cadence®

Dracula Graphical User Interface Reference

Product Version IC23.1
Septembee 2023

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

Contents

Preface

This manual assumes that you are familiar with the development and design of integrated circuits. It contains reference information about the Dracula® graphical user interface which belongs to the Dracula group of physical verification and analysis products.

The Dracula graphical user interface's executable in the IC 5.1.41 and IC 6.1.x releases is called `draculaInteractive`. You can access its functionality in standalone mode or via a DFII session. The Cadence® design framework II software refers to it in its user interface as the *Dracula Interactive* product. Refer to the previous versions for information on this product's former names.

The preface discusses the following:

- Related Documents
- Typographic and Syntax Conventions

Related Documents

For more information about the Dracula graphical user interface and other related products, you can consult the sources listed below.

- See the Virtuoso Layout Editor User Guide for information about the Design, Window, Create, Edit, Verify, Connectivity, Options, and Route menus.
- See the Cadence Installation Guide for information about how to install the product.
- If you want to use the Dracula® standalone verification product to verify your design, you should read the Dracula Reference and the Dracula User Guide. If you want to learn how to use the Dracula graphical user interface to identify and display DRC and LVS results, see also the *Dracula Graphical User Interface Tutorial*.

Typographic and Syntax Conventions

Here are some conventions used to describe menu commands.

Boxes and arrows in a sequence like the one below show you the order in which you select a command from a menu.

DRC ⇛ **Setup**

Each form shows you the system defaults:

- Filled buttons are the default selections.
- Filled-in values are the default values.

1

Introduction to the Dracula Graphical User Interface

The purpose of this chapter is to provide an overall introduction to the Dracula® graphical user interface which belongs to the Dracula group of physical verification and analysis products.

The Dracula graphical user interface's executable in the IC 5.1.41 and IC 6.x releases is called `draculaInteractive`. You can access its functionality in standalone mode or via a DFII session. The Cadence® design framework II software refers to it in its user interface as the *Dracula Interactive* product. Refer to the previous versions for information on this product's former names.

Note: Throughout this manual, when just the words "graphical user interface" is used, it refers to the Dracula graphical user interface product.

The chapter focuses primarily on the following:

- Introducing the Dracula Graphical User Interface
- Using the Dracula Graphical User Interface
- Using the Dracula Graphical User Interface on Layout Data Not in the Database

Introducing the Dracula Graphical User Interface

The graphical user interface is an interactive graphic tool for identifying and analyzing Dracula verification errors. With this tool, you can do the following:

- Display and analyze flat mode, composite mode, and hierarchical mode Design Rule Checker (DRC) errors
- Display and analyze flat mode, composite mode, and cell mode Layout Versus Schematic (LVS) errors
- Display derived layers for all Dracula products (DRC, ERC, LVS, LPE, and PRE)
- Display nets with the highest parasitic capacitance and resistance

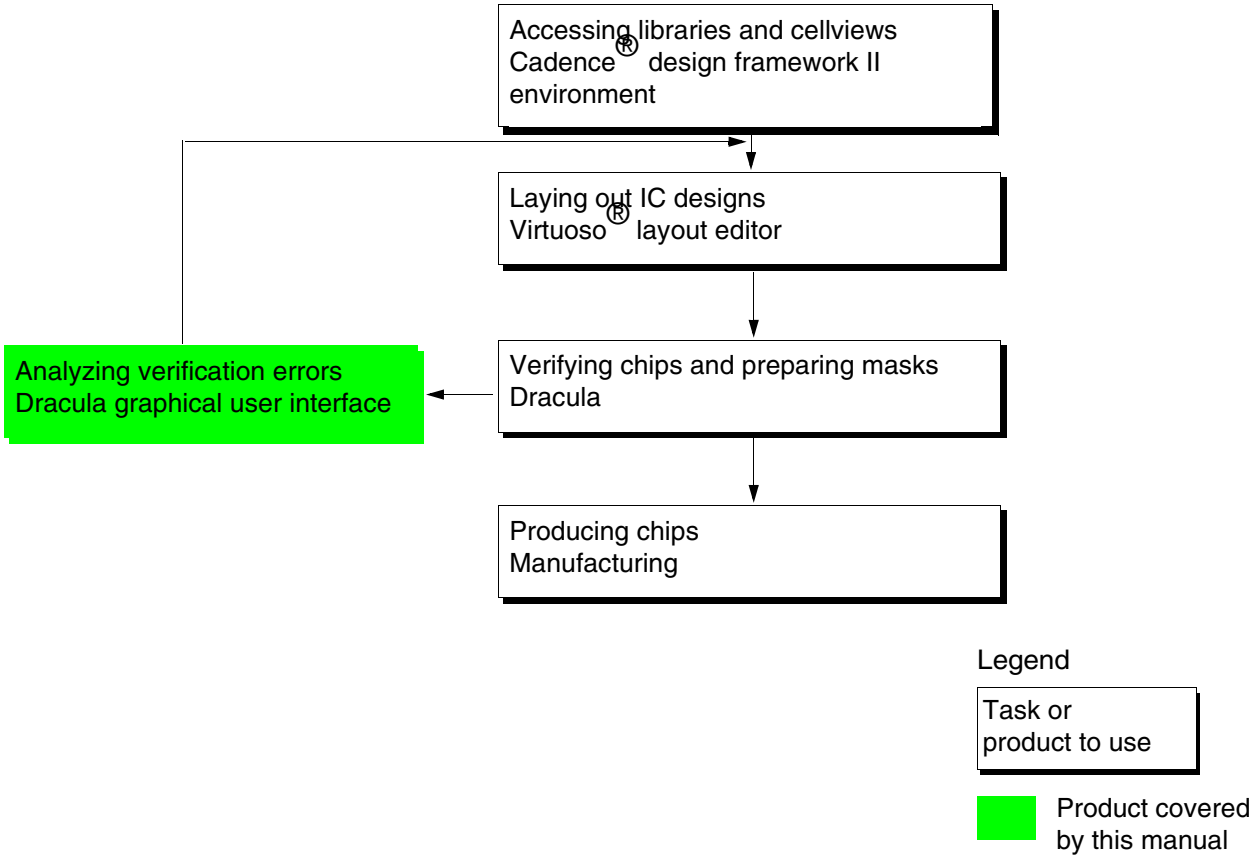
- Display the circuit netlist
- Highlight nets and devices
- Cross-display between the layout, netlist, and schematic (Virtuoso® schematic composer window)
- Display the circuit hierarchy
- Display Dracula text

The graphical user interface takes you through each verification error and keeps a record of the errors you have viewed and those you have not. If you need to interrupt an graphical user interface session, you can save the error-viewing status and resume where you left off.

This manual shows you how to display Dracula verification errors using this tool.

Verification Flow

The following chart shows the flow of tasks in the verification process and the product you use for each task. This manual covers the Dracula graphical user interface only.



Using the Dracula Graphical User Interface

The graphical user interface displays the Dracula verification data by reading files in the Dracula run directory and drawing them on the screen. This tool does not add or delete data

from the Cadence® design framework II (DFII) database. This tool can overlay the Dracula data on top of the DFII data.

Prerequisites

Do the following before running the graphical user interface:

- Use the DRE or HTV option when you run Dracula LOGLVS.
- Make sure the results of your Dracula run are in a UNIX directory on the system in which the Cadence software is running.
- Modify the Dracula information files if you moved the Dracula results directory or the netlists from one location to another or one machine to another.
- Set the `KEEPDATA` function to `DRACULAINTERACTIVE` or `YES` in the Description block of your Dracula rules file.
  - The Dracula graphical user interface supports Dracula distributed processing. You need to set the Dracula data path correctly while running the Dracula graphical user interface. To get the Dracula data, you have to set the path to master_node`/drac_link` where master_node is the directory from which you ran the Dracula distributed job.
  - Dracula distributed processing always runs with `KEEPDATA=YES`, regardless of what you specify. For a single CPU job, `KEEPDATA=NO` works correctly. For a MULTICPU job, however, specifying `KEEPDATA=NO` does not delete intermediate files such as `.DAT` after you run jobs.
  - You should not use the `/NEXT` command when compiling a Dracula rule file for Dracula distributed processing with PDRACULA.
- Netlists associated with the LVS section of a Dracula run are not normally stored in the results directory. The graphical user interface program reads the netlist location from the first line of the `CELLTABLE.HTV` file in the Dracula directory. Here is an example of the beginning of a `CELLTABLE.HTV` file.

```
/usr1/mnt3/your_path/
INV_B
netlist
          39          43
```

The first line defines the UNIX path to the netlists. If the UNIX path is a full path, you can move the Dracula results directory to another location on the same machine without any problems.

If the UNIX path is a relative path, you must move the netlists to maintain the same position relative to the Dracula directory. You can also modify the path inside the `CELLTABLE.HTV` file.

Setting up the DRC, LVS, and LPE Environments for the Dracula Graphical User Interface

Before you work with the Dracula graphical user interface, you must set up its environment. When you first start the graphical user interface, only the `Setup` commands in the DRC, LVS, and LPE menus are available to you. The rest of the commands are grayed out and cannot be selected.

When starting a new DRC run, you must do the following:

1. Start the Cadence software.
2. Open your design.
3. Select *Tools – Dracula Interactive* from the design window banner.
4. Specify your Dracula directory as follows:
   - To specify the Dracula DRC directory, select *DRC – Setup*.
   - To specify the Dracula LVS directory, select *LVS – Setup*.
   - To specify the Dracula LPE/PRE directory, select *LPE – Setup*.

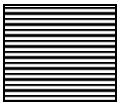
You must run *Setup* before you can use the other graphical user interface commands.

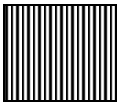
Note: The graphical user interface displays Dracula results from one directory at a time, and you can switch directories within the same run. For example, if DRC and LVS results are in different directories, you can specify either the LVS directory with the *LVS – Set Up* command or the DRC directory with the *DRC – Set Up* command.

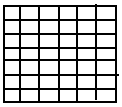
Displaying Error Flag Information

The graphical user interface uses the default display resource file (DRF) to display DRC error layers. The graphical user interface assigns unique colors to the first 23 flags, then starts again with the first color if the number of verification error layers exceeds 23.

The graphical user interface uses the following stipple patterns to display error flags of a DRC error layer.


Viewed error flags


Unviewed error flags


Current error flag

Saving and Restoring

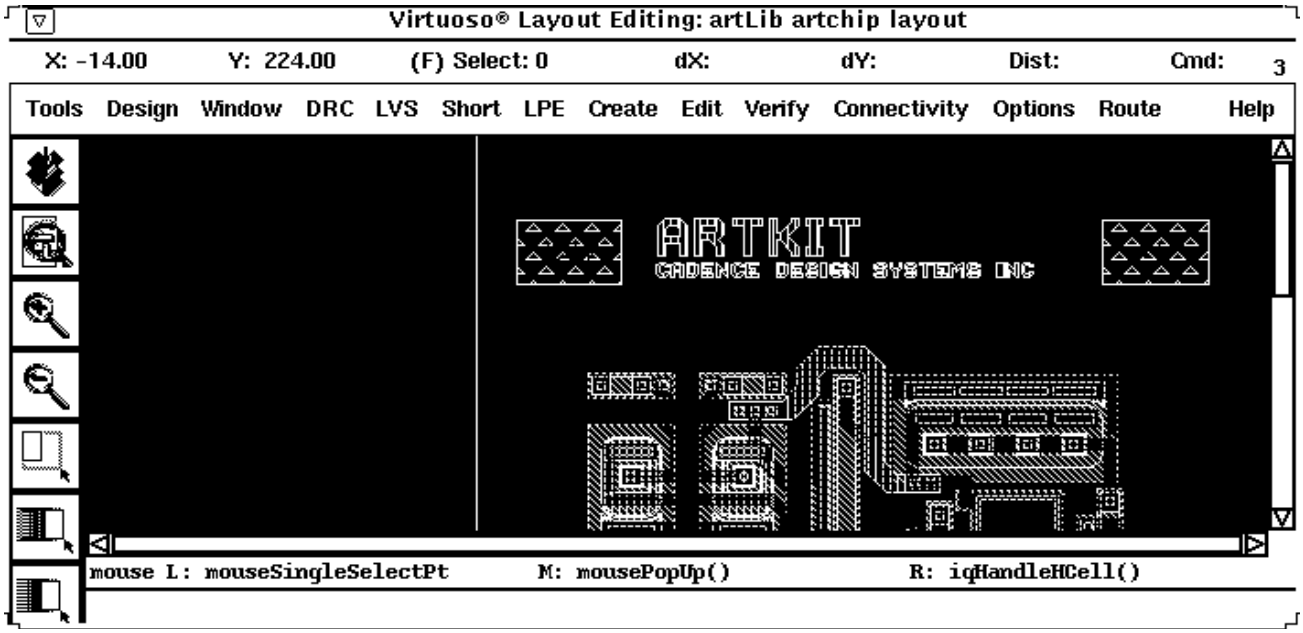
The graphical user interface keeps track of the DRC errors you have viewed by assigning errors a status of current, viewed, or unviewed. If you need to interrupt your session, you can store the error status using the *Error Status* command. See the "Error Status" section in the "DRC Commands" chapter for details about how to save and restore the error viewing status.

Using the Dracula Graphical User Interface Windows

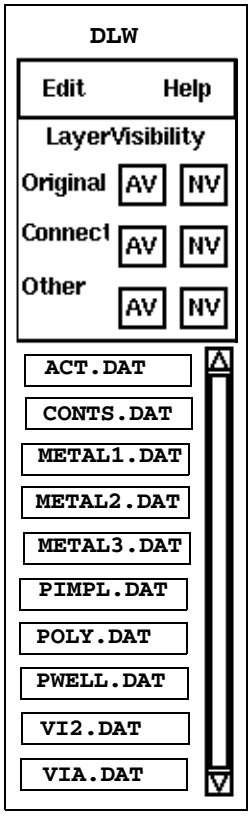
After you run *Setup*, the Dracula graphical user interface displays several windows that let you control how to display verification errors and parasitic data.

The following figures show the design windows for the graphical user interface.

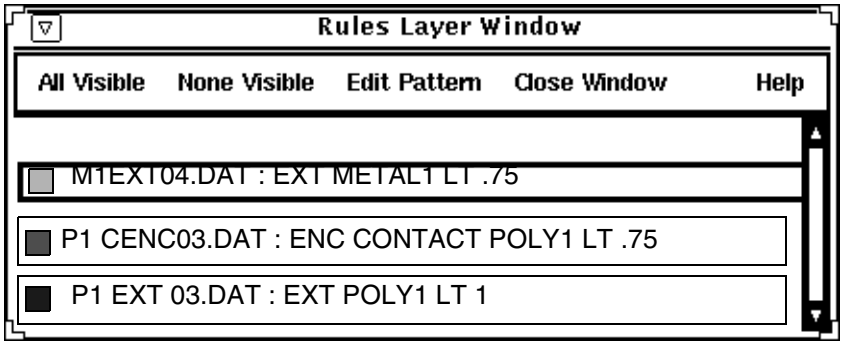
Dracula Graphical User Interface Design Window



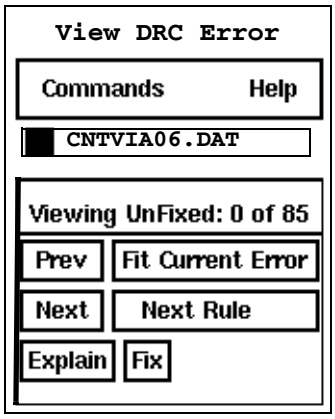
Dracula Layer Window (DLW)



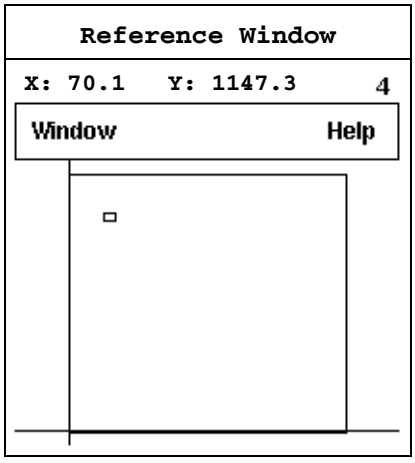
Rules Layer Window (RLW)



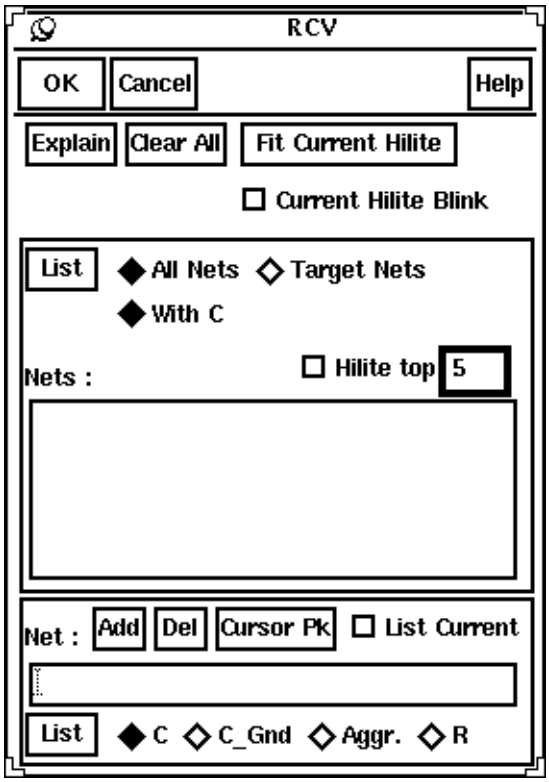
View DRC Error window



Dracula graphical user interface Reference window

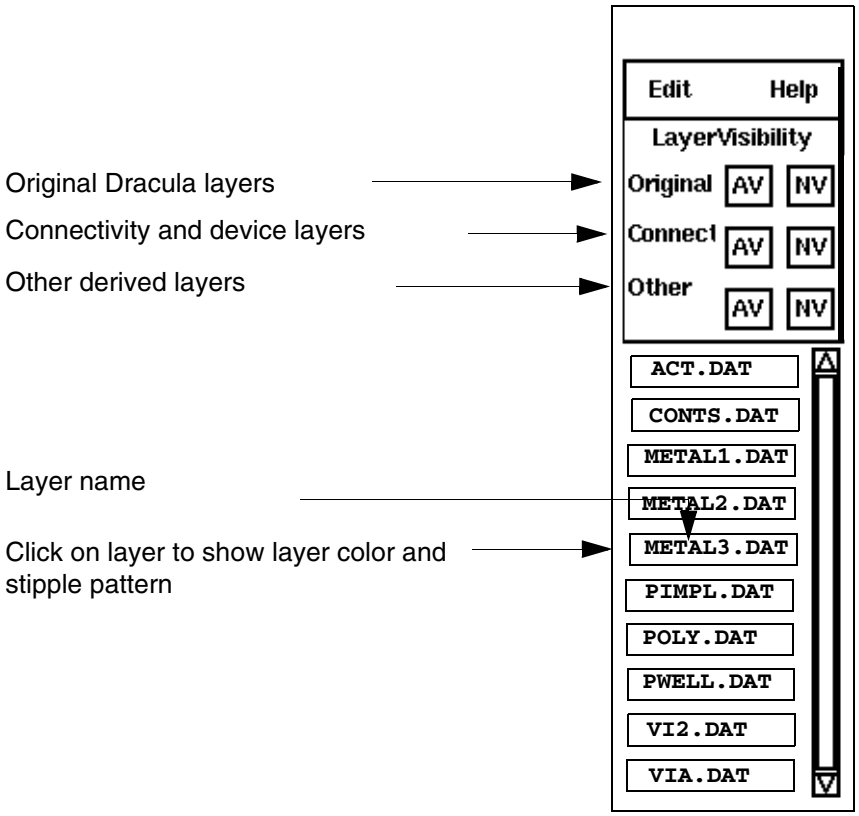


Resistance and Capacitance Viewing window (RCV)



Using the Dracula Layers Window (DLW)

To control which Dracula layers to display in the graphical user interface window, use the Dracula Layers Window (DLW). The following figure shows a sample DLW.



