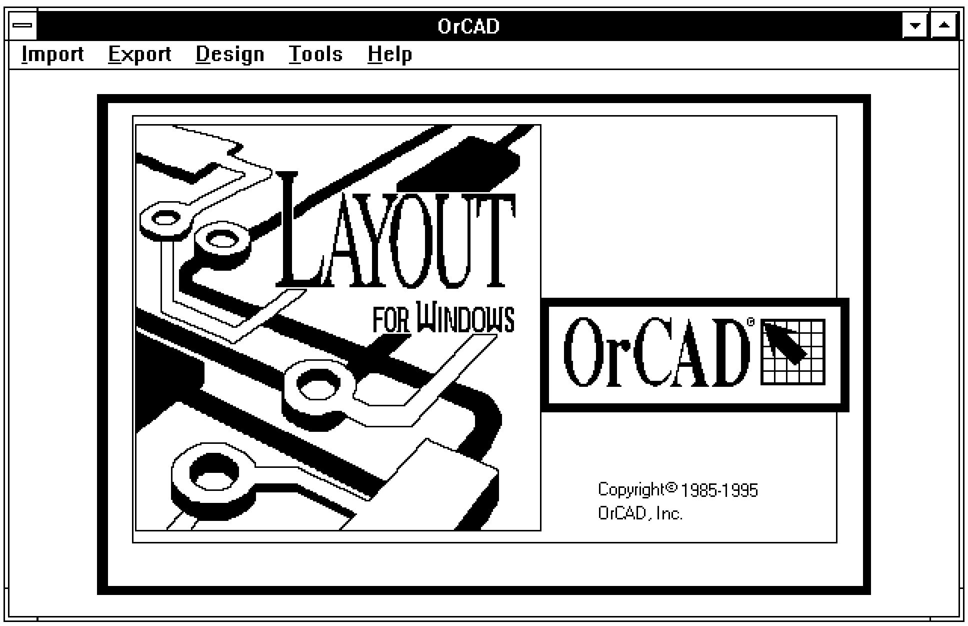
OrCAD Layout for Windows Demo Guide



OrCAD®

9300 S.W. Nimbus Ave.
Beaverton, OR 97008• USA

OrCAD DIRECT	(800) 671-9505
Administration	(503) 671-9500
Fax	(503) 671-9501
Technical support	(503) 671-9400
Bulletin board system	(503) 671-9401
Email	info@orcad.com
World Wide Web	http://www.orcad.com

Contents

Chapter 1
OrCAD Layout for Windows demo overview

Go to the table of contents.

Introduction

Welcome to the OrCAD Layout for Windows Demo, an interactive tutorial for learning and evaluating OrCAD Layout™ for Windows® software.

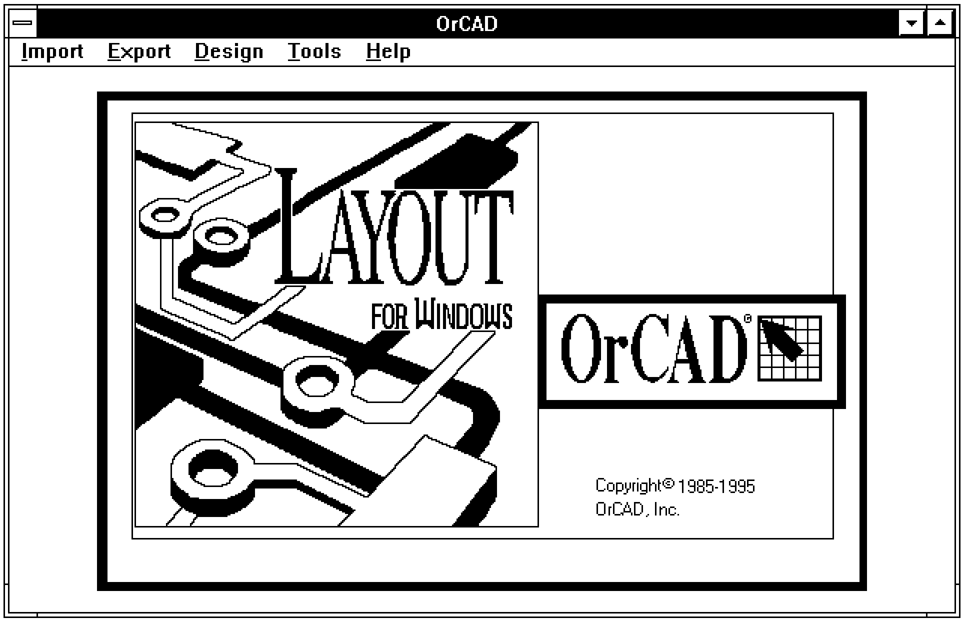
OrCAD Layout™ for Windows is a family of Windows 3.1, Windows NT, and Windows 95 printed circuit board (PCB) design tools. Each package provides a powerful, easy-to-use solution for PCB design. The OrCAD Layout for Windows product family was developed from Massteck's award-winning MaxEDA PCB Layout System.

The OrCAD Layout for Windows Demo uses many of OrCAD Layout for Windows' innovative features, but is intended only as a product overview. The software included with this package is a demonstration version; you cannot save a board design or generate photoplot files from the board files provided. The files are structured to guide you through each step in the Layout design flow.

The *OrCAD Layout for Windows Demo Guide* tells you which design file to open for each section. Every page of the guide includes pictures, explanations, and instructions that correspond to each activity.

Online help is also provided with the OrCAD Layout for Windows Demo. A complete product documentation set is provided with your OrCAD Layout for Windows purchase.

OrCAD Layout for Windows—product family



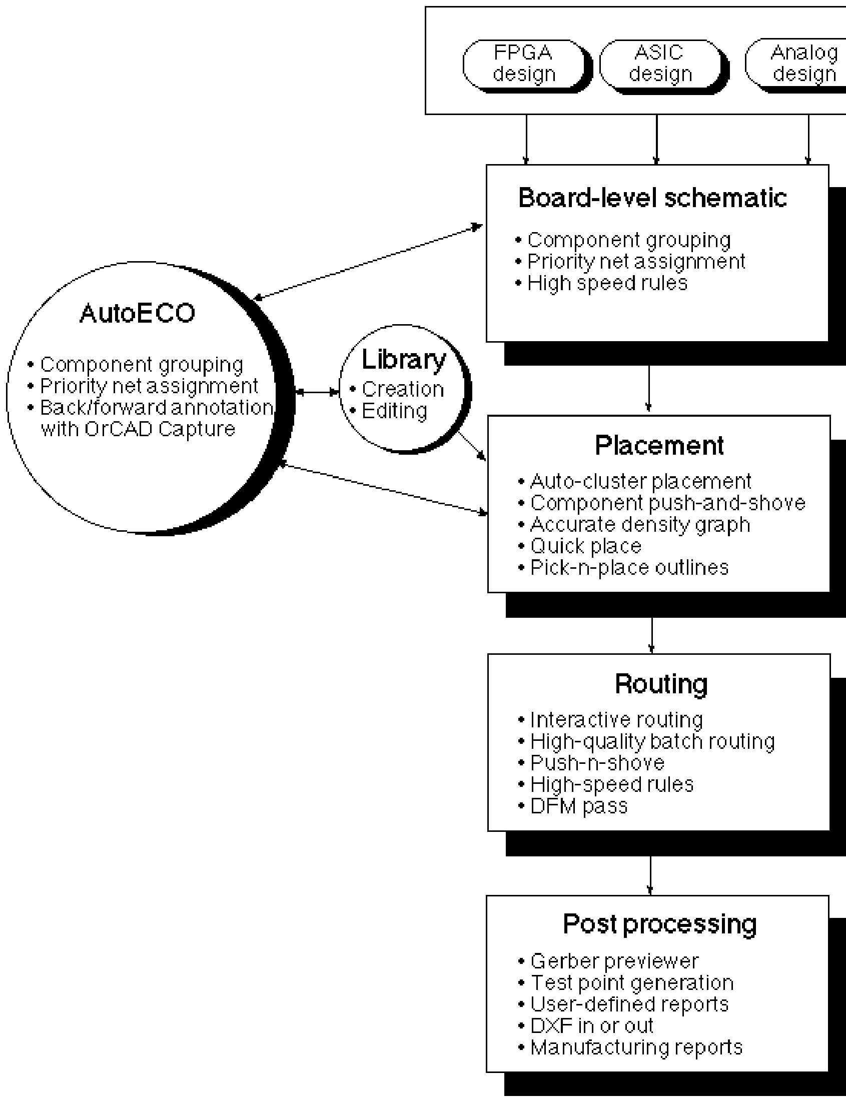
OrCAD Layout for Windows product family

- OrCAD Layout Plus for Windows
- OrCAD Layout for Windows
- OrCAD Layout Ltd. for Windows

OrCAD Layout for Windows—product family

OrCAD Layout product family features	*Layout Plus*	*Layout*	*Layout Ltd.*
Placement			
Cluster placement	•		
Density graph	•		
Manual placement	•	•	•
Matrix placement	•		
Shove component	•		
Routing			
16-layer batch router	•	•	
Add/delete connection interactively	•	•	•
AutoDFM	•	•	
AutoPath	•		
Replicate connections	•	•	•
Interactive push-n-shove	•	•	
Online DRC	•	•	•
Single-layer autorouter	•		
Post processing			
23 standard reports	•	•	•
80+ user-configurable reports	•	•	•
Automatic aperture generation	•	•	•
Gerber previewer	•	•	•
Other features			
Auto test point generation	•	•	
AutoECO	•	•	•
Intelligent copper pour	•	•	•
DXF in and out	•	•	•
Standard Windows print driver	•	•	•
1000-Connection/350 Component Limit			•
PCB translators			
PADS, PCAD, Tango, CadStar, OrCAD PCB 386+, PROTEL	•	•	•
Schematic netlist interfaces			
OrCAD SDT, OrCAD Capture, Futurenet, Viewlogic, DATA I/O SCS	•	•	•

The OrCAD Layout for Windows productivity flow



OrCAD Layout for Windows—demo files

Demo design files

Below is a list of the demo design files provided with the OrCAD Layout for Windows Demo. Each file corresponds to one or more activity sets in the demo guide. Open the design files as instructed in the demo guide and perform the corresponding exercises. You can open the files and perform the exercises in any order. Keep in mind that when you exit a file, you cannot save any changes you have made to the design.

OrCAD has provided two sets of design files with the OrCAD Layout for Windows Demo: a board design with two routing layers (these files have a *2* in the file name), and a board design with four routing layers (these files have a *4* in the file name). You need a minimum of 8 MB of RAM to run the design with two routing layers, and a minimum of 16 MB of RAM to run the design with four routing layers.

DSTART2.MAX or DSTART4.MAX

These boards contain components and nets that are ready for interactive or batch placement.

DPLACE2.MAX or DPLACE4.MAX

These boards are partially placed. You may use them for batch placement or routing.

DROUTE2.MAX or DROUTE4.MAX

These boards are placed and ready for via dispersion and interactive or batch routing.

FINISH2.MAX or FINISH4.MAX

These boards are completely routed and can be used for post processing.

RENAME2.MAX and RENAME4.MAX

These boards can be used for performing back annotation to Capture.

