cadence®

Dracula Graphical User Interface Tutorial

Product Version IC23.1
September 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 task-related 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 its 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.

- The Dracula Graphical User Interface Reference describes the Dracula graphical user interface menu commands.
- The Virtuoso Layout Editor User Guide provides information about the Tools, Design, Window, Create, Edit, Verify, Connectivity, Options, and Route menus.
- The Cadence Installation Guide tells you how to install the product.
- The Dracula Standalone Verification Reference and the Dracula User Guide tell you how to use the Dracula® standalone verification product to verify your design.

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.

Each form shows you the system defaults:

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

1

Getting Started

This chapter deals with the basics of getting started with 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.

.This chapter discusses the following topics:

About the Dracula Graphical User Interface

When you use the Dracula® product to check whether you wrote your design rules correctly or to find and fix Dracula errors in your layout, the results can be difficult to analyze. One way to find Dracula Design Rule Checking (DRC) or Layout Versus Schematic (LVS) errors is to overlay the Dracula results on your layout and zoom in on each error manually. The problem with this method is that errors can be hard to see and keep track of, particularly in a dense layout. There is no way to access specific errors automatically.

The graphical user interface helps solve this problem. With the graphical user interface commands, you control which Dracula verification errors you want to see. You can cycle through errors automatically while the graphical user interface keeps track of the errors you've seen. When you use the graphical user interface to find and keep track of Dracula verification errors, you can resolve errors faster.

To correct Dracula errors in your layout, you can use the graphical user interface with your layout data to locate these errors. The graphical user interface works with any system that you use to create your original layout data. If you use the Virtuoso® layout editor, you can display both the layout and Dracula error files in the the graphical user interface window. If you use a layout tool other than the Virtuoso layout editor, you need to open the layout window separately from the graphical user interface window.

To verify the rules in your command file, you do not need to display the layout data. In this case, you use the graphical user interface to determine whether you wrote the rules correctly and whether Dracula flagged the errors as you intended.

This tutorial shows you how to use the graphical user interface to identify and display DRC and LVS results.

How to Use This Tutorial

In this tutorial, you will learn how to

- Display DRC errors in a flat design
- Display DRC errors in a hierarchical design
- Display and analyze LVS errors
- Display parasitic resistance and capacitance (RC)

Start with "Preparing to Start the Dracula Graphical User Interface" on page 10. This section helps you learn how to set up the software you will use in each of the DRC and LVS chapters.

After you set up the software, you can work through the chapters in this tutorial in any order.

Preparing to Start the Dracula Graphical User Interface

To do this tutorial, you need to copy the sample files to your own directory. Once you've copied the files, you'll start the software and open an empty cell.

In this chapter, you'll learn how to

- Copy the sample tutorial files
- Start the Cadence software
- Use the Command Interpreter Window (CIW)
- Use forms and menus

- Create an empty cell in which you can display data in the graphical user interface
- Open the empty cell

Copying the Sample Files

To perform the steps in this tutorial, you must use the tutorial files. The files you need to run the tutorial are in the directories `dracrun` and `drchier`. The paths to the directories are shown in the table below. If the `dracrun` and `drchier` directories are not there, contact your Cadence sales representative or customer support person.

Note: In the paths listed below, *your_install_dir* is the highest level in your Cadence directory hierarchy.

Directory	Contents	Path to the directory
`dracrun`	Directory containing the flat DRC and LVS data you use for this tutorial.	*your_install_dir*`/tools/dfII/samples/tutorials/inq/dracrun`
`drchier`	Directory containing the hierarchical DRC data you use for this tutorial.	*your_install_dir*`/tools/dfII/samples/tutorials/inq/drchier`
`dracrc`	Directory containing the parasitic RC data you use for this tutorial.	*your_install_dir*`/tools/dfII/samples/tutorials/inq/dracrc`

Before you start the tutorial, do the following:

1. Create a new directory called `tutorial`.
2. Create a new directory called `tutorial/dracrun`.
3. Create a new directory called `tutorial/drchier`.
4. Create a new directory called `tutorial/dracrc`.
5. Change your working directory to the `tutorial/dracrun` directory.
6. Copy the `dracrun` files into the `tutorial/dracrun` directory.
7. Change your working directory to the `tutorial/drchier` directory.
8. Copy the `drchier` files into the `tutorial/drchier` directory.

9. Change your working directory to the `tutorial/dracrc` directory.
10. Copy the `dracrc` files into the `tutorial/dracrc` directory.

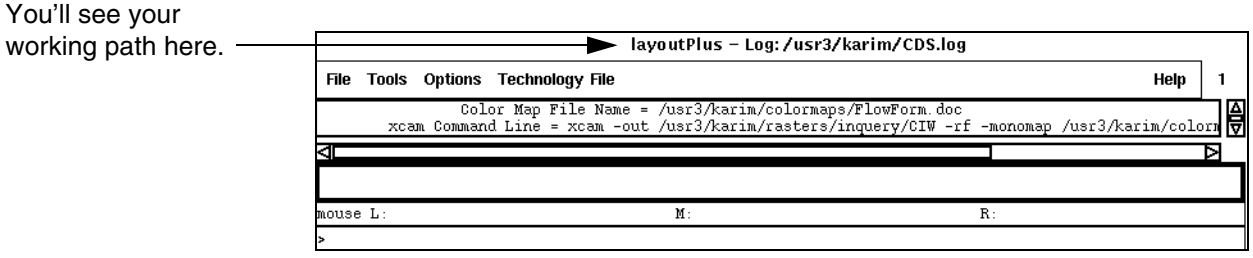
Now the tutorial files are ready for you to use them.

Starting Cadence Software

Each company has its own way to start Cadence software. Ask your system administrator how to log on and what command to type to start the software. If you have a Sun workstation, you also need to know how to start the X Window System software. Once you have this information, follow these steps.

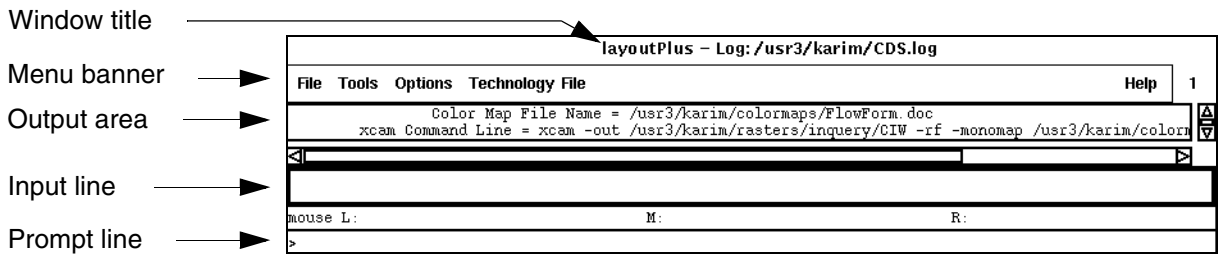
1. Log on to your system.
2. If you have a Sun workstation, start the X Window System software. If you have a different workstation, go on to the next step.
3. Type your command for starting Cadence software at the system prompt.

   In a few moments, you'll see the Command Interpreter Window (CIW). The Cadence® design framework II commands are located on menus at the top of the CIW, as shown.



Using the CIW

The CIW is the control window for the Cadence software. The following figure shows the parts of the CIW.



Window title displays the Cadence executable name and the path to the log file that records your current editing session. The log file is in your home directory.

Menu banner lets you display command menus to access the graphical user interface.

Output area displays a running history of the commands you run and their results. For example, it displays a status message when you open a library.

Input line is where you type in SKILL expressions or enter numeric values for commands.

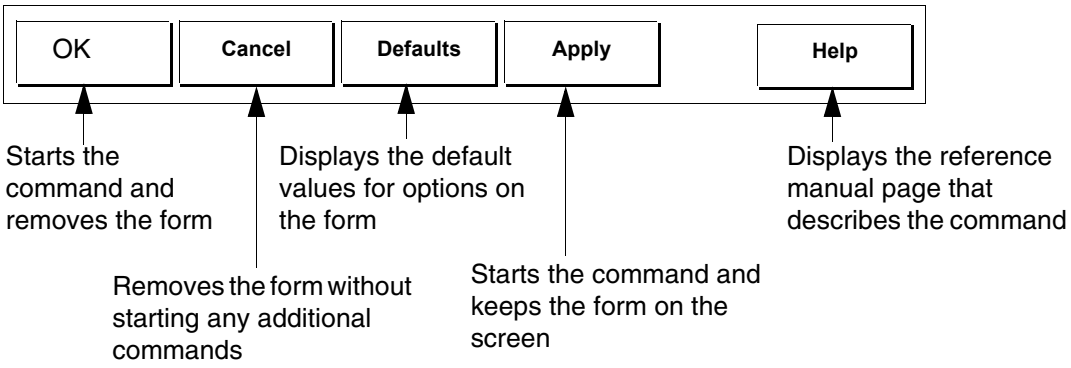
Prompt line reminds you of the next step during a command.

Using Forms and Menus

Most of the menus you use in the Cadence software are pull-down menus. Pull-down menus appear at the top of a window. You click the left mouse button on a menu title to “pull down” the menu and see the commands on it. The pull-down menus are the primary place to find commands.

Three dots (...) follow some commands. These dots mean that a “form” appears after you click on the command. Many commands have forms you use to provide additional information.

To use the following form banner buttons, you click on them with the left mouse button.



Note: Pressing `Return` has the same effect as clicking left on *OK*.

Using the Mouse

In this tutorial, “click left” means to press and release the left mouse button. If you need to click on a different mouse button, it will always be specified.

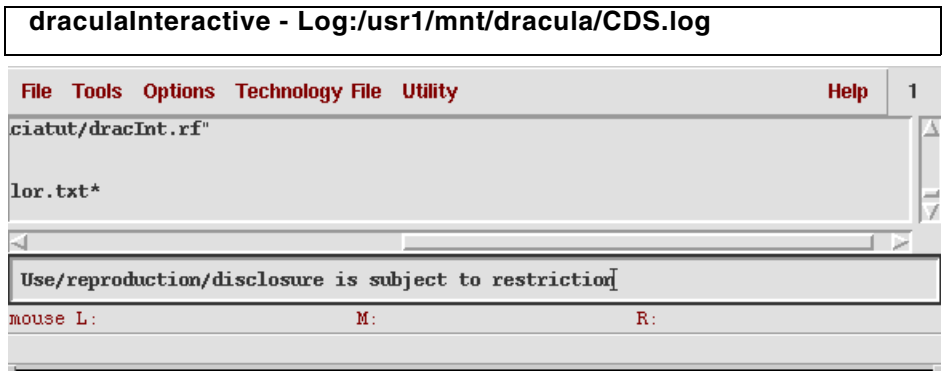
When you are asked to “select” commands from a menu, you click the left mouse button on the main menu, then on the command. For example, “Select `Library Manager` from the Tools menu” means to click the left mouse button on `Tools` to display the menu, then click the left mouse button on `Library Manager`.

When you are asked to “double click,” you click the mouse button twice without moving it.

Starting the Standalone Dracula Graphical User Interface

Instead of starting the Cadence layout software, you can start the standalone Dracula graphical user interface by typing the following command at the UNIX command line:

```
draculaInteractive &
```



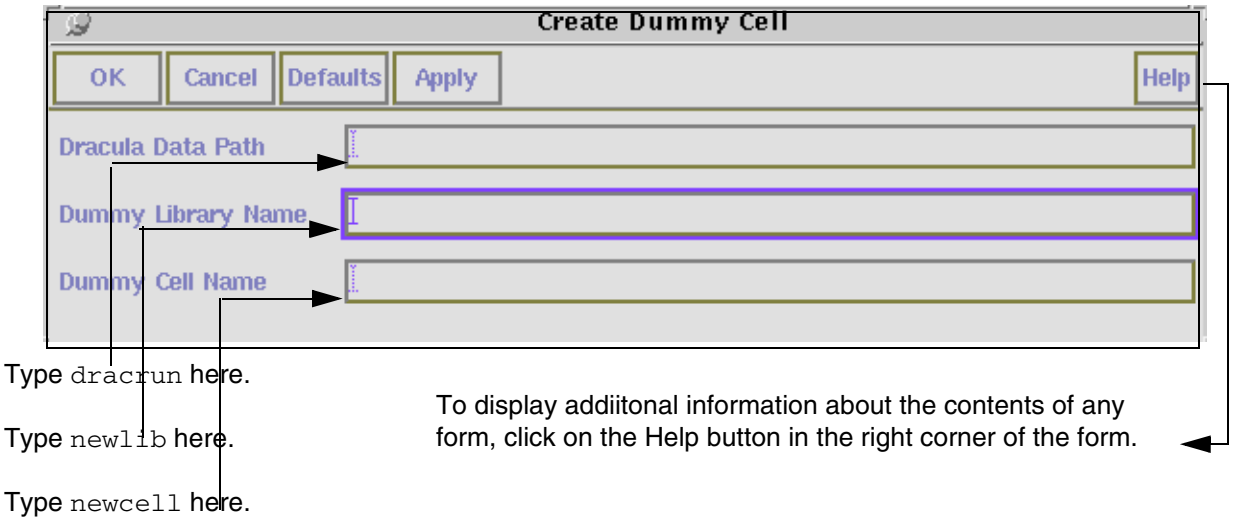
Creating an Empty Cell

Before you can display the sample Dracula data for this tutorial, you need to create an empty or “dummy” cell in which to put the data. The empty cell lets you access the graphical user interface menu commands. You use the same empty cell in each chapter of this tutorial. You need to create the empty cell only once.

1. In the `tutorial` directory, start the Cadence CIW or the Standalone Dracula graphical user interface window.
2. To display the Create Dummy Cell form of the graphical user interface, do one of the following:
   - In the CIW, type `iqCreateDummyCell` and press Return.
   - In the Dracula graphical user interface window, click on the menu command *Utility – Create Dummy Cell.*

   The Create Dummy Cell form appears.
3. In the form, type `dracrun` and the names of the new library and new cell.

Note: To move between the fields in a form, press the Tab key.



4. Click left on *OK*.

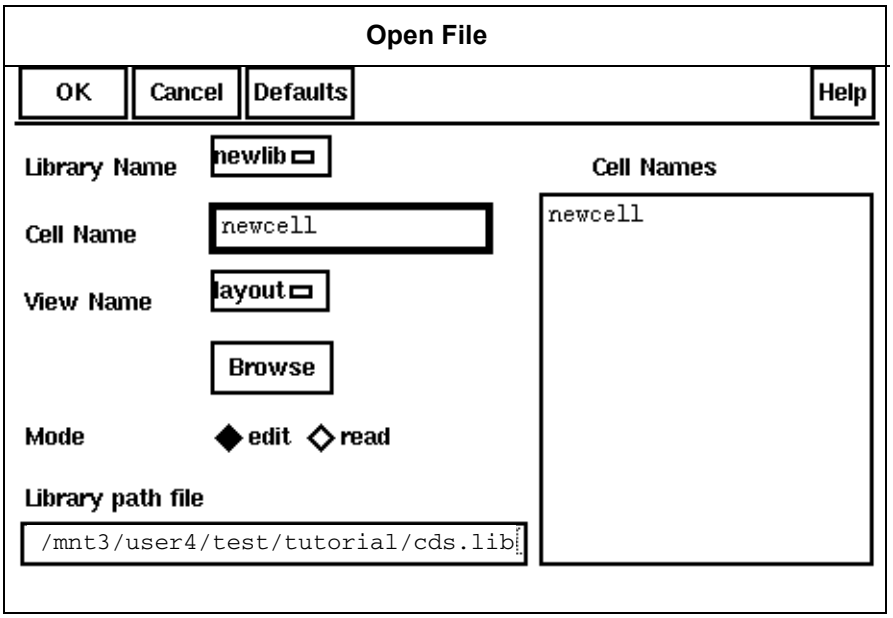
   After a few seconds, the form disappears and a message appears in the CIW to tell you that the cell was created successfully. Now that you've created the empty cell, you can open it and access the graphical user interface commands.

Opening the Empty Cell

To open the new empty cell you created, you'll use the Library Browser.

1. In the CIW, select *File – Open*.

The Open File form appears.



2. From the *Library Name* menu, select *newlib*.

   The Open File form is completed when `newcell` appears in the *Cell Name* field.

3. To open `newcell`, in the Open File form, click *OK*.

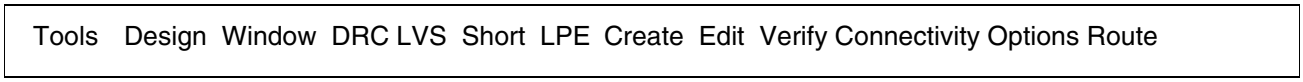
   Note: If you select a menu accidentally, click anywhere outside the menu to close it. If you select a command accidentally, click on `Cancel` in the command form to cancel the command.

   The graphical user interface window that displays the empty cell opens in *edit* mode, which means you can edit the cell. You will use this cell to display Dracula data. The menu banner in the empty cell looks like this:

4. From the menu banner, select *Tools – Dracula Interactive*.

   You use the left mouse button to select menus and commands.

The menu banner changes to display the graphical user interface menus. Except for the new menus, the Dracula graphical user interface window does not change.



Now you are ready to start any of the DRC, LVS, or LPE chapters.

Reopening an Empty Cell

At some point, you might want to close the windows and quit the tutorial. To resume the tutorial later, you can reopen the dummy cell as follows.

1. In the CIW or the graphical user interface window, click on *File – Open*.
2. When the Open File form is displayed, select `newlib` from the *Library Name* menu.
3. Be sure that `newcell` is displayed in the *Cell Name* field.
4. In the Open File form, click *OK*.
5. When the design window is open, from the menu banner, select *Tools – Dracula Interactive*.

   Now that you've got your dummy cell open again, you're ready to resume working through the tutorial.

2

Displaying DRC Errors in a Flat Design

This chapter relates to the Design Rule Checking (DRC) commands in the Dracula® graphical user interface and covers the following topics

About Flat DRC

When you run a flat DRC, Dracula creates error files for each of the design rule checks. In this chapter, you'll learn how to use the graphical user interface to display Dracula data. You'll use the the graphical user interface commands to locate and examine error flags in these files automatically, while the graphical user interface keeps track of which errors you've seen.

The procedures in this tutorial use Dracula output error files, not the original layout data. Because Dracula output data is in a different format, data you'll see in this tutorial won't look like your original layout data.

In this chapter, you will learn how to

- Start the graphical user interface DRC
- Select DRC error files to display
- Define how to display DRC errors

- Display DRC errors
- Verify DRC rules
- Keep track of which errors you've seen
- Quit the Cadence software

Starting DRC

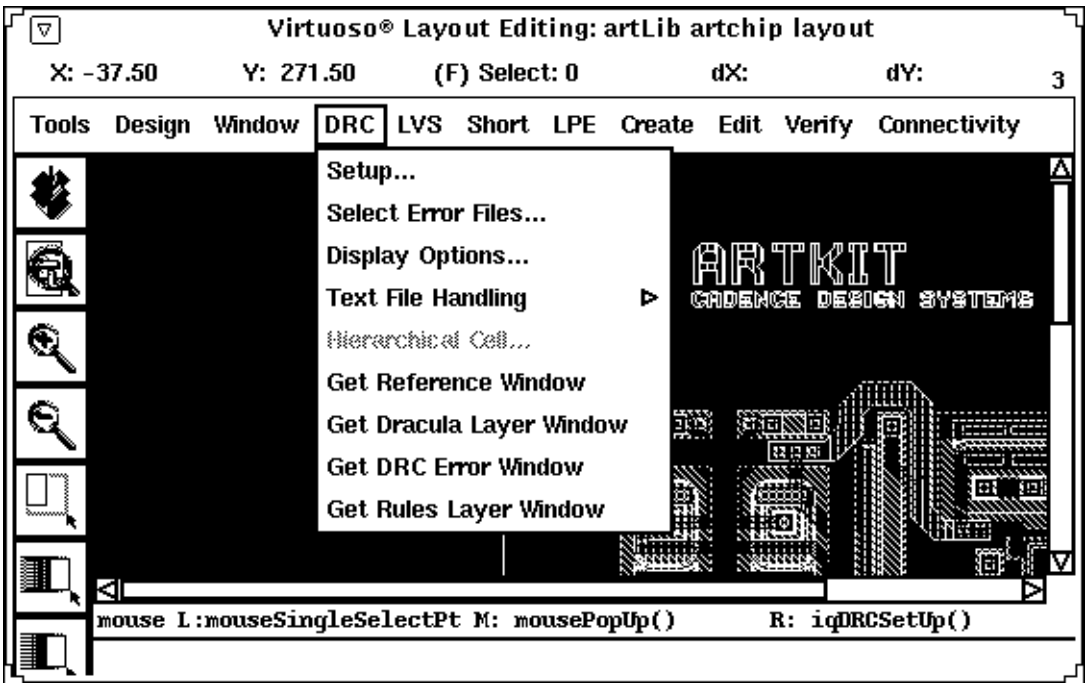
You need to create and open an empty cell, which is explained in Chapter 1, "Creating an Empty Cell." Once you have an empty cell open, you can go on with this section.