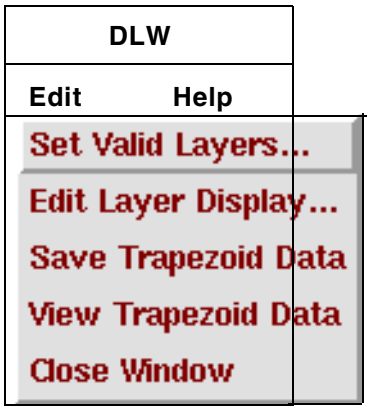
To make layers visible or invisible, use one of the options from the following table.

To make	Do this
One layer invisible	Click the middle mouse button on the layer.
An invisible layer visible	Click the middle mouse button on the layer.
All layers of a particular type invisible	Click on the NV button next to the type of layer you want to make invisible.
All layers of a particular type visible	Click on the AV button next to the type of layer you want to make visible.

To make	Do this
A layer disappear from the DLW	Click right on the layer name in the DLW.
A layer appear in the DLW	Select `Set Valid Layers` from the DLW Edit menu and use the Select Layer or Trapezoid Layer Menu button to display that layer in the DLW.

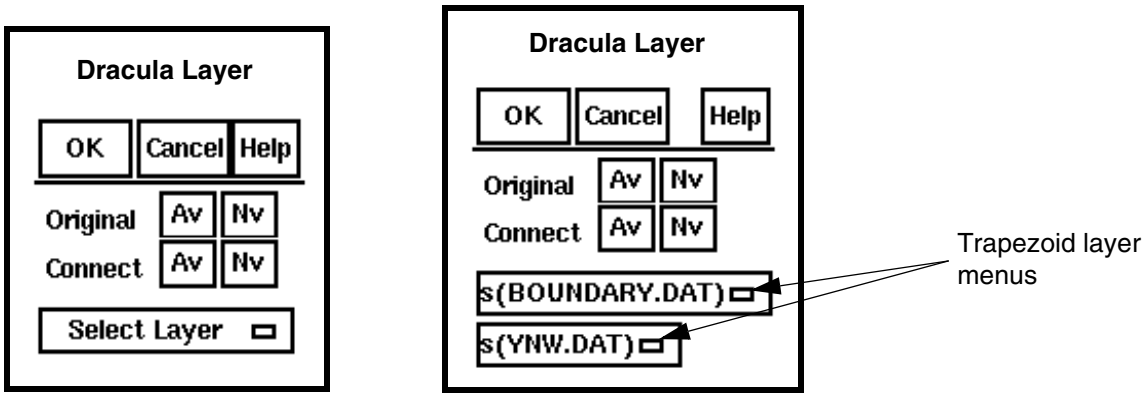
Edit Menu of the DLW

The *Edit* menu in the DLW window displays the following submenus.



Set Valid Layers

To control which layers to display in the DLW and graphical user interface design windows, choose *Edit – Set Valid Layers* from the DLW banner. This command displays one of the following forms.



Original (Av) installs all Dracula original layers in the DLW.

Original (Nv) removes all Dracula original layers from the DLW.

Connect (Av) installs all Dracula connectivity layers in the DLW.

Connect (Nv) removes all Dracula connectivity layers from the DLW.

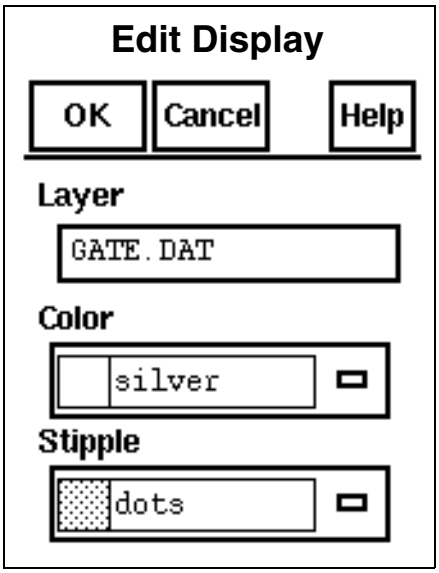
Select Layer lets you select layers which you want to be displayed and put in the DLW window. Selecting this menu item displays lists of layers in alphabetic order. The first 200 layers are listed in the first menu. If there are more than 200 layers, the remaining layers are listed in the second menu. The layer you choose is listed in the DLW and displayed in the graphical user interface design window.

Edit Layer Display

To set the graphic display of a layer

1. Select a layer in the DLW.
2. Choose *Edit – Edit Layer Display* from the DLW banner.

The *Edit Layer Display* command displays the following form.



Layer shows the layer name.

Color changes the color of the layer you select in the DLW and the graphical user interface design windows.

Stipple changes the stipple pattern of the layer you select in the DLW and the graphical user interface design window.

Save Trapezoid Data

DraculaInteractive lets you save a specified area of a layout window to a trapezoid file. This is a very useful feature for debugging because one can concentrate on a smaller amount of trapezoid data which is possibly corrupted instead of the whole trapezoid data file.

To save trapezoids in a certain layer follow these steps:

1. Set the layer you want to visible.
2. In the DLW, from the *Edit* menu, choose *Save Trapezoid Data.*

A message shows up in the CIW asking you to specify the layer in which you want to save the trapezoids.

```
Please click left mouse button in DLW to pick an entry layer.
```

Note: If no valid layer is set to visible, you get the following error message.

```
No valid layers are selected.
```

After you specify the entry layer, the trapezoids within that area are saved as trapezoid files. The prefix of those files are `IQ_` (For example, `IQ_ACTIVE.DAT`, `IQ_CONT.DAT`, and `IQ_MET.DAT`). A trapezoid gets saved as long as part of it is in the specified area.

View Trapezoid Data

Lets you view the contents of the files which you had saved using *Save Trapezoid Data*.

Close Window

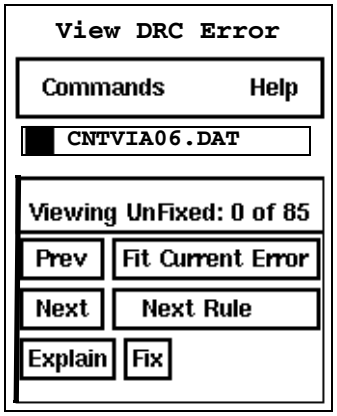
Closes the DLW window.

Using the Dracula Graphical User Interface Design Window

You use the graphical user interface design window to display your Dracula data.

Using the View DRC Error Window

To cycle through and manage DRC errors in the graphical user interface window, use the View DRC Error window. The following figure shows a sample View DRC Error window.



Commands are described in the "View DRC Error Window Commands Menu" section in Chapter 2 of this manual.

The current layer appears below the *Commands* menu button.

Prev displays the previous DRC error you viewed in the graphical user interface design window. In a hierarchical design, this button steps through only the visible HDRC errors of the selected cells.

Fit Current Error fits the current error in the graphical user interface design window.

Next displays the next error in the graphical user interface design window. In a hierarchical design, this button steps through only the visible HDRC errors of the selected cells.

If you click on the *Next* button after you displayed the last error for a rule, the next rule in the RLW with unviewed errors is displayed.

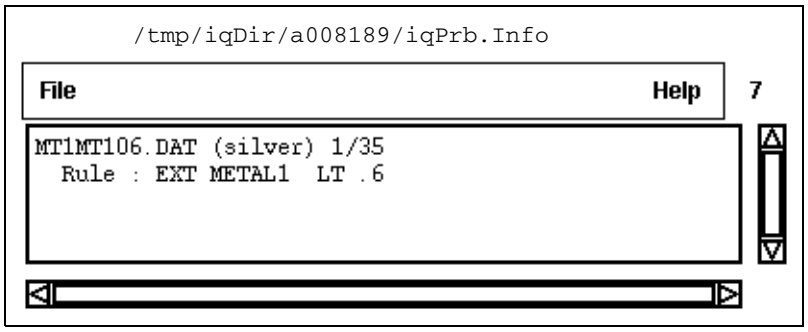
Next Rule selects the next rule in the RLW.

Explain updates the information in the Explain window with information about the current error in the graphical user interface design window. This button displays the Explain window if it is not displayed already.

The Explain window contains

- The Dracula error file name
- The error number and total number of errors for the selected Dracula error file
- The DRC rule or comment text

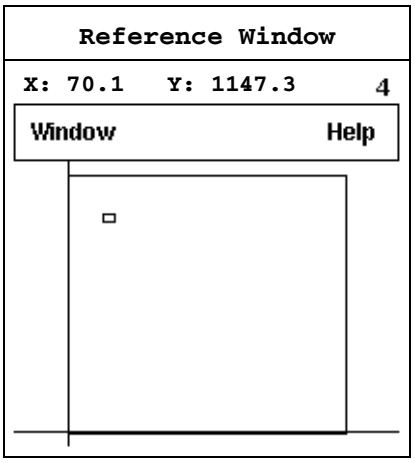
The following figure shows a sample Explain window.



Fix lets you point to a DRC error and mark it as fixed. Fixed errors do not appear in the graphical user interface design window.

Using the Reference Window

To make it easier to orient yourself in the larger graphical user interface window, the Reference window provides a smaller rectangular representation of your entire design. The following figure shows a sample Reference Window.



The following commands are available under the *Window* button in the Reference window.

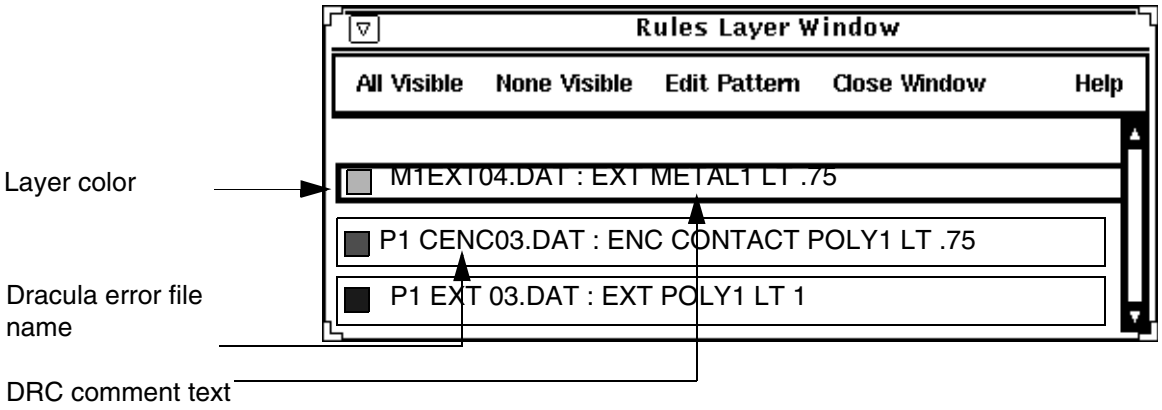
Zoom Reference Area lets you specify a rectangular area in the Reference window and zoom to the corresponding area in the graphical user interface design window.

Fit All fits your design in the Reference window.

Close closes the Reference window. To display the Reference window, select *Get Reference Window* from the DRC, LVS, or LPE menu in the graphical user interface design window or from the *Commands* menu in the View DRC Errors window

Using the Rules Layer Window (RLW)

The Rules Layer Window lets you select the Dracula error file that contains the DRC errors you want to see. The following figure shows a sample Rules Layer Window.



All Visible makes all Dracula DRC error files you selected with the *DRC – Select Error Files* command visible in the RLW.

None Visible makes none of the Dracula DRC error files you selected with the *DRC – Select Error Files* command visible in the RLW.

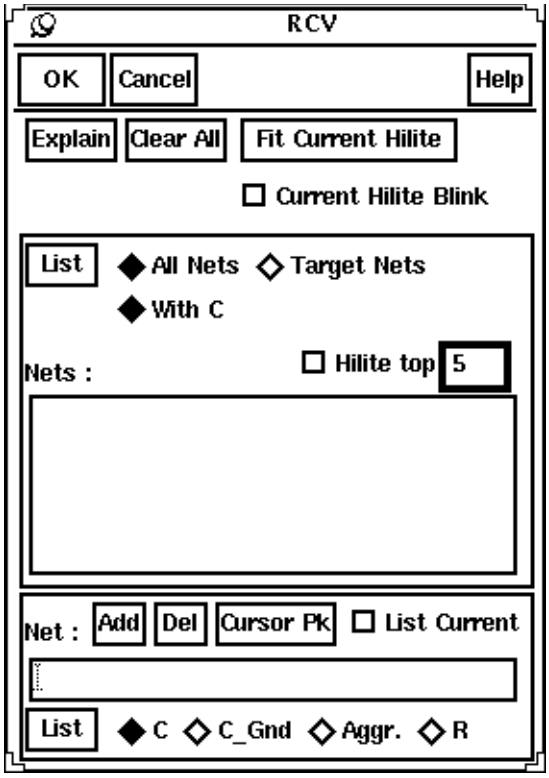
Edit Pattern lets you change the color and stipple of the DRC errors.

Close Window closes the Rules Layer Window.

To make a single Dracula error file visible, click left on its entry in the RLW. The last entry you click on becomes the current Dracula error file in the View DRC Error window.

Using the RCV Window

The Resistance and Capacitance Viewing window (RCV) lets you access the parasitic resistance and capacitance display capabilities in the layout window.



For details, see the View Parasitic section in the “LPE Commands” chapter of this manual.

Using the Dracula Graphical User Interface on Layout Data Not in the Database

Usually, the Dracula graphical user interface overlays Dracula output onto a cellview window containing the original layout data. However, you do not need to display the layout itself in the window. You only need to have a cellview window available.

In some cases, Dracula output data can be used for a layout that is not available inside the Cadence design framework II database. To use this data, you need to open a window onto which the Dracula output can be superimposed.

You can access the Dracula data and create a dummy cell of the correct cell size and data units by doing one of the following:

- Starting the CIW and typing in the SKILL function, `iqCreateDummyCell`.
- Starting the draculaInteractive window (by typing `draculaInteractive` on the command line) and using the *Utility – Create Dummy Cell* menu command

  Note: You cannot edit your data in the draculaInteractive window.

You can then open the dummy cell in the normal way, as if it contained the original Dracula layout data. Because a cell cannot exist in a Cadence design framework II database without a library, the command also creates a dummy library.

iqCreateDummyCell

```
iqCreateDummyCell()
```

Description

Creates a dummy cell and library when you type this SKILL command, in the Command Interpreter Window (CIW).

Note: When you invoke the standalone draculaInteractive tool, there will be a pull-down menu for you to use to create a dummy cell. But, if you start `layoutPlus` or `icfb`, there will be no such menu for you to use. So in that case you have to type the command `iqCreateDummyCell` in the CIW of DFII to invoke a popup menu.

