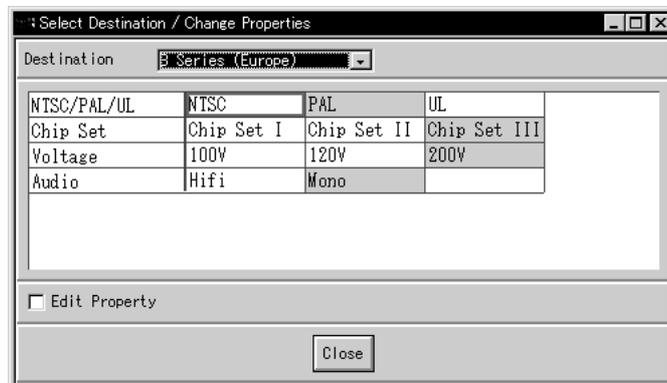


Use the variant properties edit dialog to change the display to a specific variant. Select **Attribute** → **Select Destination / Change Properties** from the menu bar to open the dialog.



See Also

See "● Displaying a Design Variant" for details.



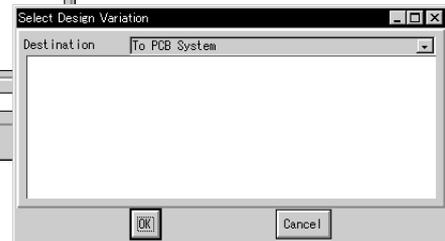
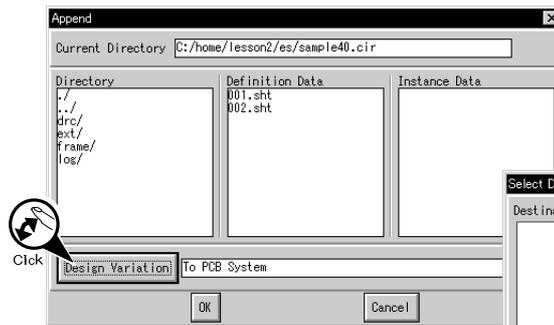
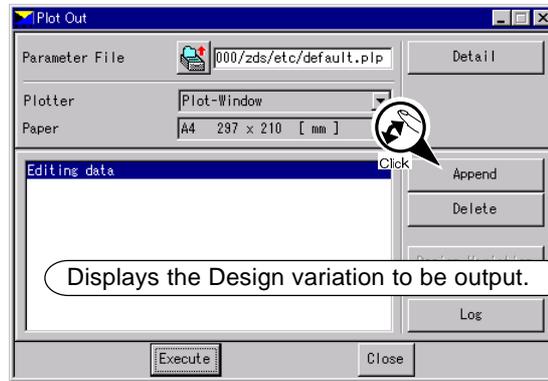
NOTE

By inputting the property viewer of [`@plotdestination`] in the frame symbol, "Design Variation Name" can be output while plotting out.

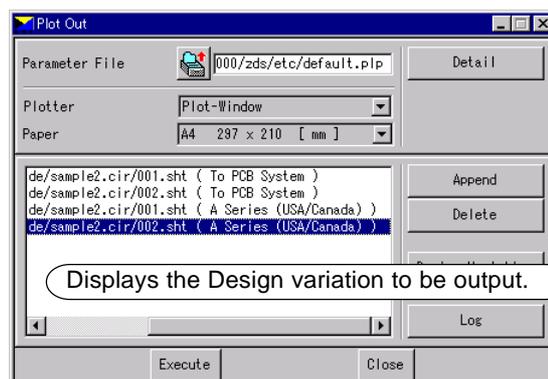
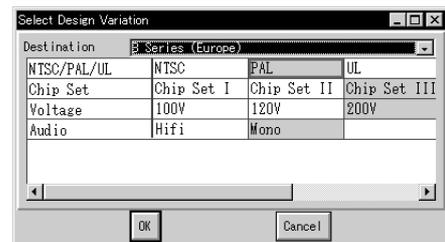
● **Printing from the Plotout dialog**

When Design variation manager has been performed, the variant to print can be specified in the plotter output dialog.

Open the plotter output dialog from the CR-5000 Design File Manager.