After you open the empty cell, the Dracula graphical user interface window appears. This graphical user interface window and the Layer Selection Window (LSW) are the only two windows open. You can ignore the LSW for now.

Accessing Dracula Error Files in the Dracula Graphical User Interface Window

In order to tell the software where to find your Dracula error files follow these steps.

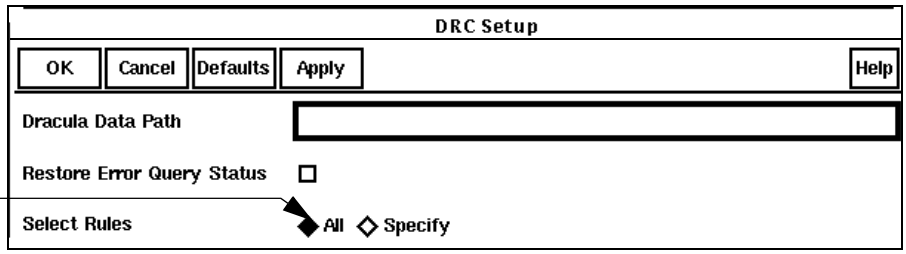
1. To display the DRC commands, click left on the DRC menu.



2. Select *Setup* from the DRC menu.

The DRC Setup form appears.

All is the default. This specifies that you want to check all the rules files.



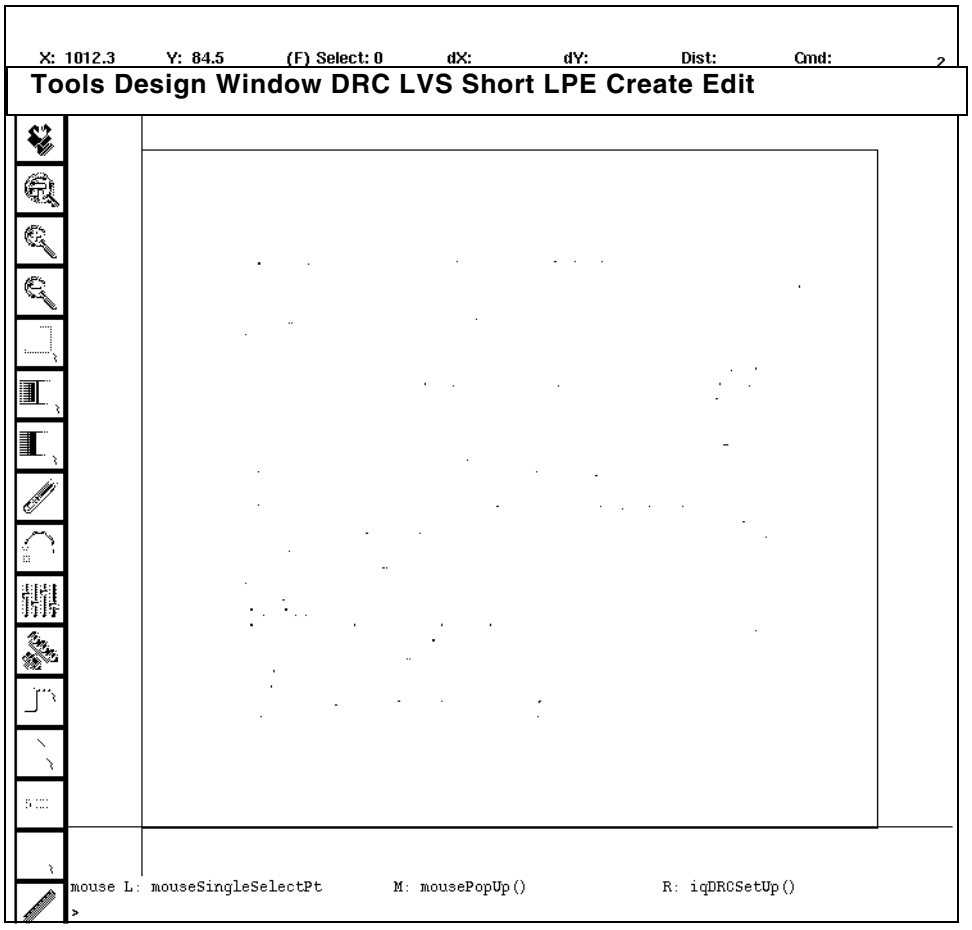
3. To tell the Dracula graphical user interface where to find your Dracula DRC files, in the Dracula Data Path field, type `dracrun`.

If you did not start the software in the directory where the Dracula DRC files are located, you must type the complete path to the files.

4. Click left on *OK*.

Note: The form takes a few seconds to disappear.

The graphical user interface window opens displaying all the error flags.



The following windows appear:

- Dracula Layer Window

  The Dracula Layer Window (DLW) appears on the left side of your screen and covers the Layer Selection Window (LSW). The DLW shows the original layers in your design after you run Dracula. You use the DLW to select the layers you want to view.

- View DRC Error window

  The View DRC Error window opens at the right side of your screen. This window lets you cycle through and manage DRC errors in the graphical user interface window.

- Reference Window

  The Reference Window opens at the bottom left of your screen. This window shows the entire the graphical user interface window on a small scale and helps you determine

where you are in the design window. You can use the `Window – Zoom Reference Area` command to have the the graphical user interface window zoom to the area you specify in the Reference Window.

- Rules Layer Window

  The Rules Layer Window opens at the bottom of your screen. This window lists DRC error files and lets you select errors you want to view in the graphical user interface window.

You will use these windows later in this tutorial. For now, you can ignore them.

Now the graphical user interface knows where to find your Dracula DRC error files. Next you'll select individual error files you want to display.

Defining How to Display DRC Errors

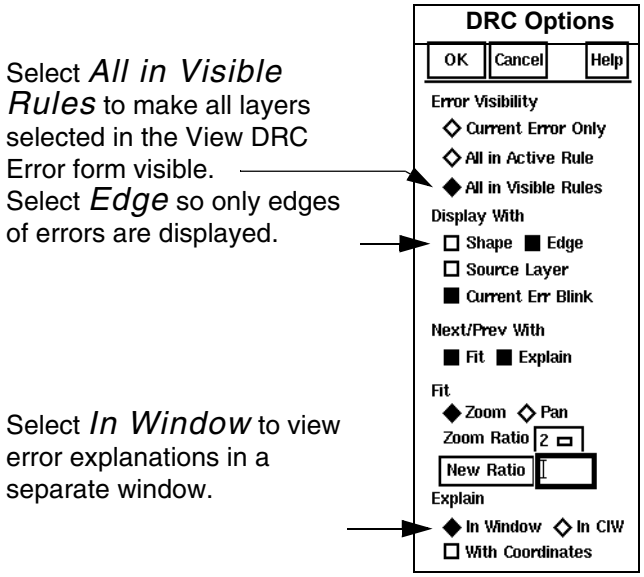
To help you distinguish different types of errors, you set the way you want error flags to appear in the graphical user interface window. For this chapter, you'll display error flag outlines.

Displaying Error Flags

1. To set how error flags are displayed, from the DRC menu, select *Display Options*.

   The DRC Options form appears.

2. Select the following options.



The graphical user interface window is automatically updated when you select display options. You might want to leave this form up so you can quickly change how error flags are displayed. To close the form, click *OK*.

You just specified how you want errors to be displayed. Next you'll display the errors.

Displaying DRC Errors

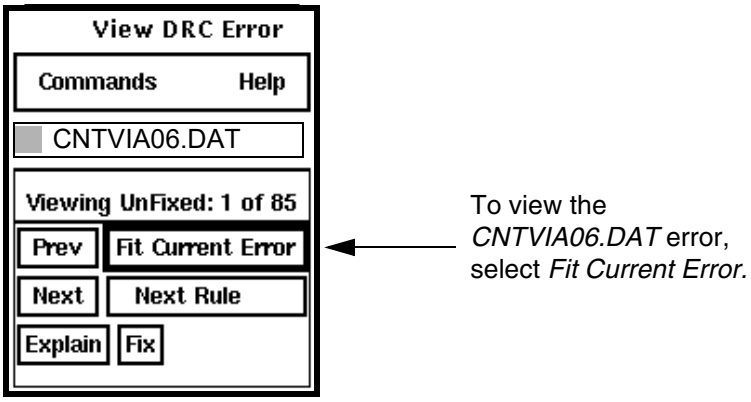
A Dracula run often generates several DRC error files. To minimize the number of error flags in the graphical user interface window, you might want to display only a few errors at one time. For this chapter, you'll zoom in on each contact-to-via error.

Selecting Error Files

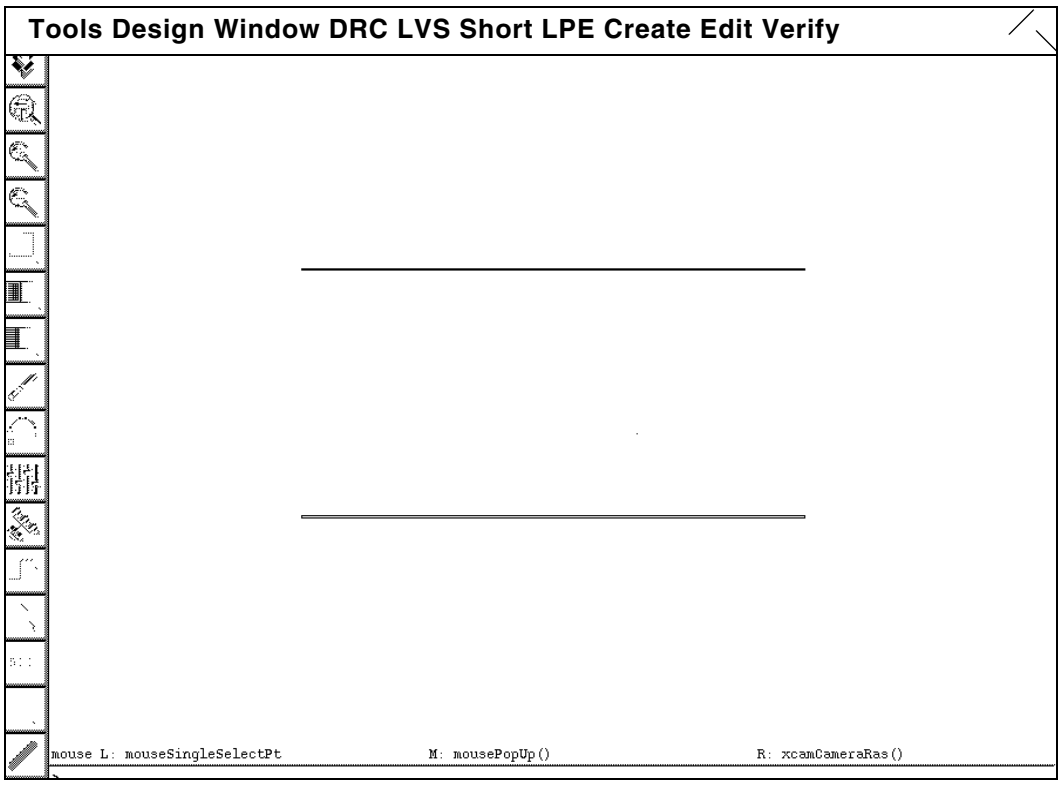
1. In the Rules Layer Window, click left on *CNTVIA06.DAT*.

This makes the *CNTVIA06.DAT* error your current error file.

2. In the View DRC Error window, select *Fit Current Error*.

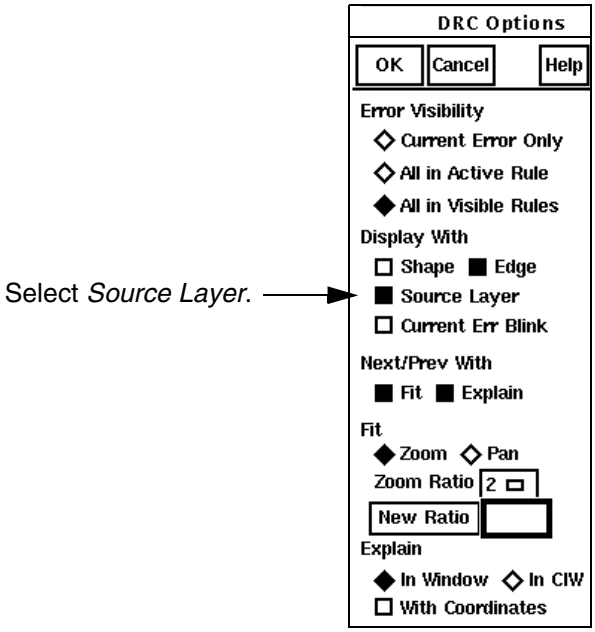


The first contact-to-via error appears in the graphical user interface window.

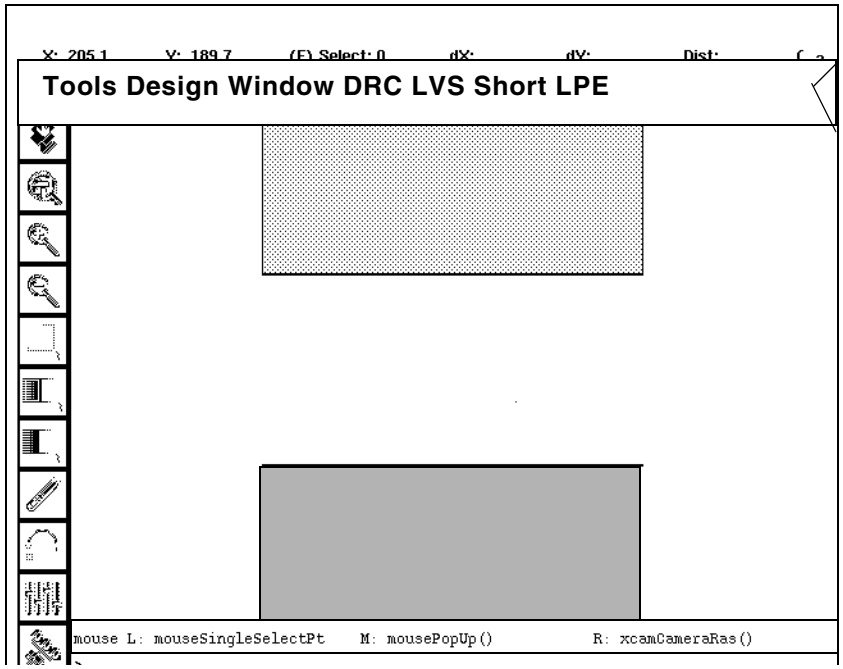


A window explaining the error appears in the upper right corner of your screen. You can ignore this window for now.

1. To display the layout layers associated with this error, from the Display With field on the DRC Options form, select *Source Layer*.



The graphical user interface window is automatically updated to show the source layers for the *CNTVIA06.DAT* error. In this case, the layers that are CONTACT and VIA, the layers that are the source of the error.



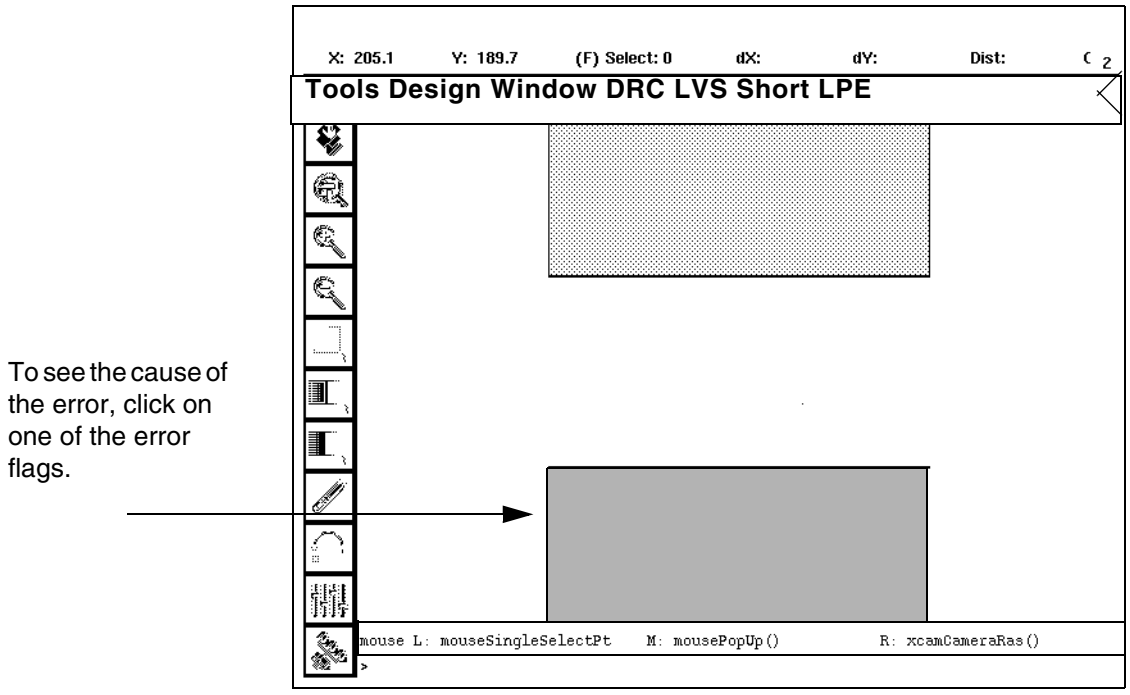
You've just displayed the Dracula layers for an error flag. Next you'll examine and verify the cause of these contact-to-via spacing error flags.

Verifying DRC Rules

Now that you've seen the contact-to-via spacing error, you're ready to determine the cause of the problem.

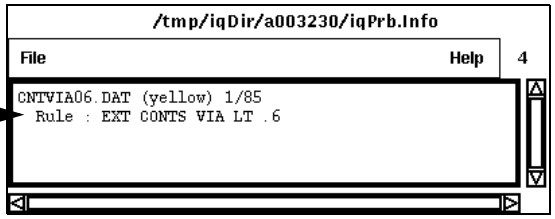
Displaying DRC Rules

1. In the View DRC Error window, click left on Explain.
2. Click left on one of the DRC flags in the graphical user interface window.



The error window at the top of the screen explains the cause of the error.

You see that this error is caused by contact-to-via spacing of less than 0.6 microns.



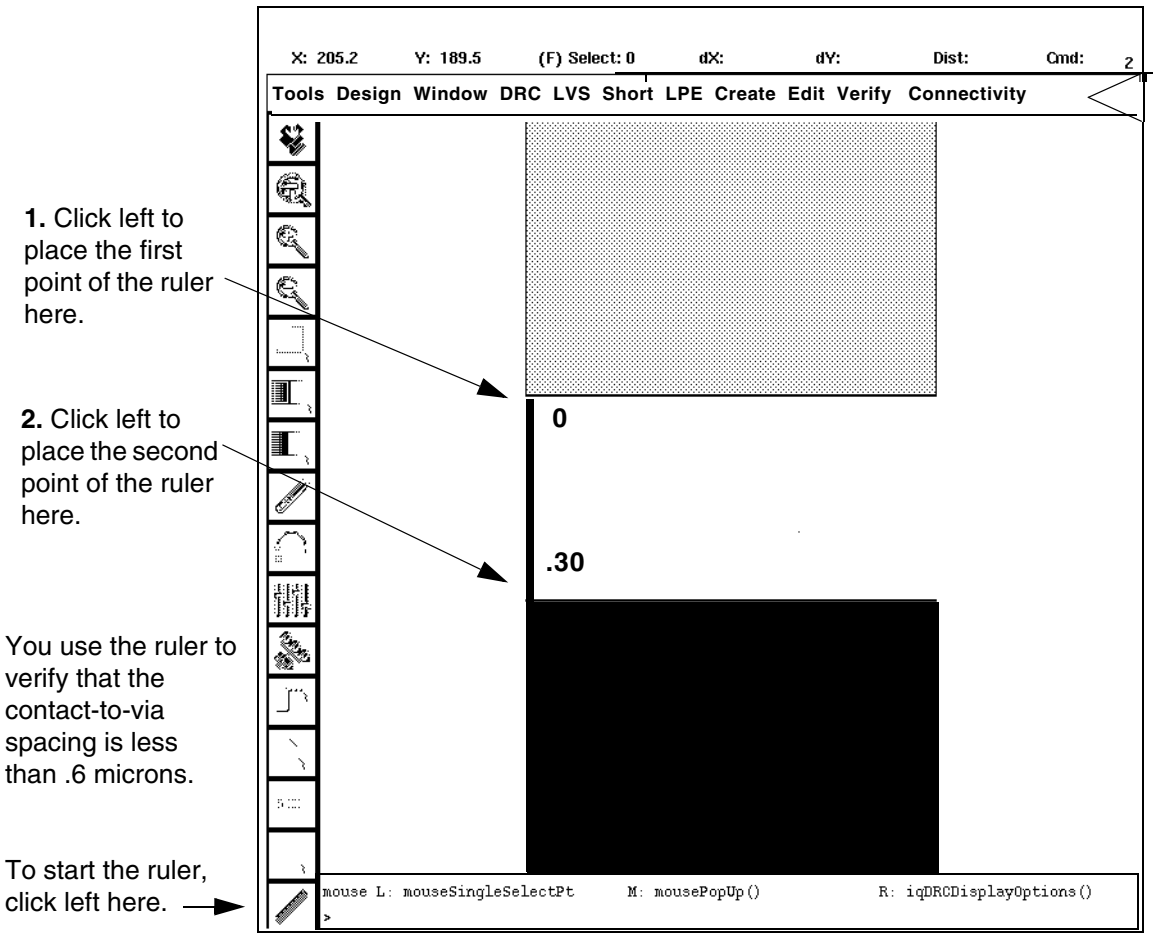
3. To close the text file, from the File menu, select *Close Window*.