Benchmark design files

Below are the results of two benchmarks that OrCAD has run. Use these results to evaluate board size versus routing time. Both of the designs were routed using a Pentium P90 with 32 MB of RAM and running under Windows NT 3.5

Benchmark design files		
Statistics	*Benchmark 1*	*Benchmark 2*
Board area	31.60	232.10
Equivalent IC's (16-pin)	100.70	488.40
Number of route layers	4	4
Percent routed	100%	100%
Connections	770	3932
Number of vias	686	2930
Total route time	1 hour 16 minutes	7 hours 41 minutes

OrCAD Layout for Windows design environment

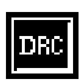
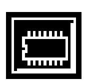
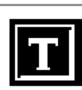
Before you begin running the demonstration, review this section to familiarize yourself with the Layout design environment, including the Design window, the toolbar, and the menus. To view the environment, you can open the demo file DSTART2.MAX or DSTART4.MAX.

Note You need a minimum of 16MB of RAM to run DSTART4.MAX.

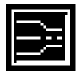
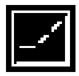
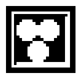
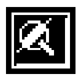
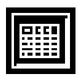
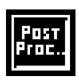
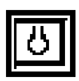
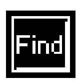
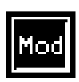
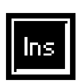
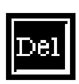
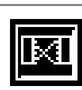
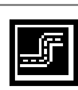
Note If the system prompts you to add a new netlist, choose No.

When you open the file, you will see the Layout Design window. The toolbar runs across the top of the window. The buttons on the toolbar provides access to the most frequently used Layout commands.

Toolbar

Tool	*Name*	*Description*
	Design Rules Check	Enable online design rules checking for interactive routing or component editing.
	Component Tool	Insert, move, edit, or delete components in the design.
	Connection Tool	Insert or delete connections in the design.
	Pin Tool	Insert, move, edit, or delete pins in the design.
	Obstacle Tool	Insert, move, edit, or delete obstacles such as electrical copper or lines.
	Text Tool	Insert, move, edit, or delete text.
	Error Tool	Find and query spacing and design rule violations.

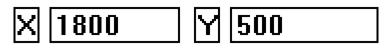
Tools on the toolbar (1 of 2).

Tool	*Name*	*Description*
	Manual Route with Shove	Route interactively using the shove algorithm.
	Manual Route	DRC enabled - Route interactively without using the shove algorithm. DRC disabled - manual routing.
	Initialize Color	Change or check the color of a layer, or make a layer invisible.
	Initialize Query	Change or check an object's attributes.
	Spreadsheet Tool	List the available spreadsheets.
	Post Processing	List the post processing options, including the Post Process spreadsheet.
	Refresh Copper Pour	Recalculate the copper pour based on changes made to the board since the last copper pour operation.
	Find	Find coordinates or components by name.
	Modify	Display an editor that allows you to modify a selected item.
	Insert	Copy a selected item.
	Delete	Delete a selected item.
	Reconnect Enabled	Enable the instantaneous reconnect environment (Layout Plus only).
	Auto Path	Route automatically using the shove algorithm while placing vias manually (Layout Plus only).

Tools on the toolbar (2 of 2).

In addition to the command buttons, the toolbar also provides information about the status of the board.

Current screen coordinates



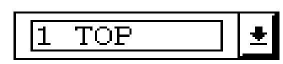
The coordinates that correspond to the current location of the cursor display on the toolbar making it easy to locate and place objects.

Current grid

The current grid spacing displays on the toolbar.

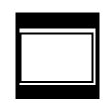
Current layer

The active layer (the layer on which you are currently working) and its color display on the toolbar.



Postage stamp view

A miniature outline of the board, used to determine your current view in relation to the PCB, also displays in the toolbar. You can change the view by pressing the left mouse button and dragging across the area that you want to view.

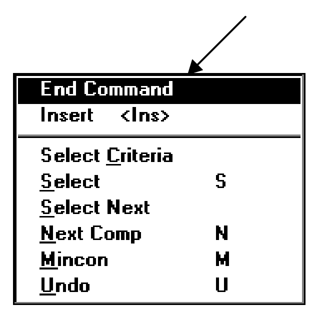


Menus

You can access OrCAD Layout for Windows commands using convenient popup and pull-down menus.

Layout displays unique popup menus for each tool selected.

Popup menu



- **Display popup menus using a two-button mouse.**

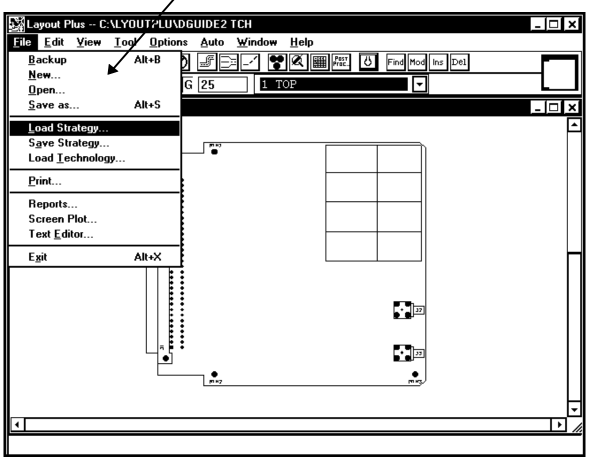
  If you are using a two-button mouse, display popup menus by pressing the right mouse button.

- **Display popup menus using a three-button mouse.**

  Layout assumes that you are using a two-button mouse. You can specify that you are using a three-button mouse by choosing User Preferences from the Options menu and selecting the Massteck Legacy mouse style option.

  If you are using a three-button mouse, display popup menus by pressing the middle mouse button.

Pull-down menu



Go to the next chapter.

Go to the table of contents.

Chapter 2
Footprint library

Go to the table of contents.

Go to the last page of the previous chapter.

Creating footprints

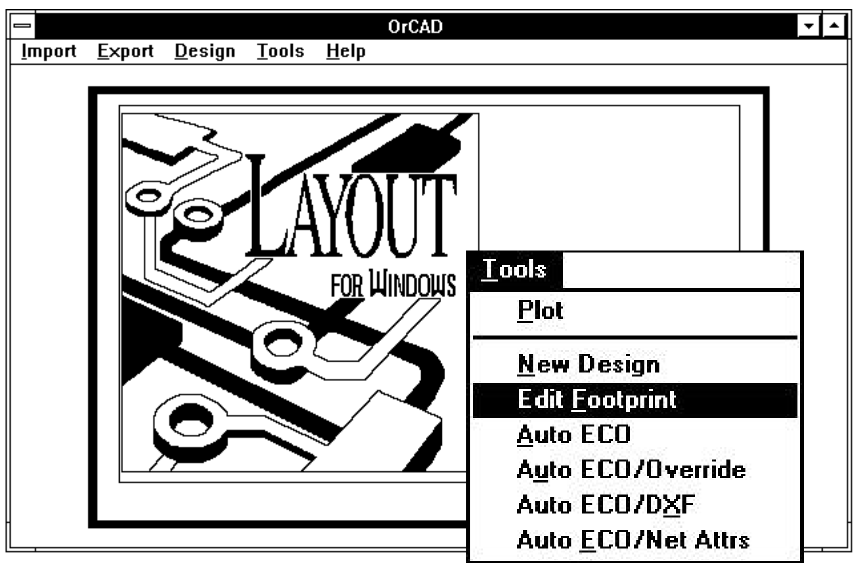
This portion of the demonstration shows you how easy it is to create physical footprints for the components used in the PCB layout process. Footprints consist of three basic types of objects: thrucodes (padstacks), obstacles (silkscreens, assembly drawing data, keep-outs, etc.), and text.

Opening the footprint library

- **Open Layout.**

  You can open Layout by double-clicking on the OrCAD Layout for Windows demo icon.

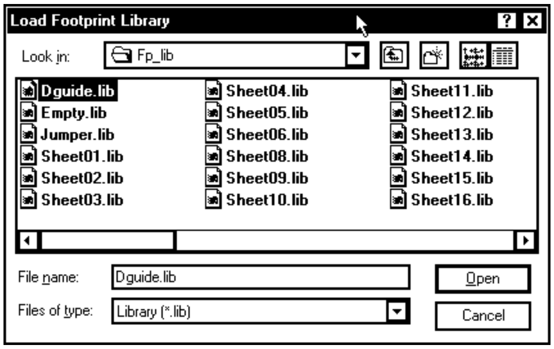
- **In the Layout frame, choose Edit Footprint from the Tools menu.**



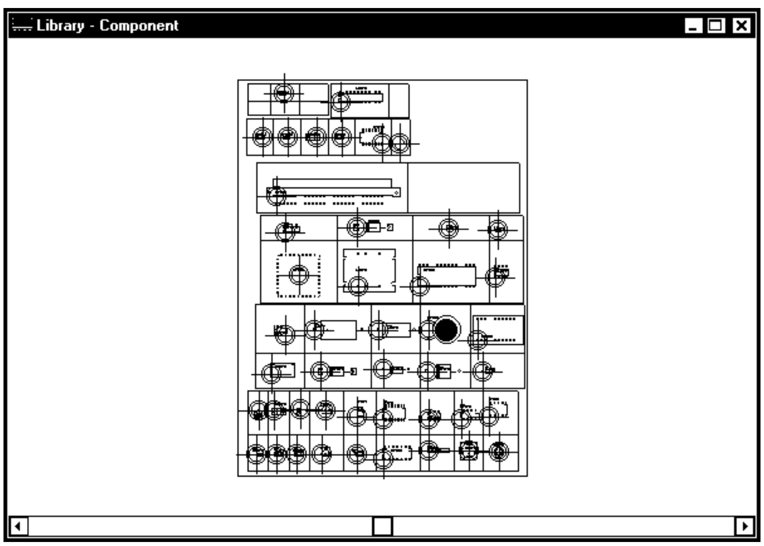
- **Open the library.**

  In the Load Footprint Library dialog box, select the file GUIDE.LIB and choose the OK button.

Note If the system prompts you to add a new netlist, choose No.



Layout's libraries are built on sheets. Each sheet may contain similar footprints, such as resistors, or all the footprints for a specific project.

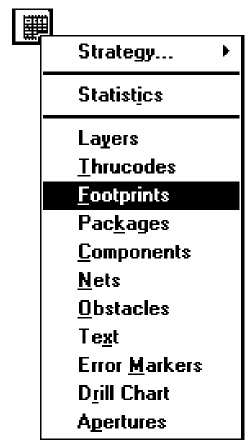


Viewing footprints

There are two ways to view a footprint during the design process. You can look at it graphically in the Library window, or you can look at the data associated with the footprint using the spreadsheets. One method may be more convenient than the other depending on the type of activity you are performing. Typically, when creating footprints or editing their obstacles or text, use the Library window. When editing multiple pad locations or thrucodes, use the spreadsheets.

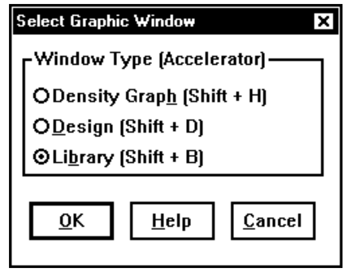
- **View the footprints in the spreadsheets.**

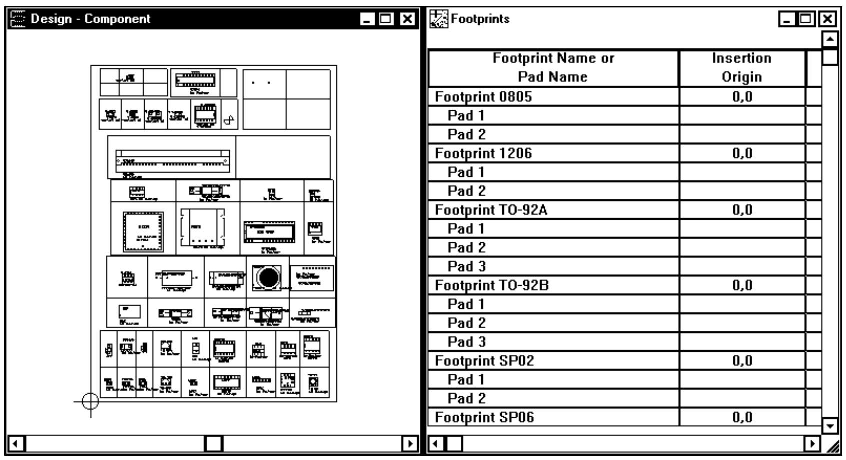
Choose the Spreadsheets toolbar button. Choose the Footprints, Components, or Thrucodes spreadsheet to view footprint data.