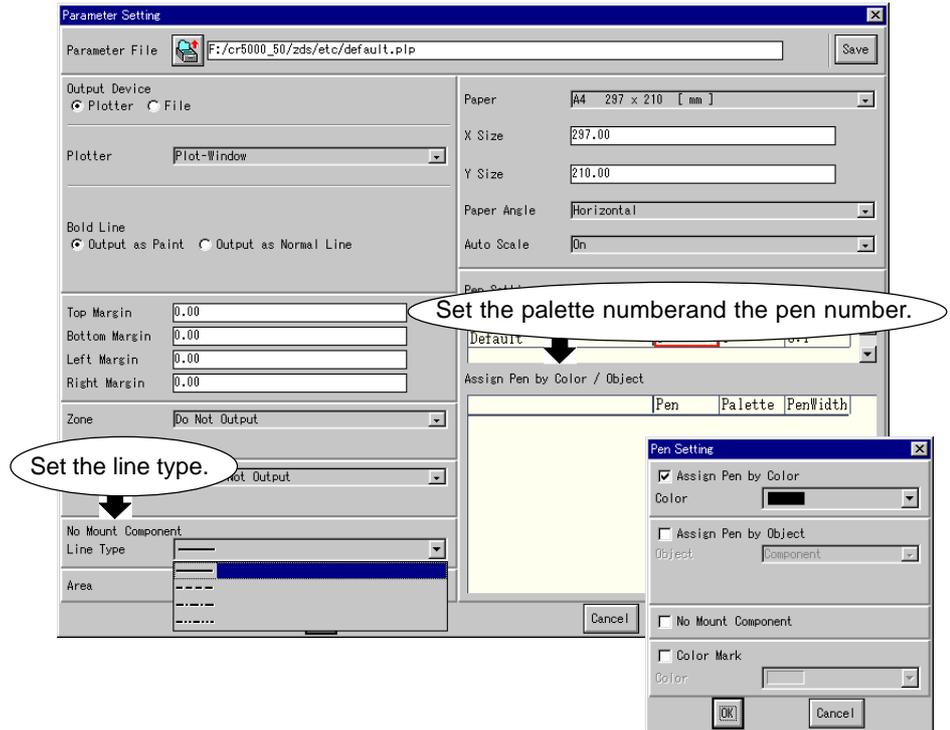
Select the Destination of Design variation.



*Printing No Mount Components

For the no mount components, you can set the following printing ways from the "PlotOut" dialog:

- Set the line type.
- Set the palette number and the pen number.



Set the pen number to 0 when you want to hide and print a no mount component.



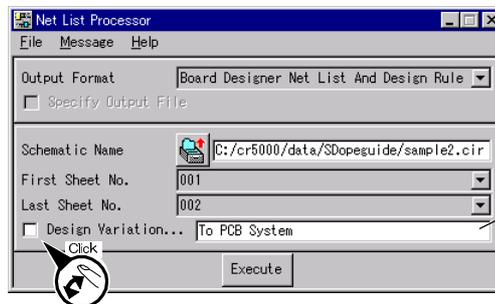
2. Outputting Design Variations (Netlist)

Variant-specific netlists, component lists, and similar can be output from the netlist processor for schematic sheets that use of Design variation manager.

● Outputting Design Variations (Netlists, Component Lists)

To start the netlist processor from the Sheet Editor, select the schematic directory then select **Utilities** → **Netlist Processor** from the menu bar.

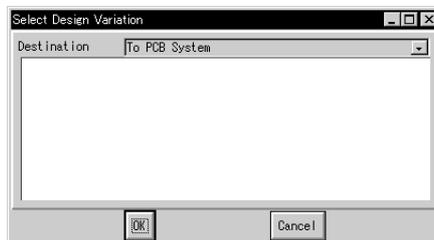
To start the netlist processor from the CR-5000 Design File Manager, select **Tool** → **Actions** → **Netlist Processor** from the menu bar.



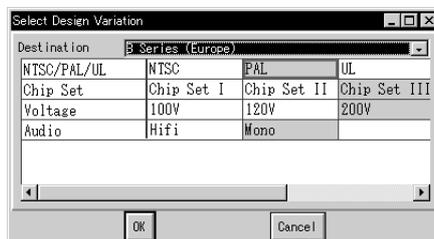
Displays the variant to be output.



Click the Design variation radio button.



Select the Destination of Design variation.





Displays the Design variation to be output.

NOTE

For net lists passed to PCB design, select “To PCB System” for design variations. Also, when performing back annotation from a PCB design to the schematic, execute it without specifying design variation. Since “To PCB System” is used to exchange data between PCB design and schematic, if the command is executed by selecting design variation, data mismatch may occur.

Even though “Not Mount On PCB” property is set to “YES”, the component and net information are output. For details, refer to “3. Interface with the PCB Design Tool”.