To verify that the error explanation reflects the problem in the design, you'll measure the space between the contact and via Dracula layers.

4. Click left on the ruler at the bottom of the icon menu on the left of the graphical user interface window.

Note: The locations of the commands in your icon menu might be different. To see the command name associated with an icon, move the cursor over the icon.

5. Use the ruler to measure the space between the shapes.



You see that this spacing is less than the required 0.6 microns.

1. To remove the ruler, go to the Misc menu and select *Clear Rulers*.

Now that you know how to display and verify errors, you need a way to keep track of which errors you've seen. Next you'll display the error status report, which helps you automatically keep track of which errors you've seen.

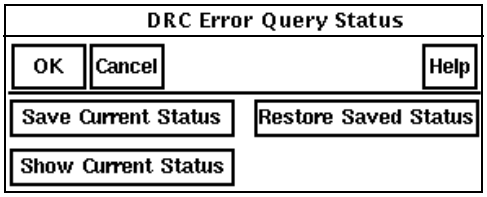
Keeping Track of Errors You've Seen

To keep track of all the errors that resulted from your Dracula DRC run, you use the error status report. This report is a text file that provides information about these errors.

Displaying the Error Status Report

1. To display the error status report, from the View DRC Error window, select *Commands – Error Status*.

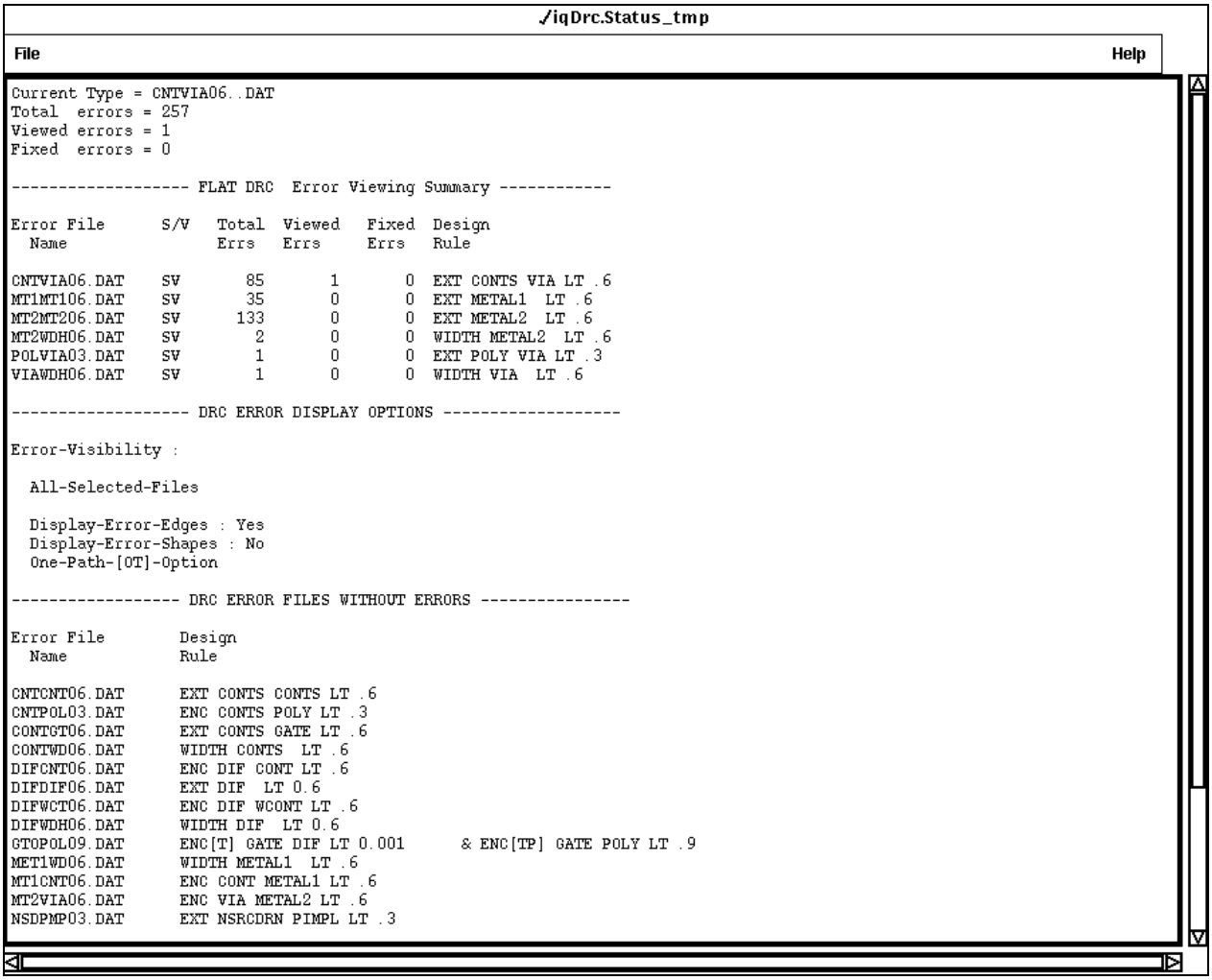
   The DRC Error Query Status form appears.



   Note: You can use this form to save your place in the error file so you can start with the same error when you begin the next session.

2. To display the error status report, click left on *Show Current Status.*

The status text file appears.



3. To close this text file, from the File menu, select *Close Window*.

Quitting the Tutorial

At this point, you might want to continue working with the graphical user interface. If you'd like to continue, you can leave the windows you used for the tutorial on the screen or you can close them. If you want to close the windows, go to the "Closing Windows" section.

When you are ready to quit the Cadence software, go to "Quitting Cadence Software" on page 33.

In this section, you'll learn about

- Closing windows
- Quitting Cadence software

Closing Windows

If you want to continue working with the graphical user interface, but would like to close some of the windows, follow these steps.

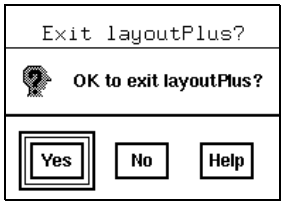
1. To close the graphical user interface window, from the Window menu, select *Close*.
2. To close a text window, such as the error status report window, from the File menu, select *Close Window*.

Quitting Cadence Software

To quit the Cadence software, follow these steps.

1. From the File menu in the CIW, select *Exit*.

   A dialog box appears asking if you want to exit.



2. Click left on *Yes*.

   The graphical user interface closes all the windows and returns you to the system prompt.

Summary

In this chapter, you learned how to use the graphical user interface to identify and display DRC errors in a flat design. Specifically, you learned to

- Start the DRC command from the graphical user interface's menu
- Choose which errors to view
- Set how you want error flags to appear in the the graphical user interface window
- Display DRC errors
  - Display the layers associated with the error flags
  - Verify the cause of DRC errors
- Use the error status report to keep track of which errors you've seen
- Quit the Cadence software

3

Displaying DRC Errors in a Hierarchical Design

This chapter deals with Design Rule Checking (DRC) errors in the Dracula® graphical user interface and covers the following topics:

About Hierarchical DRC

When you run hierarchical DRC, Dracula creates error files for each of the design rule checks as for flat checking, but it reports them cell-by-cell for cell-based errors rather than flattening them. The cells are called Hcells, and error viewing needs to deal directly with this extra structure.

In this chapter, you'll learn how to use the Dracula graphical user interface to display hierarchical Dracula data. You'll use the graphical user interface commands to automatically

locate and examine the error flags in these files, while the graphical user interface keeps track of which errors you've seen. You'll also learn how to work with Hcells.

The procedures in this tutorial use Dracula output error files, not the original layout data. Because Dracula output data is in a different format, data you'll see in this tutorial won't look like your original layout data.

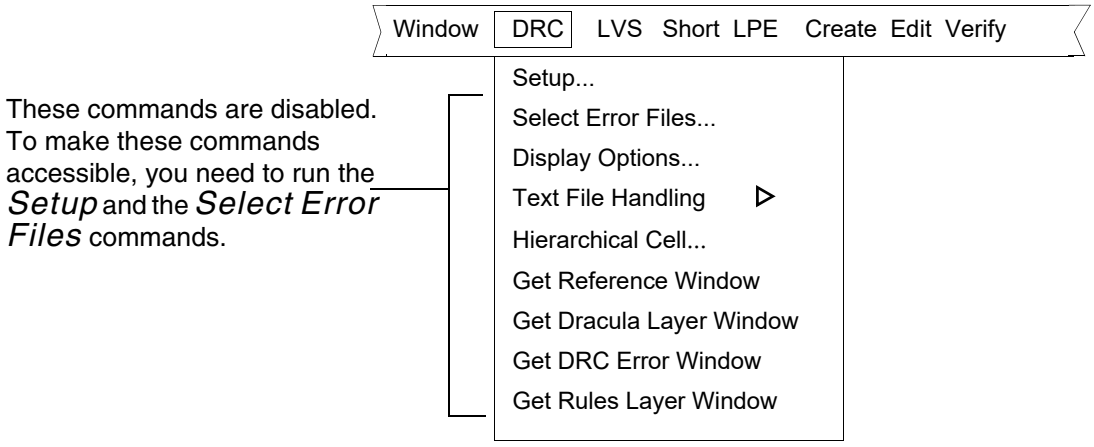
Starting DRC from the Graphical User Interface

Before you start this section, you need to create and open an empty cell, which is explained in the first chapter of this tutorial. If you have not yet started the graphical user interface software yet, read Chapter 1, "Getting Started."

After you open the empty cell, the graphical user interface window appears. The graphical user interface window and the Layer Selection Window (LSW) are the only two windows open. You can ignore the LSW for now.

Now you'll tell the software where to find your Dracula error files.

1. To display the DRC commands, click left on the DRC menu.

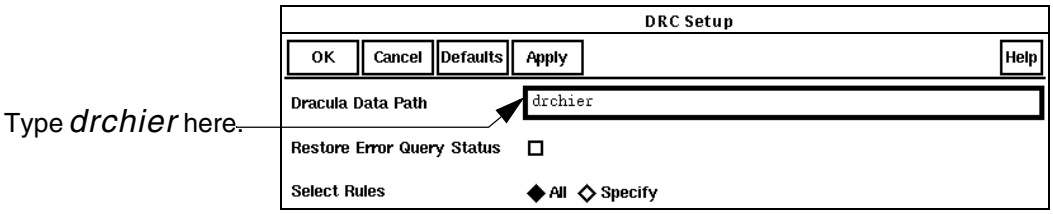


2. From the DRC menu, select *Setup*.

   The DRC Setup form appears.

3. To tell graphical user interface where to find your Dracula DRC files, in the Dracula Data Path field, type *drchier*.

If you did not start the software in the directory where the Dracula DRC files are located, you must type the complete path to the files.



4. Click left on *OK*.

Now the graphical user interface knows where to find your Dracula DRC error files.

The following windows appear. These windows are described in the section of the previous chapter.

- Dracula Layer Window
- View DRC Error window
- Reference Window
- Rules Layer Window

You'll use these windows later in this tutorial.

Next you'll select the error files you want to display.

Selecting Hierarchical DRC Error Files

A Dracula run often generates several hierarchical DRC error files. To minimize the number of error flags in the graphical user interface window, you probably want to display only a few of the error files at once.

Next you'll learn how to tell the graphical user interface which files to display. For this section, you'll need to select the files with *metal2-to-metal2* errors.

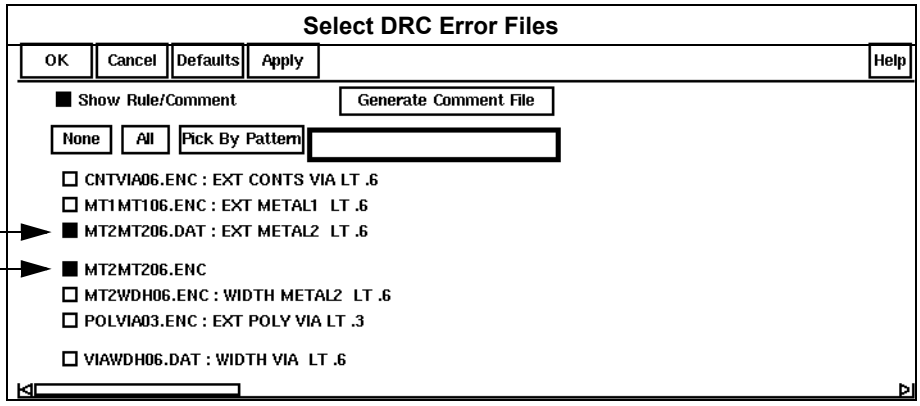
1. From the DRC menu, select *Select Error Files*.

   The Select DRC Error Files form opens.

2. On the Select DRC Error Files form, select Show Rule/Comment.

3. Keep selected only the *MT2MT206.DAT* and *MT2MT206.ENC* error files.

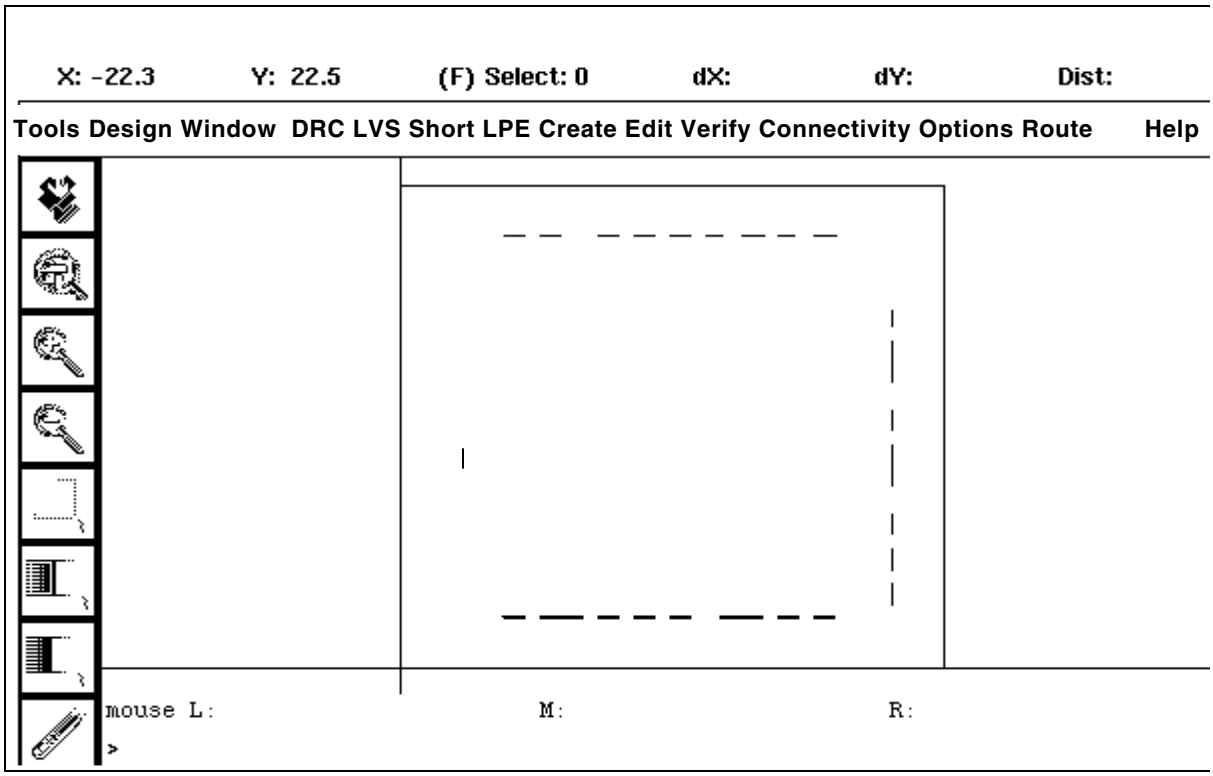
Leave only the *metal2-to-metal2* error files selected.



Note: The *.ENC* error files are Hcell and composite-to-Hcell errors. The *.DAT* error files are composite errors.

4. Click left on *OK*.

The Rules Layer Window contains only the error files you chose. The graphical user interface window shows the *contact-to-via* errors and *metal1-to-metal1* errors.



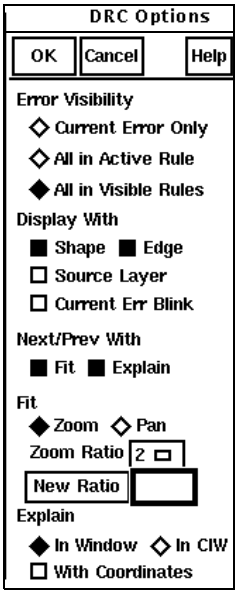
Now that the graphical user interface knows which error files you want to examine, you're ready to display the error flags.

Defining How To Display Hierarchical DRC Errors

To help you distinguish different types of errors, you set the way you want error flags to appear in the graphical user interface window. For this chapter, you'll choose how you want the error flags to appear. You can leave the default settings if you want.

1. To change how the error flags are displayed, from the DRC menu, select *Display Options*.

   The DRC Options form appears.



   The window is updated automatically when you select different options.

2. Select *All in Active Rule*.

   The graphical user interface window is updated to show the *MT2MT206.DAT* errors only because this is the active rule in the Rules Layer Window.

3. Select *All in Visible Rules.*

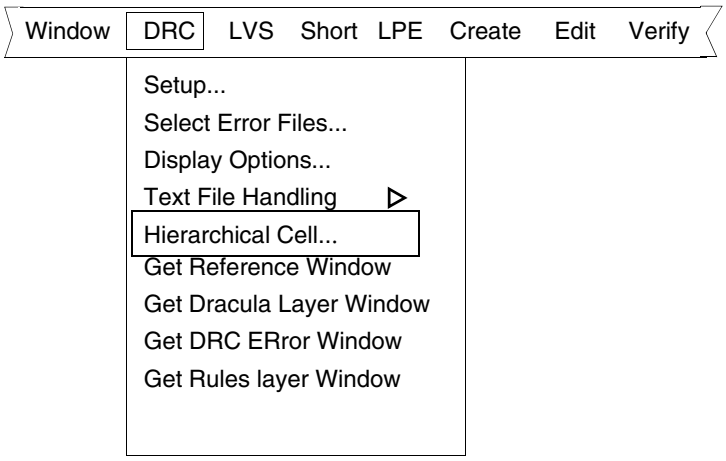
   The graphical user interface window is updated again to show all the *MT2MT206.DAT* and *MT2MT206.ENC* errors because these are all visible rules in the Rules Layer Window.

You've now defined how the *MT2MT206.DAT* and *MT2MT206.ENC* error flags appear. Next you'll select an Hcell with errors.

Selecting Hcells with Errors

Before you can see the hierarchical DRC errors in the *CNTVIA06.ENC* file, you need to select the Hcell in which the errors occur.

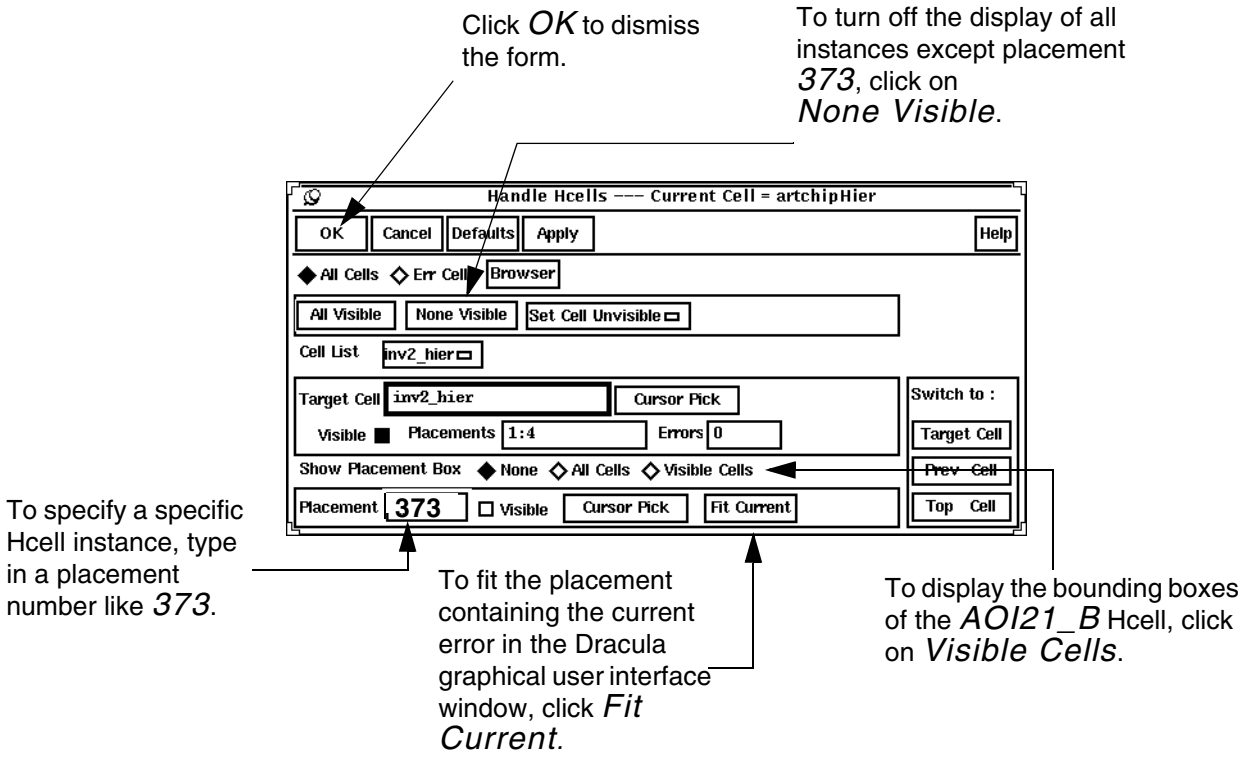
1. From the DRC menu, use Select Error Files to add the *CNTVIA06.ENC* error file to the Rules Layer Window.
2. In the Rules Layer Window, click on *CNTVIA06.ENC*.
3. From the DRC menu, select *Hierarchical Cell*.