- **View the footprints in the Library window.**

  Choose Graphics Windows from the Window menu and select the Library option from the Select Graphic Window dialog box.





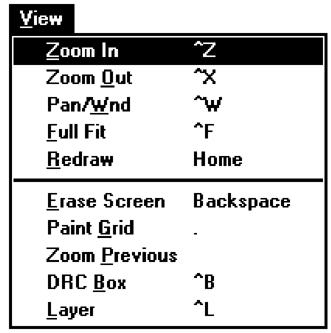
Creating a new footprint

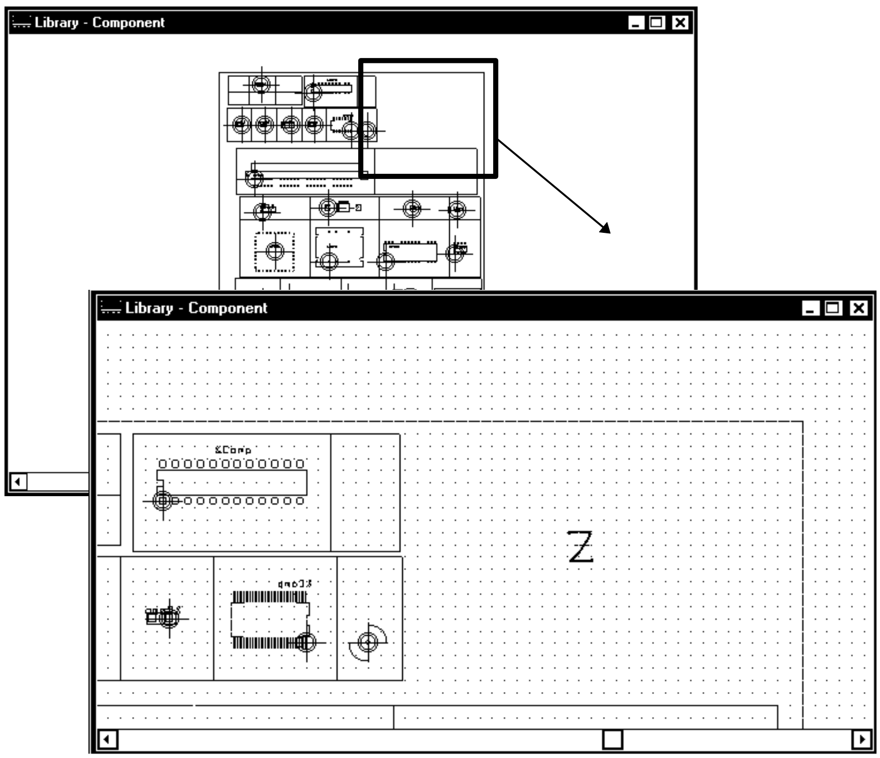
In Layout, you can create custom footprints to add to your library by following the steps below.

- Create a matrix to house the new part.
- Set up system grids for placement and routing.
- Add and identify the pins.
- Add silkscreen and other outlines.
- Add text.
- Choose the insertion datum.

- **Zoom in to the target location.**

Choose Zoom In from the View menu and zoom in to the top right corner of the library sheet.



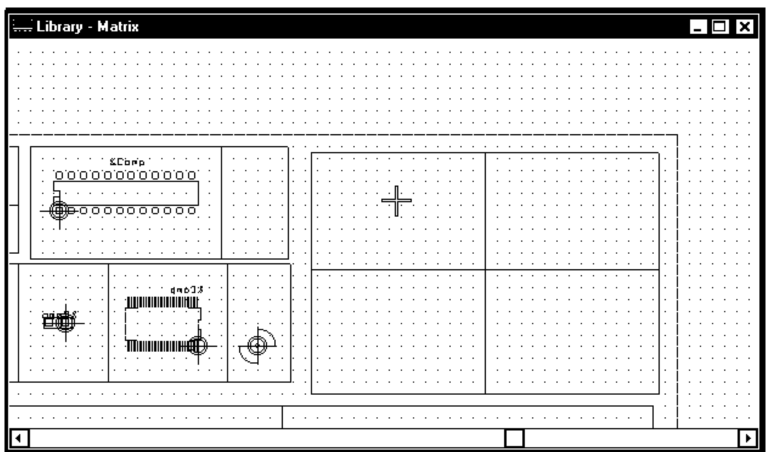


- **Create a matrix.**

  Each library sheet is divided into a user-defined matrix. A single footprint exists within each cell.

  Choose Matrix from the Tool menu to begin creating the matrix. Place the cursor in the upper left corner of the magnified area. Pressing the left mouse button, drag the cursor diagonally to the lower right corner and release.

  Click in the matrix and drag the cursor. Notice that as you move the cursor, new lines appear within the defined block enabling you to add as many cells to the area as you wish. Click with the left mouse button when you have created four matrix cells as shown below.



- **Select system grids.**

In Layout, you can set several system grids to facilitate the design process. The Routing and Via grids are used for place and route. The Dot, Place, and Detail grids are used throughout the design process.

The *Routing Grid* establishes a grid for routing.

The *Via Grid* establishes the grid on which vias are placed. It usually has the same value as the routing grid.

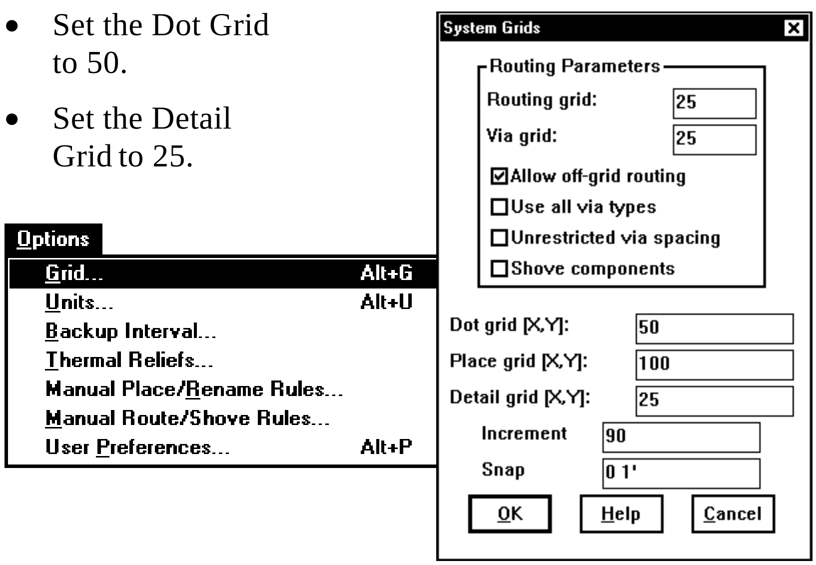
The *Dot Grid* is a visual reference grid that can be turned on and off by the user.

The *Place Grid* is the grid on which the pins are placed. It is also used for moving parts on the board.

The *Detail Grid* is the grid on which you create and move the graphics for a footprint such as the silkscreen and assembly data.

It is important to establish the grid you are going to use for pins before you begin creating the footprint. To set the grids for the design, choose Grids from the Options menu, and enter the following values in the System Grids dialog box:

- Set the Place Gridto 100.
- Set the Dot Grid to 50.
- Set the Detail Grid to 25.

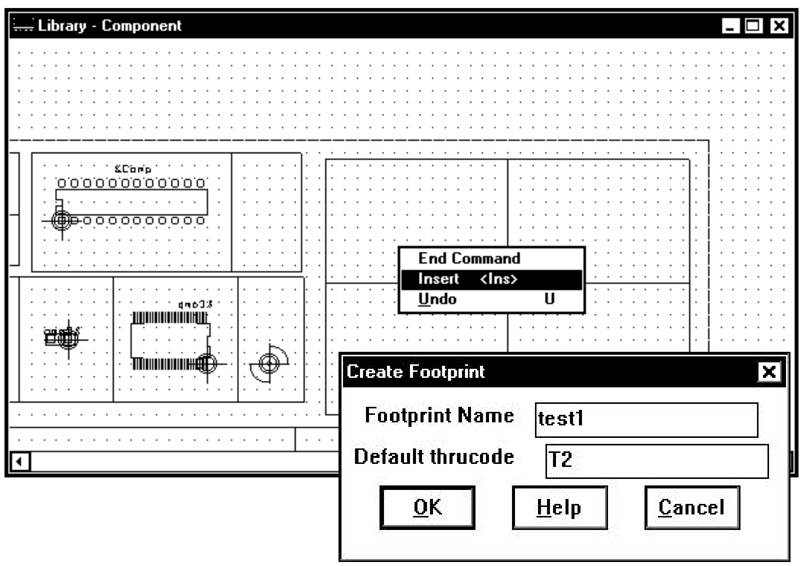


Adding pins to the footprint

- **Create the first pin.**

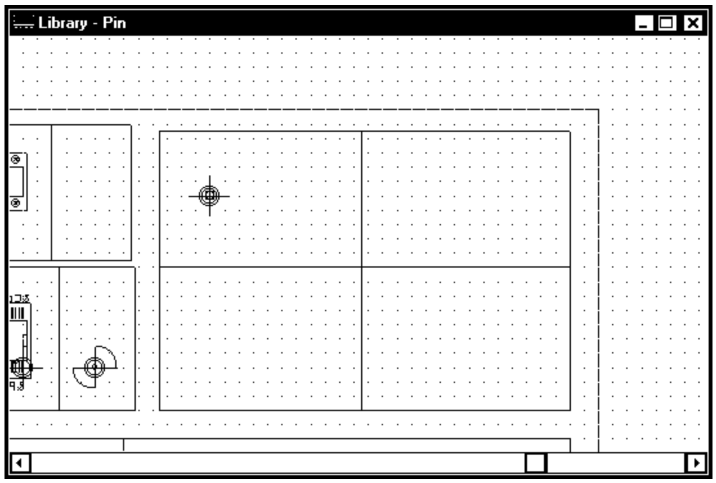
  Choose the Component toolbar button and place the cursor within one of the matrix cells you have defined. From the popup menu, choose Insert. The Create Footprint dialog box displays.

  In the Create Footprint dialog box, enter *test1* as the footprint name, and enter *T2* as the default thrucode. The matrix cell is labeled with the footprint name you enter. All of the graphics, thrucodes, and text within that cell become a part of that footprint. Choose the OK button to exit the dialog box.



- **Place the first pin.**

  Position the pin over the desired location and click the left mouse button to place it.