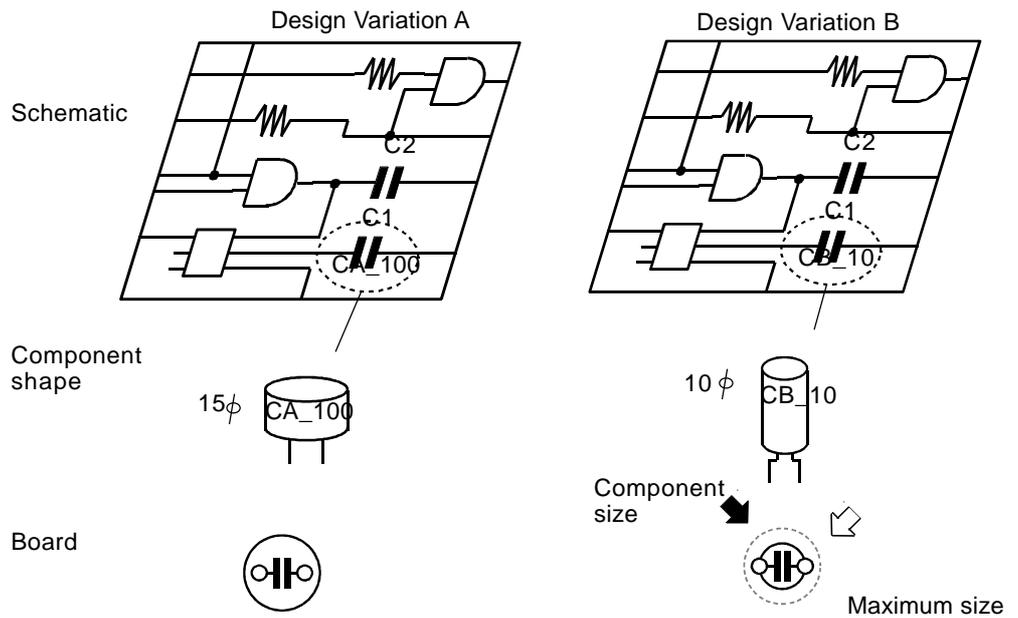
3. Interface with the PCB Design Tool

● To PCB System

Design Variation manager cannot be used to produce schematic sheets in which the net connection information is different for different Design variations. However, if the net connection information is the same but components (part names) are different, Design variation manager can be used for variants that use components with the same pin positions and shapes but with different sizes.

EXAMPLE

If the schematic (netlist) is the same but the components used differ depending on the Destination of Design variation, the component sizes may differ. The diagram below shows an example.



	To PCB System	Variation A	Variation B	Variation C
C1	CA_100	CA_100	CB_10	CA_220
Component size	15 φ	15 φ	10 φ	12 φ
C2	CA_470	CB_100	CA_10	CA_470
Component size	12 φ	10 φ	10 φ	12 φ

↑
Component size used to create board

↖ ↗
Maximum component size

Creating the board using the shape that gives the largest component size (variation A if C1, variation C if C2) enables the board for any variant to be produced simply by changing the components to be mounted, without problems (such as components unable to operate, connections, etc.) occurring.

Accordingly, by setting components for the variant specified as "To PCB System", the netlist (NDF) and design rules (RUF) can be created when the board is created.

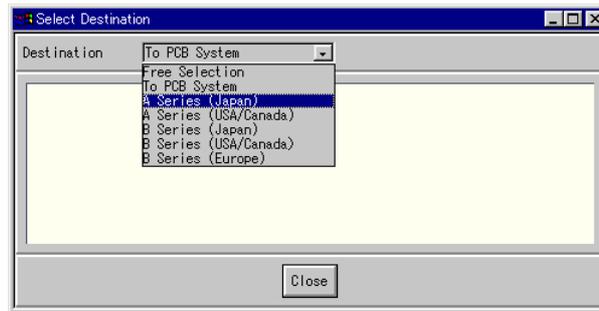
(For component mounting, use the component list and other information output for the specific variant.)

● Displaying the "To PCB System"

(The values displayed on the schematic when the file is opened are the "To PCB System".)

Use the Design variation properties edit dialog to change the schematic sheet display to the "board values" or to check the property values.

Select **Attribute** → **Select Destination** from the menu bar .

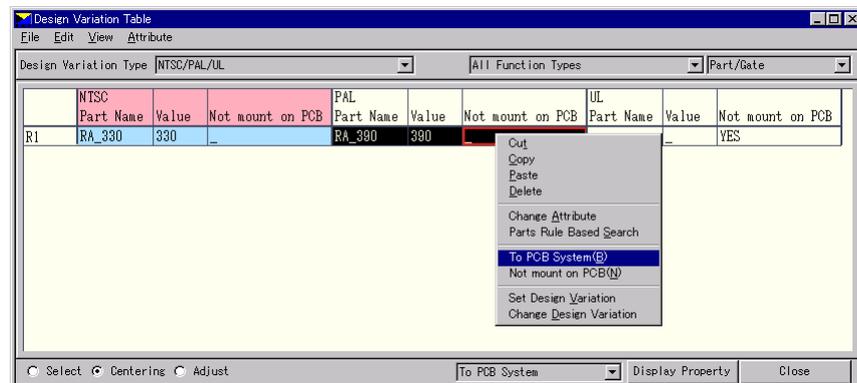


Select **To PCB System** from the variant list.

● Setting the "To PCB System"

The Design Variation Table dialog is used to set the " To PCB System" in the Sheet Editor.

Red: Design Variation being displayed and edited in the Sheet Editor
 Blue: Design Variation with property values set
 Black: Design Variation with the "To PCB System" settings



Select the cell where the property value to be edited is, then from the menu bar select [Property] or from the Assist Menu set the [To PCB System].