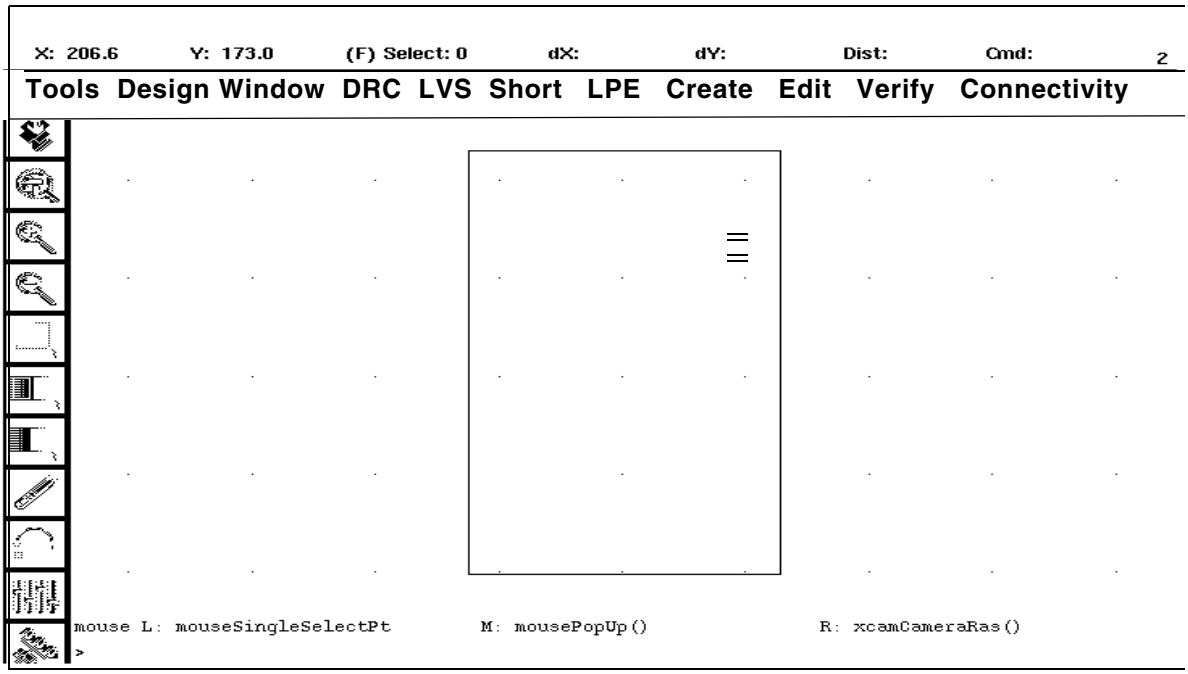
The Handle Hcells form appears. For this tutorial, you'll select the *A0121_B* Hcell.

4. Fill out the Handle Hcells form as shown.

The graphical user interface window is updated automatically when you select these options.



The graphical user interface window is updated to show placement *373* of the *AOI21_B* Hcell only.

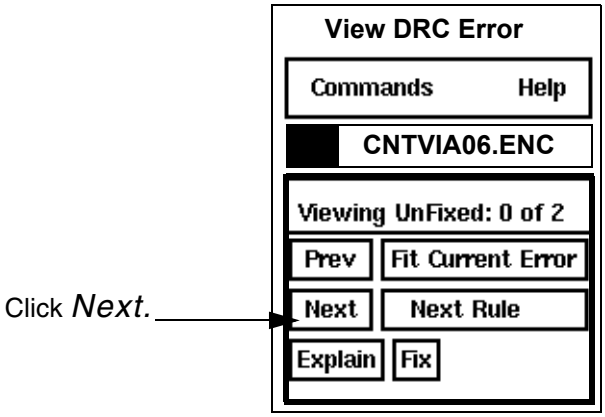


Next you'll display the error flags in the Hcell you selected.

Displaying Hcell Errors

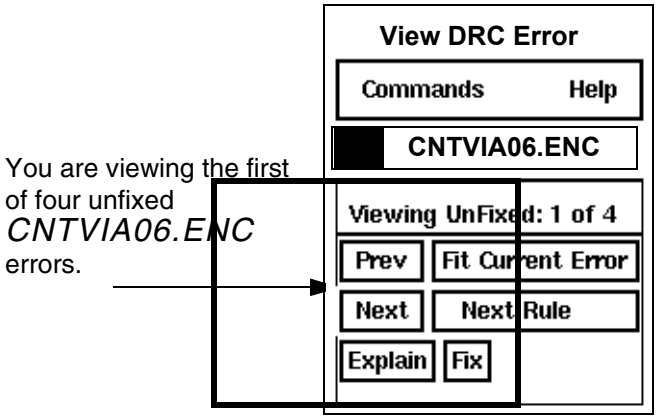
You can now display the contact-to-via error flags in the *AOI21_B* Hcell. You'll zoom in to the errors in that Hcell.

1. From the View DRC Error window, click left on *Next*.



The graphical user interface window zooms to the first error in the current Hcell.

The View DRC Error window is updated to show that you are viewing the first of four unfixed errors.



A window explaining the error opens at the top of your screen. You'll use it later in this tutorial.

2. To display the Dracula layers associated with the Hcell error flags, from the DRC Options form, select *Source Layer*.

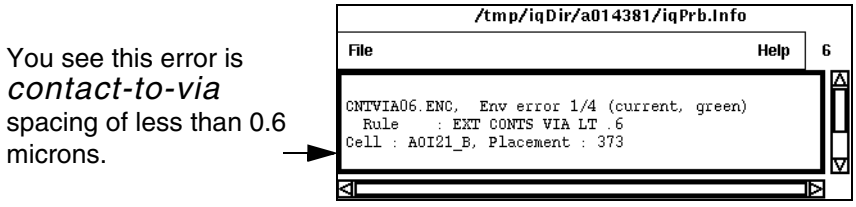
The graphical user interface window is updated automatically to show the source layers for the first error.

Notice that the DLW highlights source layers displayed in the graphical user interface window.

Verifying DRC Rules

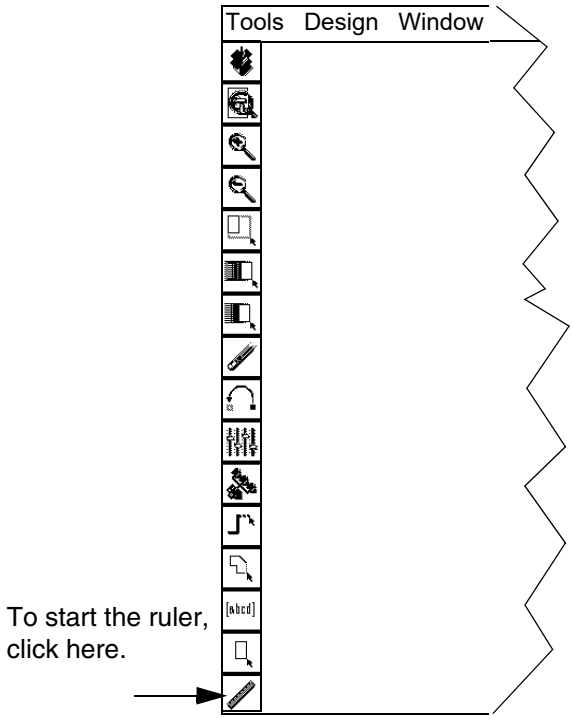
Now that the error flags are displayed, you can identify the rule that caused these DRC flags.

The text window at the top of your screen gives an explanation of the DRC error you selected in the *AOI21_B* Hcell.

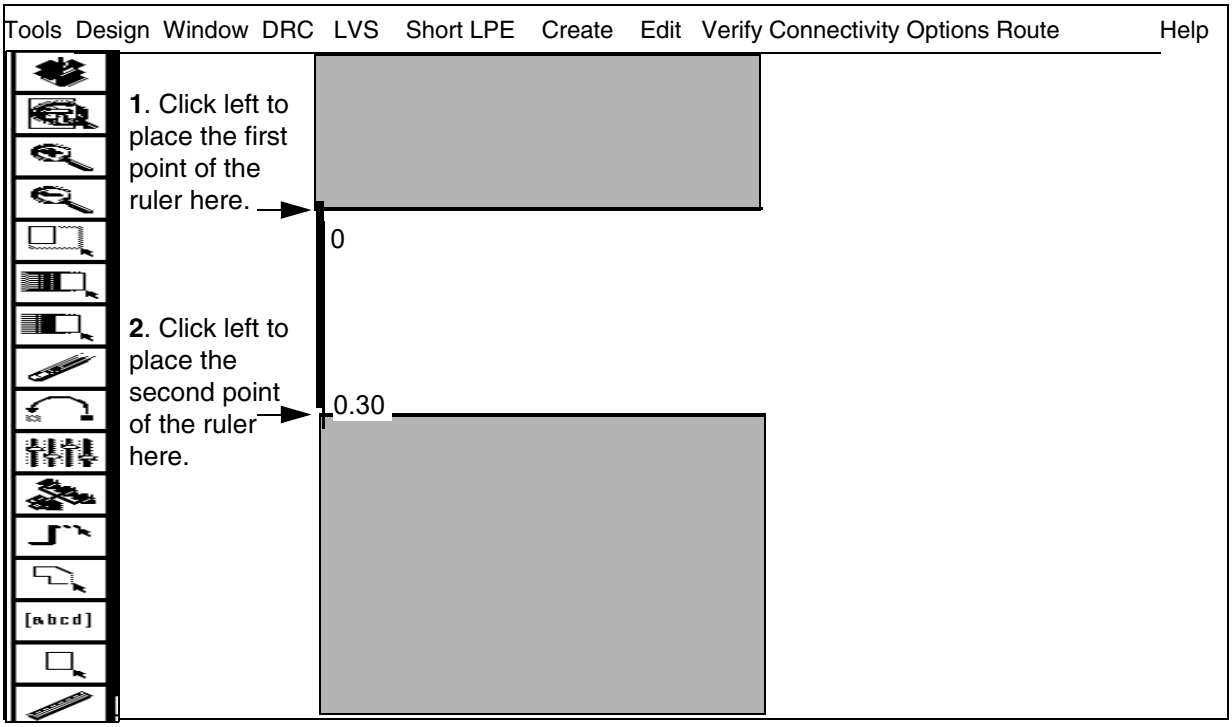


To verify that the error explanation reflects the problem in the design, you'll measure the space between the contact and via Dracula layers.

1. Click left on the ruler at the bottom of the icon menu on the left of the graphical user interface window.



2. Measure the distance between the edges of the shapes on each layer as shown in the graphic below.



You see that this spacing is less than the required 0.6 microns.

3. To remove the ruler, select *Window > Clear All Rulers*.

4. From the View DRC Error window, select *Next*.

The graphical user interface window zooms in to the next error in the current Hcell. Notice that the View DRC Error form shows that you are now viewing the second of four unfixed *CNTVIA06.ENC* errors.

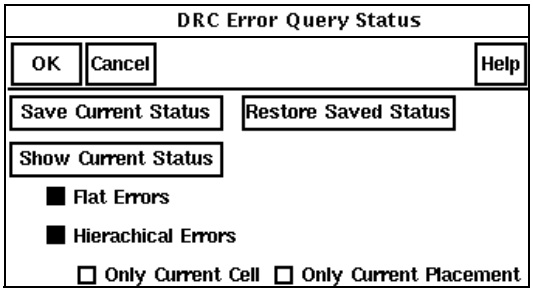
Next you'll learn how to keep track of the errors you've seen.

Keeping Track of Errors You've Seen

To keep track of all the errors that resulted from your hierarchical DRC run, you use the error status report. This report is a text file that provides information about all the errors that resulted from your hierarchical DRC run.

1. To display the error status report, from the View DRC Error window, select *Commands – Error Status*.

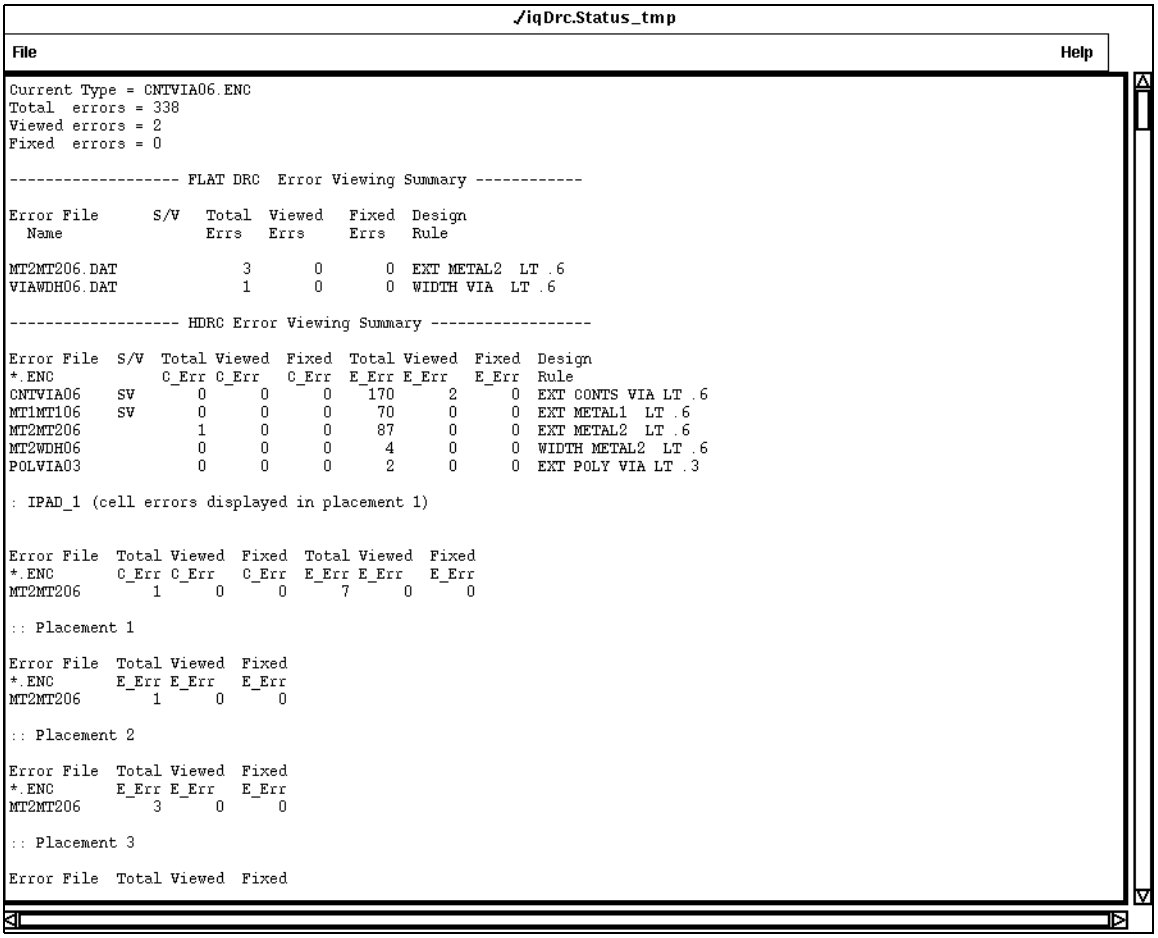
The DRC Error Query Status form appears.



Note: You can also use this form to save your place in the error file so you can start with the same error when you begin the next session.

2. To display the status report, click left on *Show Current Status*.

The status text window appears.



Now that you have seen the error status report containing information about all errors resulting from your hierarchical DRC run, you can close this text window.

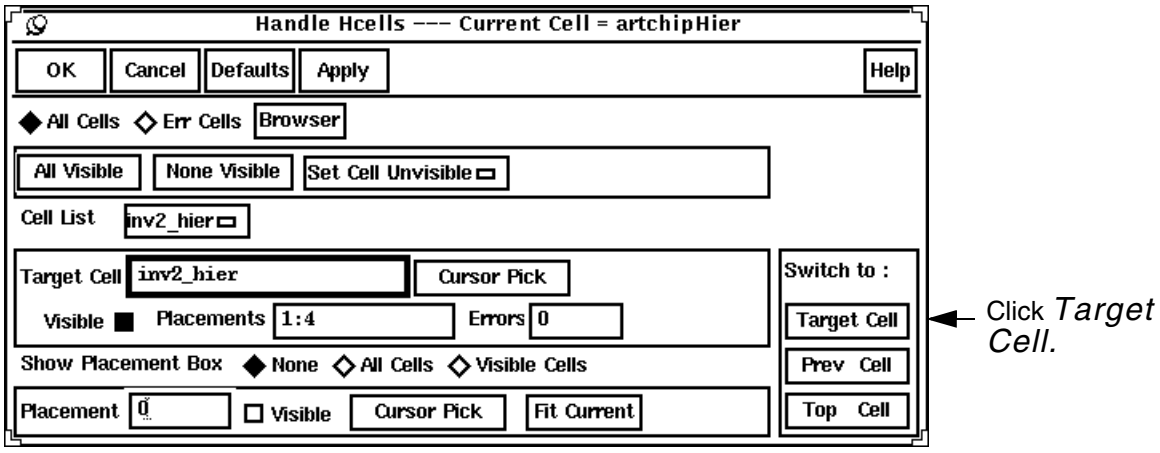
1. To close the error status report, from the *File* menu, select *Close Window*.

Displaying Hcell Masters

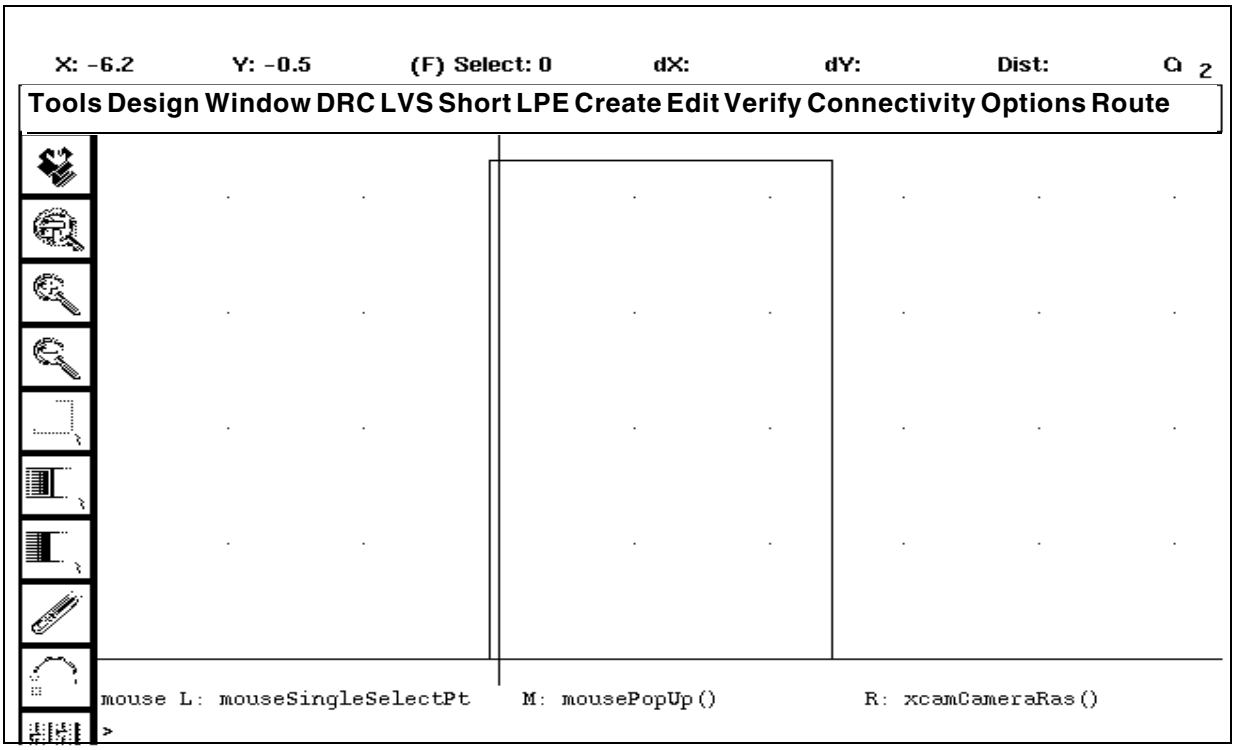
When you display Hcells, the graphical user interface window can get cluttered, which makes it hard to see individual Hcells. To reduce the amount of data in the graphical user interface window, you can display the Hcell masters only. When there is less data in the graphical user interface window, you can locate and evaluate errors faster.