Arguments

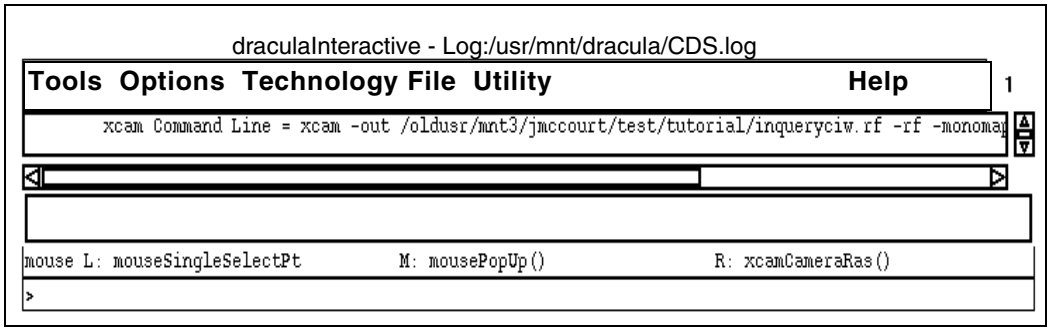
None

Create Dummy Cell

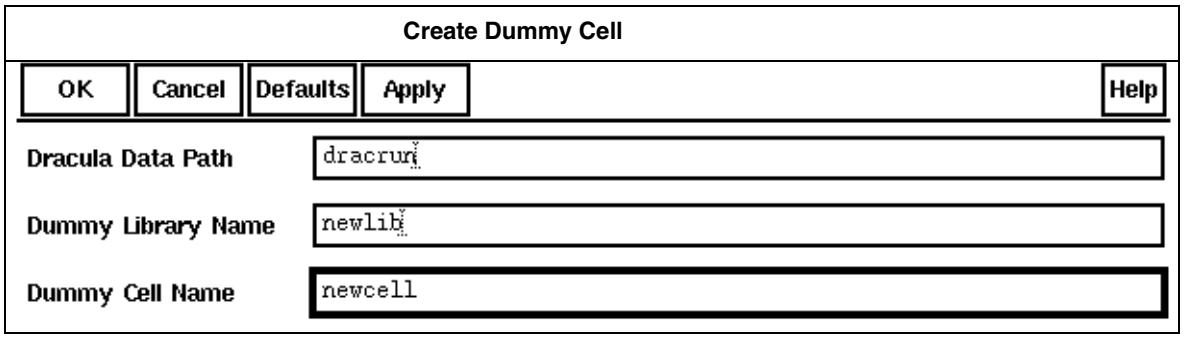
Start the Dracula graphical user interface by typing the following command at the UNIX command line:

```
draculaInteractive &
```

A draculaInteractive command window appears. Click on `Utility - Create Dummy Cell`. This command creates a dummy cell and library.



Create Dummy Cell Form



Dracula Data Path is the path to the directory containing the Dracula data you want to view.

Dummy Library Name is the name you choose for the dummy library containing the dummy cell.

Dummy Cell Name is the name you choose for a dummy cell in the dummy library. This cell will have the attributes of the master cell in the Dracula directory you specified.

Creating a Dummy Cell with the Dracula Graphical User Interface

To create a dummy cell, follow these steps.

1. To display the Create Dummy Cell form, do one of the following:
   - In the CIW, type `iqCreateDummyCell`.
   - In the draculaInteractive window, click on the graphical user interface's menu *Utility – Create Dummy Cell*.
2. Type the required path and names.
3. In the form banner, click on *OK*.
4. Continue initializing the graphical user interface as normal, using the cell and library names you specified in the form.

Reopening a Dummy Cell

You can reopen the dummy cell as follows.

1. In the CIW or the Dracula Interactive window, click on *File – Open*.
2. When the Open File form is displayed, select the name of your dummy library.
3. Be sure the name of your dummy cell is displayed in the *Cell Name* field. If not, click on the dummy cell name in the *Cell Names* list box.
4. In the Open File form, click *OK*.

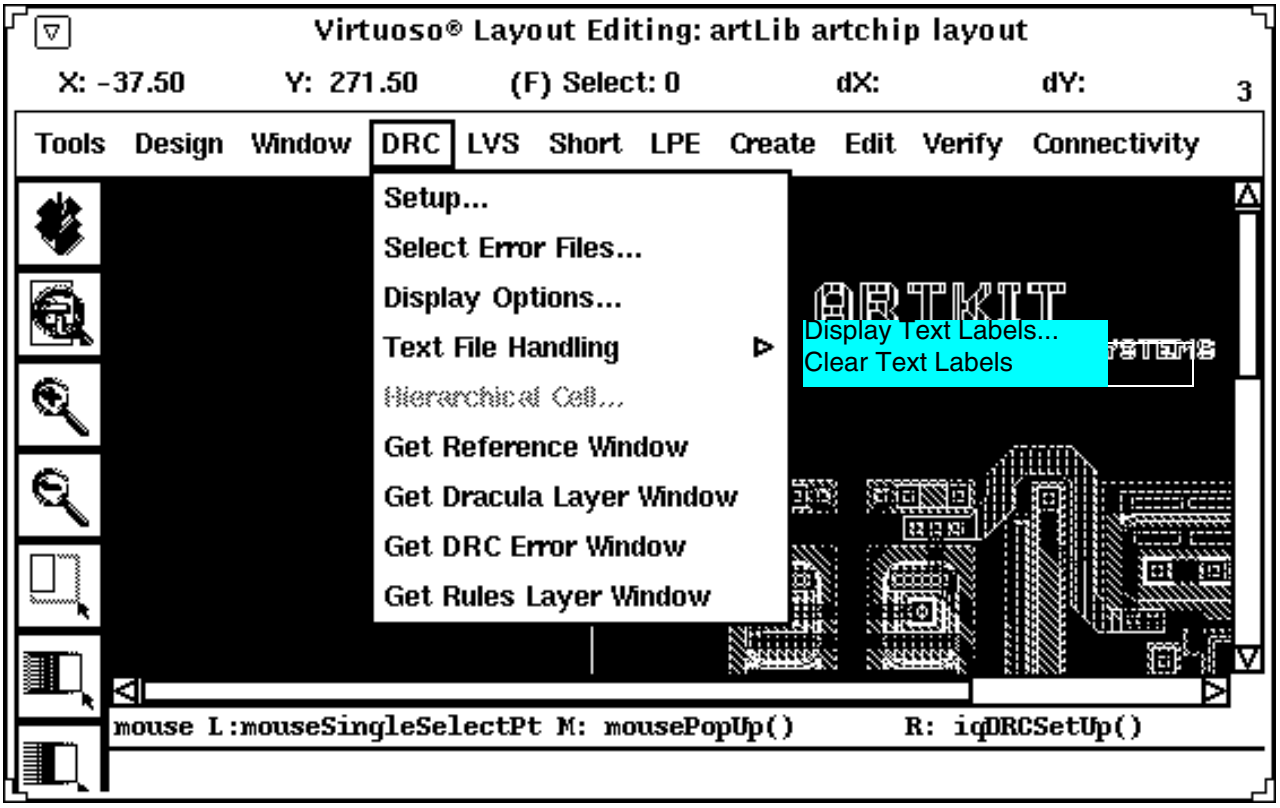
2

DRC Commands

This chapter discusses the DRC Menu.

DRC Menu

The DRC menu lets you view DRC errors, including HDRC errors. The menu contains the following commands:



Setup defines a Dracula directory and restores a previous run.

Select Error Files lets you select the error files you want to display.

Display Options controls how to display DRC errors.

Text File Handling lets you display text layers.

Hierarchical Cell lets you choose which Hcells to display and how to display them.

Get Reference Window displays the reference window.

Get Dracula Layer Window displays the DLW.

Get DRC Error Window displays the View DRC Error window.

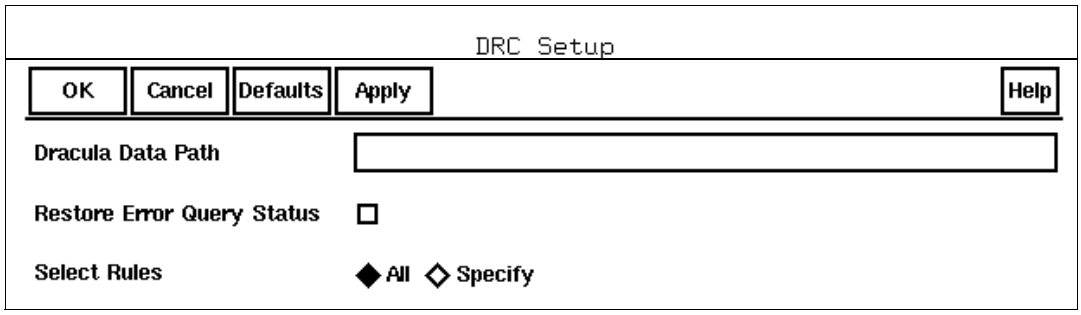
Get Rules Layer Window displays the Rules Layer Window.

Setup

DRC ⇛ **Setup...**

Lets you define the path to the Dracula directory containing the data you want to see. You must specify a directory before you can use any of the Dracula graphical user interface's DRC commands. This command also lets you restore the viewed and unviewed DRC error status from a previous Dracula graphical user interface analysis.

DRC Setup Form



Dracula Data Path is the path to the directory that contains the Dracula error files that you want to display. You can enter either a path relative to the current directory or a full path. The default is the current directory.

Restore Error Query Status restores the error status from a previous interactive debugging run.

When you save the error status using the `Error Status` command under the Commands menu of the View DRC Error window, the Dracula graphical user interface stores the following information in the `iqDRC.Status` file.

- Which display options you selected
- Which Dracula error files you displayed
- Which errors you displayed in each error file
- Which errors you marked as fixed

If you want to save more than one interactive debugging run, you can rename `iqDRC.Status` before doing each new save. When you restore a run, the graphical user interface looks for the `iqDRC.Status` file. Therefore, if you have changed the file name, you must rename it `iqDRC.Status` before you restore that specific Dracula interactive debugging run.

Select Rules controls which Dracula error data is displayed in the graphical user interface window and the RLW.

- ◈ **All** displays all of the Dracula error files.
- ◇ **Specify** displays the Select Error Files form to let you choose which Dracula error files you want to display.

Using DRC Setup

To set up your Dracula data and display it in the graphical user interface, follow these steps.

1. Select *Setup* from the DRC menu.

   The DRC Setup form appears.
2. Type in the path to the Dracula directory.
3. To restore the error status of a previous run, click on *Restore Error Query Status*.
4. To specify which Dracula errors to display, set Select Rules to *All* (the default) or *Specify*.
5. Click on *OK*

Select Error Files

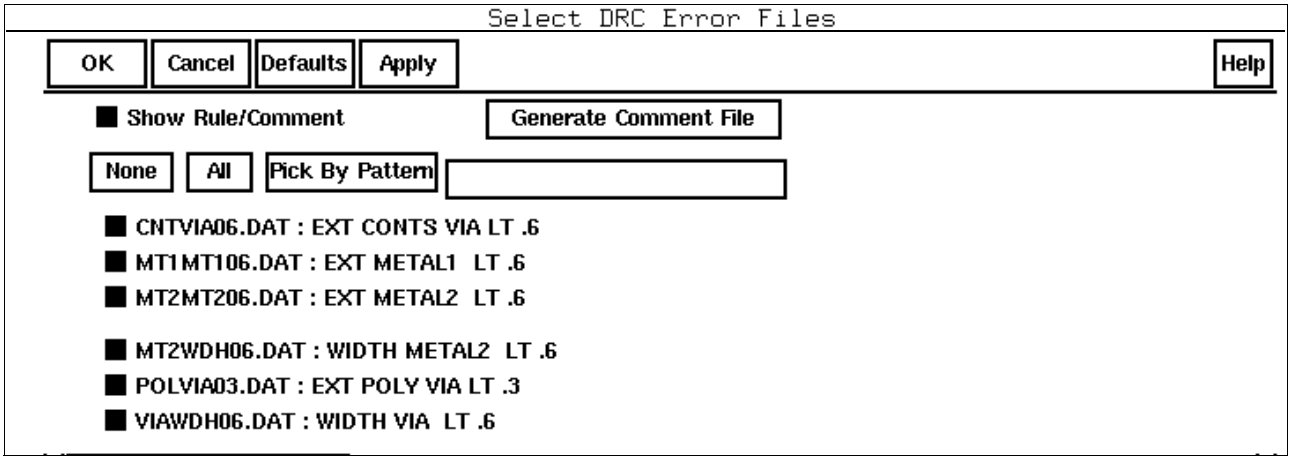
DRC ⇛ **Select Error Files...**

Lets you select the DRC error files you want to display.

Prerequisites

You must specify the Dracula directory with the `Setup` command.

Select DRC Error Files Form



▣ **Show Rule/Comment** specifies whether or not to display the comments associated with the Dracula error files you select. When you select this option, it reads the `drc.comment` file in the Dracula data path and displays the comments that attach to rules in the Select DRC Error Files form. If a comment does not attach to a rule, this option displays the Dracula rule text after the rule name.

Generate Comment File creates a new `drc.comment` file in the Dracula directory and overwrites the existing `drc.comment` file. To add comments to the `drc.comment` file, read the file in a text editor and add the comments that you want to appear in the Select DRC Error Files form. Click the *Show Rule/Comment* option off and then on again to display the new comments in the form.

Following are a few examples of entries found in a `drc.comment` file.

```
D52110 : 5.2.1 N-well width <3.0u
D52211 : 5.2.2 N-well spacing <5.2u
```

```
D52312 : 5.2.3 P-well implant blk width <2.0u
D52413 : 5.2.4 P-well implant blk spacing <2u
```

The rule file might also need editing to work with the following conventions.

- Comments for a rule must start on the same line as the rule and can continue for consecutive lines after the rule.
- Two consecutive semicolons in the beginning of a line specify that the line is a continuation of the comment in the previous line.

For example:

```
EXT[T] PINN NOUT LT 6.2 OUT D5311A 26      ;1.1.1 P+ inside
   ;;N-well TO N+
   ;;outside N-well
   ;;<6.2u
NOT XPSUB SPTAP UPSUB OUT D5316 34   ;2.2.2 P tap density
   ;;>500u
AND CONTACT GATP12 E0710 OUT D5710 85   ;3.3.3 contact
   ;;crosses/overlaps
   ;;poly gates also
   ;;checked by 7.5/7.6
```

None deselects all Dracula error files listed in the form.

All selects all the Dracula error files listed in the form.

Pick By Pattern selects all the Dracula error files with file names containing the alphanumeric and wildcard text you type in the *Pattern* field.

▣ *file_name* lists the Dracula error files in the Dracula Path you specified when you ran *DRC – Setup.*

Using Select Error Files

To select error files to display in the graphical user interface's design window, follow these steps.

1. Choose *Select Error Files* from the DRC menu.

   The Select DRC Error Files form appears and gives you four choices.

   - ❑ To deselect all files, click on *None*.
   - ❑ To select all files, click *All*.
   - ❑ To select files by pattern, type the pattern in the field and click *Pick By Pattern*.

- To select individual files, click on the files you want.

2. Click on *OK*

Display Options

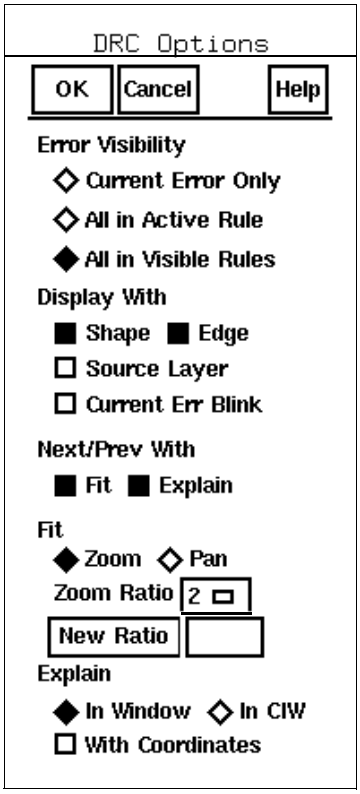
DRC ⇛ **Display Options...**

Lets you control how to display DRC errors in the graphical user interface design window.

Prerequisites

Before you can use this command, you must use *Select Error Files* to select the error files you want to view.

DRC Options Form



Error Visibility specifies which errors to display.

- ◇ **Current Error Only** displays only the last (current) error.
- ◇ **All in Active Rule** displays all the errors in the current active file.

◈ **All in Visible Rules** displays all the errors in all the visible files you selected.

Display With determines how errors are highlighted.

▣ **Shape** highlights the polygon area between the edges that indicate a DRC violation.

▣ **Edge** highlights the line edges that indicate a DRC violation.

☐ **Source Layer** displays the Dracula layers associated with the DRC violation.

☐ **Current Err Blink** controls whether the current error blinks.

Next/Prev With controls how to display the next and previous error you specify with the *Next* and *Prev* buttons in the View DRC Error window.

▣ **Fit** fits the next or previous error in the graphical user interface design window.

▣ **Explain** updates the text in the Explain window when you select the next or previous error.

Fit controls the window display size relative to the next or previous error.

◈ **Zoom** zooms in on the next or previous error.

◇ **Pan** pans to the next or previous error.

Zoom Ratio controls the window display size relative to the current error displayed. To see more of the area that surrounds the current error, specify a higher value in this field. The default value is 2.

New Ratio adds a user-defined zoom ratio value to the *Zoom Ratio* cyclic field. To add a user-specified Zoom Ratio value, type the value you want in the field and click on the *Zoom Ratio* button.

Explain determines how the information about a DRC violation is displayed.

◈ **In Window** displays the information about a DRC violation in a separate Explain window.

◇ **In CIW** displays the information about a DRC violation in the CIW.

☐ **With Coordinates** displays the coordinates of a DRC violation in the Explain window or the CIW.

Using Display Options

Many of the DRC options take effect as soon as you click on them. To control how errors appear in the graphical user interface design window, you can leave the DRC Options form on your screen and experiment with the different options.

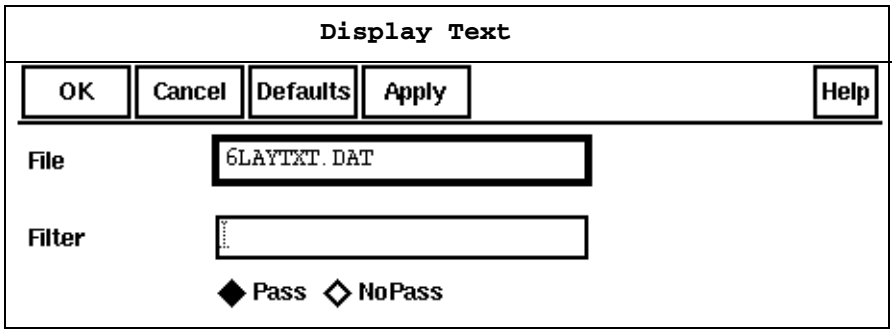
1. Select *Display Options...* from the DRC menu.
2. Select the display options you want to set.
3. Click on *OK* or *Cancel* to remove the DRC Options form

Display Text Labels

DRC ⇛ **Text File Handling** ⇛ **Display Text Labels...**

Displays text labels associated with the cellview. This command is also available on the LVS and LPE menus.