- **Create the next pin.**

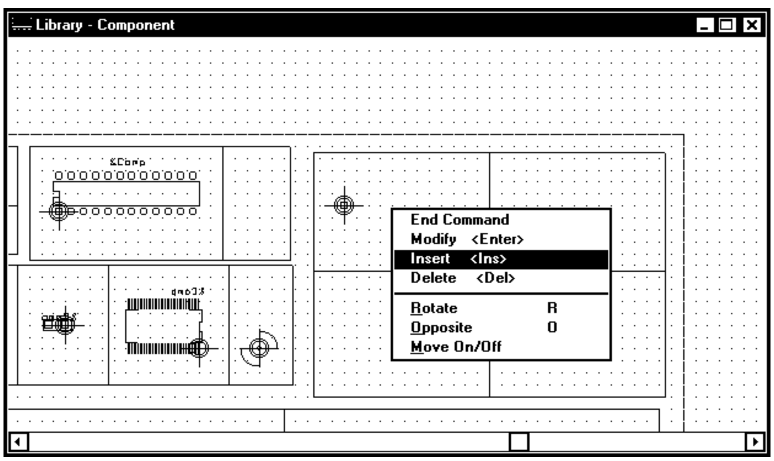
  Choose the Pin toolbar button. Pressing the SHIFT key, select the first pin you created by clicking on it with the left mouse button.

  Choose Insert from the popup menu again. You now have a second pin attached to the cursor.

- **Place the next pin.**

  Position the new pin 500 mil in the X direction as measured from the first pin. At the bottom of the screen, the status bar displays the coordinates for the new pin, DX:500, DY:0, DIST:500.

  In Layout, pin names can be numeric, alpha, or alphanumeric, and can be placed in any order. For example, you can name the pads 1, 7, 8, and 14 to fit a 4-pin oscillator that is numbered for a 14-pin part. Pin names *must* correspond to the pin names used in the schematic parts.



Note By default, Layout names the pins in numerical order beginning with the number 1. You must change the pin names in Layout to match the pin names in the schematic.

Viewing the pins on the footprint

- **Magnify the pin area.**

  Choose Zoom In from the View menu.

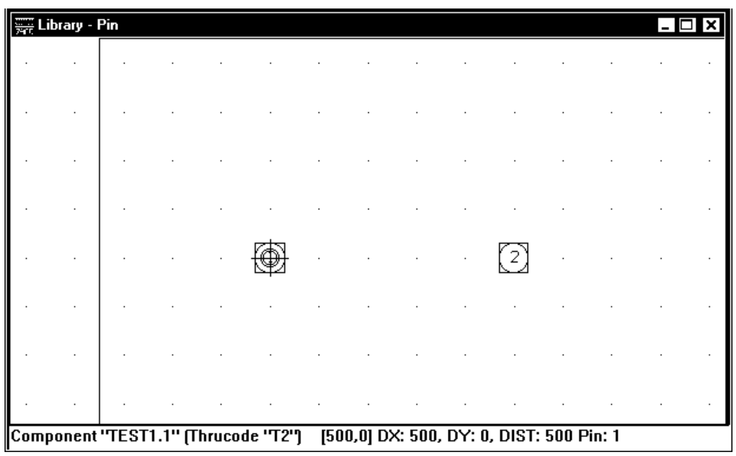
- **View the pin coordinates in the status bar.**

  When you place your cursor on a pin, the status bar displays the footprint's name and coordinates.

  Now, there are two pins on the footprint.

  PIN1 at X 0, Y 0

  PIN2 at X .500, Y 0.



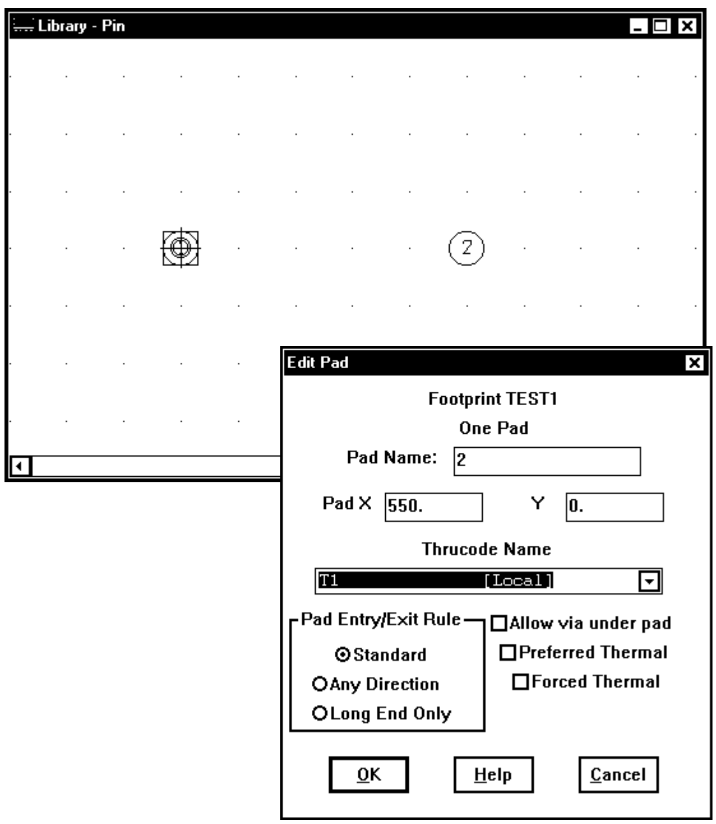
Editing the component pad

- **Edit the component pad.**

Using the Edit Pad dialog box, you can assign a thrucode to the pad. You can also specify exact coordinates for the pad, a powerful feature for parts that have pins spaced on a fine or irregular grid. For rectangular surface mount pads, you can restrict the route entrance to the long end only.

Pressing the SHIFT key, select the second pin you created by clicking on it with the left mouse button. Then, press the ENTER key. The Edit Pad dialog box appears.

Select thrucode *T2* from the Thrucode Name drop list. By selecting thrucode T2, you change Pin 2 from square to round.

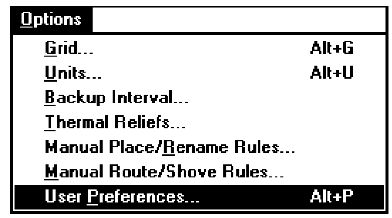


Setting display preferences for the footprint

In Layout you can set a variety of user preferences and save them with the board design.

- **Open the User Preferences dialog box.**

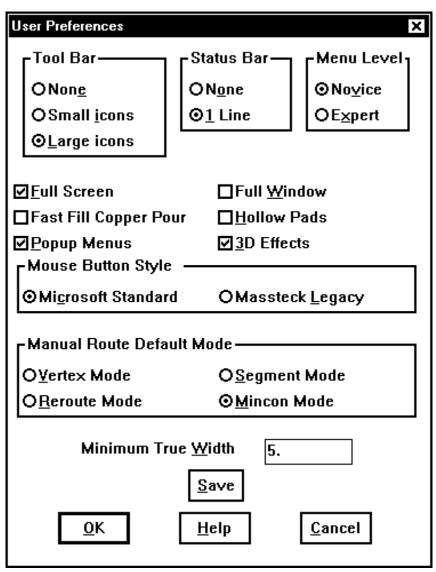
  Choose User Preferences from the Options menu.



- **Edit the options to set your display preferences.**

  Change the following settings and choose the OK button to apply them.

  - If you prefer solid pads, deselect the Hollow Pads check box.
  - To display a width for all lines on the board that are five millimeters wide or greater, enter 5 in the Minimum True Width text box.
  - To see component heights in your library, select the 3D Effects check box.



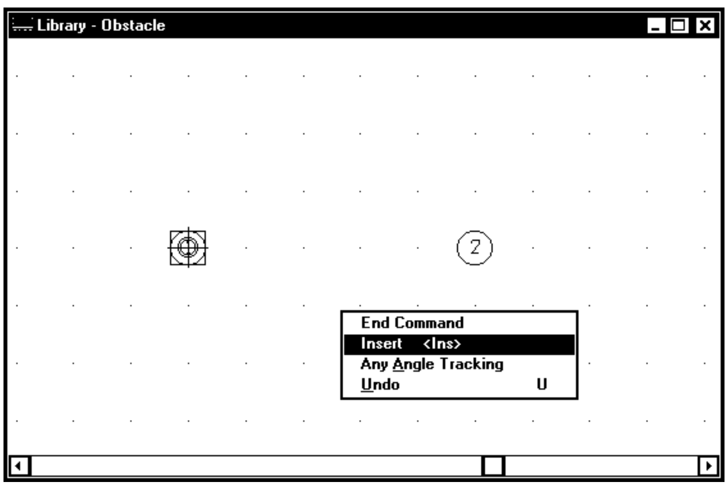
Creating silkscreens, assembly drawings, and placement outlines

In OrCAD Layout for Windows, you create silkscreens, assembly drawings, and placement outlines for your footprint using obstacles.

- **Begin drawing the obstacle.**

Choose the Obstacle toolbar button. Place the cursor in the Library window. From the popup menu, choose Insert.

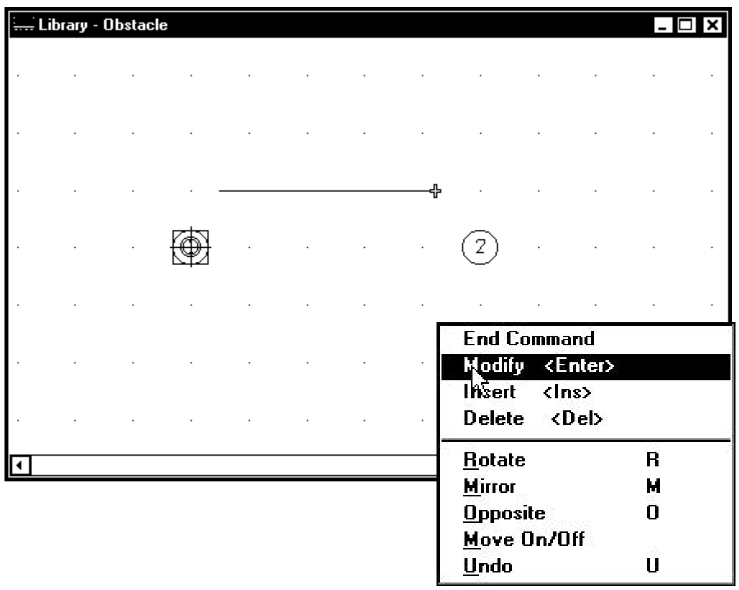
Click the left mouse button to begin drawing. Use the arrow keys to move the cursor, drawing the obstacle. Press the SPACEBARto stop drawing.



- **Define the obstacle.**

  As you input an obstacle, you must specify the type of obstacle that you want to create and the target layer for the obstacle.

  Choose Modify from the popup menu to display the Edit Obstacle dialog box.



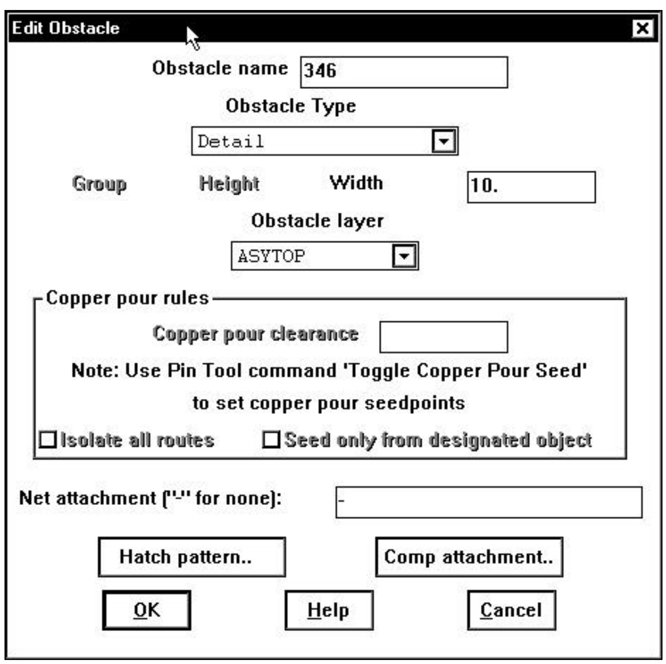
Using the Edit Obstacle dialog box, select the type of obstacle you want to create, including those used by the placement and routing routines and those used for silkscreens and assembly drawings. You can also set physical attributes such as width, layer, and hatch pattern.