1. From the DRC menu, select *Hierarchical Cell*.
2. From the Handle Hcells form, click the *Target Cell* button as shown below.

   The Handle Hcells form has all options you specified earlier in this chapter.



The graphical user interface window is redrawn to show the via and contact Dracula layers in the context of the INV cell rather than the whole chip.



Quitting the Tutorial

If you'd like to continue, you can leave the windows you used for the tutorial on the screen, or you can close them. To close the windows, go to the "Closing Windows" section.

When you are ready to quit the Cadence software, go to the "Quitting Cadence Software" section.

Summary

In this chapter, you learned how to use graphical user interface to identify and display DRC errors in a hierarchical design. Specifically, you learned to

- Start DRC from the graphical user interface
- Select hierarchical DRC error files
- Define how to display hierarchical DRC errors

- Select Hcells with errors
- Display Hcell errors
- Verify DRC rules
- Keep track of errors you've seen
- Display Hcell masters
- Quit the tutorial

4

Analyzing LVS Errors

This chapter deals with Layout Versus Schematic (LVS) in the Dracula® graphical user interface product. This chapter covers the following topics:

About LVS

Layout Versus Schematic (LVS) analysis is often the most time-consuming phase of the design verification process. The devices that cause errors in the layout can be difficult to locate if you have to manually zoom in and out to check each device listed in the LVS discrepancy report.

The graphical user interface's LVS commands help you analyze and locate LVS errors faster. You can display the schematic netlist in one window and the corresponding Dracula layout in another window. You can also select data in one window, such as nets or devices in the netlist window, and display corresponding information in another window.

This chapter introduces you to some of the ways you can use the graphical user interface to display and identify Dracula LVS errors. The procedures help you find the short in the tutorial data.

Note: There are many ways to identify LVS errors. Once you become familiar with the graphical user interface, the approach you use for your own designs might be different.

In this chapter, you'll learn how to

- Start the Dracula graphical user interface
- Interpret the discrepancy report
- Check netlist connectivity
- Display and highlight LVS errors
- Quit the Cadence software

Starting the Dracula Graphical User Interface's LVS

Create and open an empty cell before you start this section. Chapter 1, "About the Dracula Graphical User Interface" explains how to create and open an empty cell.

Important

Before you start graphical user interface's LVS command, you also need to edit the first line of the *CELLTABLE.HTV* file in your */tutorial/dracrun* directory to specify your complete directory path. You need to edit this file to display the netlist window.

Important

To edit the *CELLTABLE.HTV* file, follow these steps.

1. In your *dracrun* directory, use a text editor to open the *CELLTABLE.HTV* file.
2. Replace the path in the first line of the file with your full tutorial directory path:

   <your_working_directory_path>`/tutorial/dracrun/`

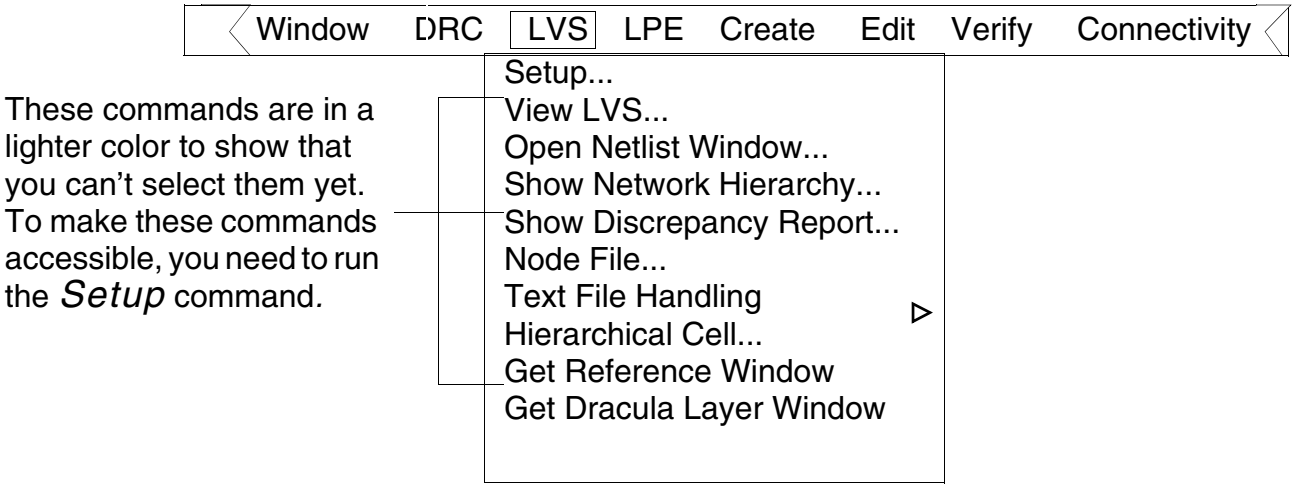
Note: Don't forget to add the final "/" character.

3. Save and exit the *CELLTABLE.HTV* file.

Accessing Dracula Error Files

Now you'll start the graphical user interface and tell the software where to find your Dracula error files. If you haven't started graphical user interface yet, read Chapter 1, "About the Dracula Graphical User Interface."

1. To display the LVS commands, click left on the LVS menu.

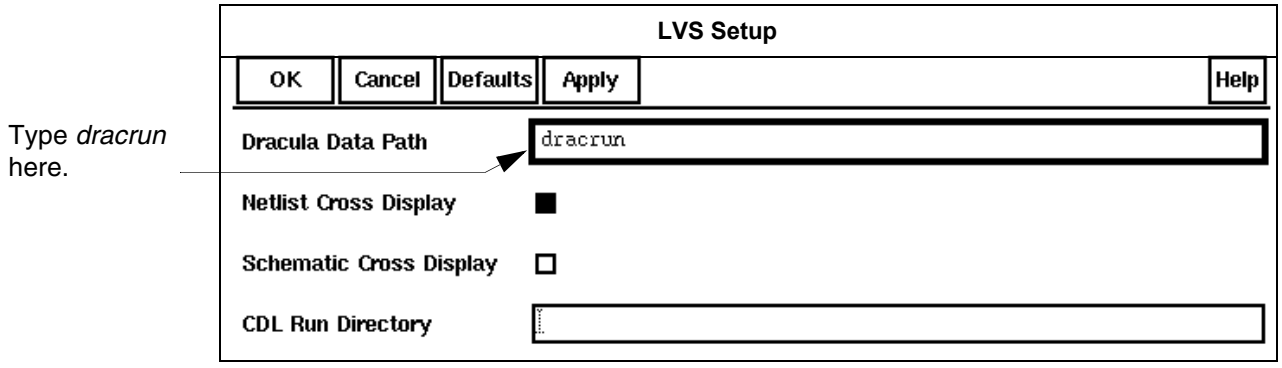


2. Select *Setup* from the LVS menu.

   The LVS Setup form appears.

3. To tell the graphical user interface where to find your Dracula LVS files, type *dracrun* for the Dracula Data Path.

   If you did not start the software in the directory where the Dracula LVS files are located, you must give the complete path to the files.



4. Click left on *OK*.

Note: The form takes a few seconds to disappear.

The View LVS form appears.

Now the graphical user interface knows where to find your Dracula LVS error files. Next you'll open the discrepancy report that lists LVS errors.

Interpreting the Discrepancy Report

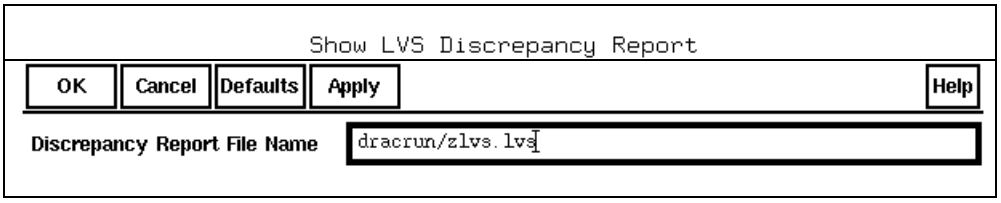
When you run Dracula LVS, the software creates a discrepancy report that lists inconsistencies between the layout and schematic data. You use the information in the discrepancy window to determine which data you want to display in the netlist and graphical user interface windows. See the Dracula Reference Manual for a description of the discrepancy report format.

You can use the graphical user interface to find and display the devices listed in the discrepancy report. Next you'll use graphical user interface to automatically display specific devices to help you locate the error in the sample Dracula layout.

Displaying the Discrepancy Report

1. From the LVS menu, select *Show Discrepancy Report*.

   The Show LVS Discrepancy Report form appears.



2. Click left on *OK.*

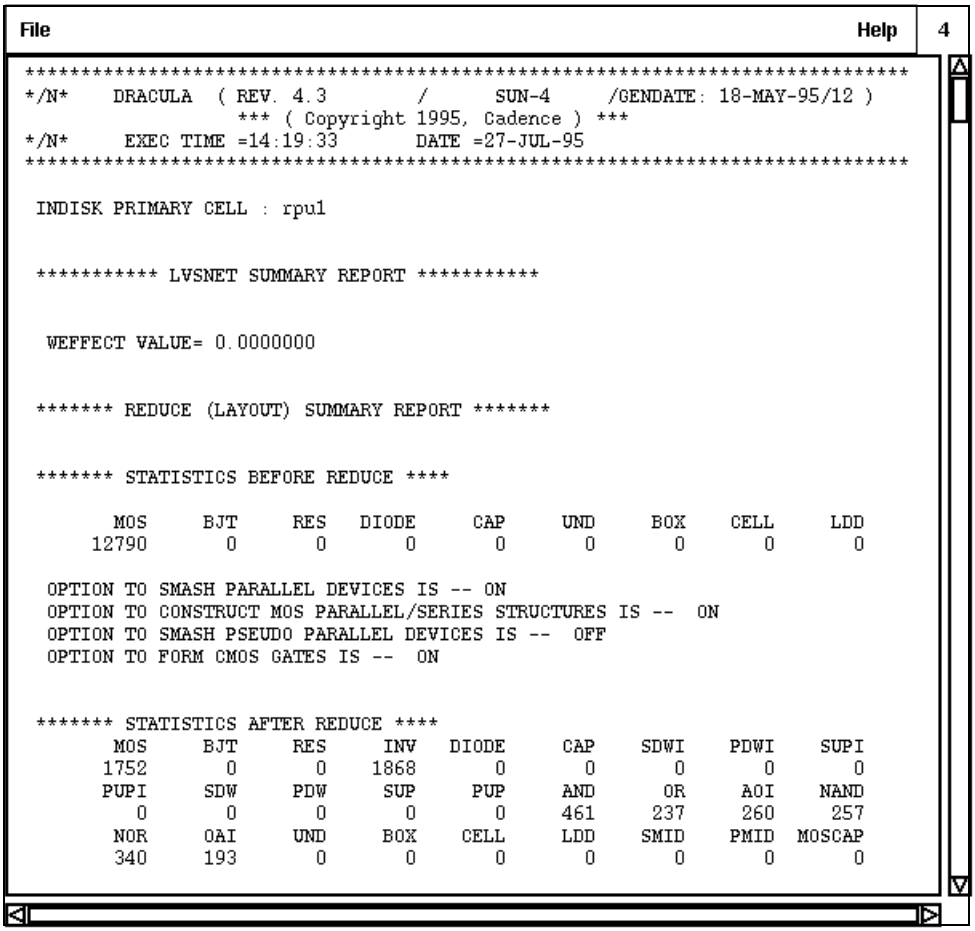
Important

When you are working on your own designs, the first thing you need to do is check the REDUCE SUMMARY REPORT section of the LVS report. If LVS reduces devices on the schematic and layout differently, you need to examine the connections to the devices and resolve the differences. If you do not resolve the differences, LVS cannot match the networks.

Important

For example, the following figure shows resistors that do not reduce identically. LVS reduces the schematic resistors in series to one resistor. However, because the layout resistors are connected to another node on the center net, LVS does not reduce them.

The discrepancy report appears in a new window.



3. Check the REDUCE statistics in the report.

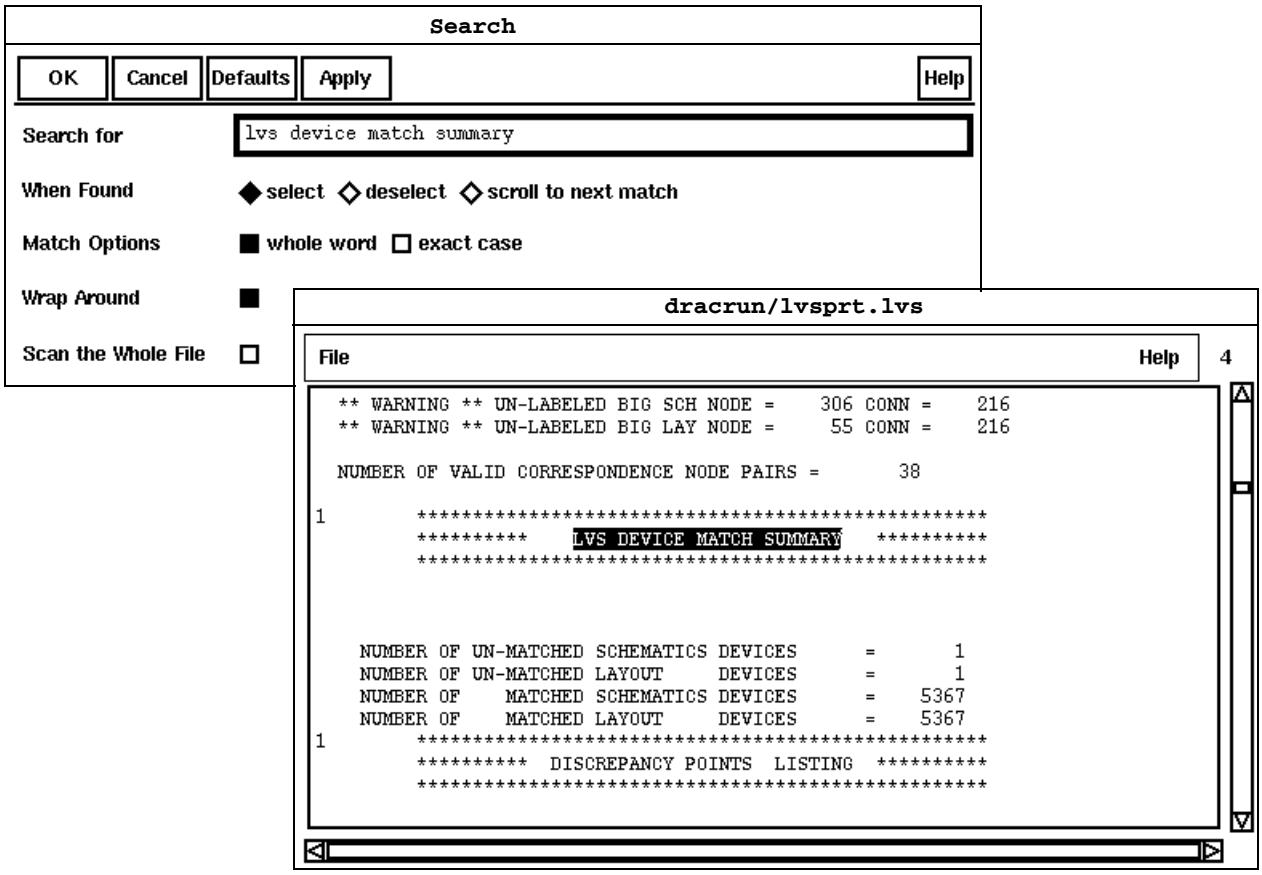
The layout had 12790 MOS devices. REDUCE combined them into logic gates, leaving 1752 MOS devices uncombined.

The schematic had 11378 MOS devices. REDUCE combined them to make the same logic gates it found in the layout and left the same number of uncombined MOS devices.

4. Scroll down to the LVS DEVICE MATCH SUMMARY.

You can see in the next figure that the layout and schematic each have one unmatched device. You use the discrepancies that follow to find out which devices are not matched. You use the highlight and cross-probing capability of the graphical user interface to find the unmatched devices in the layout and schematic.

You can use *File – Search* in the discrepancy report window to search for any text string in the discrepancy report.

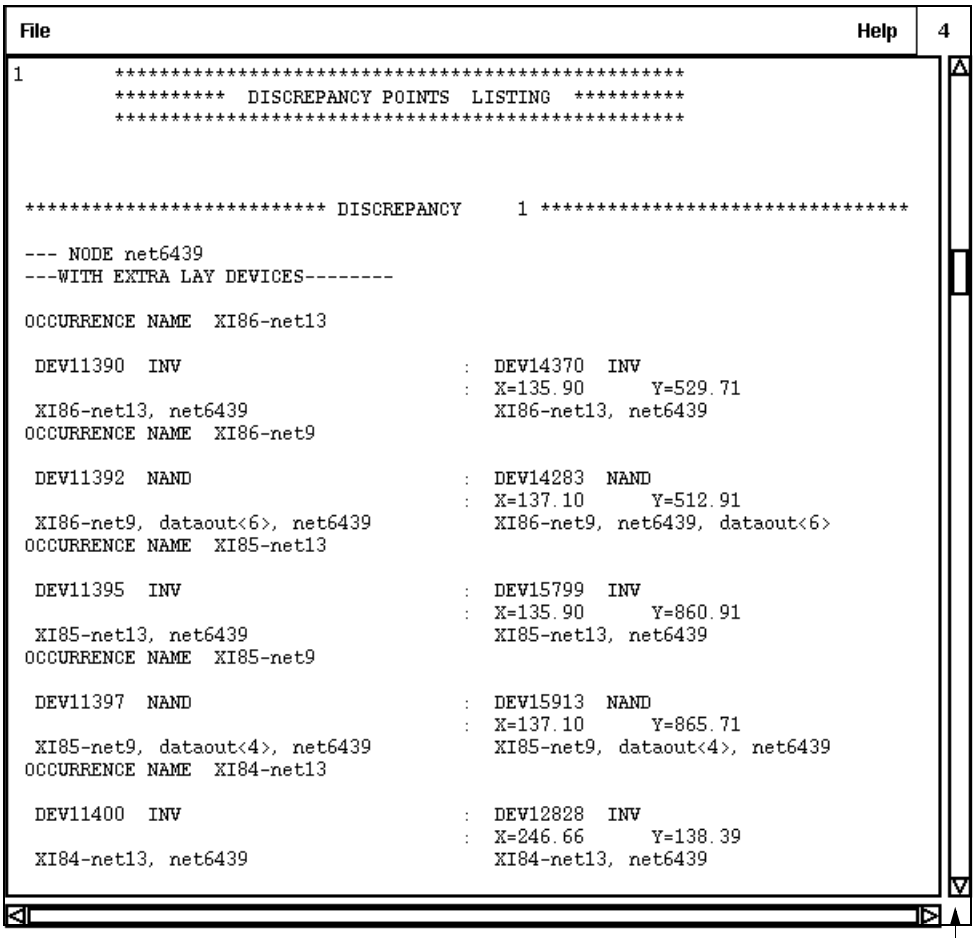


5. Scroll the discrepancy report until you see DISCREPANCY 1 at the top of the window.

Discrepancy 1 reports a matched node with extra layout devices.

Discrepancy 2 reports a matched node with an extra schematic device.

Discrepancies 3 and 4 are matched nodes with unmatched devices.



To scroll the discrepancy report window, click left here and hold.

Discrepancy 1 is a matched node, net6439, with extra layout devices. The extra layout device is listed at the end of the discrepancy:

```
***** UN-MATCHED *****      : ?DEV15181  NAND
                             :  X=1079.40     Y=706.50
                                X_rp-X_r2-w000062,
                               X_rp-X_r2-w000304, net6439
```

Discrepancy 2 is a matched node, dataout<3>, with extra schematic devices. The extra schematic device is listed toward the end of the discrepancy:

```
?DEV13735  NAND         :***** UN-MATCHED *****
           X_rp-X_r2-w000062, dataout<3>,
           X_rp-X_r2-w000304
```

The unmatched devices are noted with ? preceding their names. The name of the unmatched layout device is the internal name given by Dracula.

Discrepancies 3 and 4 are nodes, X_rp-X_r2-w000304 and X_rp-X_r2-w000062, with unmatched devices, both layout and schematic.

These discrepancies indicate that nodes net6439 and dataout<3> might be connected incorrectly.

6. To make room on the screen for the other windows, iconify the discrepancy report window.

   Now that you've used the discrepancy report to identify the devices that aren't matched correctly, you'll use the graphical user interface to locate the unmatched devices by highlighting the nets and devices in the layout and in the netlist.