Display Text Form



File specifies the name of the Dracula text label file. For cell mode data, the default is *6LAYTXT.CEL*. For composite mode data, the default is *6LAYTXT.DAT*.

Filter lets you choose which text labels you want to display by specifying one or more text strings. Separate the text strings with a space. If you leave this field blank, all text labels in the Dracula text label file are displayed.

◈ **Pass** displays only text labels that contain the text strings specified for *Filter*.

◇ **NoPass** displays only text labels that do not contain the text strings specified for *Filter*.

Using Display Text Labels

To display text labels, follow these steps.

1. From the DRC menu, select *Text File Handling – Display Text Labels*.
2. Type the Dracula text label file name.
3. To display text labels, use one of the following methods:

- To display only certain text labels, type text strings contained in these text labels in *Filter* and click on *Pass*.
- To exclude text labels from being displayed, type text strings contained in these text labels in *Filter* and click on *NoPass*.
- To display all text labels, leave *Filter* blank.

4. Click on *OK*.

Clear Text Labels

DRC ⇛ **Text File Handling** ⇛ **Clear Text Labels...**

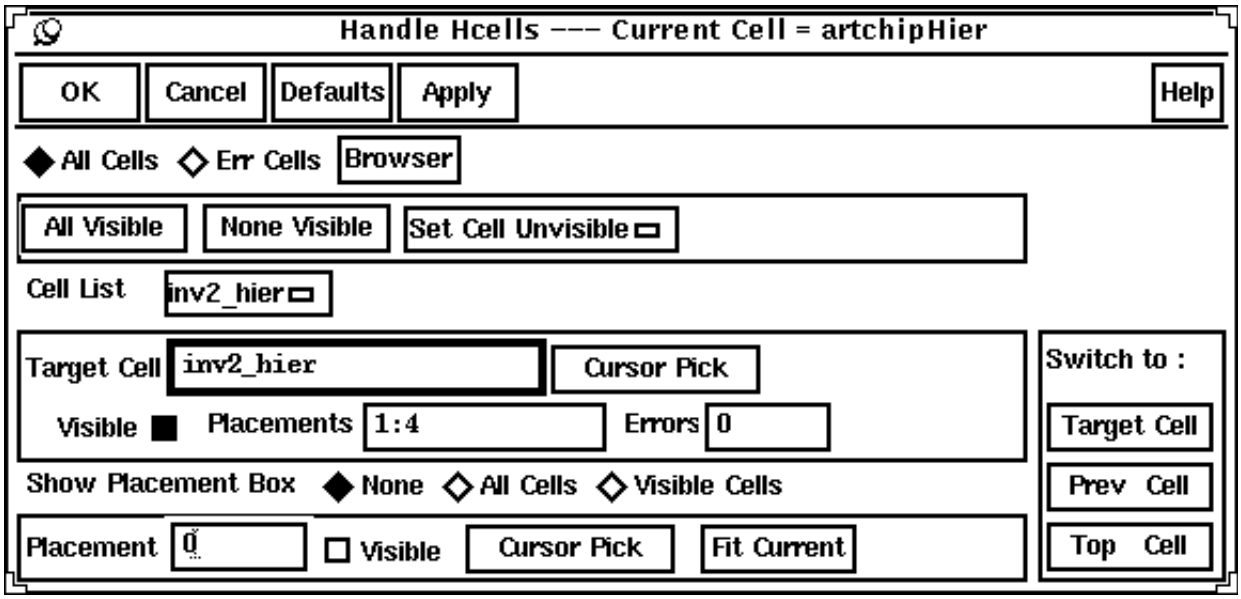
Removes the text labels from the current window.

Hierarchical Cell

DRC ⇛ **Hierarchical Cell...**

Lets you control which Hcells to display in the Dracula graphical user interface design window. This command is also available in the LVS and LPE menus.

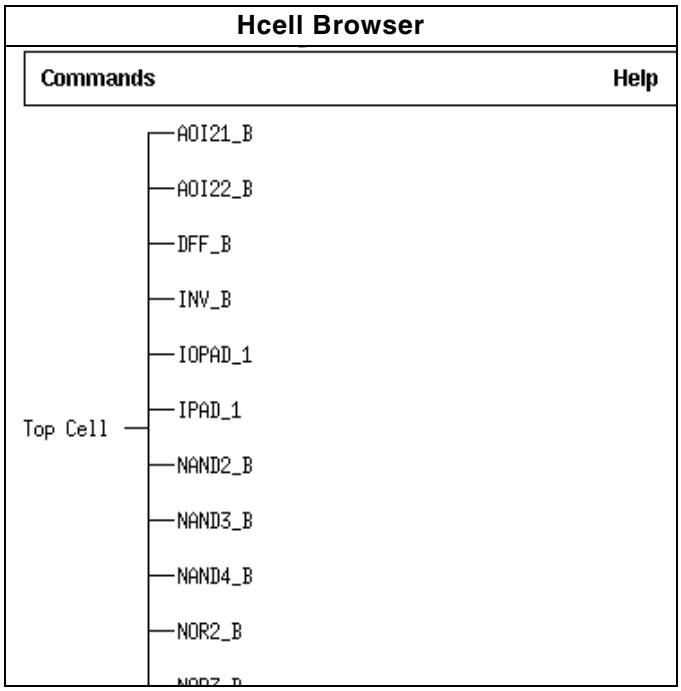
Handle Hcells Form



◈ **All Cells** lists all of the Hcells in the *Cell List* cyclic field.

◇ **Err Cells** lists only the Hcells with errors in the *Cell List* cyclic field.

Browser displays a hierarchical cell browser that you can use to display Hcells and information in the Handle Hcells form. The following figure shows a sample Browser window.



The graphical user interface's Hcell Browser lets you do the following:

- To expand the Hcell hierarchy display, click on a cell name.
- To display the Hcell in the graphical user interface window, hold down the middle mouse button and select *Switch to*.
- To unselect the Select option below the *Target Cell* field, hold down the middle mouse button and select *Unselect*.
- To activate the Select option below the Target Cell field, hold down the middle mouse button and choose *Select*.
- To update the Target Cell, Placements, and Errors fields in the Handle Hcells form, hold down the middle mouse button and select *Show Attributes*.

To close the graphical user interface's Hcell Browser window, select *Commands – Close HcBrowser.*

All Visible displays all of the Hcells in the Cell List cyclic field.

None Visible turns off the displayed Hcells in the Cell List cyclic field.

Switch to lets you choose which cell to switch to in the graphical user interface window.

Target Cell switches to the target Hcell listed in the Target Cell field.

Prev Cell switches to the previous Hcell you displayed.

Top Cell switches to the top level cell.

Cell List lists the Hcells that you can display in the graphical user interface window.

Set Cell Unvisible lets you remove a cell from the Cell List cyclic field.

Target Cell is the name of the cell to switch to when you use the *Target Cell* button.

Cursor Pick lets you point to an Hcell and select the Hcell name in the *Target Cell* field.

Visible displays or turns off the display of the cell you specified in the *Target Cell* field.

Placements displays the Hcell placement numbers of the Hcell in the *Target Cell* field.

Errors displays the number of errors in the Hcell in the *Target Cell* field.

Show Placement Box controls the display of the Hcell bounding boxes.

◈ **None** displays none of the Hcell bounding boxes.

◇ **All Cells** displays all of the Hcell bounding boxes.

◇ **Visible Cells** displays the bounding boxes of displayed Hcell.

Placement lets you specify an Hcell placement number. This option is useful if you want to display a specific Hcell instance. You can type in a placement number or use the *Cursor Pick* button to fill in the *Placement* field.

☐ **Visible** controls whether the placement is displayed. If selected, the Hcell that corresponds to the Hcell placement number in the *Placement* field is displayed.

Cursor Pick lets you point to an Hcell and fill in the Hcell placement number in the Placement field.

Fit Current fits the Hcell that corresponds to the Hcell placement number in the Placement field in the graphical user interface design window.

See the *Dracula Graphical User Interface Tutorial* for more information about how to use the Hierarchical Cells command.

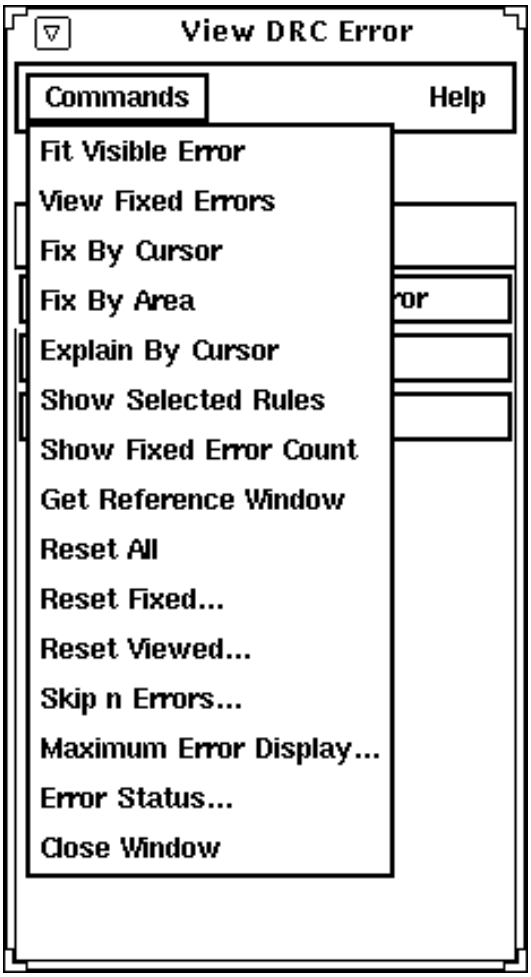
Get Reference Window

DRC ⇛ **Get Reference Window**

Displays the Reference window.

View DRC Error Window Commands Menu

The View DRC Error Window commands let you control how to display DRC errors in the graphical user interface design window. The menu contains the following commands.



For information about the buttons on the View DRC Error window, see the Using the View DRC Error Window in chapter 1 of this manual.

Fit Visible Error zooms the graphical user interface design window to display all visible errors.

View Fixed Errors displays the DRC errors you marked as fixed in the graphical user interface design window.

View UnFixed Errors appears in the Commands menu after you select *View Fixed Errors*. *View UnFixed Errors* displays the DRC errors you marked as unfixed in the graphical user interface design window.

Fix By Cursor lets you point to a DRC error in the graphical user interface design window and mark it as fixed.

Fix By Area allows users to mark the DRC errors as fixed in a certain area.

Explain by Cursor lets you point to a DRC error and get information about that error.

Show Selected Rules displays the RLW.

Show Fixed Error Count displays the number of errors you marked as fixed.

Get Reference Window displays the reference window.

Reset All resets all DRC error files to unviewed.

Reset Fixed changes the status of DRC errors in the Dracula error files from fixed to unfixed.

Reset Viewed changes the status of DRC errors in the Dracula error files from viewed to unviewed.

Skip n Errors is the increment to skip errors when you display them in the graphical user interface design window.

Maximum Error Display is the maximum number of errors to display in all DRC error files.

Error Status lets you query, save, or restore the error status.

Close Window lets you close the View DRC Error window.

Fit Visible Error

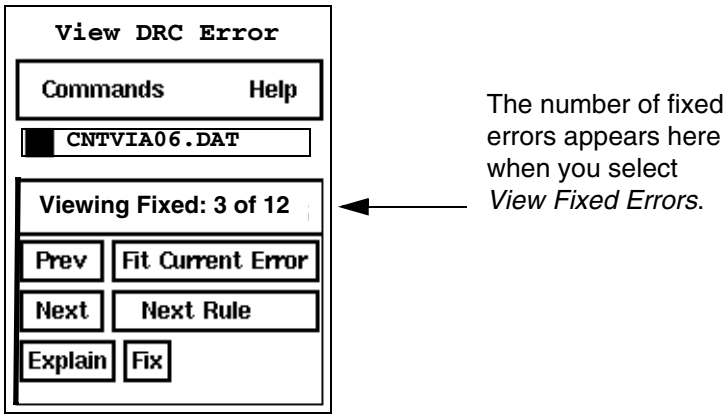
Commands ⇛ **Fit Visible Error**

Zooms the graphical user interface design window to display all visible errors.

View Fixed Errors

Commands ⇛ **View Fixed Errors**

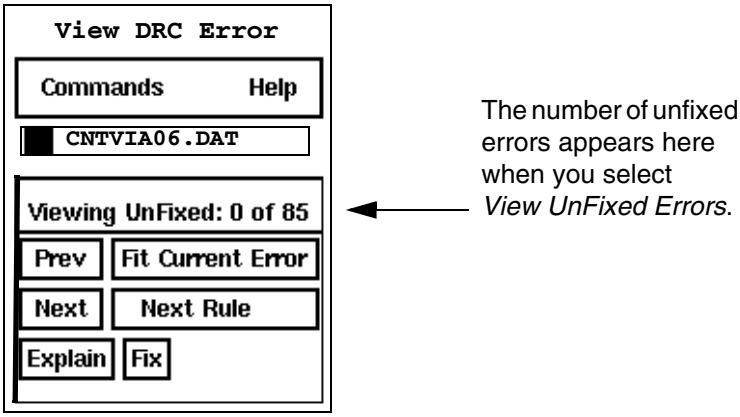
This is the default in the Commands menu of the View DRC Error window. *View Fixed Errors* displays the DRC errors you marked as fixed in the graphical user interface design window. The View DRC Error window shows the number of fixed errors that you are viewing. See the following example.



View UnFixed Errors

Commands ⇛ **View UnFixed Errors**

Appears in the Commands menu after you select *View Fixed Errors*. *View UnFixed Errors* displays the DRC errors you marked as unfixed in the graphical user interface design window. The View DRC Error window shows the number of unfixed errors that you are viewing. See the following example.



Fix by Cursor

Commands ⇛ **Fix by Cursor**

Lets you point to a DRC error in the graphical user interface design window and mark it as fixed. Fixed errors do not appear in the graphical user interface design window after you fix them.

UnFix by Cursor

Commands ⇛ **UnFix by Cursor**

Appears in the Commands menu after you select *View Fixed Errors*. *UnFix By Cursor* lets you point to a DRC error in the graphical user interface design window and mark it as unfixed. Unfixed errors are displayed in the graphical user interface design window.

Explain by Cursor

Commands ⇛ **Explain by Cursor**

Lets you point to a DRC error and get information about that error in the Explain window or in the CIW, depending on the option you selected with the `Display Options` command.

Fix By Area

Commands	⇛	Fix By Area

Allows users to mark the DRC errors as fixed in a certain area.

To use this function, follow these steps:

- Click left on *Fix By Area* and a message appears in the CIW asking you to specify the lower-left and upper-right corners of the area.
- Use the mouse to select the the lower-left and upper-right corners of the area in which you want ot mark the DRC errors as fixed.

Note: A DRC error is marked as fixed as long as part of that DRC error is in the specified area. To unfix the DRC errors by area, click on *View Fixed Errors* and the items on the Commands menu changes as follows:

View Fixed Errors --------> View Unfixed Errors

Fix By Cursor ----------> UnFix By Cursor

Fix By Area ----------> UnFix By Area

Note: Click on *UnFix By Area* to unfix DRC errors by area and follow the steps you need to do are the same as those given under *Fix By Area*.

Show Selected Rules

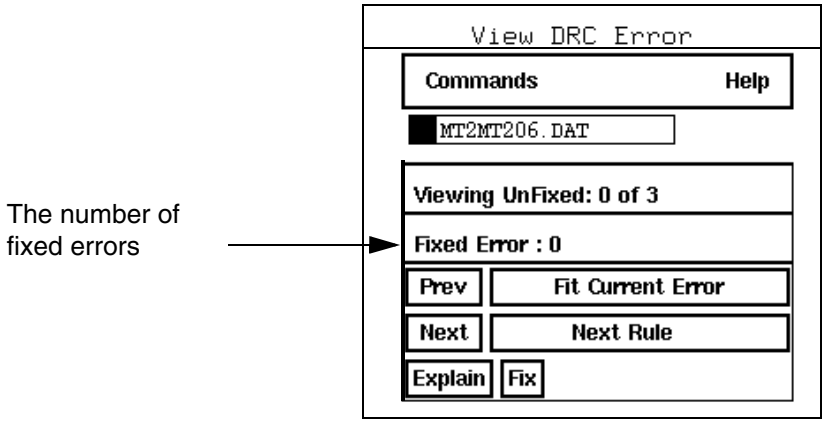
Commands ⇛ **Show Selected Rules**

Displays the RLW.

Show Fixed Error Count

Commands ⇛ **Show Fixed Error Count**

Displays the number of errors you marked as fixed with the *Fix* button with the *Fix by Cursor* or *Fix by Area* commands. See the following example.



Hide Fixed Error Count

Commands ⇛ **Hide Fixed Error Count**

Removes the display of the number of errors you marked as fixed with the *Fix* button with the *Fix by Cursor* or *Fix by Area* commands.

Get Reference Window

Commands ⇛ **Get Reference Window**

Displays the reference window.

Reset All

Commands	⇛	Reset All

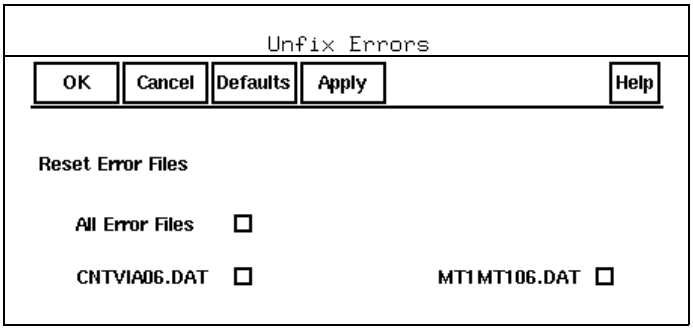
Resets all DRC error files to unviewed.

Reset Fixed

Commands ⇛ **Reset Fixed...**

Resets the status of DRC errors in the Dracula error files.