Specify the following options to create a silkscreen obstacle for components that appear on the top layer of the board.

- Select Detail from Obstacle Type drop list.
- Set the Obstacle Width to 10.
- Select SSTOP from the Obstacle Layer drop list.

Specify the following options to create an assembly drawing for components that appear on the top layer of the board.

- Select Detail from the Obstacle Type drop list.
- Set the Obstacle Width to 10
- Select ASTOP from the Obstacle Layer drop list.



- **Create a placement outline.**

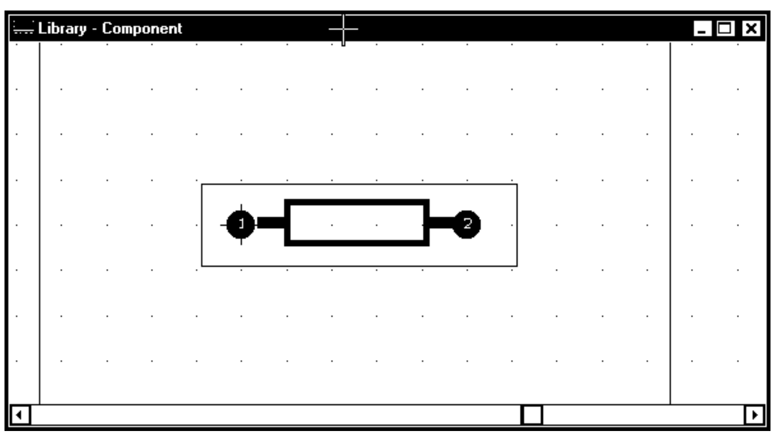
Layout's interactive and automatic placement utilities look for a placement outline. The outline is used to maintain a specified distance between parts.

For surface mount parts, this outline should be large enough to provide sufficient space between parts, eliminating solder shadowing and facilitating the post-assembly inspection process.

A placement outline is created using the same method that you used to create the silkscreen and assembly drawing.

Begin drawing an obstacle, select it, and choose Modify from the popup menu. Modify the options in the Edit Obstacle dialog box to create the outline.

- Select Place Outline from the Obstacle Type drop list.
- Change the target layer option. You can draw the placement outline on the top layer, the bottom layer, the top and bottom layers, or on Conn (all layers).



Adding text to components

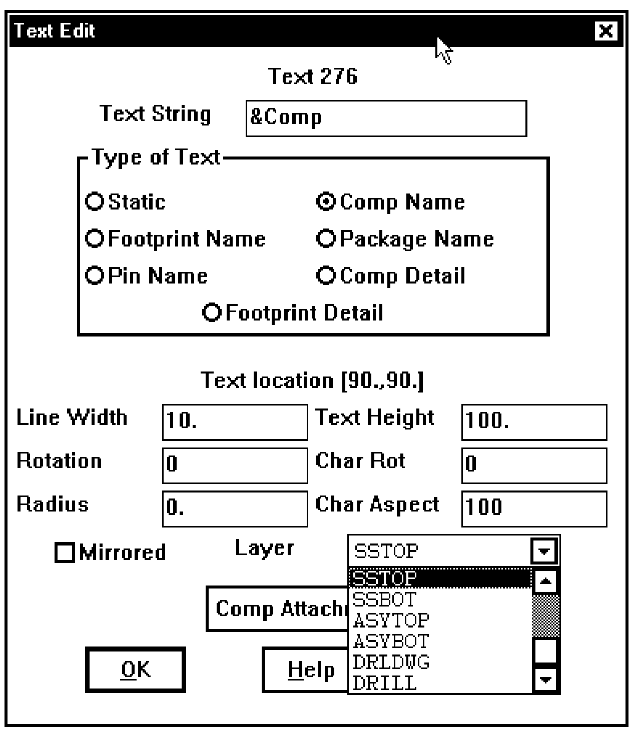
- **Add a reference designator.**

Choose the Text toolbar button. Position your cursor over the Library window and choose Insert from the popup menu. Modify the options in the Text Edit dialog box to create the text.

- For the text type, select Comp Name
- Specify a text width of 10.
- Specify a text height of 100.
- Select the silkscreen top layer (SST) from the Layer drop list.

You now have the text string *&Comp* attached to your cursor. Position it above the footprint and click the left mouse button to place it.

When the component is placed on the design, the text string *&Comp* is replaced with the appropriate reference designator.

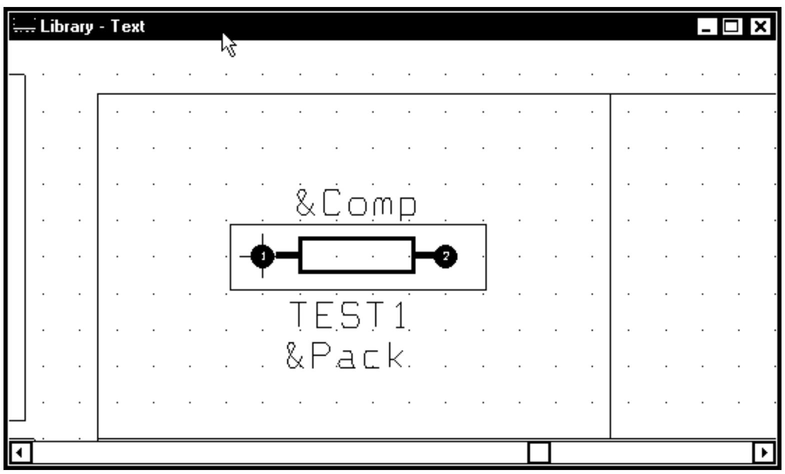


- **Create additional labels for the footprint.**

To create additional labels for the footprint, modify the text type and layer options in the Text Edit dialog box as described in the previous exercise, *Add a reference designator*.

The reference designators (represented here by the keyword *&Comp*) can be placed in different locations and can be different sizes.

Note The text string *&Pack* or *No Package* may appear instead of the package label.



Setting an insertion origin

In OrCAD Layout for Windows, you must specify an insertion origin as well as a graphic origin. The location of the part as specified in the insertion report is calculated using the insertion origin.

For a through-hole part, the graphic origin is typically pin 1.

Note In the surface mount footprints supplied by OrCAD, the graphic origin is at the centroid of the part by default.

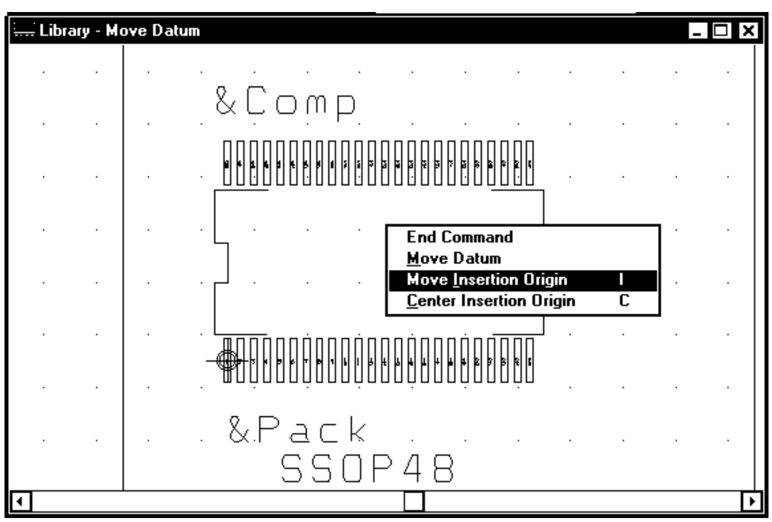
- **Locate the footprint.**

Press the TAB key and enter the footprint name *SSOP48*.

- **Specify the insertion origin.**

Choose Move Datum from the Tool menu. Position the small cursor at the desired location for the insertion datum and choose Move Insertion Origin from the popup menu.

Position the cursor at the desired location and click the left mouse button.



Managing the footprint library

Once you create the first footprint, you can build the library and use it to organize and manage footprints. Using the footprint library, you can do the following.

- Copy footprints to create new footprints.
- View footprints.
- Use Query and the Footprints spreadsheet to access and modify footprint data.

Copying footprints

The easiest way to create another component footprint is to copy and modify an existing footprint.

- **Zoom out.**

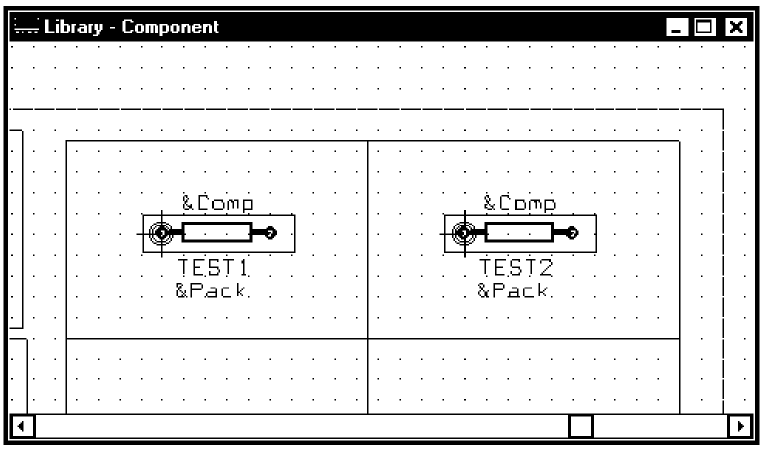
Choose Zoom Out from the View menu and click once on the screen so that you can see the footprint and an empty matrix cell.

- **Copy the footprint.**

Choose the Component toolbar button. Select the footprint and choose Modify from the popup menu.

In the Create Footprint dialog box, accept the default name for the new footprint or enter a new name.

Click the left mouse button to place the copy of the footprint within a blank cell. Press the HOME key to redraw the screen.



Viewing obstacle types in the library

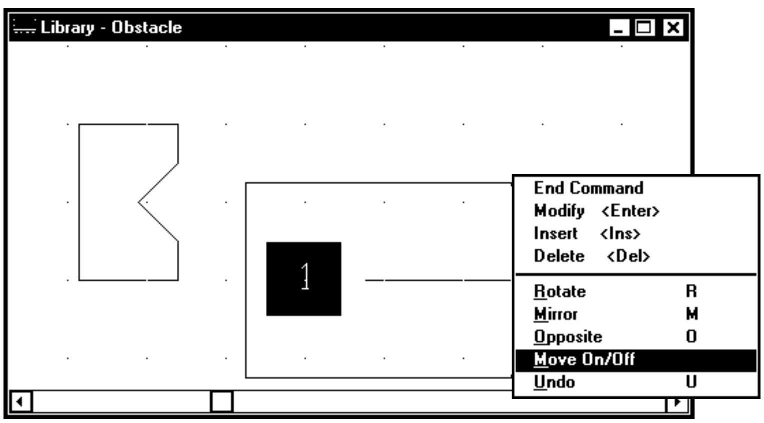
There are many types of obstacles available to view in the demo library, DGUIDE.LIB. This section illustrates and describes some of these obstacles.

- **View an insertion outline.**

An insertion outline is added to a footprint to represent the size of the auto-insertion head.