Checking Netlist Connectivity

The netlist window lists the schematic netlist. This window makes it easier for you to compare the netlist connectivity to the Dracula layout data so you can identify inconsistencies.

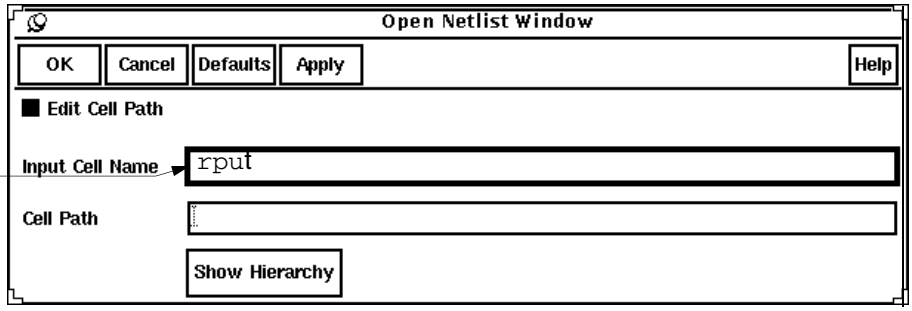
You identified dataout<3> as a net with an extra schematic device and net6439 as a net with an extra layout device. Now you'll display these nets in the netlist window and in the layout window to see where they appear.

Opening the Netlist Window

1. From the LVS menu, select *Open Netlist Window*.

   The Open Netlist Window form appears.

`rpu1` is the top-level cell name.

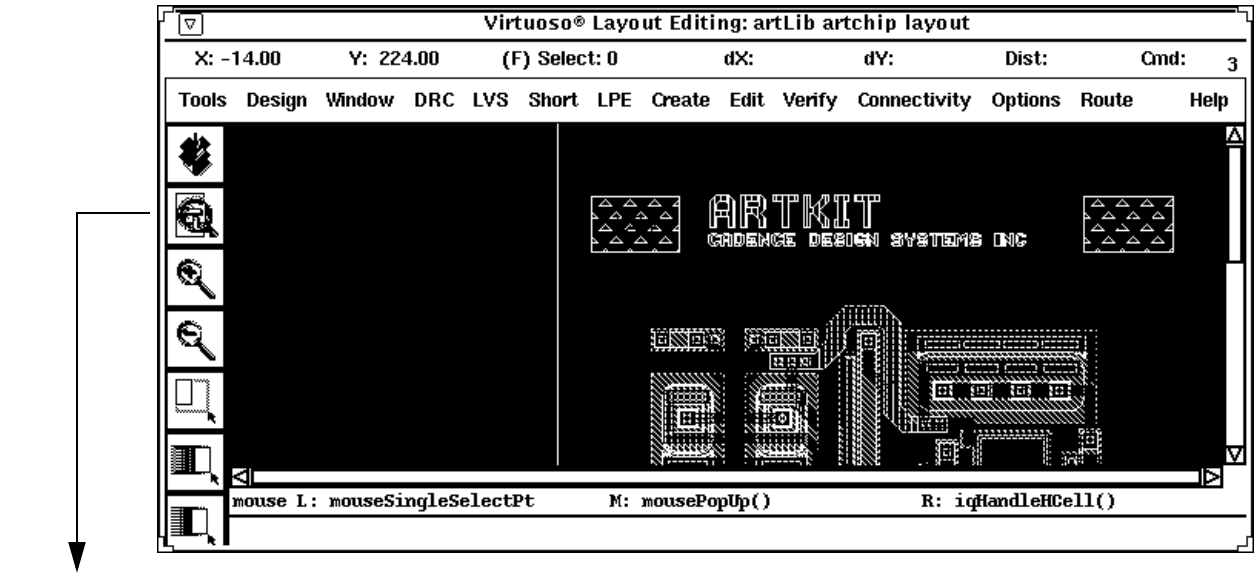


2. Click left on *OK*.

The netlist window appears. If the netlist window doesn't appear, see "Starting the Dracula Graphical User Interface's LVS" on page 52 for details about what you need to do.

3. To minimize how much the windows overlap, resize the layout and netlist windows and place them side by side.
4. In the graphical user interface window, from the icon menu select *Fit Edit*.

   Note: The locations of the commands in your icon menu might be different. To identify a command name in the icon menu, move the cursor over an icon.



Now you are ready to show the nets.

Displaying and Highlighting LVS Errors

The discrepancy report lists discrepancies between schematic and layout devices. Based on this information, you'll highlight the discrepancies in both the graphical user interface and netlist windows to help you identify the error.

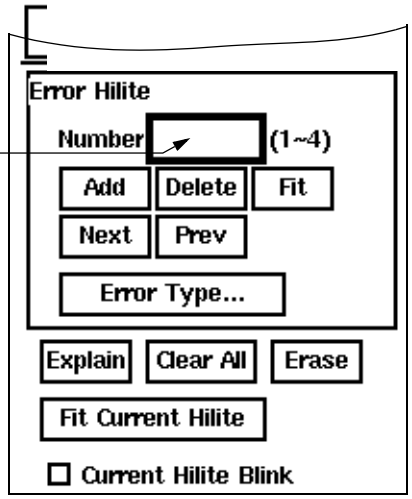
Displaying Nets and Devices in the Dracula Graphical User Interface and Netlist Windows

Now you'll highlight nets and devices in the graphical user interface and netlist windows.

1. In the *Error Hilite* section of the View LVS form, in the *Number* field type 1.

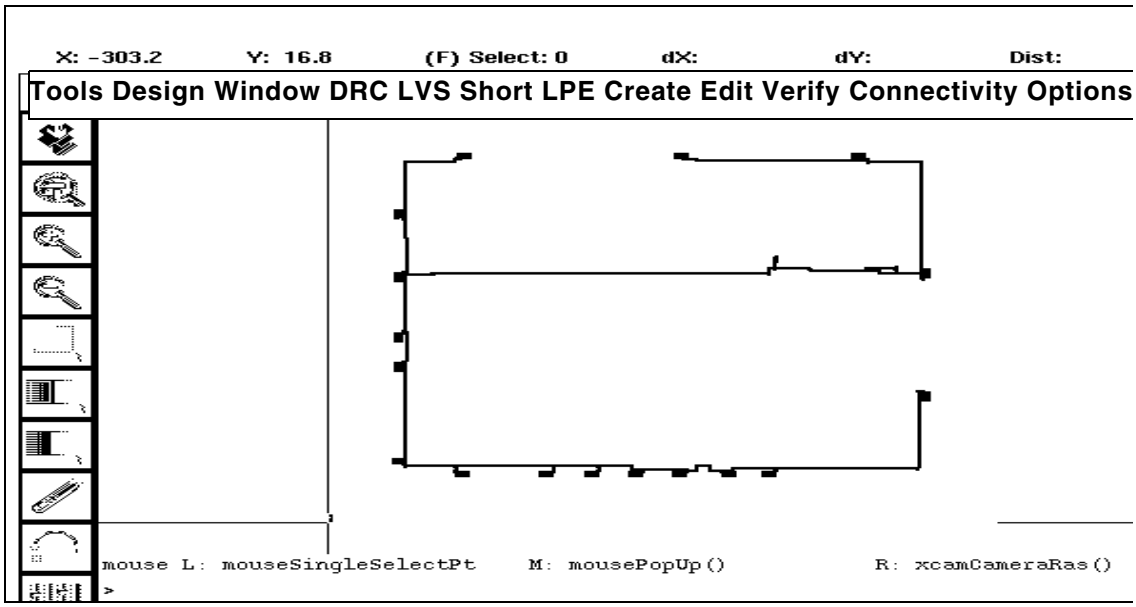
When you looked at the discrepancy report, you saw that discrepancy 1 listed net6439 as a node with an extra layout device. Extra layout devices often indicate a misconnection in the layout, so you want to display this node in the graphical user interface window.

To display discrepancy 1, the node with extra layout devices, type *1* here.



2. Click on *Add*.

The node with extra layout devices, net6439, appears in the graphical user interface window. This node is also highlighted in the netlist window.



Because extra layout devices indicate a misconnection in the layout, you want to display any extra layout devices in the graphical user interface window.

Highlighting Unmatched Devices in the Dracula Graphical User Interface Window

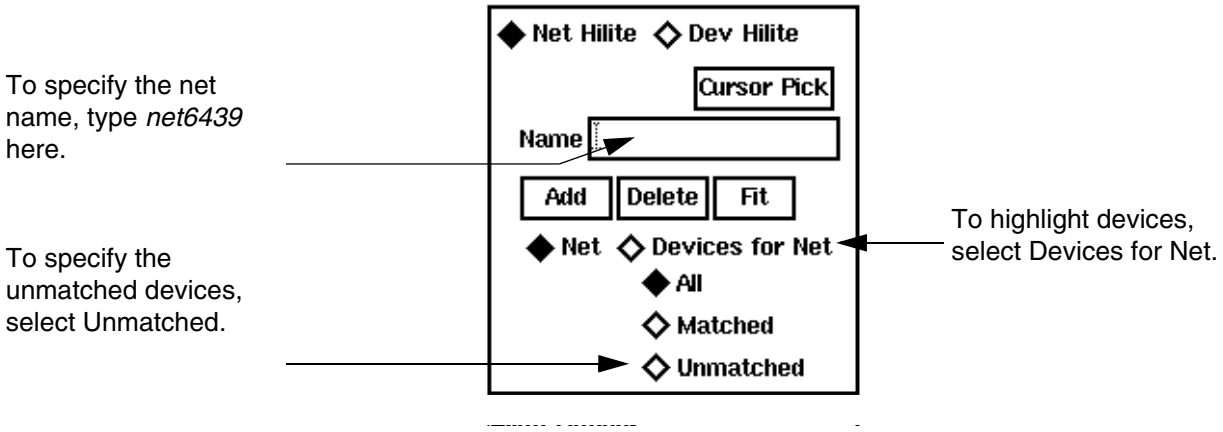
In the discrepancy report, you saw the following discrepancies:

- A node with an extra layout device
- A node with an extra schematic device
- Unmatched devices

You've already highlighted the node with an extra layout device. Now you will highlight the unmatched device for that node.

1. Select *Net Hilite* from the View LVS form.

2. To highlight the unmatched devices on net6439, set the following options.:



3. Click left on *Add*.

The display for net6439 changes color and the unmatched devices are highlighted in white. The unmatched devices aren't matched in the netlist, so they aren't highlighted in the netlist window.

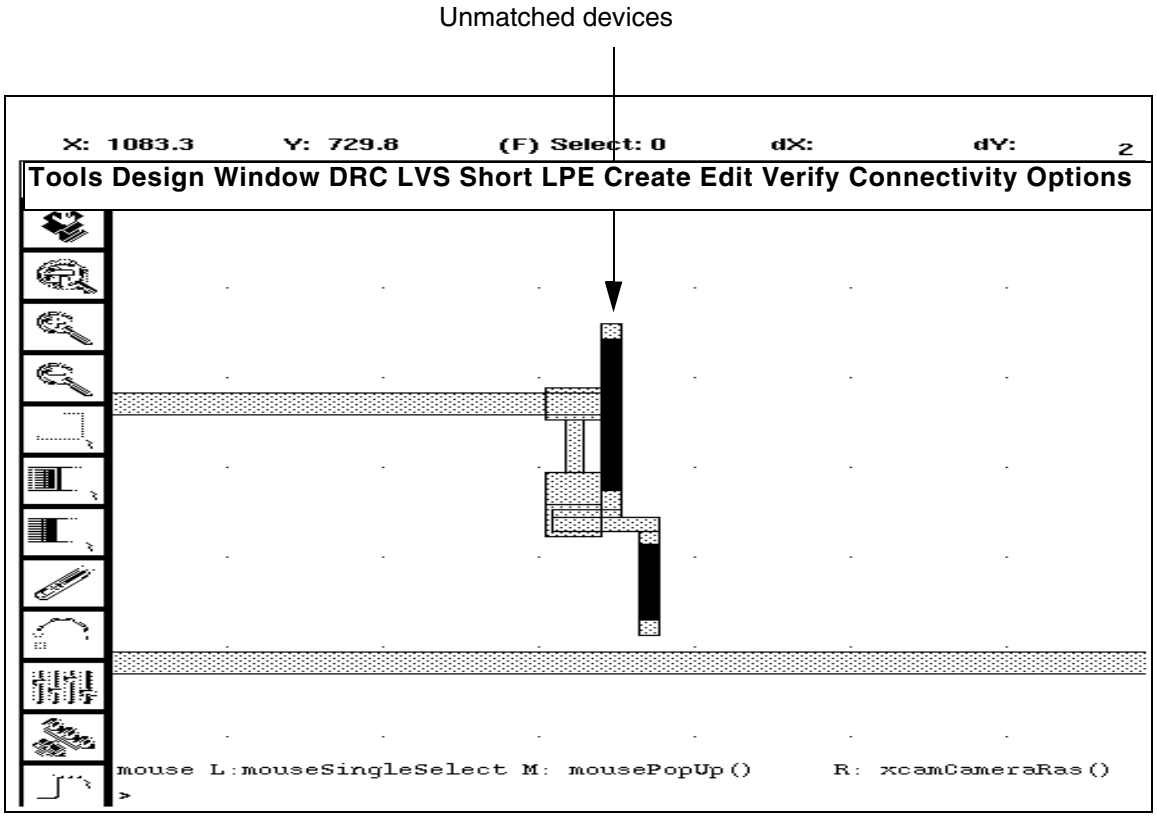
Zooming In on the Unmatched Devices

To zoom in on the area around the unmatched devices in the graphical user interface window, follow these steps.

1. To view the unmatched devices, do one of the following:

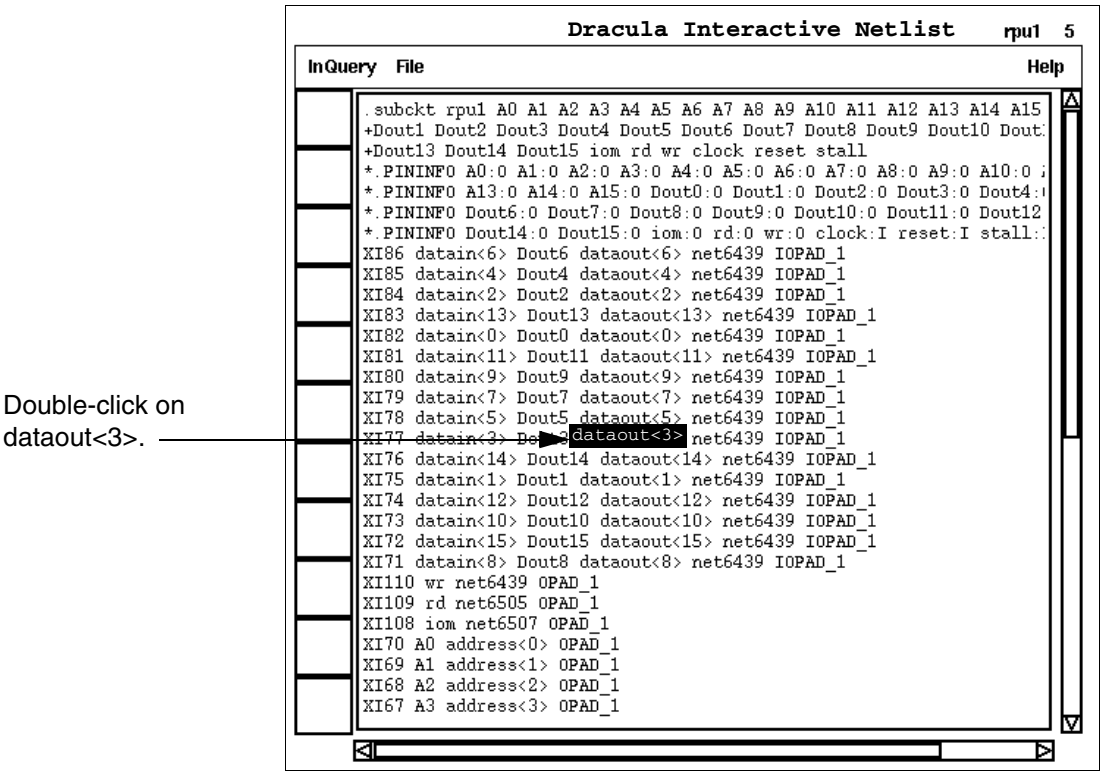
- ❑ To zoom in around the unmatched devices, from the View LVS form, select *Fit Current Hilite*.
- ❑ To get a larger view, use the Zoom Out icon.

The graphical user interface window zooms in on the area you specified. The enlarged image you see in this window makes it easier to see where the short occurs.



Now you can cross-probe the dataout<3> net from the netlist window to display it in the graphical user interface window.

2. To select dataout<3>, double click left on it in the netlist window.



3. To identify where dataout<3> appears in the netlist window, select *Dracula Interactive – Display Net*.

The dataout<3> net is highlighted in a different color in all of the locations where it appears in the netlist window. The layout net matched to dataout<3> is highlighted in the graphical user interface window.

When you select a matched net in the netlist window, the graphical user interface window displays the layout node it matches. When you select a matched net in the graphical user interface window, the netlist window displays the schematic node it matches.

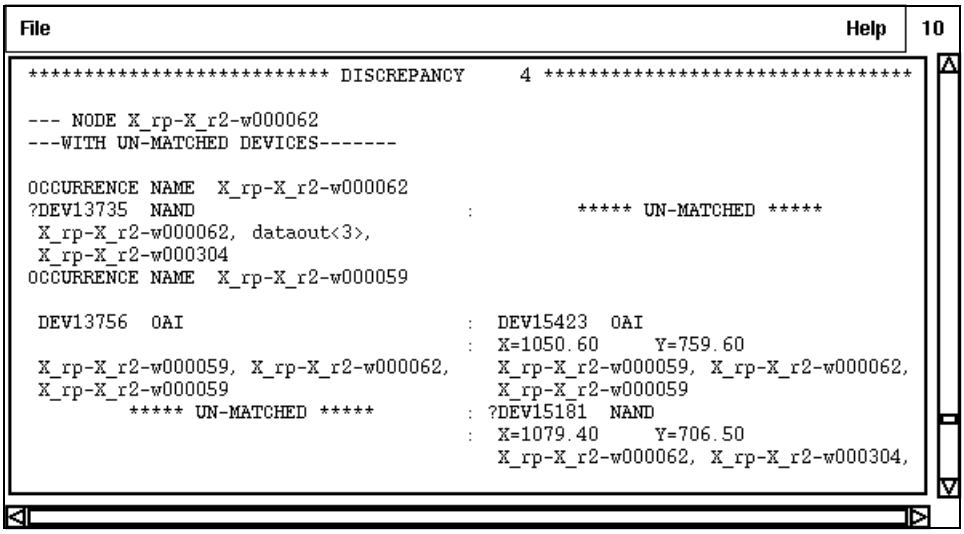
Identifying the Unmatched Devices

Now that you've highlighted the two nets with errors you'll use other discrepancies to help identify the problem. Typically, it is easier to analyze LVS discrepancies for nets that have only a few devices.

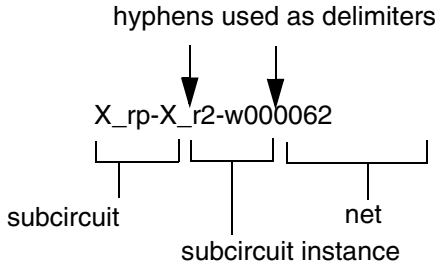
1. Reopen the discrepancy report window.

2. Scroll down in the discrepancy report window to Discrepancy 4, node X_rp-X_r2-w000062 with un-matched devices.

This discrepancy shows the unmatched schematic device and the unmatched layout device.



You can use the hierarchy information in the report to navigate through the netlist. Hierarchical names are listed as follows:



3. Iconify the discrepancy report window.

4. Double-click left on X_rp in the netlist window.

X_rp is the top-level instance. The dataout<3> and net6439 nets are inputs to X_rp.

5. To descend one level in the netlist hierarchy, select *Show Subcircuit* from the graphical user interface menu in the netlist window.

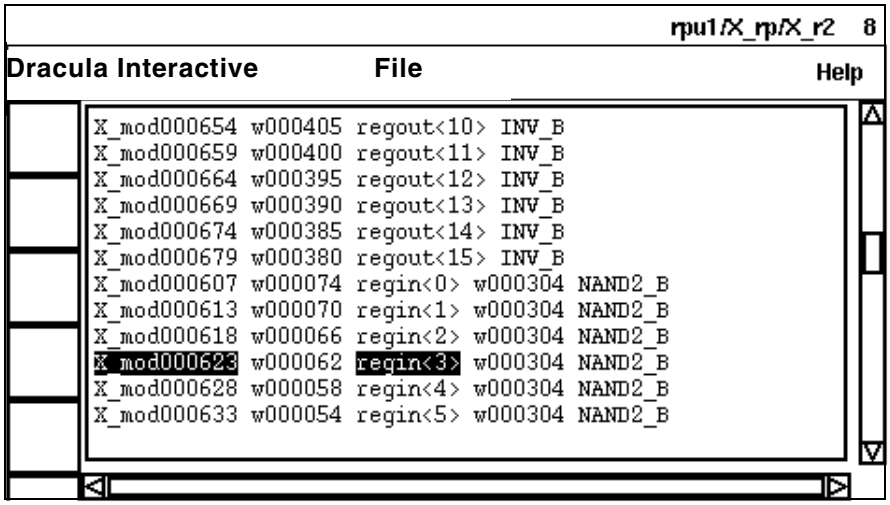
The netlist window displays the next lowest level. The subcircuit nets connected to dataout<3> and net6439 are highlighted in the same colors as in the parent.