Unfix Errors Form



Reset Error Files lets you reset the status of DRC errors in the Dracula error files. Click on *OK* or *Apply* after you choose the following options.

All Error Files chooses all of the error files listed in the form.

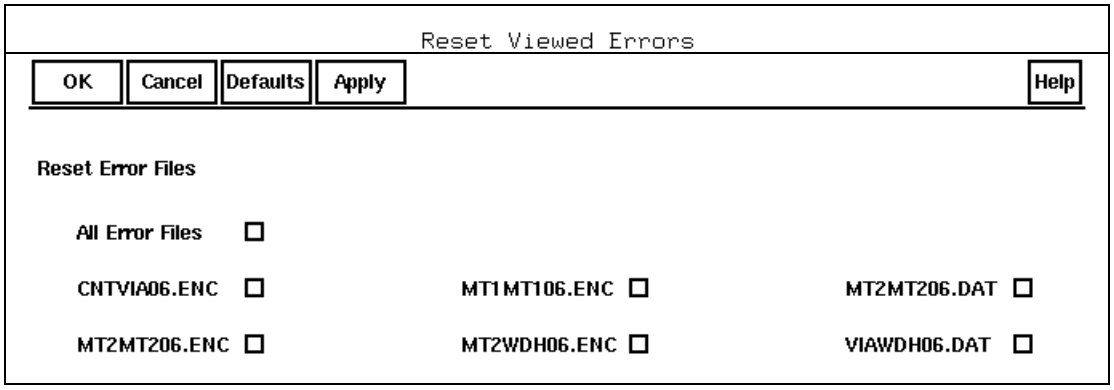
file_name lets you choose the status of DRC errors in the Dracula error files you want to change from fixed to unfixed.

Reset Viewed

Commands ⇛ **Reset Viewed...**

Lets you reset the status of DRC errors in the Dracula error files.

Reset Viewed Errors Form



Reset Error Files lets you reset the status of DRC errors in the Dracula error files. Click on *OK* or *Apply* after you choose the following options.

All Error Files chooses all of the error files listed in the form.

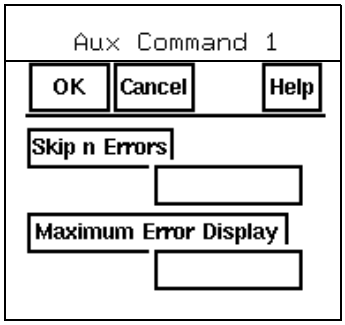
file_name lets you choose the status of DRC errors files you want to change from viewed to unviewed.

Skip n Errors/Maximum Error Display

Commands ⇛ **Skip n Errors...**

Commands ⇛ **Maximum Error Display...**

Aux Command 1 Form



Skip n Errors is the increment to skip errors when you display them in the graphical user interface design window. Suppose that the View DRC Errors window indicates that you displayed 1 of 10 errors. If you type 3 in the field and click on the *Skip n Errors* button, error 4 of 10 is displayed in the graphical user interface design window. If you click on the *Skip n Errors* button again, error 7 of 10 is displayed in the graphical user interface design window. You can type a positive or negative value in the field.

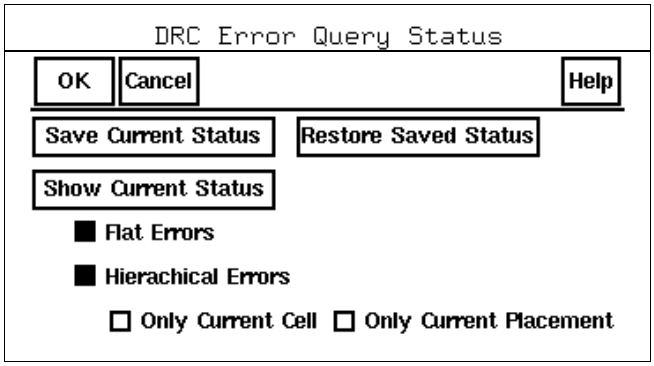
Maximum Error Display is the maximum number of errors to display in all DRC error files. Suppose that your design has 100 total errors. If you type 25 in the field and click on the *Maximum Error Display* button, you can only cycle through the first 25 errors. The default is 1000.

Error Status

Commands ⇛ **Error Status...**

Lets you query, save, or restore the error status.

DRC Error Query Status Form



Save Current Status saves the current DRC error viewing status in a file named `iqDRC.Status` in the working directory. If a status file already exists, this graphical user interface overwrites it.

Restore Saved Status restores the DRC viewing status. The option is useful if you want to resume the error-viewing status from the previous graphical user interface run.

Show Current Status displays the number of errors currently loaded, the number viewed so far, and the number not yet viewed for each file that has been active during this session.

Note: The following form fields apply only to hierarchical designs.

- ▣ **Flat Errors** shows only the flat DRC errors in the error status window.
- ▣ **Hierarchical Errors** shows only the hierarchical DRC errors in the error status window.
  - ☐ **Only Current Cell** displays only the hierarchical errors in the current cell in the error status window.

☐ **Only Current Placement** displays only the hierarchical errors in the current cell placement in the error status window.

Using Error Status

To use *Error Status*, follow these steps.

1. Selec*t Error Status* from the View DRC Error window.

   The Error Status form appears.

2. To see the current error status, click on the *Show Current Status* button.

   A new window appears, listing the DRC Output Error Summary, DRC Error Display Options, and DRC Error Files Without Errors.

3. To save the status of the viewed and unviewed errors, click on the *Save Current Status* button.

4. To restore a saved error status, click on the *Restore Saved Status* button.

   After restoring the status, you can use *Fit Next Error* to display the next unviewed error in the file that was the active file when you saved the status.

5. Click on *OK*.

Close Window

Commands ⇛ **Close Window**

Lets you close the View DRC Error window.

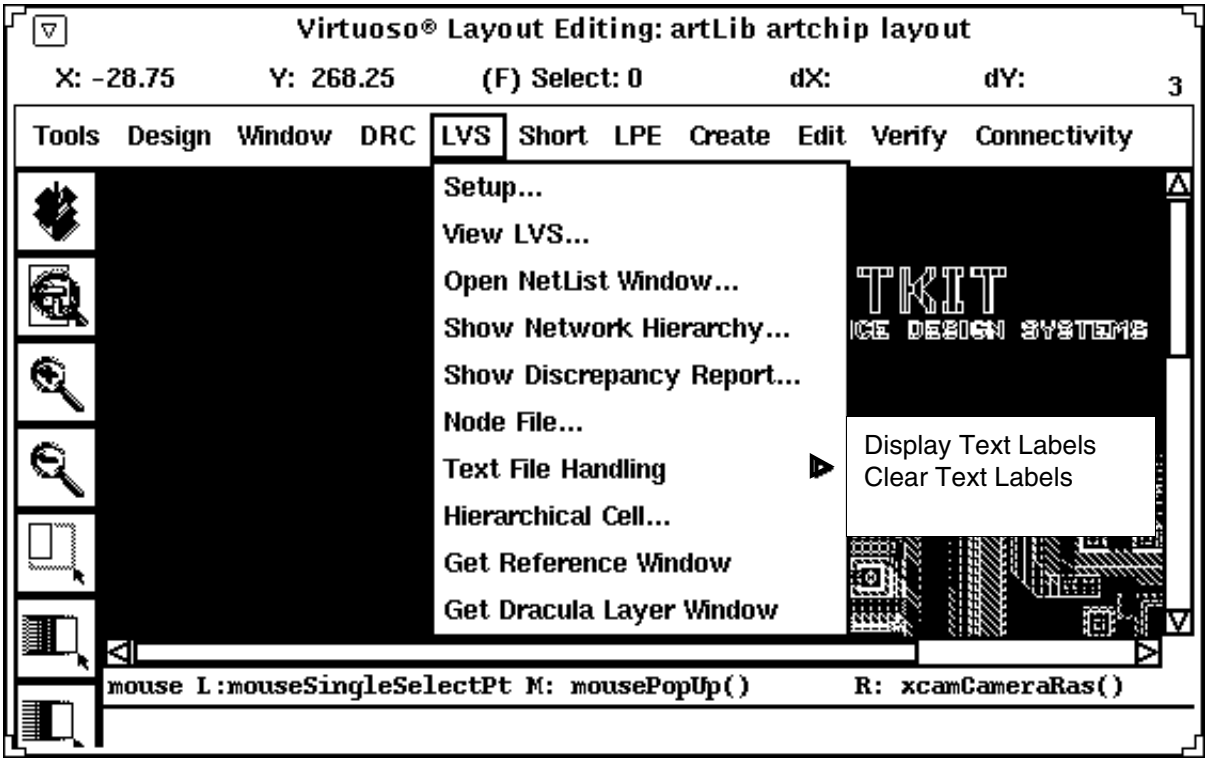
3

LVS Commands

This chapter discusses the LVS Menu commands.

LVS Menu

The LVS menu lets you display layout versus schematic errors.



Setup defines a Dracula directory and sets parameters.

View LVS... lets you set display options and get information about nets, devices, and errors.

Open Netlist Window opens a new window containing netlist information.

Show Network Hierarchy displays the circuit hierarchy.

Show Discrepancy Report displays the LVS discrepancy report.

Node File... displays the Node File form.

Text File Handling lets you display text layers.

Hierarchical Cell lets you choose which Hcells to display and how to display them.

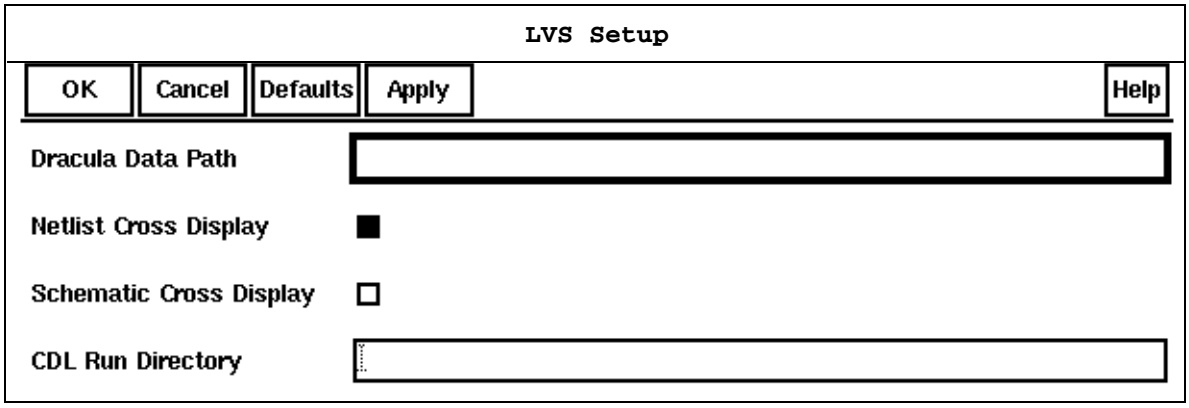
Get Reference Window displays the reference window.

Setup

LVS ⇛ **Setup...**

Lets you define the path to the Dracula directory containing the data you want to query. *Setup* also lets you select display options for the analysis.

LVS Setup Form



Dracula Data Path specifies the path to the directory containing Dracula data. You can enter a relative path or a full path.
Default: current directory

Netlist Cross Display allows corresponding nets or devices in the netlist window to be highlighted when you highlight nets or devices in the layout window. When you highlight nets or devices in the netlist window, the graphical user interface highlights the corresponding nets or devices in the layout window.

Note: The graphical user interface debugging tool supports cross-display of CDL, SPICE™, Verilog®, TEGAS®, and SILOS® netlists only.

Schematic Cross Display allows corresponding devices or nets in the Cadence Composer schematic window to be highlighted when you highlight nets or devices in the layout window. When you highlight nets or devices in the schematic window, the graphical user interface highlights corresponding nets or devices in the layout window.

CDL Run Directory specifies the run directory used in a previous CDLOUT translator run to generate a netlist from a Cadence schematic. This directory contains cross-reference information and other data needed to cross-probe between the graphical user interface and

the Cadence environment. Specify a CDL run directory only if you used CDLOUT to generate the netlist for LVS and you want to perform schematic cross-probing.

Note: You must specify the HTV command in LOGLVS before you process your schematic data with the CIR or NVER commands if you want `draculaInteractive` to perform cross probing between the layout and schematic, or vice-versa.

Using LVS Setup

You can change the setup options any time while you are running the graphical user interface.

1. Select *Setup* from the LVS menu.

   The Setup form appears.

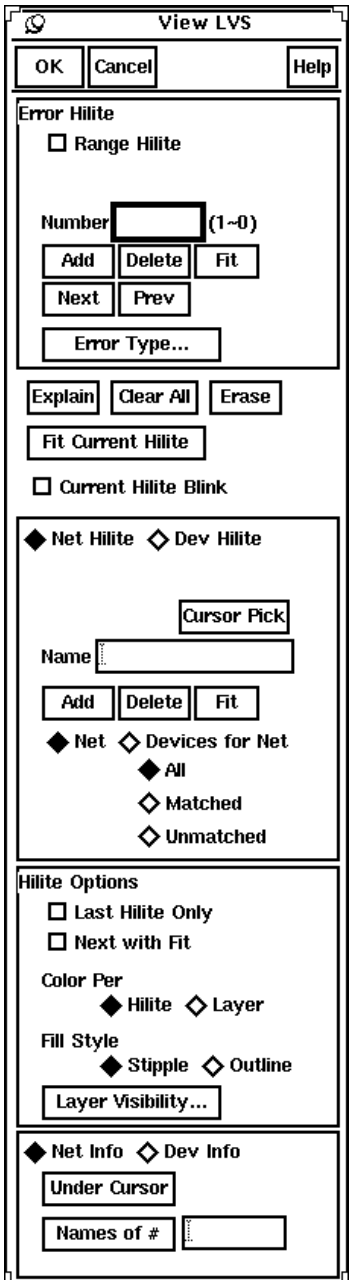
2. Type in the path to the Dracula directory.
3. To highlight a net or device in both the layout and netlist windows, click on *Netlist Cross Display*.
4. To highlight a net, device, or error in both the layout and schematic windows, click on *Schematic Cross Display*.
5. To cross-reference the schematic, type in the name of the CDL Run Directory, which was created when you ran the CDLOUT program to create the netlist.
6. Click on *OK*.

View LVS

LVS ⇛ **View LVS...**

Lets you set display options and get information about nets, devices, and errors. This command is also available on the LPE menu.

View LVS Form



Error highlighting section

Error, net, and device highlighting utility section

Net and device highlighting section

Highlighting options section

Net and device information section

Error Highlighting Section

Number lets you specify one or more errors to display. The valid range of numbers for your LVS job is listed to the left of the Number field. Separate the error numbers with a space. The graphical user interface highlights the selected errors.

If you selected the *Setup – Netlist Cross Display* option, the graphical user interface also highlights the errors in the netlist window. If you selected the *Setup – Schematic Cross Display* option, the graphical user interface also highlights the errors in the schematic window.

Add highlights the error specified with Number.

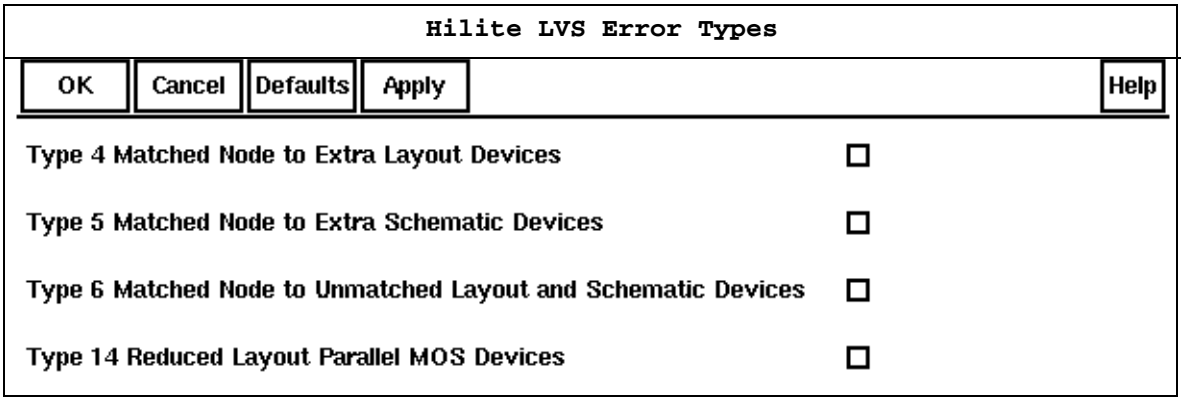
Delete unhighlights the error specified with Number.

Fit zooms in or out to encompass the last displayed error.

Next highlights the next error.

Prev highlights the previous error.

Error Type opens the Hilite LVS Error Types form in which you select one or more error types to highlight. For each type that contains an error, the graphical user interface lists a Dracula error type number and a brief description. Select the error types you want to display and click *OK*. To reset the Hilite LVS Error Types form to its default state, click *Defaults*.



The graphical user interface treats errors displayed by number and errors displayed by type separately. If you selected an error by number and that error is part of an error type also being displayed, when you clear the error type, the error selected by number is not affected.

Highlighting Utility Section

Options in this section apply to the error, net, or device highlighted with the error highlighting section or the net and device highlighting section.

Explain displays the message associated with a highlighted net or device.

To get information about a net or device, follow these steps.

1. Click *Explain* on the View LVS menu.

   The graphical user interface debugging tool prompts you to point to an item.

2. Click on the net or device you want.

   The graphical user interface displays a text message like this one in a viewfile window.

   ```
   LVS error type 2 (Device 3, MOSXP8G, hd2)
   ```

Clear All removes any highlighting from the view and the netlist.

Erase erases the highlighted item you click on. The item is not removed from the database.

Fit Current Hilite zooms in or out to encompass the last highlighted item.

☐ **Current Hilite Blink** makes the last item you highlighted blink.