Choose the Obstacle toolbar button. Press the TAB key to display the Find Coordinate or Component Name dialog box. Enter the component name *DO900* (d,o,nine,zero,zero) and choose the OK button. The screen scrolls to the component in the demo library and centers it on the screen.

In the component, the odd-shaped obstacle that surrounds pin 1 is an insertion outline. Pressing the SHIFT key, select the insertion outline and choose Move On/Off from the popup menu. Move the insertion outline away from the component so that you can see it clearly. Click the left mouse button to place it.



- **View a via keep-out.**

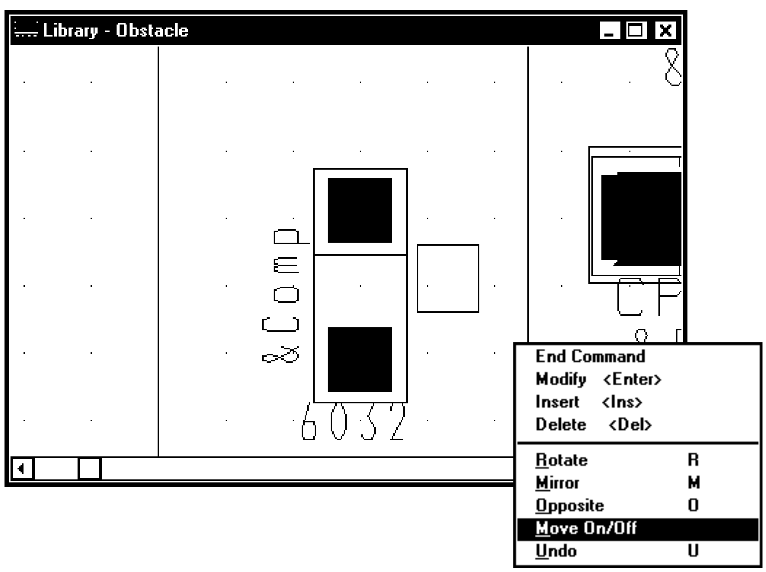
Via keep-outs are used to restrict vias from being placed under surface-mount discrete parts.

Similar obstacles, called route keep-outs are added to a footprint to restrict tracks. For example, you may want to avoid tracks under a component on the top layer of the board, but may want to route under the component on all other layers.

To view the via keep-out, choose the Obstacle toolbar button. Press the TAB key to display the Find Coordinate or Component Name dialog box. Enter the component name *6032* and choose the OK button. The screen scrolls to the component and centers it on the screen.

The small rectangle between the pads is the via keep-out.

Pressing the SHIFT key, select the via keep-out and choose Move On/Off from the popup menu. Move the via keep-out away from the component so that you can see it clearly. Click the left mouse button to place it.



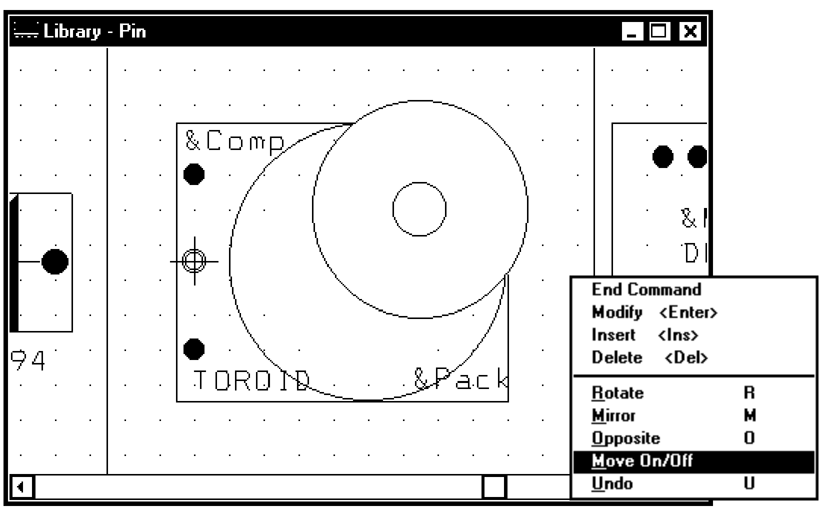
- **View a copper obstacle.**

When attached to a pad, copper can create a heatsink under a power part. Or, copper can create an odd-shaped pad for a special application.

When the copper is attached to the pad it becomes an integral part of the pad. If the pad is moved the copper moves with it. If the pad is attached to a net, then the copper automatically becomes part of that net.

Choose the Obstacle toolbar button. Press the TAB key to display the Find Coordinate or Component Name dialog box. Enter the component name *Toroid* and choose the OK button. The screen scrolls to the component and centers it on the screen.

Choose the Pin toolbar button. Select PIN3 (the larger pin) and move it. Notice that the copper obstacle moves with the pin.

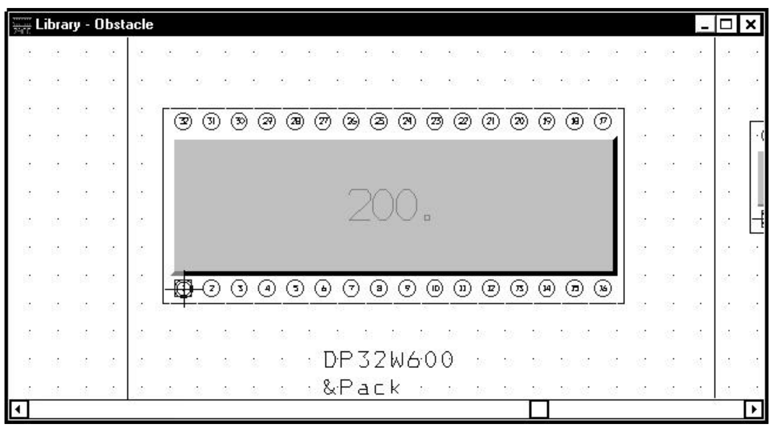


- **View a component height obstacle.**

For designs in which components with height restrictions need to be restricted to specific areas of the board, you can create component keep-outs to keep those components out of areas where they do not fit. Component height keep-in and keep-out obstacles are defined in the footprint library.

Choose the Obstacle toolbar button. Press the TAB key to display the Find Coordinate or Component Name dialog box. Enter the component name *DP32W600* and choose the OK button. The screen scrolls to the component and centers it on the screen.

The grayed area on the footprint is a component height that can be built into a library part.



Using the Query tool

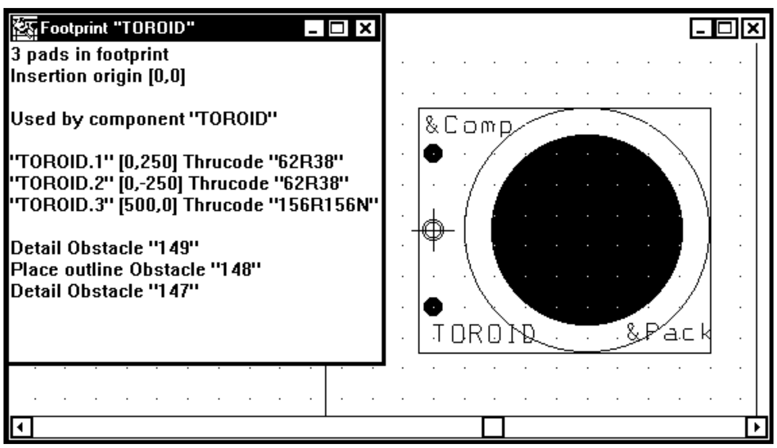
One of the most advanced features in OrCAD Layout for Windows is Query. Using Query, you can access and edit almost any element in the board design.

- **Query a footprint.**

For footprints, the Query window displays information such as quantity, pad labels, thrucodes, and location.

Choose the Query toolbar button. The Query window appears in the upper left corner the graphics window.

Choose the Component toolbar button and select the Toroid footprint in the library. Information about the footprint appears in the Query window.



- **Choose End Command from the popup menu.**

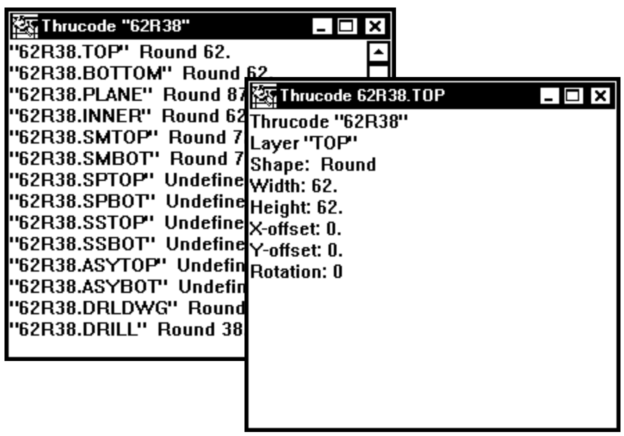
You must choose End Command from the popup menu before you do the next exercise.

- **Use Query to edit a component thrucode.**

You can edit a single thrucode layer or all thrucode layers from the Query window by accessing the Edit Thrucode Layer and Edit Thrucode dialog boxes. OrCAD Layout for Windows offers a variety of pad shapes as well as the ability to offset the drill from the center of the pad.

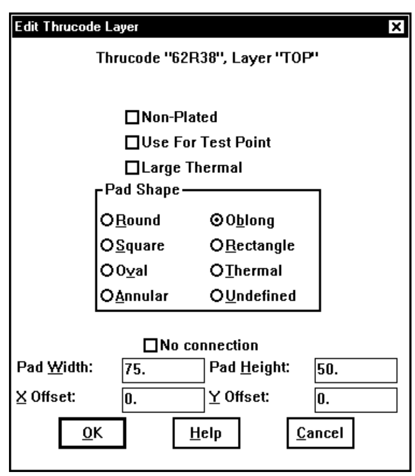
In the Query window, select *62R38*. Then, select *62R38.TOP*.

Note When selecting items in the Query window, click on the word in quotation marks. Any data in quotation marks can be modified.



Tip If you scroll down, you can see a list of footprints using this thrucode.

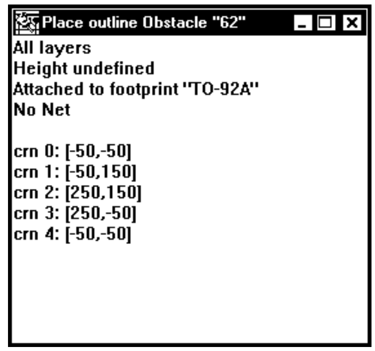
Press the ENTER key to display the Edit Thrucode Layer dialog box. Change the thrucode to 75x50 Oblong.



- **Use Query to view obstacle data.**

  From the Query window, it is also possible to check the coordinates and parameters of any obstacle attached to a footprint.

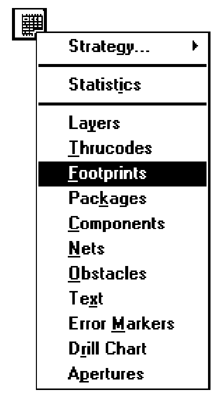
  - Again, select the Thrucode *62R38* in the Query window.
  - Scroll down and select *Attached to footprint "TO-92A."*
  - Select *62,* which is the Place Outline obstacle.



Using the Footprints spreadsheet to edit footprints

As OrCAD Layout for Windows and its predecessor MaxEDA have always been Windows programs, the full functionality of spreadsheets is available. Using spreadsheets makes it easy to perform tedious tasks such as moving entire rows of pads in slight increments.