6. In the netlist window scroll down to X_r2.

X_r2 is highlighted in the same color as the dataout<3> net, which is one of its inputs.

7. Double click left on X_r2 in the netlist window.
8. From the graphical user interface menu in the netlist window, select *Show Subcircuit*.

   The NAND2_B device X_mod000623 is connected to nets w000062 and regin<3>. The regin<3> node is highlighted in the same color as dataout<3>, indicating that it is the same net as dataout<3>.



The discrepancy report lists the connections to a device in “output input input” order. In the discrepancy report, discrepancy 4 shows the unmatched layout device, DEV13735, connected to the following nets:

- X_rp-X_r2-w000062
- dataout<3>
- X_rp-X_r2-w000304

The output of DEV15181, the unmatched layout device, is connected to the same net, X_rp-X_r2-w000062, as the output of the unmatched schematic device. Its inputs are connected to

- X_rp-X_r2-w000304, the same net as one of the inputs to the unmatched schematic device
- net6439

Based on the discrepancy report you can conclude that DEV15181 should be connected to dataout<3> instead of net6439 and should match schematic device X_mod000623.

Quitting the Tutorial

At this point, you might want to continue working with graphical user interface. If you'd like to continue, you can leave the windows you used for the tutorial on the screen or you can close them. If you want to close the windows, go to the "Closing Windows" section.

When you are ready to quit the Cadence software, go to the "Quitting Cadence Software" section.

Summary

In this chapter, you learned how to use graphical user interface to analyze LVS errors. Specifically, you learned to

- Start Dracula graphical user interface
- Interpret the discrepancy report
- Check netlist connectivity
- Display and highlight LVS errors
- Quit the Cadence software

5

Displaying Parasitic Resistance and Capacitance

This chapter discusses parasitic resistance and capacitance in the Dracula® graphical user interface product. The chapter's main focus is in the following areas:

About LPE/PRE

When you run an LPE or PRE job, Dracula creates files for each of the types of capacitance and resistance extracted. In this chapter, you'll learn how to use graphical user interface to display Dracula parasitic extraction data. You'll use the graphical user interface commands to locate and examine parasitics in these files automatically.

The procedures in this tutorial use Dracula output data files, not the original layout data. Because Dracula output data is in a different format, data you'll see in this tutorial won't look like your original layout data.

In this chapter, you will learn how to

- Start the Dracula graphical user interface's LPE
- List nets with the highest parasitic capacitance
- Display nets of your choice
- Display nets with parasitic area and fringe capacitance

- Display nets with parasitic coupling capacitance
- Display parts of a parasitic capacitor
- List nets with the highest parasitic resistance
- Display parts of a parasitic resistor

Starting LPE

Before you start this section, you need to open an empty cell, which is explained in "About the Dracula Graphical User Interface" section in Chapter 1. Once you have an empty cell open, you can go on with this section.

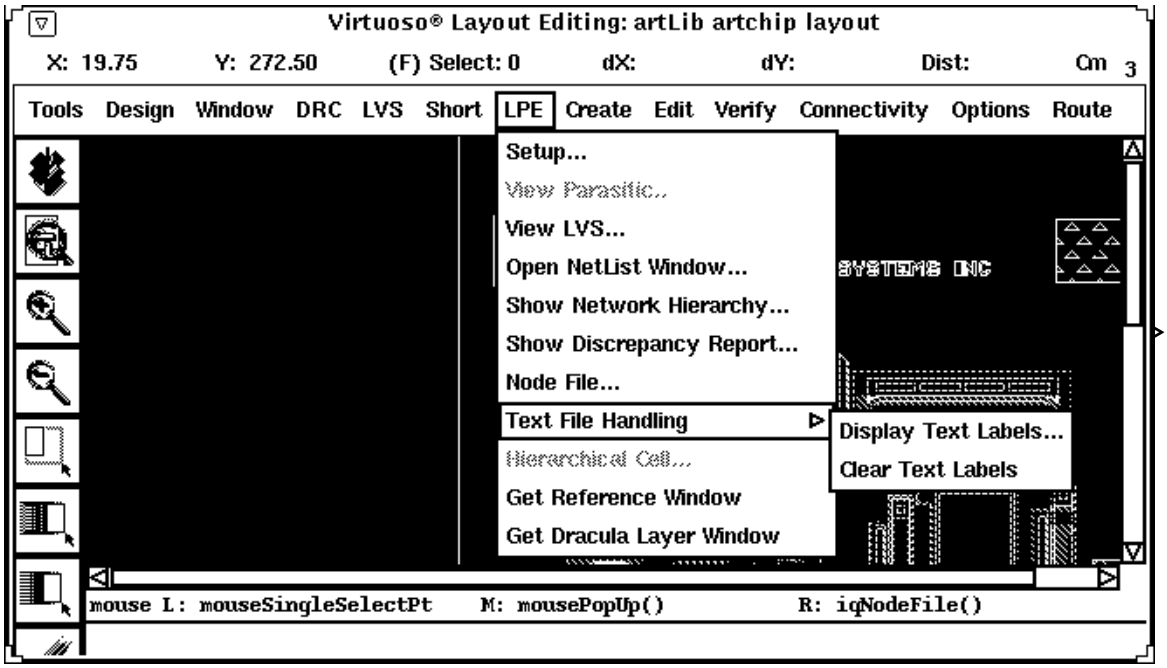
After you open the empty cell, the graphical user interface window appears. The graphical user interface window and the Layer Selection Window (LSW) are the only two windows open. You can ignore the LSW for now.

Looking at Dracula RC Data in the Dracula Graphical User Interface

Now you'll tell the software where to find your Dracula RC data files.

1. Select *Tools – Dracula Interactive* if necessary.

2. To display the LPE commands, click left on the LPE menu.

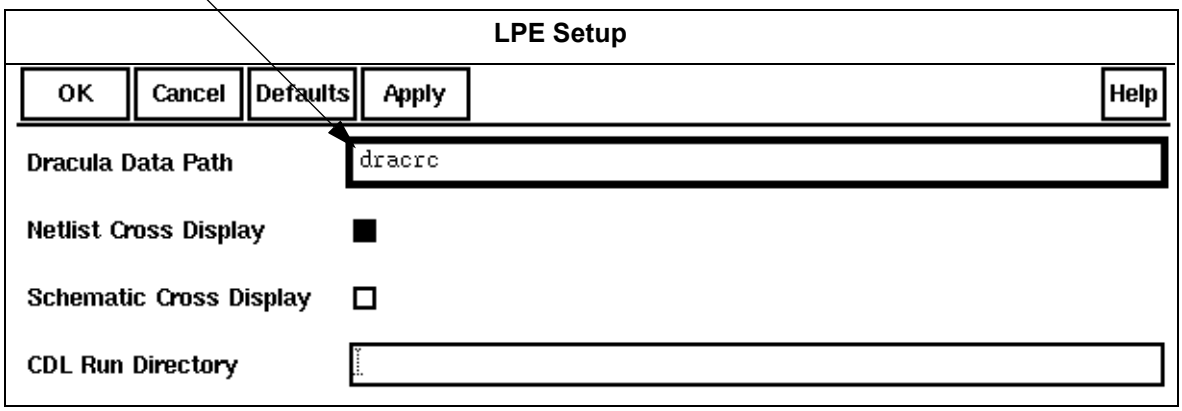


Except for the *Setup* submenu, you cannot enable any of the other submenus in the LPE pull-down menu until you perform step 3.

3. Select *Setup* from the LPE menu.

The LPE Setup form appears.

Type *dracrc* here.



4. To tell the graphical user interface where to find your Dracula LPE/PRE files, in the Dracula Data Path field, type *dracrc*.

If you did not start the software in the directory where the Dracula LPE/PRE files are located, you must type the complete path to the files.

5. Click left on *OK*.

Note: The form takes a few seconds to disappear.

The RCV window appears on the right.

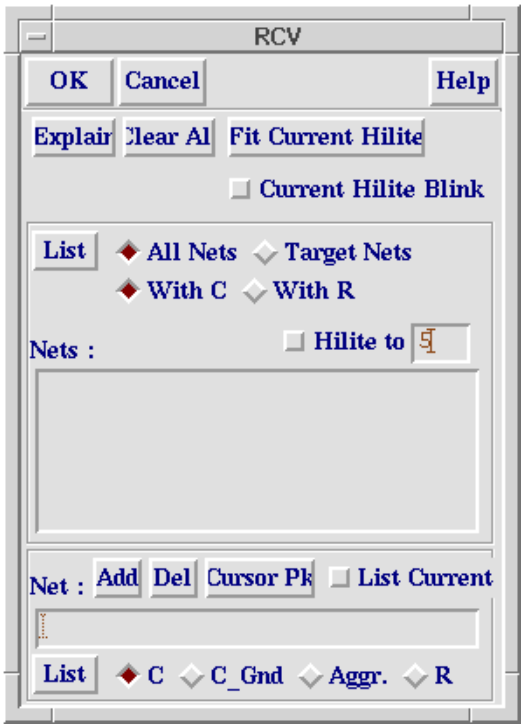
Now the graphical user interface knows where to find your Dracula LPE/PRE RC data files. Next you'll list nets with parasitic capacitance.

Viewing Parasitic Capacitance

In this section, you'll list nets with parasitic capacitance, select a net, and view its capacitance components.

Listing Nets with the Highest Parasitic Capacitance

1. To list nets with the highest parasitic capacitance, from the RCV form, select *All Nets* and *With C*.



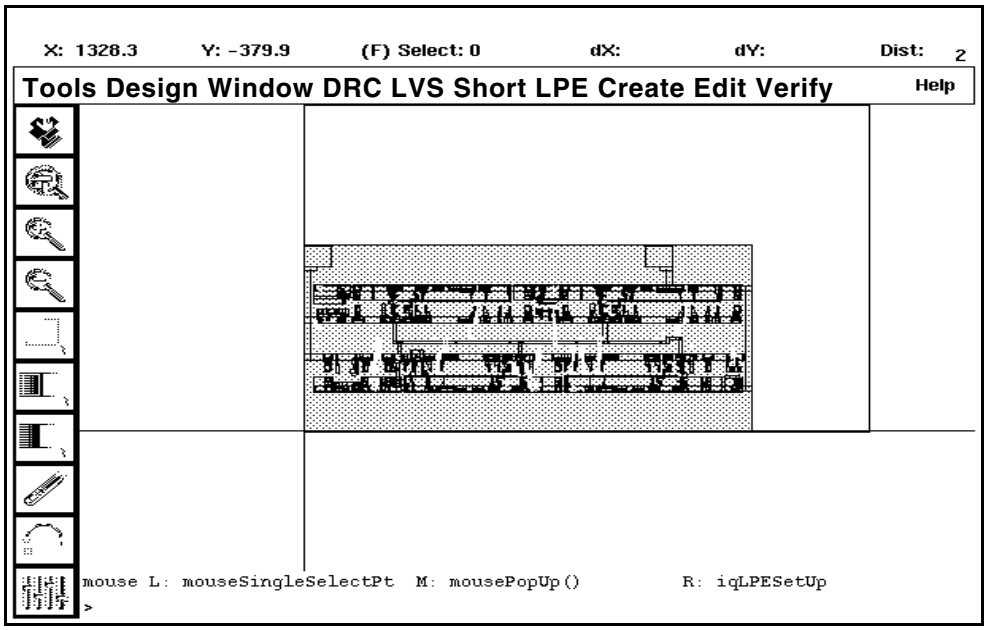
2. Click the top *List* button.

   The Nets list box lists original nets in order from highest to lowest capacitance. You can scroll through the list of nets with the scroll bar.

3. To highlight the nets with the highest capacitance, select *Hilite top*.

   The five nets with the highest capacitance are highlighted in the layout window, because *Hilite top* is set to 5 by default. You can highlight up to nine nets with the highest capacitance by filling in a number of your choice, from 1 to 9, in the *Hilite top* field.

4. To zoom in on the highlighted nets, click *Fit Current Hilite*.



   Now that you've listed all the nets with the highest capacitance, you're ready to list specific nets to view.

Listing Specific Nets

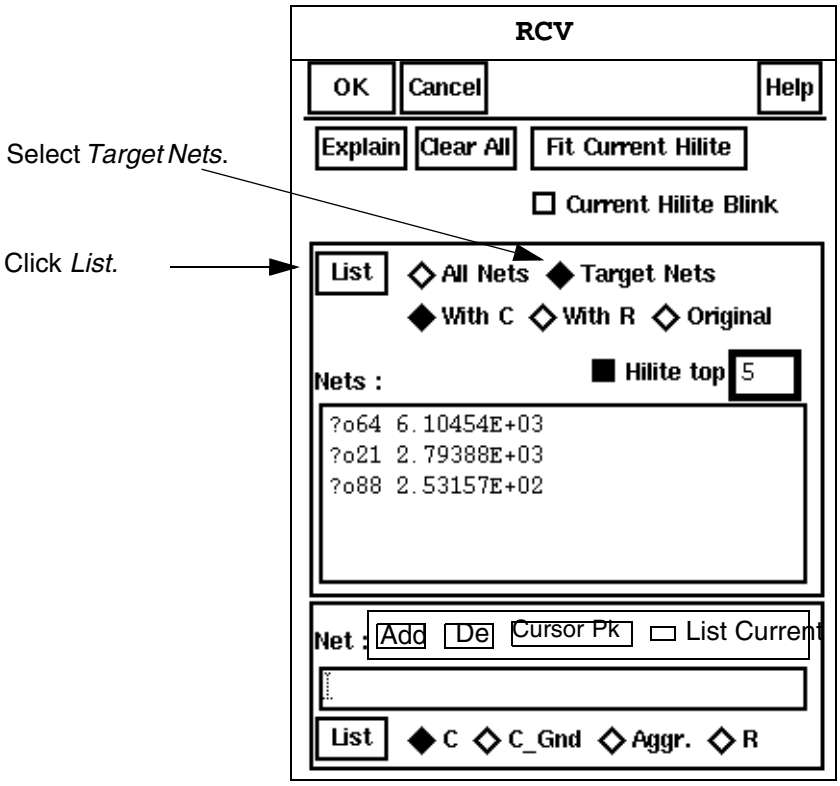
When you only want to look at certain nets, you can list the nets you want to display in an ASCII text file.

1. With a text editor such as vi, emacs, or textedit, create a text file in the *dracrc* subdirectory that contains the following lines:

```
?o64
?o21
?o88
```

2. Save the text file as *TARGET.DAT*.
3. In the RCV form, select *Target Nets,* then click the top *List* button.

   The nets you listed in *TARGET.DAT* are displayed in the Nets list box.



4. In the RCV form, experiment by selecting *With C* and *List*; *With R* and *List;* and *Original* and *List*.

You just listed nets that can be displayed. Next you'll select a single parasitic capacitance net to display.

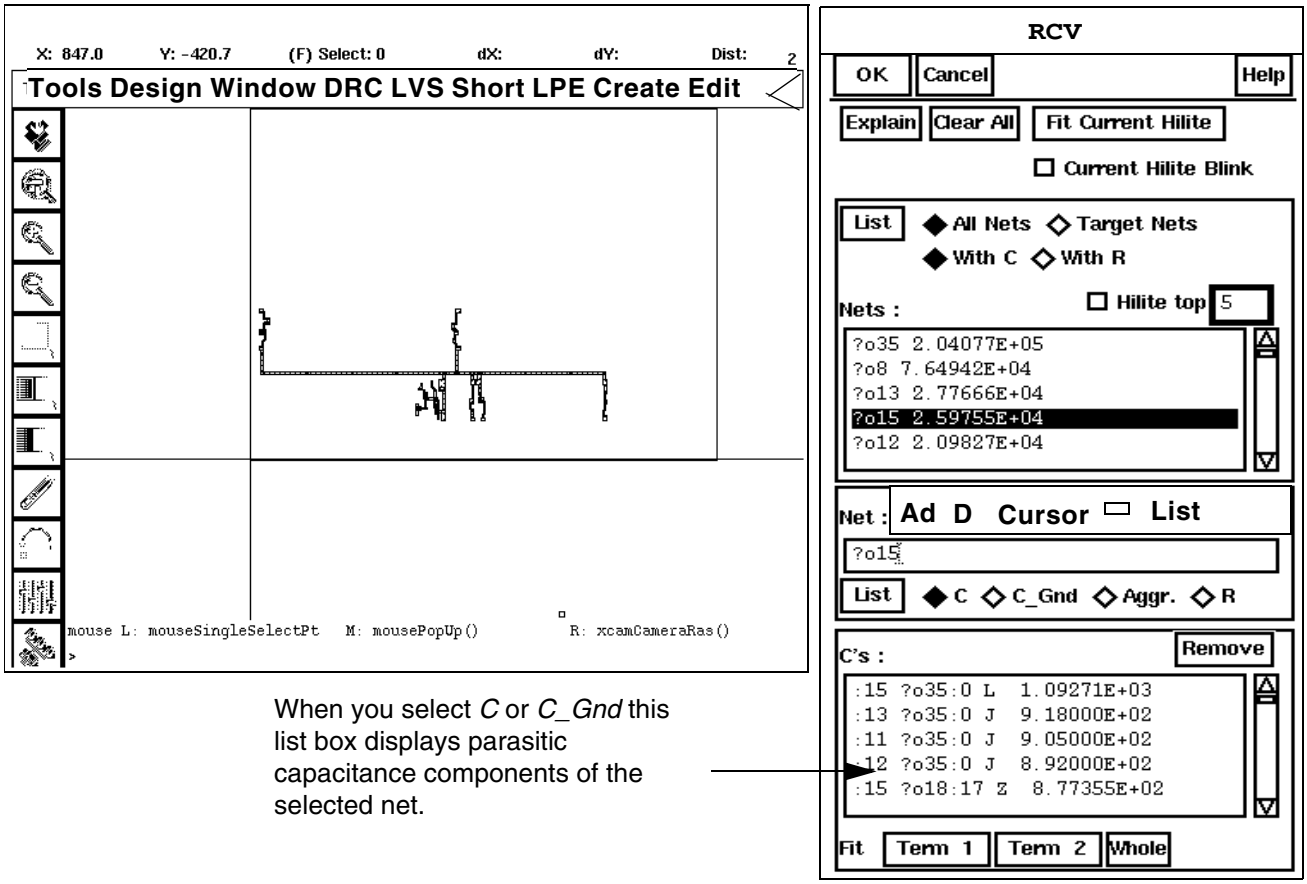
Displaying Capacitance Components for a Net

To display capacitance components, you select a net from the Nets list box. You will then list the capacitance components for this net and highlight them.

1. In the RCV form, click *Clear All*.
2. Select *All Nets* and *With C*, then click *List*.

3. To select a net with components to view, click on the fourth net listed in the Nets list box, original net 15 (?o15).

The net is highlighted in the graphical user interface layout window. Because *C* is selected, a C's list box opens to list the components of parasitic capacitance to the target net.



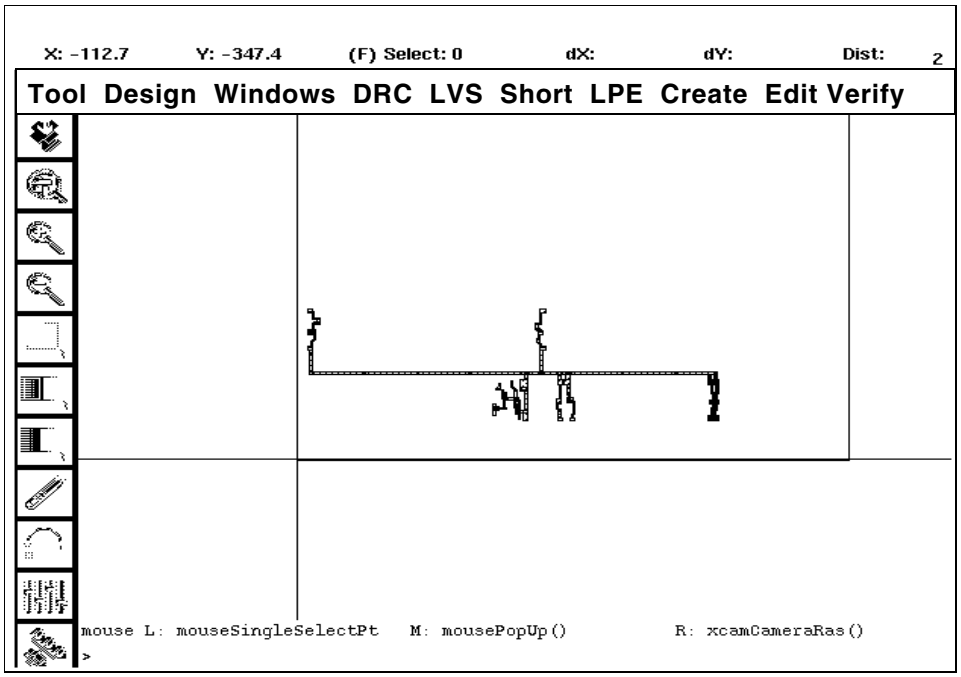
When you select *C* or *C_Gnd* this list box displays parasitic capacitance components of the selected net.