Net and Device Highlighting Section

This section lets you highlight nets and devices in the layout window. If you display the device or net in a netlist window, and you selected the *Setup – Netlist Cross Display* option, the graphical user interface also highlights the nets or devices in the netlist window. If you selected the *Setup – Schematic Cross Display* option, the graphical user interface also highlights the nets or devices in the schematic window.

◈ **Net Hilite** customizes the net and device highlighting section for net highlighting.

◇ **Dev Hilite** customizes the net and device highlighting section for device highlighting.

Cursor Pick lets you point to the net or device you want to highlight. If there are multiple items under the cursor, a list box appears. When you choose an item, the graphical user interface displays the item name in the Name field.

Name specifies the schematic name of the net or device you want to highlight. You can also type a layout net number preceded with a question mark (?o for original nets or ?p for subnets) or a layout device number preceded with ?dev.

Add highlights the net or device you specified with Name, *Cursor Pick*, or *All*/*Matched*/*Unmatched*.

Delete unhighlights the net or device you specified with Name, *Cursor Pick*, or *All*/*Matched*/*Unmatched*.

Fit zooms in or out to encompass the last highlighted item.

◈ **Net** turns on net highlighting when *Net Hilite* is selected. When *Dev Hilite* is selected, this button becomes **Device**, and turns on device highlighting.

◇ **Devices for Net** turns on device highlighting for a selected net. When *Dev Hilite* is selected, this button becomes **Nets for Device**, and turns on net highlighting for a selected device. Select *All*, *Matched*, or *Unmatched*.

- ◈ **All** highlights all the devices associated with a net if *Devices for Net* is selected. It highlights all the nets associated with a device if *Nets for Device* is selected.
- ◇ **Matched** highlights matched devices associated with a net if *Devices for Net* is selected. It highlights matched nets associated with a device if *Nets for Device* is selected.
- ◇ **Unmatched** highlights unmatched devices associated with a net if *Devices for Net* is selected. It highlights unmatched nets associated with a device if *Nets for Device* is selected.

Highlighting Options Section

☐ **Last Hilite Only** displays or makes visible only the current item (last net, device, or LVS error highlighted). Previously highlighted items are unhighlighted.

☐ **Next with Fit** highlights the item you have just selected and adjusts the view to encompass all error shapes.

Color Per determines whether the layers are displayed in different colors.

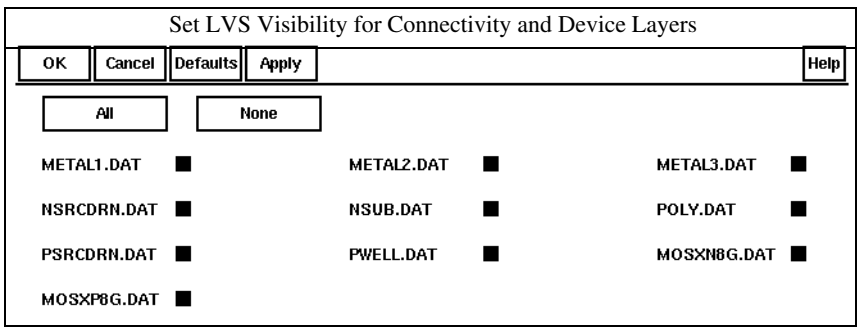
- ◈ **Hilite** highlights all of a net or device on all layers in a single color. The graphical user interface always uses the same color to highlight the current device or net. When you highlight a new net or device, the previously highlighted net or device changes to the next available color. The color in the netlist window matches the highlighted net in the layout or schematic windows when you use this option.
- ◇ **Layer** highlights each layer of a net or device in a different color. The graphical user interface outlines the current net or device with a white line. Note that there is no relationship between the color in the netlist and layout windows when you use this option. The colors in the schematic and netlist windows still match.

Fill Style determines how the error shapes are to be displayed.

- ◈ **Stipple** displays the highlighted object with a solid fill and no outline.
- ◇ **Outline** displays the highlighted object with an unfilled outline.

Note: The graphical user interface uses the color, fill-style, and stipple pattern of the layer purpose pairs *hilite-drawing1* through *hilite-drawing9* and *y1-drawing* through *y9-drawing* to highlight the first 18 nets or devices. If there are more than 18 nets or devices to be highlighted, the graphical user interface uses the layer-purpose pairs cyclically, starting from *hilite-drawing1* again. The graphical user interface reserves *hilite-drawing* for the last highlighted net or device and *y0-drawing* for the nets or devices in the netlist that are not found in the layout.

Layer Visibility lists all the layers that are part of a net or device on the Set LVS Visibility for Connectivity and Device Layers form. You can turn the display on or off for any of these layers. If you turn off a layer for a net, you affect only its visibility. The graphical user interface still recognizes the connectivity through that layer.



Use the Set LVS Visibility for Connectivity and Devices Layers form to set layer visibility as follows:

- To see selected layers, click on the layer names to turn them off or on.

  You can use this option to improve visibility by turning off global layers, such as *PWELL.DAT,* that obscure the highlighting of other layers.

- To see all layers, click *All*. To turn off all layers, click *None*.
- To reset layer visibility to the default, click *Defaults*.
- Click on *OK*.

When you run Dracula LVS, you must save all layers forming nets and devices using the Dracula *KEEPDATA* command. *KEEPDATA=YES* is the Dracula default, which saves the device and interconnect layers for the graphical user interface.

The visibility defaults can be set by defining the *iqSetVisibilityDefaults* variable in the *.cdsinit* as follows:

```
iqSetVisibilityDefaults = '( ( "METAL.DAT" t )
          ( "POLY.DAT" t )
          ( "NSD.DAT" nil ) )
```

Layers not listed in the *iqSetVisibilityDefaults* variable are turned on. If you do not define the variable in *.cdsinit*, all layers are turned on. After LVS Setup, you can change the value of *iqSetVisibilityDefaults* by typing *iqSetVisibility* in the CIW. For example

```
iqSetVisibility('(("CMET.DAT" t) ("POLY.DAT" nil)))
```

Net and Device Information Section

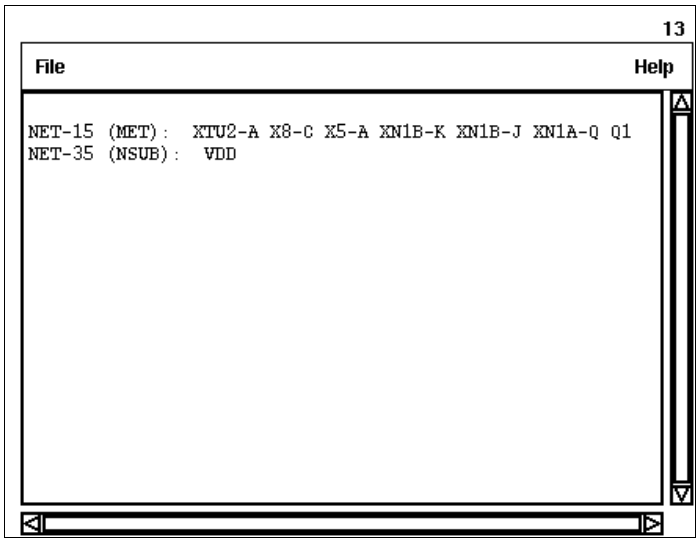
This section displays the number, layer, and name of the net or device you select.

◈ **Net Info** displays the number, layer, and name of the net you select with *Under Cursor* or *Names of #*.

◇ **Dev Info** displays the number, layer, and name of the device you select with *Under Cursor* or *Names of #*.

Under Cursor selects the net or device you point to with the cursor. When you click on *Under Cursor*, the graphical user interface prompts you to point to the net or device you want.

The graphical user interface displays a view window with the number, layer, and names of the specified nets or devices. If more than one net or device appears under the cursor, the graphical user interface displays the information for all the nets or devices under the cursor.



Names of # selects nets or devices by number. Click on *Names of #* and type in the number.

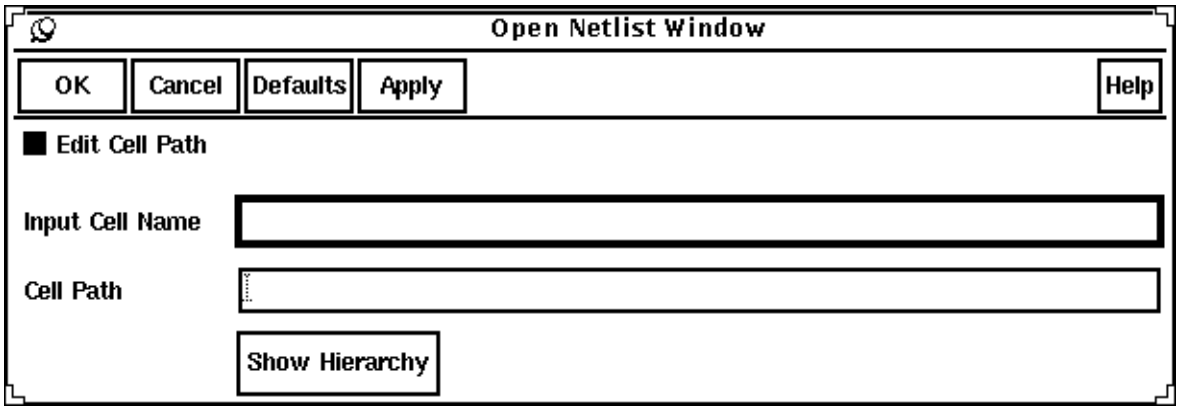
The graphical user interface displays a view window with the number, layer, and names of the specified nets or devices.

Open Netlist Window

LVS ⇛ **Open Netlist Window...**

Opens a new window containing the circuit netlist. This command is also available on the LPE menu.

Open Netlist Window Form



Input Cell Name is the name of the cell whose netlist you want to see.

Edit Cell Path allows users to edit the cell path of the input cell name to be viewed. If the **Edit Cell Path** checkbox is disabled, the default path is used to search for the specified cell.

Show Hierarchy displays a browser window that shows the cell hierarchy. For more information about how to about how to display the cell hierarchy, see "Show Network Hierarchy" on page 82.

Using Open Netlist Window

To open the netlist window, follow these steps.

1. Select *Open Netlist Window* from the LVS menu.
2. Select the netlist you want to view using one of the following methods:
   - ❑ To enter the name manually, type in the name of the cell netlist you want to view.

- To see the cells in the hierarchy, click on *Show Hierarchy*.
  (For more information about how to display the cell hierarchy, see the "Show Network Hierarchy" on page 82.)

The graphical user interface opens a browser window or uses an existing browser window to display the hierarchy of the circuit you selected.

3. To find the cell you want, expand the hierarchy.
4. To select the cell name, choose *Select* from the pop-up menu.

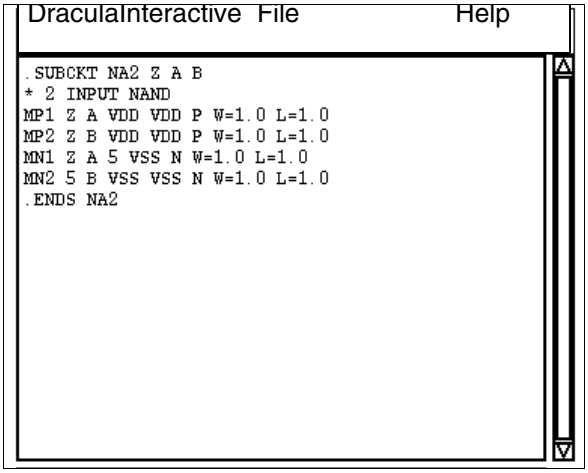
   The graphical user interface enters the cell name into the *Input Cell Name* field.
5. Click on *OK*.

   The graphical user interface opens a view window and displays the netlist for the selected cell.

   If you selected the *Netlist Cross Display* option of the *Setup* command and you highlight a net or device in the layout window, the graphical user interface highlights the corresponding net or device (if any) in the netlist window.

Note: You can also open the netlist directly from the browser window with the *Open Netlist* command.

Example



Show Network Hierarchy

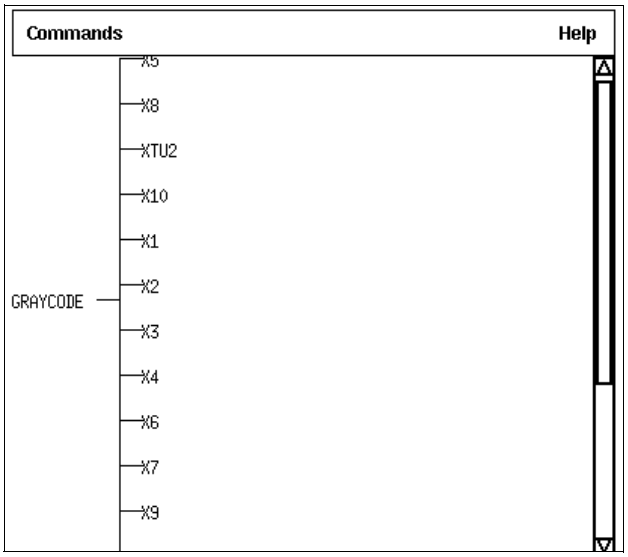
LVS ⇛ **Show Network Hierarchy...**

Opens a browser window that displays the hierarchy of the circuit netlist so you can select the cell name for the netlist you want. The browser hierarchy contains only references to cells, not references to nets and devices. This command is also available on the LPE menu.

Prerequisites

You need a Dracula netlist and a hierarchy.

Show Network Hierarchy Browser Window



Using Show Network Hierarchy

To display the network hierarchy, follow these steps.

1. Select *Show Network Hierarchy* from the LVS menu.

   The graphical user interface opens a browser window or uses an existing browser window to display the hierarchy of the circuit in the current window.

2. To find the cell you want, expand the hierarchy so you can see all of the cell names.

- Click on the circuit name with the left mouse button and hold to pop up the function menu.
- Choose the *Expand Cell* command and release.

3. Display a netlist for a subcircuit using either of the following methods:

- To display a netlist in a new window, click on the cell name and choose *Open Netlist* from the pop-up menu.

The graphical user interface opens a view window containing the netlist for the circuit you chose.

- To enter a cell name in an Open Netlist Window form, click on the cell name and choose *Select* from the pop-up menu.

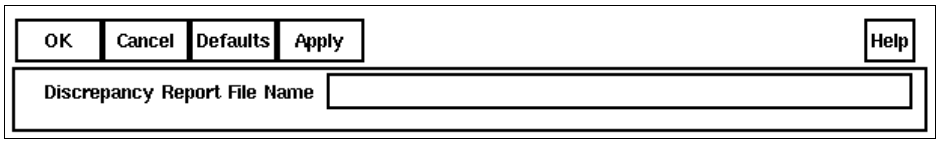
4. To exit from the hierarchy browser window, select *Close Browser* from the Commands menu in the window.

Show Discrepancy Report

LVS ⇛ **Show Discrepancy Report...**

Opens a new window displaying the LVS discrepancy report. This command is also available on the LPE menu.

Show Discrepancy Report Form



Discrepancy Report File Name is the LVS discrepancy report you want to display.

Example

```
********** LVSNET SUMMARY REPORT **********




                    DATE : 13-Dec-88
                    TIME : 12:04:32



                    ***** LVS CHECK *****

                    PRINTLINE      =     1000
                    WPERCENT(MOS)  =    0.000 %
                    LPERCENT(MOS)  =    1.000 %
      ***************************************************
      *********   CORRESPONDENCE NODE  PAIRS   **********
      ***************************************************



        SCHEMATICS              LAYOUT          PAD TYPE


  VDD                 1   VDD                35   P
  VSS                 2   VSS                 8   G
  CA                  8   CA                  2   I
  CO                  7   CO                  1   I
  MC1                 9   MC1                 3   I
  Q1B                 3   Q1B                 4   O
  Q2B                 4   Q2B                 6   I
  Q3B                 5   Q3B                48   I
  Q4B                 6   Q4B                47   I
  SCT1               10   SCT1                5   I
  SCT2               11   SCT2                7   I
  SCT3               12   SCT3               66   I
  SCT4               13   SCT4               65   I

NUMBER OF LAY. PADS READ =       13   DISCARDED =        0
NUMBER OF SCH. PADS READ =       13   DISCARDED =        0


NUMBER OF VALID CORRESPONDENCE NODE PAIRS =       11

      ***************************************************
      **********    LVS DEVICE MATCH SUMMARY   **********
      ***************************************************



  NUMBER OF UN-MATCHED SCHEMATICS DEVICES      =       0
  NUMBER OF UN-MATCHED LAYOUT     DEVICES      =       0
  NUMBER OF    MATCHED SCHEMATICS DEVICES      =      81
```

Node File

LVS	⇛	Node File...

Creates and edits the `TARGET.DAT` file or other text files containing lines of node names. This feature can be used to generate the list of target nets used by the LVS View Parasitic form.

See the section on Node File on page 116 for more information.

Display Text Labels

LVS ⇛ **Text File Handling** ⇛ **Display Text Labels...**

Displays text labels associated with the view.

For details, see the “Display Text Labels” section in the “DRC Commands” chapter.

Clear Text Labels

LVS ⇛ **Text File Handling** ⇛ **Clear Text Labels...**

Removes the text labels from the current window.

Hierarchical Cell

LVS ⇛ **Hierarchical Cell...**

Lets you control which Hcells to display in the graphical user interface design window.

For details, see the “Hierarchical Cell” section in the “DRC Commands” chapter.

Get Reference Window

LVS ⇛ **Get Reference Window**

Displays the Reference window.

Get Dracula Layer Window

LVS ⇛ **Get Dracula Layer Window**

Displays the DLW (Dracula Layer Window).