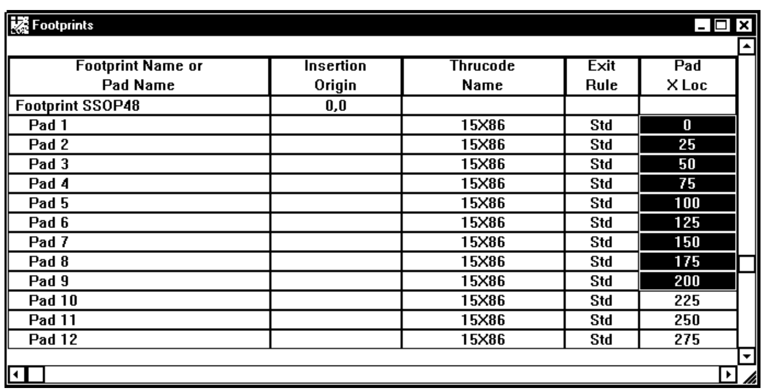
- **Open the Footprints spreadsheet.**



Choose the Spreadsheets toolbar button and choose Footprints from the menu that appears.

- **Select a footprint.**

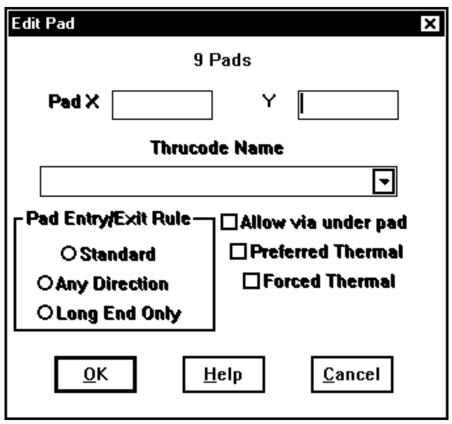
Press the TAB key and enter SSOP48. Pressing the left mouse button, drag the cursor to select several pads in the Y Loc column.


Footprints

Footprint Name or Pad Name	Insertion Origin	Thrucode Name	Exit Rule	Pad X Loc
Footprint SSOP48	0,0			
Pad 1		15X86	Std	0
Pad 2		15X86	Std	25
Pad 3		15X86	Std	50
Pad 4		15X86	Std	75
Pad 5		15X86	Std	100
Pad 6		15X86	Std	125
Pad 7		15X86	Std	150
Pad 8		15X86	Std	175
Pad 9		15X86	Std	200
Pad 10		15X86	Std	225
Pad 11		15X86	Std	250
Pad 12		15X86	Std	275

- **Edit the footprint pads.**

  Press the ENTER key or choose Modify from the popup menu. In the Edit Pad dialog box, notice that the number of pads selected appears at the top of the editor. Any change you make affects all of the selected pads. As you selected the pads in the Y Loc column, the cursor is blinking within the Y location field. If you desire, you can now move the selected pads in increments as small as a few thousandths of an inch.



Working with application windows in Layout

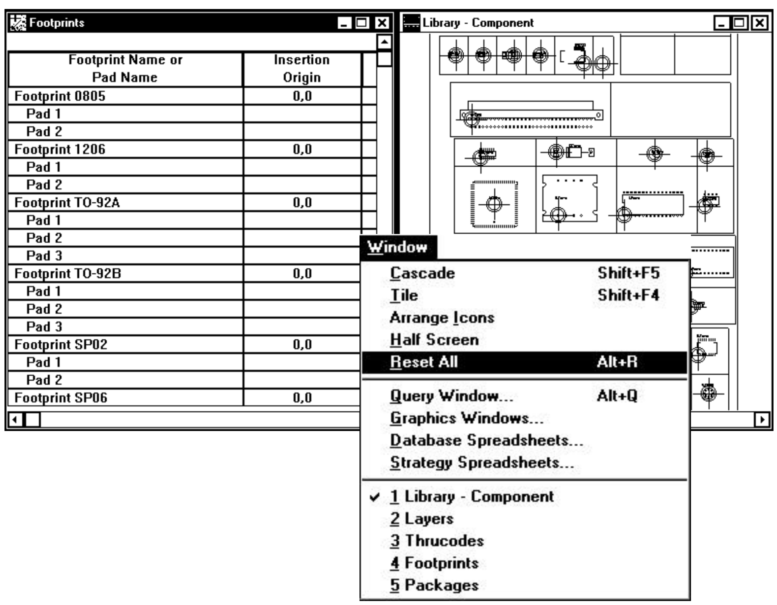
OrCAD Layout for Windows is a Windows-based design tool. Therefore, most of the functions of the graphic windows within the software follow standard Windows convention.

- **Tile the windows on the screen.**

  Choose Tile from the window menu.

- **Reset the windows on the screen.**

  Choose Reset All from the Window menu to close the Footprints spreadsheet and return the graphics design window to full display.



Exiting the library

When you have finished with the Footprint library, choose Exit from the File menu.

Go to the next chapter.

Go to the table of contents.

Chapter 3
Component placement

Go to the table of contents.

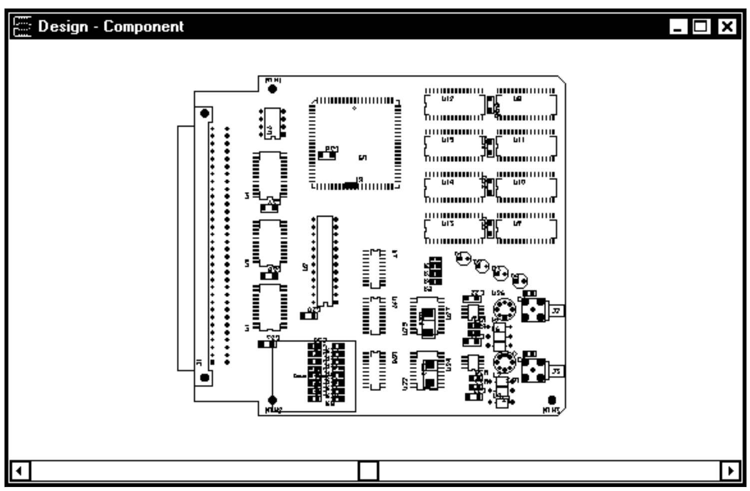
Go to the last page of the previous chapter.

Component placement

You are now ready to create a Printed Circuit Board (PCB). OrCAD Layout for Windows provides users with the most advanced layout features available on the market today.

In Layout, the designer has the ability to specify design rules in the schematic, such as component locations, net spacing criteria, component group information, net widths, and routing layers.

Note If you wish to skip the interactive placement section of the demo and use only the autoplacement section, skip to *Autoplacement* in this chapter.



Opening a new design

- **Choose New Design from the Tools menu.**

- **Select a technology file.**

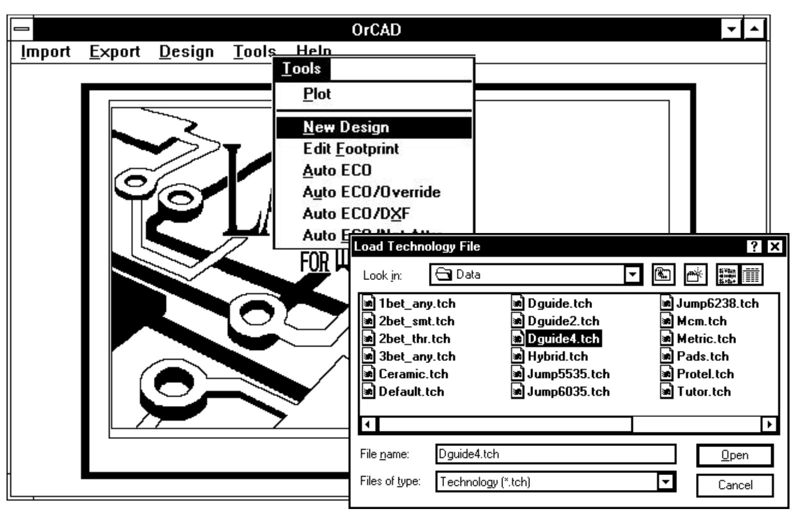
When you create a new design in Layout, you must select a set of design instructions called a technology file. A technology file can include design rules, drawing formats, dimensions, preplaced components, tooling holes, and the board outline. Technology files save time, especially when designing multiple boards that share common formats and design rules. They can serve as a company standard for design rules such as layer stacks, padstacks, track widths and spacing, Gerber output standards, and report formats. Virtually any design rule that is common to some or all of your projects can be included in a technology file.

For the demo, select the technology file DGUIDE2.TCH or DGUIDE4.TCH when prompted

---

Note You need a minimum of 16MB of RAM to run DGUIDE4.TCH.

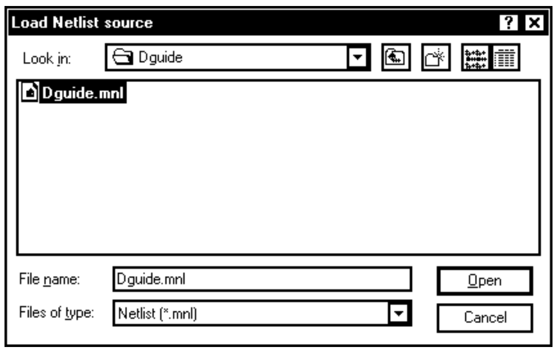
Note If you do not want to go through the process of creating a new design, choose Design from the menu bar and select the DPLACE2.MAX or DPLACE4.MAX file from the list presented. Then, skip to the section *Preparing to place components on the board* in this chapter.



- **Select a design netlist.**

The design netlist is extracted from the schematic. For this demo, select DGUIDE.MNL from the list presented in the Load Netlist source dialog box.

Note If the system prompts you to add a new netlist, choose No.



- **Name the design.**

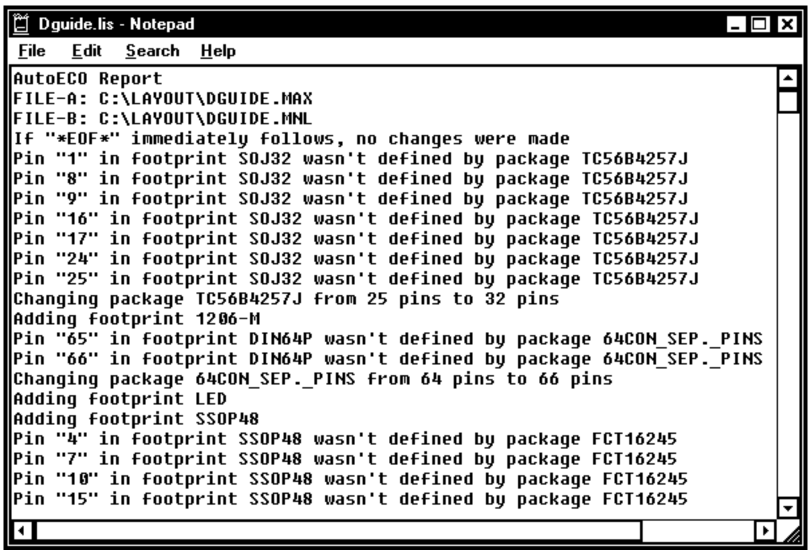
By default, the design assumes the name of the netlist source. Choose OK to choose the default name for the design.

- **Review the AutoECO report.**

When you create a design, Layout saves an ASCII report file generated from the AutoECO (Automatic Engineering Change Order) utility. Any errors that occur during the data merge are reported in the file.

Open the report DGUIDE.LIS in an editor such as Notepad. Take a moment to scan the report.