4. To highlight an area or fringe component of parasitic capacitance to the target net

- ❑ Be sure *C* is selected.
- ❑ In the C's list box, scroll down and click on the following component:

  `:20 ?o18:11 Z 7.59495E+02`

The current component is highlighted in white.

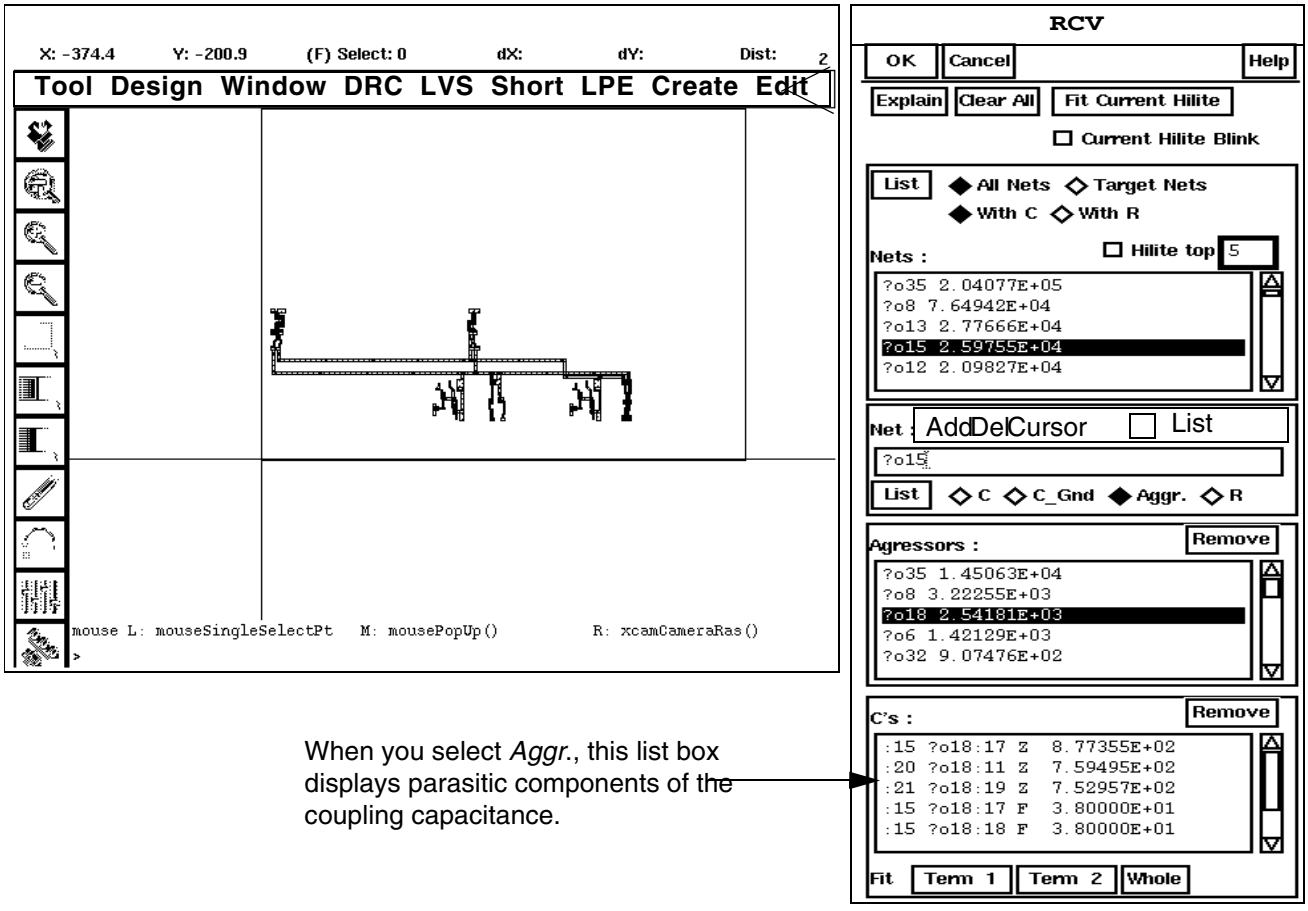


5. To unhighlight the component, click Remove next to the C's list box.
6. To highlight a component of capacitance to ground for the target net
   - Select *C_Gnd*.
   - In the C's list box, scroll to the top and click on the following component:

     :16 ?o8:17 K 2.59250E+02
7. To highlight a component of coupling capacitance for the target net
   - Select *Aggr.*
   - Click on the following component in the Aggressors list box.

     ?o18 2.54181E+03

A corresponding C's list box lists the different capacitance components for this coupling capacitance.



When you select *Aggr.*, this list box displays parasitic components of the coupling capacitance.

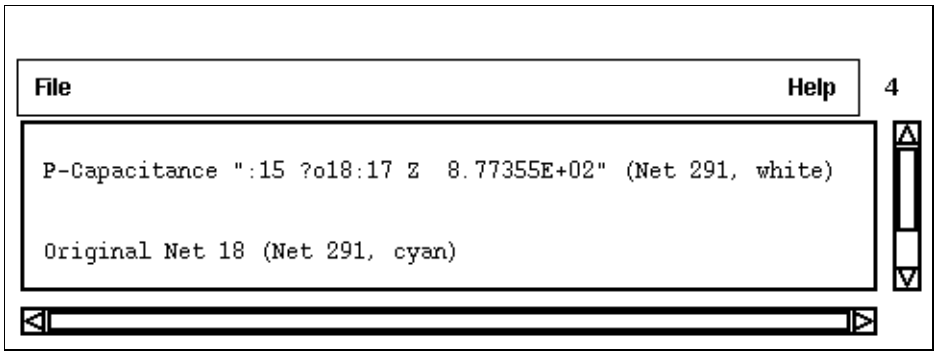
Now that you are able to view the different parasitic capacitance components for a selected net, you might want to find out what a component is and how it was extracted.

Getting Information About Capacitance Components

1. To list the extraction rule that created a capacitance component, in the RCV form
   - Click on the following component in the C's list box under the Aggressors list box:

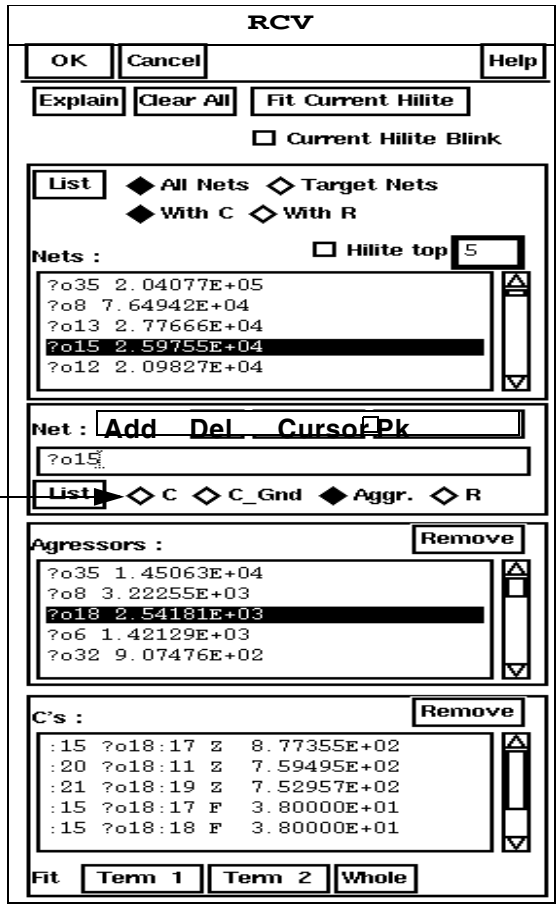
     :15 ?o18:17 Z 8.77355E+02
   - Click *Explain*.

- Click on the highlighted shape in the layout window.



If you need room on the screen, you can make this window smaller or close it.

2. List the area and fringe components of net ?o15 by clicking the *C* button.

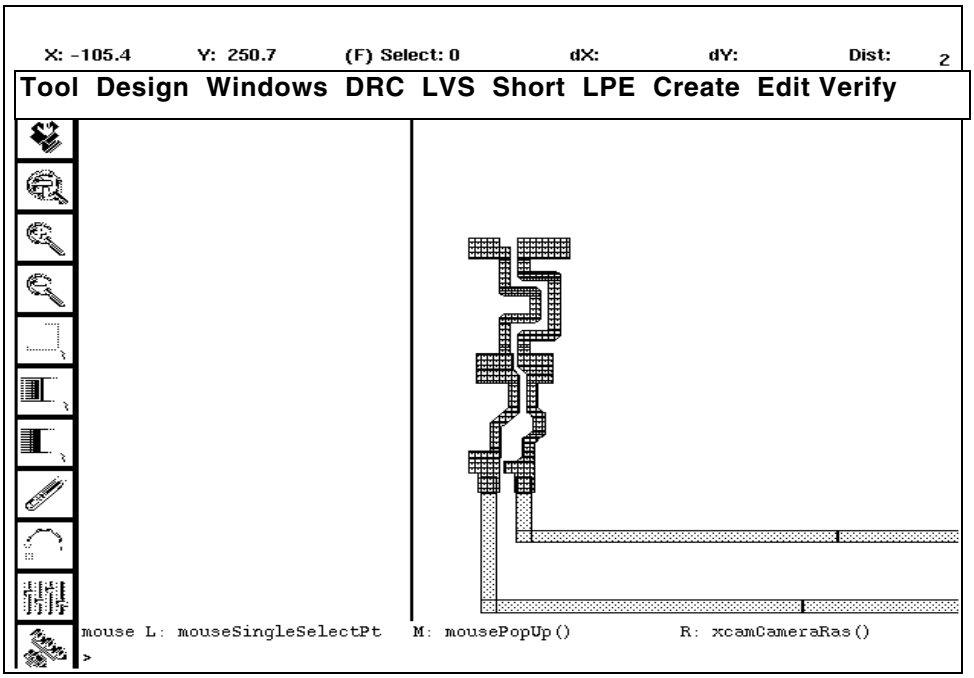


Click the *C* button.

3. In the C's list box, scroll down to the following component and click on it.

```
:21 ?o18:19 Z 7.52957E+02
```

4. To see how the parts of a parasitic capacitor were extracted, use *Term 1*, *Term 2*, and *Whole* to highlight the terminals of the capacitor listed in the C's list box.



5. Click *Clear All*.

   When you're finished seeing how Dracula extracted parasitic capacitors, you're ready to display parasitic resistance.

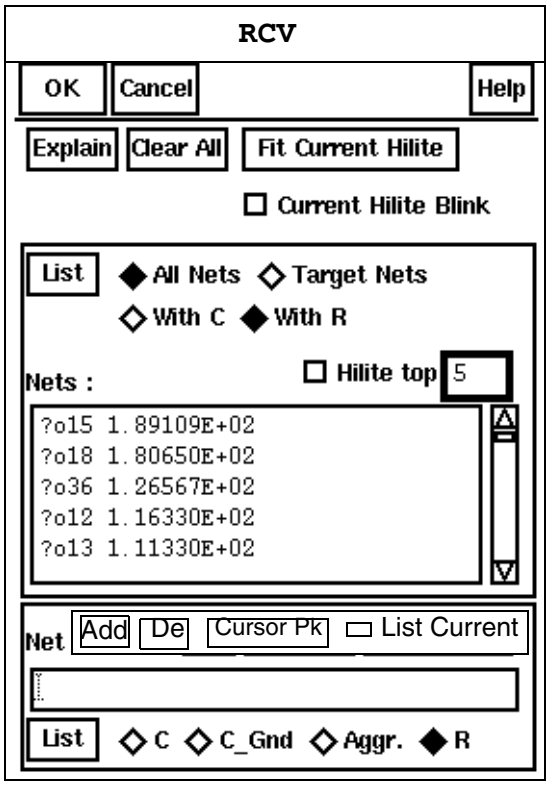
Viewing Parasitic Resistance

Now that you've looked at nets with parasitic capacitance, you're ready to look at nets with parasitic resistance. You'll follow very similar procedures.

Listing Nets with the Highest Parasitic Resistance

1. To list nets with the highest parasitic resistance, from the RCV form, select *All Nets* and *With R*.

2. Click *List*.

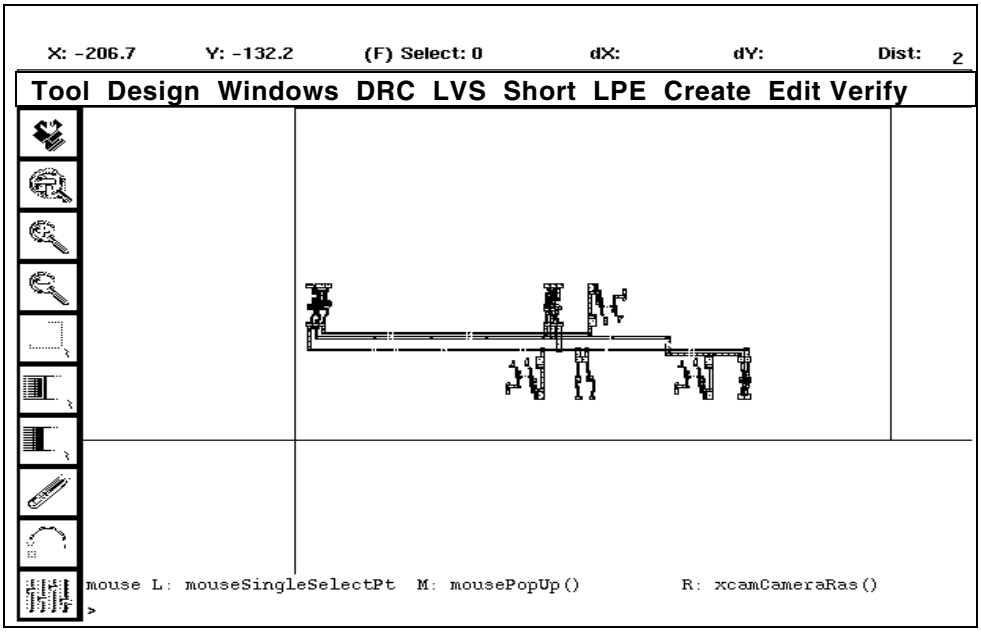


The Nets list box lists original nets in order from highest to lowest resistance. You can scroll through the list of nets with the scroll bar.

3. To highlight the nets with the highest resistance, select *Hilite top*.

The five nets with the highest resistance are highlighted in the layout window, because *Hilite top* is set to 5 by default. You can highlight up to nine nets with the highest resistance by filling in a number of your choice, from 1 to 9, in the *Hilite top* field.

4. To zoom in on the nets, click *Fit Current Hilite*.

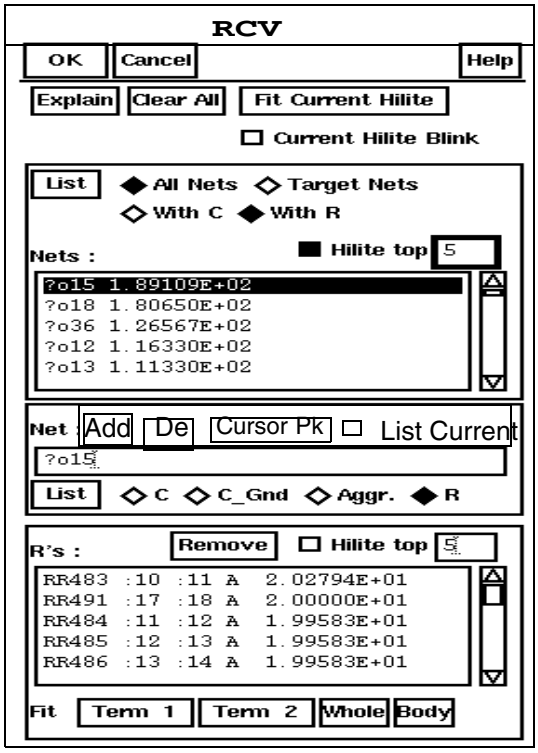


5. Click *Clear All*.

Displaying Resistance Components for a Net

1. Click on ?o15, the first net listed in the Nets list box.

The net is highlighted in the graphical user interface layout window. Because *R* is selected, an R's list box opens to list the components of parasitic resistance to the target net.



2. To highlight a component of parasitic resistance to the target net, click on RR483, the first item in the R's list box.

   The current component is highlighted in white.

3. To remove the highlighting, click the Remove button for the R's list box.

Getting Information About Resistance Components

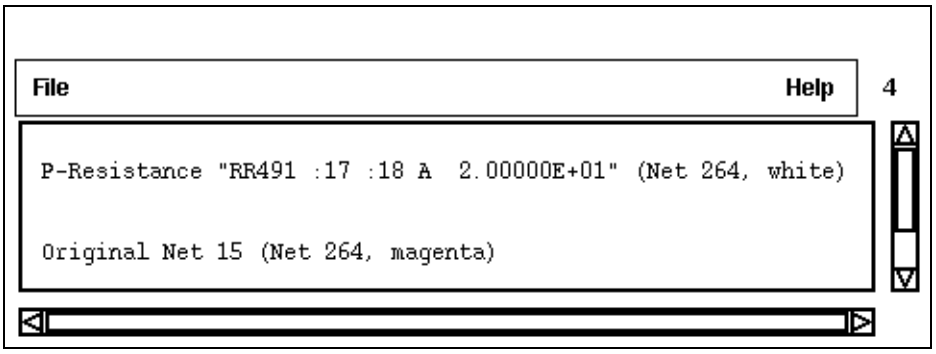
1. Click on RR491, the second component in the R's list box.

   The first is unhighlighted, and the second is now highlighted in white.

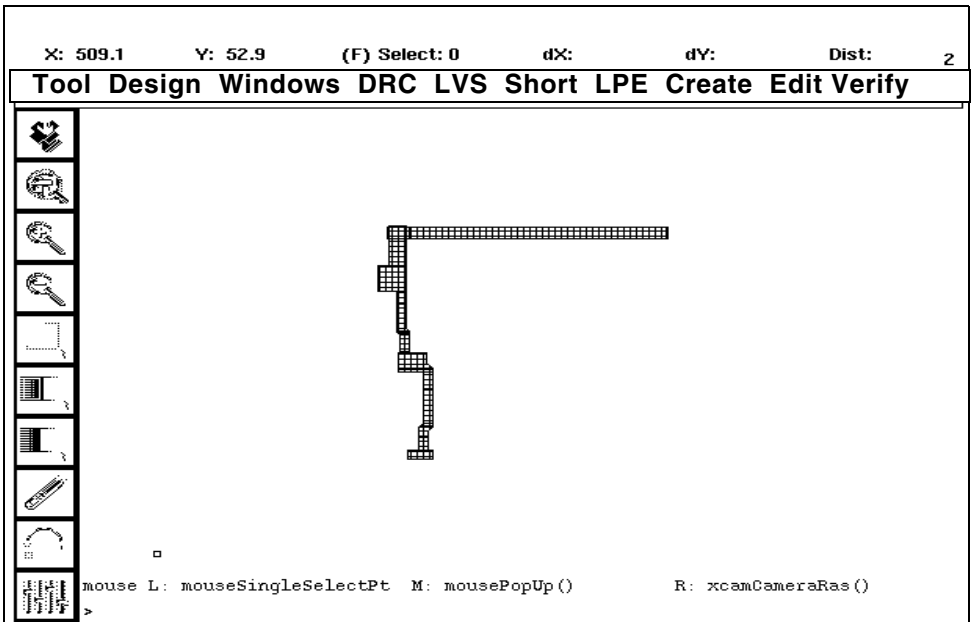
2. To list the extraction rule that created a resistance component, in the RCV form

   - Click *Explain*.

- Click on the highlighted shape in the layout window.



3. To view the parts of a parasitic resistor, use *Term 1*, *Term 2*, *Body,* and *Whole* to highlight the terminals of the resistor listed in the R's list box.



Quitting the Tutorial

At this point, you might want to continue working with the graphical user interface. If you'd like to continue, you can leave the windows you used for the tutorial on the screen, or you can close them. If you want to close the windows, go to the "Closing Windows" section.

When you are ready to quit the Cadence software, go to the "Quitting Cadence Software" section.

Summary

You've now finished the *Dracula Graphical User Interface Tutorial.*

In this chapter, you learned how to use graphical user interface to list and display parasitic capacitance and resistance in a design. Specifically, you learned to

- Start the graphical user interface's LPE
- List nets with the highest parasitic capacitance
- Display only the nets listed in a text file
- Display nets with parasitic area, fringe, and coupling capacitance components
- Get information about capacitance components
- List nets with the highest parasitic resistance
- Get information about resistance components
- Quit the tutorial

Index

A

B

C

D

E

F

H

I

K

L

M

N

U

V

W