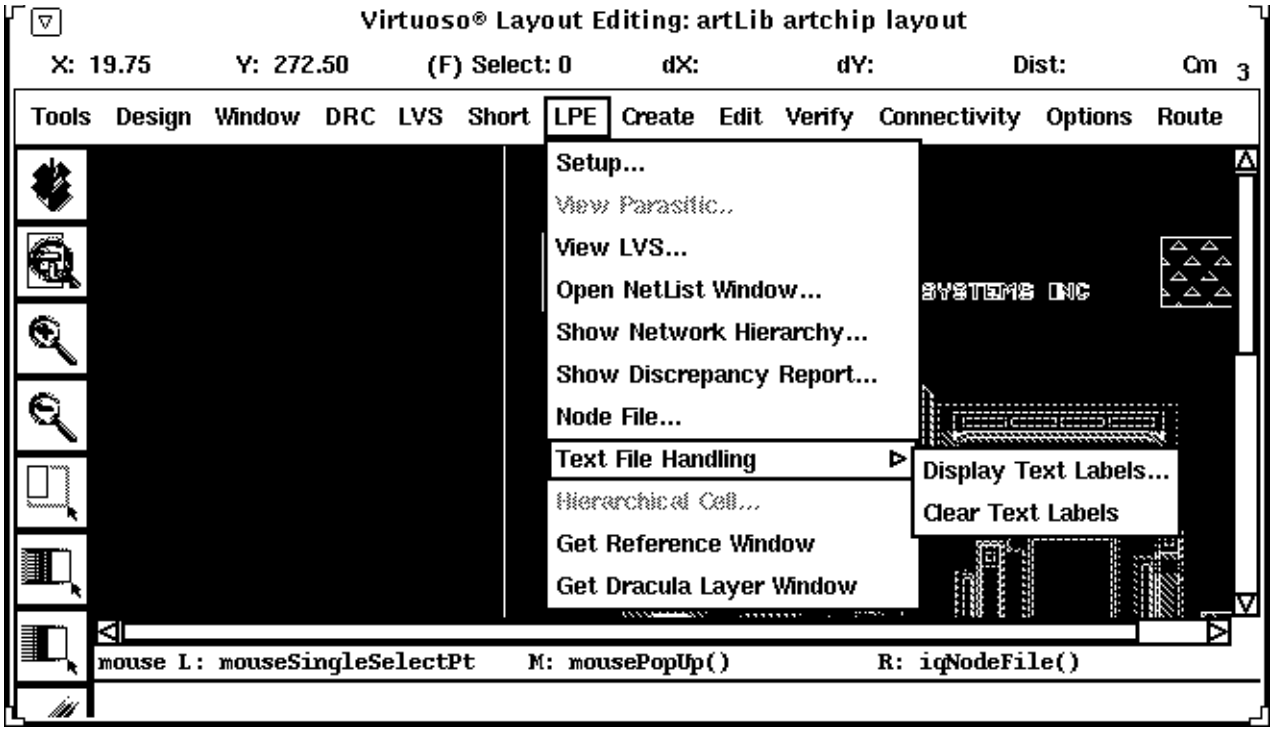
4

LPE commands

This chapter discusses the LPE Menu.

LPE Menu

The LPE menu lets you display resistance and capacitance parasitics on nets. You must have parasitic data from a Dracula LPE/PRE run to display these nets. You can also display layout versus schematic errors.



Setup defines a Dracula directory and sets parameters.

View Parasitic lets you specify which parasitic resistance and capacitance elements to view.

View LVS lets you set display options and get information about nets, devices, and errors.

Open Netlist Window opens a new window containing netlist information.

Show Network Hierarchy displays the circuit hierarchy.

Show Discrepancy Report displays the LVS discrepancy report.

Text File Handling lets you display text layers.

Hierarchical Cell lets you choose which Hcells to display and how to display them.

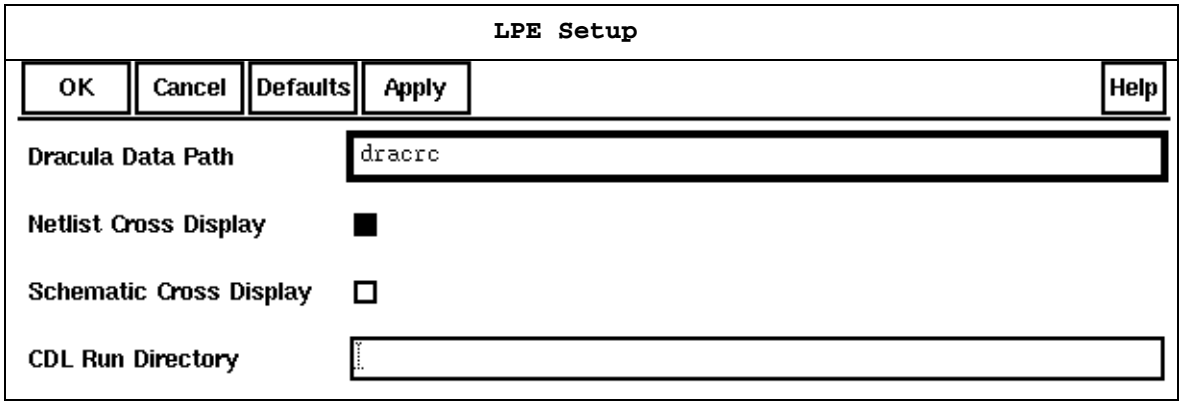
Get Reference Window displays the reference window.

Setup

LPE ⇛ **Setup...**

Lets you define the path to the Dracula directory containing the data you want to query. *Setup* also lets you select display options for the analysis.

LPE Setup Form



Dracula Data Path specifies the path to the directory containing Dracula data. You can enter a relative path or a full path.
Default: current directory

Netlist Cross Display allows corresponding nets or devices in the netlist window to be highlighted when you highlight nets or devices in the layout window. When you highlight nets or devices in the netlist window, the Dracula graphical user interface highlights the corresponding nets or devices in the layout window.

Note: The graphical user interface supports cross-display of CDL, SPICE™, Verilog®, TEGAS®, and SILOS® netlists only.

Schematic Cross Display allows corresponding devices or nets in the Cadence Composer schematic window to be highlighted when you highlight nets or devices in the layout window. When you highlight nets or devices in the schematic window, the graphical user interface highlights corresponding nets or devices in the layout window.

CDL Run Directory specifies the run directory used in a previous CDLOUT translator run to generate a netlist from a Cadence schematic. This directory contains cross-reference information and other data needed to cross-probe between the graphical user interface and

the Cadence environment. Specify a CDL run directory only if you used CDLOUT to generate the netlist for LVS and you want to perform schematic cross-probing.

Note: You must use the CASE command in LOGLVS if you want cross-probing. This command turns on case sensitivity, to preserve the case of the original text. Names in the files generated by LOGLVS, such as the net/device, are the same as upper/lower case and the net/device names in the input netlist.

Using LPE Setup

You can change the setup options any time while you are running the Dracula graphical user interface.

1. Select *Setup* from the LPE menu.

   The Setup form appears.

2. Type in the path to the Dracula directory.
3. To highlight a net or device in both the layout and netlist windows, click on *Netlist Cross Display*.
4. To highlight a net, device, or error in both the layout and schematic windows, click on *Schematic Cross Display*.
5. To cross-reference the schematic, type in the name of the CDL Run Directory, which was created when you ran the CDLOUT program to create the netlist.
6. Click on *OK*.

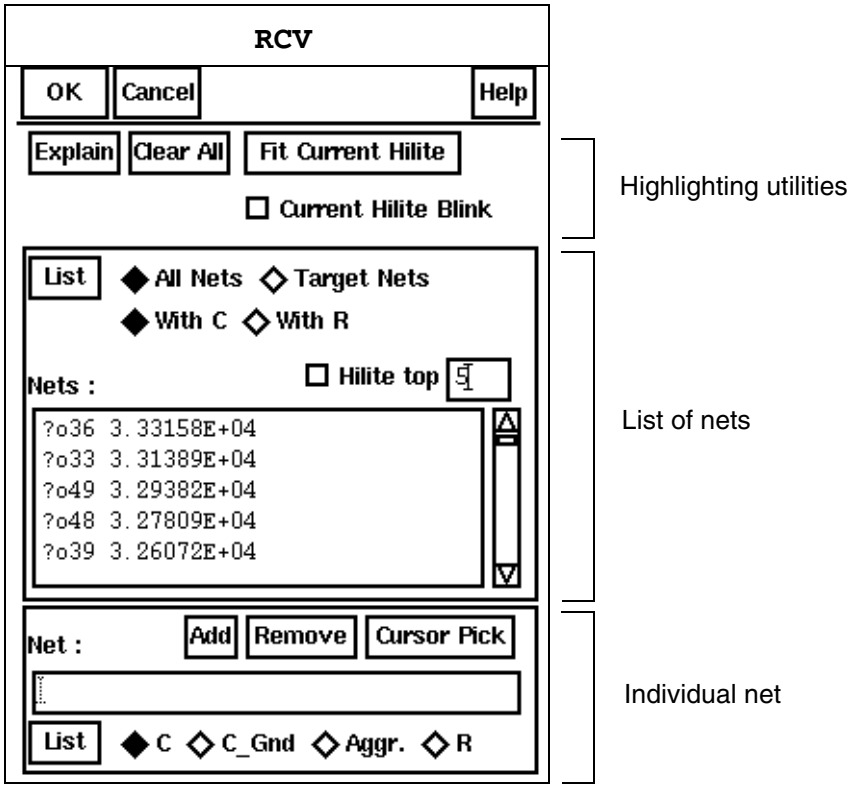
View Parasitic

LPE ⇛ **View Parasitic...**

Lets you select parasitic components to display.

RCV Form

The Resistance and Capacitance Viewing window (RCV) lets you access the parasitic resistance and capacitance display capabilities in the layout window.



Highlighting Utilities

Explain lets you click on a net in the layout window and display information about that net in a text window.

Clear All removes any highlighting from the view and the netlist.

Fit Current Hilite zooms the graphical user interface design window in or out to display the currently highlighted item.

☐ **Current Hilite Blink** makes the last item you highlighted blink.

List of Nets

The following options let you select the nets with parasitic elements to display in the Nets field:

◈ **All Nets** lets you display all nets with either parasitic capacitance (*With C* selected) or parasitic resistance (*With R* selected).

◇ **Target Nets** lists the nets in the *TARGET.DAT* file. You can create this text file to display resistance and capacitance. You can display parasitic capacitance (*With C* selected), parasitic resistance (*With R* selected), or the original *TARGET.DAT* file (*Original* selected).

◈ **With C** displays nets with parasitic capacitance, sorted by value from highest to lowest.

◇ **With R** displays nets with parasitic resistance, sorted by value from highest to lowest.

◇ **Original** (for Target Nets only) displays the *TARGET.DAT* file, unsorted.

List lists the nets you selected with *All Nets*, *Target Nets*, *With C*, *With R*, or *Original* in the Nets field.

☐ **Hilite top** highlights the number of nets you specify, up to a maximum of 9, that have the highest parasitic resistance or capacitance value. For example, if you set *Hilite top* to 5 and select *With C*, the five nets with the highest capacitance values are highlighted and listed in the Nets list box.

Nets lists the nets being displayed.

Individual Net

Net is the original net name of the net you want to display. You can select a net name from the list of nets in the Nets field, type in an original net number starting with ?o or a subnet number starting with ?p, or use *Cursor Pick* to select a net.

Add highlights the net with the name you specify in the Net field.

Remove unhighlights the net with the name you specify in the Net field.

Cursor Pick lets you point to the net you want to highlight. If there are multiple nets under the cursor, a list box appears. When you choose a net, the graphical user interface displays the item name in the Net field.

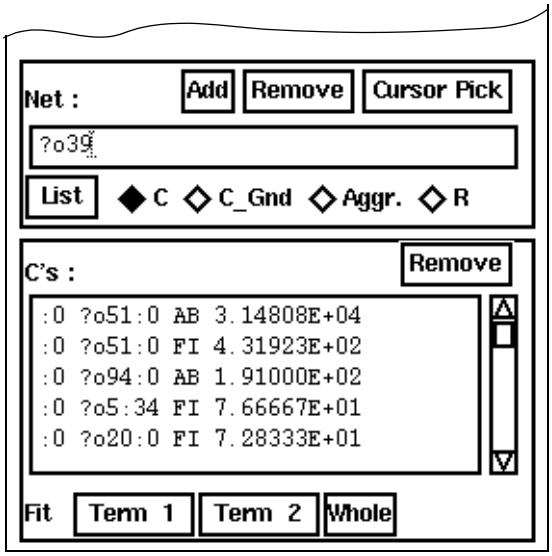
List options select the components of the parasitic shapes to display for the net listed in the Net field. When you select an option and click on *List*, the RCV form is extended with another list box, which lists the components.

Note: If you included the *LUMPCAP=YES* or *REDUCER=YES* functions in your rules file when you ran your extraction job, you can list only the C_LUMP and R parasitic components.

If you did not use either of these functions, the components you can list are C, C_GND, Aggr., and R.

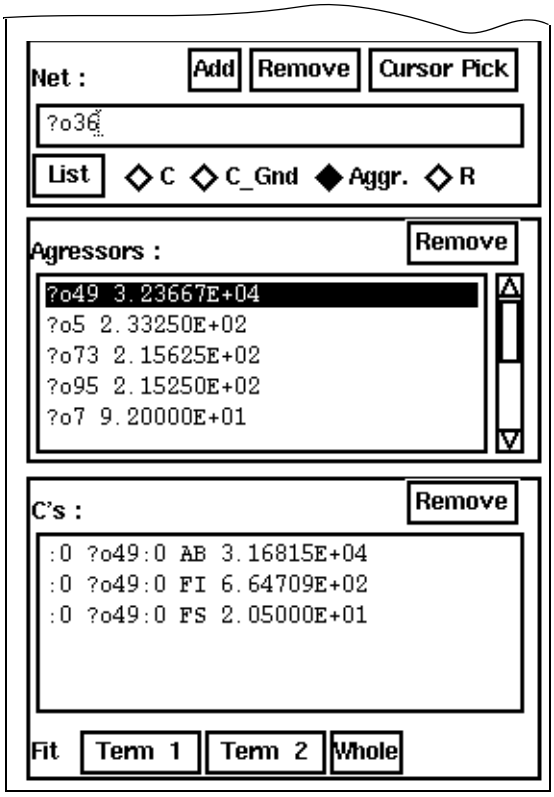
◈ **C** lists all the parasitic capacitance components of the net (*PARASITIC CAP*, *FRINGE CAP*) in the format for capacitance components.

◇ **C_Gnd** lists all capacitance from the net to ground in the format for capacitance components.



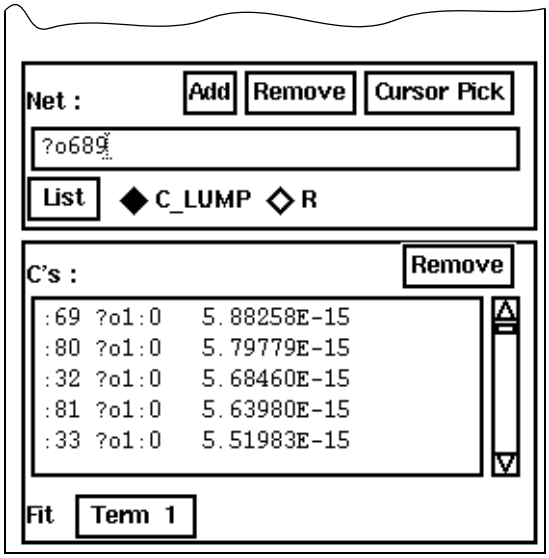
List box for C and C_Gnd

◇ **Aggr.** lists all aggressor nets, which are nets with cross-coupled capacitance to the net, in the format for original net capacitance. Aggressor nets are listed in a separate list box. You can click on an aggressor net in this list box to get a list of capacitance components between the net and the aggressor net, which are listed in the format for capacitance components.



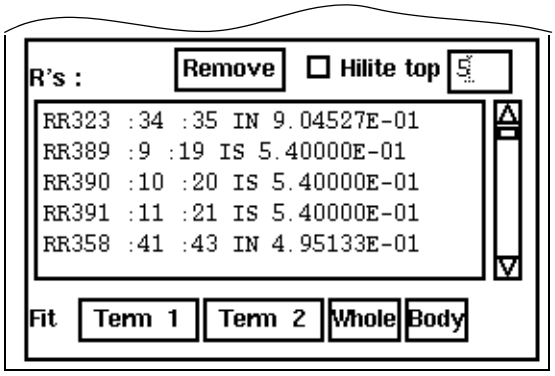
List boxes for Aggr.

◇ **C_LUMP** lists all capacitance from the net to ground in the format for capacitance components.



List box for C_lump

◇ **R** lists all the parasitic resistance components of the net in the format or resistance components.



List box for R

Remove unhighlights the parasitic component in the layout.

Fit options control how to display parasitic data in the layout window.

Term 1 fits the first terminal of the current capacitor or resistor in the layout window. For capacitors, this terminal is on the “subject” net.

Term 2 fits the second terminal of the current capacitor or resistor in the layout window. For capacitors, this terminal is on the “aggressor” net.

Whole fits all the highlighted parasitic data in the layout window.

Body fits the resistor body in the layout window.

Displaying Parasitic Resistance and Capacitance

To view parasitic resistance and capacitance, do the following:

1. From the Dracula graphical user interface window, select *LPE – Setup*.
2. Type the Dracula Data Path.
3. Click on *OK*.

The graphical user interface displays the RCV form.

How Information Is Listed

The RCV form lists parasitic resistance and capacitance for nets in different formats, depending on what type of parasitic RC you are viewing.

Original Net Capacitance

Each entry has the following format:

```
original_net_name cap_value
```

original_net_name	Original net name, beginning with ?o.
cap_value	Capacitance value of the original net.

For example

?o49 3.29382E+04

Capacitance Components

Each entry has the following format:

```
subject_subnet original_net:subnet cap_type cap_value
```

subject_subnet	The subnet number of the net listed in the Net field.

original_net	Original net name of the aggressor net, beginning with ?o.
subnet	The subnet number of the aggressor net involved in the capacitance.
cap_type	Capacitance type specified with PARASITIC CAP[*type*] or FRINGE CAP[*type*].
cap_value	Value of the capacitance between the subnets.

For example

```
:0 ?o49:0 AB 3.16815E+04
```

Original Net Resistance

Each entry has the following format:

```
original_net_name res_value
```

original_net_name	Original net name.
res_value	Resistance value of the original net.

For example

```
?o23 1.10643E+01
```

Resistance Components

Each entry has the following format:

```
resistor :subnet1 :subnet2 res_type res_value
```

resistor	Name of the resistor device.
subnet1	Subnet of the original net you selected in the left column, representing one end of the resistor.
subnet2	Subnet of the other end of the resistor.
res_type	Resistance type specified with PARASITIC RES[*type*].
res_value	Resistance value between the two subnets.

For example

RR323 :34 :35 IN 9.04527E-01

Highlighting Capacitance Parasitics

To highlight capacitance parasitics on nets in the layout window, follow these steps.

1. Specify which nets you want listed by doing one of the following:
   - ❑ To list all nets with parasitic components, select *All Nets*.
   - ❑ To list the nets you specified in the *TARGET.DAT* file, select *Target Nets*.
2. Select *With C*.
3. To list nets with the highest capacitance, select *Hilite top* and type in a number of nets to highlight, from 1 to 9.
4. Click *List*.

   The nets are listed in the Nets list box, in the format for original net capacitance.
5. Highlight the capacitance you want to display by doing one of the following:
   - ❑ Type an original net name in the Net field.
   - ❑ Click on a net name in the Nets list box.
     The net name appears in the Net field.

   The net is highlighted in the layout window.

Listing and Highlighting Capacitance Components

To list and highlight capacitance components of nets in the layout window, follow these steps.

1. To list all parasitic capacitance components of the net you specified, select *C*.

   The nets are listed in the C's list box, in the format for capacitance components.
2. To list all capacitances to ground for the net you specified, select *C_LUMP* if you used *LUMPCAP=YES* or *REDUCER=YES*, or *C_GND* if you did not.

   The nets are listed in the C's list box in the same format as the *C* option. The value is the capacitance between the net and ground. For example

```
:0 ?o23:33 AC 1.58750E+01
```

3. To list all aggressor nets that have coupling capacitance to the net you specified, select *Aggr*.

   The nets are listed in the Aggressors list box, in the format for original net capacitance. The value is the capacitance between the aggressor net and the subject net. For example

   ```
   ?o49 3.23667E+04
   ```

4. To highlight a *C* or *C_Gnd* component, click on the component in the C's list box.

   The component is highlighted in the layout window. You can use *Explain* to show the net name and list the capacitance extraction rule that generated this component.

   You can unhighlight the component by clicking *Remove*.

5. To highlight an *Aggr* component, click on the component in the Aggressors list box. The outline of the aggressor net is highlighted.

   Note: When you click on an aggressor net, the C's list box lists the capacitance components. Click on the component in the C's list box to highlight an aggressor component. You can use *Explain* to show the net name and list the capacitance extraction rule that generated this component.

Highlighting Resistance Parasitics

To highlight resistance nets in the layout window, follow these steps.

1. Specify which nets to list by doing one of the following:

   - To list all nets with parasitic components, select *All Nets*.
   - To list the nets you specified in the *TARGET.DAT* file, select *Target Nets*.

2. Select *With R*.
3. To list nets with the highest resistance, select *Hilite top* and type in a number of nets to highlight, from 1 to 9.
4. Click *List*.

   The nets are listed in the Nets list box, in the format for original net resistance.

5. Highlight the resistance you want to display by doing one of the following:

   - Type an original net name in the Net field.

- Click on a net name in the Nets list box.
  The net name appears in the Net field.

The net is highlighted in the layout window.

6. To list all parasitic resistance for the net you specified, select *R* and click *List*.

   The nets are listed in the *R's* list box, in the format for resistance components.

7. To highlight a resistor, click on an entry in the *R's* list box. The resistor is highlighted in the layout window.

   You can use *Explain* to show the net name and list the resistance extraction rule that generated this component. You can use *Term 1*, *Term 2*, *Body*, and *Whole* to look at the body, terminals, or complete resistor.

View LVS

LPE ⇛ **View LVS...**

Lets you set display options and get information about nets, devices, and errors.

For details, see the "View LVS" section in the LVS Commands" chapter.

Open Netlist Window

LPE ⇛ **Open Netlist Window...**

Opens a new window that contains the circuit netlist.

For details, see the "Open Netlist Window" section in the "LVS Commands" chapter.

Show Network Hierarchy

LPE	⇛	Show Network Hierarchy...

Opens a browser that displays the hierarchy of the circuit netlist so you can select the cell name for the netlist you want. The browser hierarchy contains only references to cells, not references to nets and devices.

For details, see the "Show Network Hierarchy" section in the "LVS Commands" chapter.

Show Discrepancy Report

LPE ⇛ **Show Discrepancy Report...**

Opens a new window displaying the LVS discrepancy report.

For details, see the "Show Discrepancy Report" section in the "LVS Commands" chapter.

Display Text Labels

LPE ⇛ **Text File Handling** ⇛ **Display Text Labels...**

Displays text labels associated with the view.

For details, see the "Display Text Labels" section in the "DRC Commands" chapter.

Clear Text Labels

LPE ⇛ **Text File Handling** ⇛ **Clear Text Labels...**

Removes the text labels from the current window.

Hierarchical Cell

LPE ⇛ **Hierarchical Cell...**

Lets you control which Hcells to display in the Dracula graphical user interface design window.

For details, see the "Hierarchical Cell" section in the "DRC Commands" chapter.

Get Reference Window

LPE ⇛ **Get Reference Window**

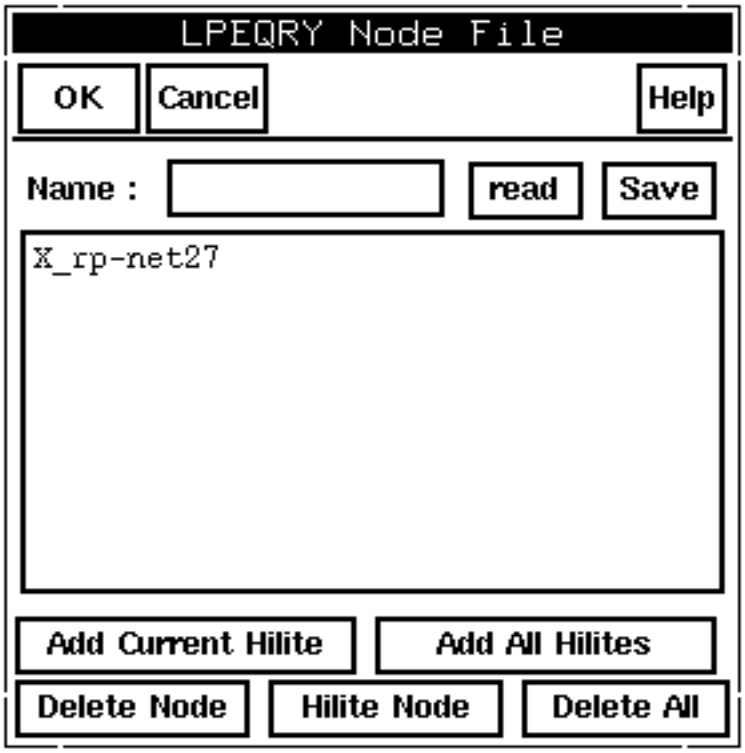
Displays the Reference window.

Node File

LPE	⇛	Node File

Creates and edits the `TARGET.DAT` file or other text files containing lines of node names. This feature can be used to generate the list of target nets used by the LPE View Parasitic form.

To bring up the Node File form, choose *Node File* in the LVS or LPE pull-down menu.



Name contains the filename of the node file. If / exists in the filename, it represents a regular Unix filename with a path. If there is no / in the filename, it specifies a file which resides in the Dracula Data Path, which has been entered during LVS/LPE setup.

The name of the file used for the LPE function is *TARGET.DAT.* Node names will appear in the list box in layout name format.

Read reads the content of the node file typed in the *Name* field into the list box. If there are nodes already in the list box when you click on *Read*, the nodes in the file which were not originally in the list box are added.

Save saves the contents of the list box to the node file.

Add Current Hilite adds the node name of the current net hilite to the list box. This can be used when the hilite is done before bringing up the Node File form.

Add All Hilites adds the node names of all net hilites to the list box. This can be used when the hilites are done before bringing up the Node File form.

Delete Node deletes a node selected it in the list box.

Hilite Node hilites a selected node it in the list box.

Delete All deletes all nodes from the list box.

5

Short Locator

This chapter will focus on the use of the Short Locator and how it works, as well as the Short Locator form for the Dracula graphical user interface.

Locating Shorts

The Short locator lets you locate shorts in a layout following an Extract run. If you want to spread the work over several sessions, you can save and restore partial runs. To help you identify the short location, you can add labels or zoom in.

Prerequisites

Before you can locate a short, you must do the following:

- Identify the net in which the short exists. The short location program handles only one shorted net at a time, although that net might be the result of many intended nets shorted together.
- Make sure all *connect* layers are available. Exception: If a layer is attached to a net at only one point, as in the case of a *tub* or *well*, the absence of the layer does not affect the connectivity of the rest of the net. In this case, all shorts can still be isolated.

How Short Location Works

A shorted net results from a mistake in labeling nets or from two nets placed so close together that they touch a misplaced contact. The Short Locator analyzes the shorted net using all labels, interconnect polygons, and contacts associated with the net. In its simplest mode, the

Short Locator attempts to isolate each short to a single polygon. It assumes that the section of the shorted net under the label is part of the net defined by that label. Using the labels in the graphics data, the Short Locator determines which parts of the shorted net belong to one net, and which belong to another. Any part of the net that appears to belong to more than one label is identified as a short.

If these labels do not isolate the short to a small enough section of the net, you can add more labels or delete misleading labels. When you do this, you do not change the original graphics data. You change only the Short Locator database.

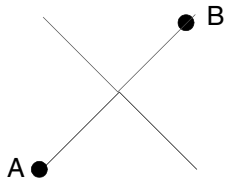
Placing Labels

To place a label properly, you must know which part of the shorted net is really part of the intended nets. Pads are obvious places to put labels. Also, if you know the circuit well, you can label the part of a net with long runs.

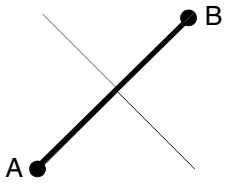
You must also consider how much to separate labels. The farther apart the labels, the more information they supply. This separation is not a measurement of physical distance across a circuit, but of logical distance along the net. You can place labels physically next to each other providing they are at opposite ends of the shorted net.

You do not have to place labels on the part of the net that contains the short. You can place them anywhere on the net. In most cases, it is better to place the labels on a part of the net that does not contain the short.

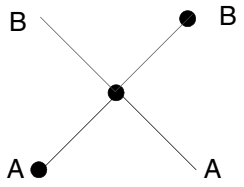
Consider the following example:



The labels A and B on the two shorted nets let the Short Locator isolate the short to the path between A and B as follows.



Additional labels on the path help only if they are very close to the short. If instead you put more labels on the part of the net that does not contain the short, the isolation becomes precise.



Defining Active Layers

Active layers are the layers that you can label. You can define which of the interconnect layers are active. The differences between active and inactive layers are as follows:

- You can add labels to active layers. You cannot add labels to inactive layers. It is best to set global layers such as *tub* and *substrate* to be inactive so you can add labels to shapes on layers over them without chance of confusion.
- Active and inactive layers are displayed in different colors and fill patterns. If you set global layers to be inactive, you can choose to display the layers with more muted colors than other interconnect layers. Setting colors this way lets you see interconnect layers that would otherwise be hidden by *tub* and *substrate*.

Displaying Information

Information about the shorted net appears on top of any existing display in all windows containing the *layout* view of the cell you are processing.

The Short Locator uses the same fill colors and fill patterns as the predefined system layers. You can change the color and fill patterns by changing the system layers using the Layer menu in the CIW.

The five layers and their uses are as follows:

hilite drawing

Displays the text labels.

hilite drawing1

For active layers, displays the part of the net that contains the short.

hilite drawing2

For active layers, displays the intended net. All shapes forming the net, regardless of the layer from which they came, are displayed in the same color.

hilite drawing3

For inactive layers, displays the part of the net that contains the short.

hilite drawing4

For inactive layers, displays the original net. All shapes forming the net, regardless of the layer from which they came, are displayed in the same color.

Usually, you set your display as follows:

- Turn off the inactive layers or set them to more muted colors and lighter fill patterns than the active layers.
- Set the shorted part of the net to a higher intensity color and a more solid fill pattern than the full net.
- Set the text label color to show above everything else.

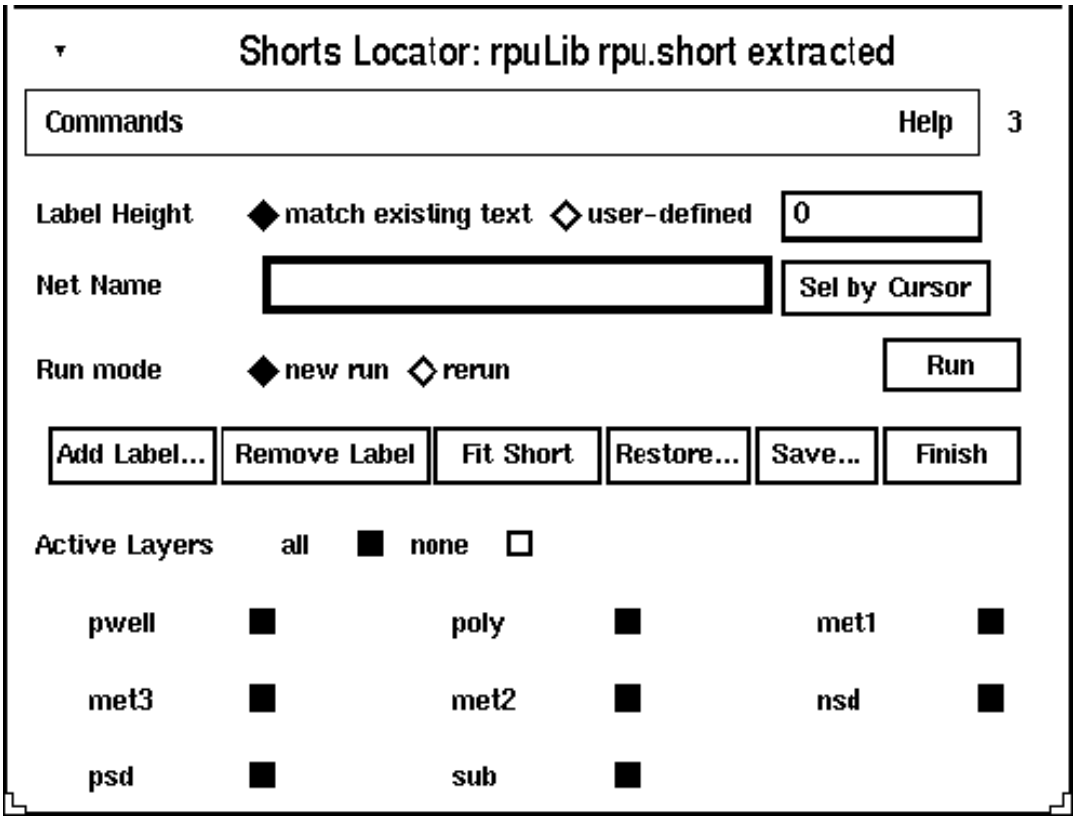
Saving and Restoring a Short Locator Run

Because the isolation of a short is an iterative process, you might need to stop before you are satisfied with the result and complete your work later. For this purpose, you can save and restore a run.

When you save the run, you define the file name and location. When you restore a run, you can open any file name you have saved. The contents of the file are in text format. Although you can edit the file using a conventional text editor, the integrity of the restored run cannot be guaranteed once you edit the file.

The saved information does not contain the entire status of the run, but includes enough data to return to that status with one iteration of the Short Locator. That iteration is performed automatically when you restore the run.

Short Locator Form



Label Height defines the height of the characters for the label text you enter. You can either accept the default height or type a height.

Net Name specifies the name of the net you want to analyze. You can type the name or browse a list of names by clicking *Select by Cursor*.

Select by Cursor lets you browse each net. If more than one net is under the cursor, a list box listing all nets appears. The Short Locator enters the name of the net you select in the name field.

Run Mode lets you select whether you wish to continue work on a previous Short Locator run or start work on a new run or.

Run starts the run.

Add Label adds a label to the net being analyzed.

Remove Label removes an existing label from the net being analyzed.

Fit Short zooms in on the bounding box surrounding the identified short.

Restore retrieves and lets you work from previously saved Short Locator data.

Save lets you save current data.

Finish cleans up memory and gets rid of old data so you can start work on a new run.

Active Layers lists circuit layers that are part of the interconnect. You can choose whether layers are active or inactive. Making a layer inactive turns off the highlighting of that layer. This is useful for layers such as a substrate, which otherwise highlights the entire circuit and interferes with selecting the net with a mouse. Inactive layers are included in the short analysis.

Using the Short Locator

1. Choose *Short – Start New Run*.

   The Short Locator form appears.
2. Accept the default label height or type the height you want.
3. To make layers active or inactive, click on the layers to turn them on or off.
4. Specify the net name in one of the following ways:
   - Type the name in the field.
   - Click *Select by Cursor*.

     The CIW prompts you to click on the net you want.

     If more than one net appears under the cursor, a list box appears listing all of the nets.

   Note: You might inadvertantly also pick up the substrate, well, or some other global layer if those layers are left active.

   - In the list box, click on the net you want and click *OK*.

     The name of the net you selected appears in the *Net Name* field.
5. To locate the short, click on *OK.*

   Note: If you click *Apply* and subsequently try to specify a new net name, The graphical user interface displays a dialog box warning you that you must end one run before starting another.

6. To refine the location of the short, select *Short – Add Label*.
7. End the run. To keep your changes, choose *Short – Save and Finish*.

   To discard your changes, choose *Short – Finish without Saving.*

Index

A

C

D

E

F

G

H

I

L

M

N

O

P

R

S

T

U

V

W