Note The message, "*Pin* in *footprint* wasn't defined by *package*" refers to an unused pin.




```
AutoECO Report
FILE-A: C:\LAYOUT\DGUIDE.MAX
FILE-B: C:\LAYOUT\DGUIDE.MNL
If "*EOF*" immediately follows, no changes were made
Pin "1" in footprint SOJ32 wasn't defined by package TC56B4257J
Pin "8" in footprint SOJ32 wasn't defined by package TC56B4257J
Pin "9" in footprint SOJ32 wasn't defined by package TC56B4257J
Pin "16" in footprint SOJ32 wasn't defined by package TC56B4257J
Pin "17" in footprint SOJ32 wasn't defined by package TC56B4257J
Pin "24" in footprint SOJ32 wasn't defined by package TC56B4257J
Pin "25" in footprint SOJ32 wasn't defined by package TC56B4257J
Changing package TC56B4257J from 25 pins to 32 pins
Adding footprint 1206-M
Pin "65" in footprint DIN64P wasn't defined by package 64CON_SEP._PINS
Pin "66" in footprint DIN64P wasn't defined by package 64CON_SEP._PINS
Changing package 64CON_SEP._PINS from 64 pins to 66 pins
Adding footprint LED
Adding footprint SSOP48
Pin "4" in footprint SSOP48 wasn't defined by package FCT16245
Pin "7" in footprint SSOP48 wasn't defined by package FCT16245
Pin "10" in footprint SSOP48 wasn't defined by package FCT16245
Pin "15" in footprint SSOP48 wasn't defined by package FCT16245
```

OrCAD Layout for Windows—component placement

Preparing to place components on the board

When a netlist is loaded, the parts are located in a pile in the lower left hand corner of the board, just outside the outline. Layout offers a variety of options to place the parts on the board.

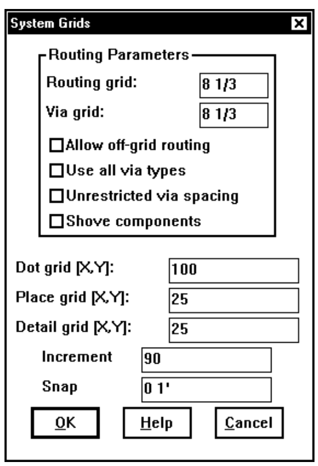
- **Check your system grids.**

Before you begin placing the components, make sure you have specified grids for placement and routing.

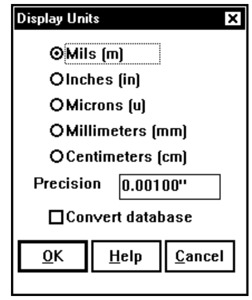
Choose Grids from the Options menu to display the System Grids dialog box.

- Set the Dot Grid to 100.
- Set the Place Grid to 25.

See For information the grids used in OrCAD Layout for Windows, see *Creating a footprint* in *Chapter 2: Footprint library*



- **Select display units.**



In OrCAD Layout for Windows, it is possible to select one of several unit values for the design. The unit value used can be changed at any time. For example, you can route the board in mils and then confirm pad locations within footprints in millimeters.

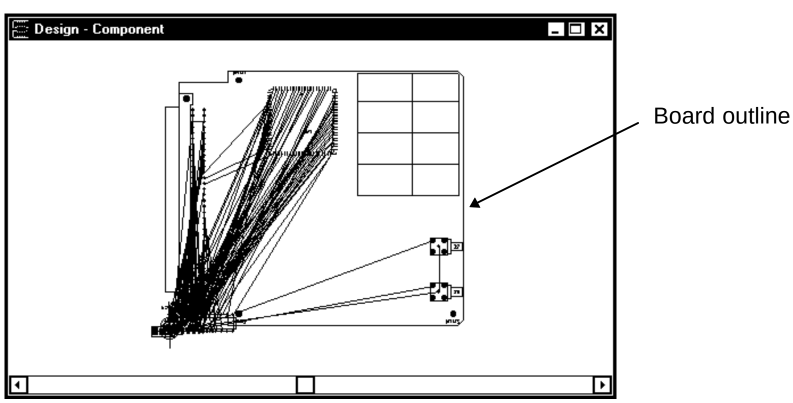
To set the current unit value, choose Units from the Options menu to display the Display Units dialog box. For the purpose of the demo, do not change the value from the default *Mils.*

- **View the board outline.**

The board outline serves as a component and route keep-in on all layers. Separate keep-ins and keep-outs can be added to any or all layers to meet specific needs.

For example, if there is a hole in the middle of your board, you can use keep-outs to prevent components and routes from being placed in that area. You can also use a route keep-in or keep-out to make sure digital signals do not route through the middle of an analog circuit.

See For information on component and route keep-ins and keep-outs, see *Creating obstacles* in *Chapter 2: Footprint library*

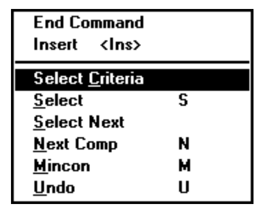


Placing components individually

The first interactive option described involves placing parts individually.

Before you begin, turn off the ratsnest display by choosing the Reconnect/Connect toolbar button (it is the second toolbar button from the left). It is important to turn off the ratsnest during interactive placement. Some interactive placement commands do not work properly if the ratsnest is enabled. When disabled, the ratsnest is only visible while components are being moved.

- **Select components to place individually.**

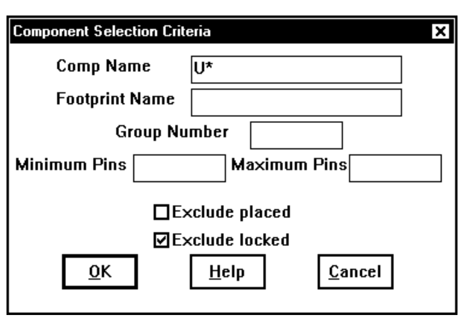


Choose the Component toolbar button. Place your cursor in the Design window and choose Select Criteria from the popup menu. Enter the letter *U** in the Comp Name text field to specify all components that begin with U.

- **Place the components.**

Choose Next Comp (next component) from the popup menu, or press the N key. Each time you choose the Next Comp command, one component beginning with the letter *U* snaps to the cursor in the order in which they are specified in the schematic netlist.

To practice placing components, drag a couple of components outside of the board outline and click the left mouse button to place them.



Placing components in groups

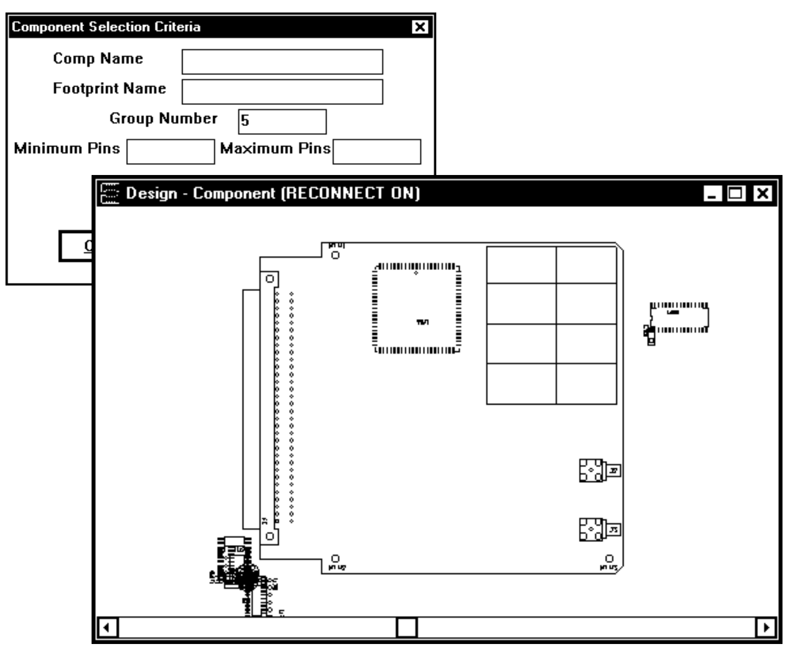
The schematic netlist that you are using for this demo includes an attribute that has assigned some components to groups. In Layout, you can use this attribute to place parts by the group number assigned in the schematic. When you specify a group number in the Component Selection Criteria dialog box, the parts assigned to the group snap to the cursor, allowing you to place them on the board.

- **Select components to place as a group.**

Choose the Component toolbar button. Place your cursor in the Design window and choose Select Criteria from the popup menu. Enter *5* in the Group Number text field to specify group *5* for placement. The group of parts snaps to the cursor.

- **Place the group of components.**

Dragging the cursor, position the group of components to the right of the matrix pattern in the top right corner of the PCB. Click the left mouse button to place the components. You will use these components for the next exercise.



Placing components using a matrix

In Layout, you can also place parts in a matrix. The matrix placement algorithm is useful for placing groups such as memory arrays and discrete components.

You can create a matrix of any size, anywhere on the board. For information on creating a matrix, follow the directions in *Preparing to create a new footprint* in *Chapter 2: Footprint Library*.

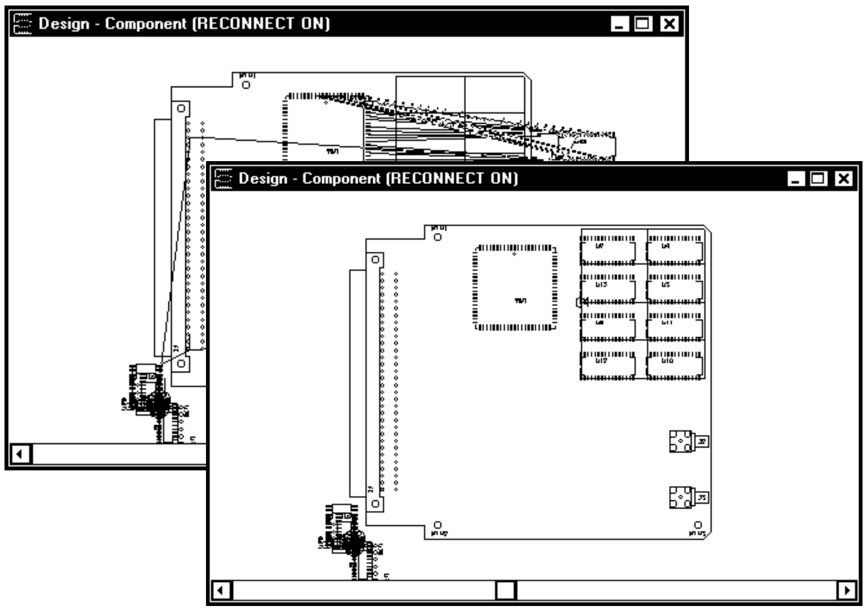
For the purposes of this demo, use the matrix created for this exercise, which is located in the upper right corner of the board.

- **Select the components to place in the matrix.**

Choose the Component toolbar button. Pressing the left mouse button, drag a box around the group of components you placed to the right of the board in the previous exercise.

- **Place the components in the matrix.**

  Choose Matrix Place from the popup menu or press the X key to place the selected components within the matrix defined in the upper right section of the board.



Placing components using clusters

Another OrCAD Layout for Windows component placement tool is cluster placement.

Using clusters is a fast and intelligent way to place a board. Clusters are component groups that are formed to simplify placement. As with groups, clusters allow you to move multiple parts at once. If clusters are created manually, then, like groups, they can represent specific circuits and can be placed quickly in the appropriate area of the board.

The size of the circle that displays to represent a cluster equals the combined area of all of the components within that cluster. Use the circle as a visual aid to ensure that there is enough room to place the cluster in a given area of the board.

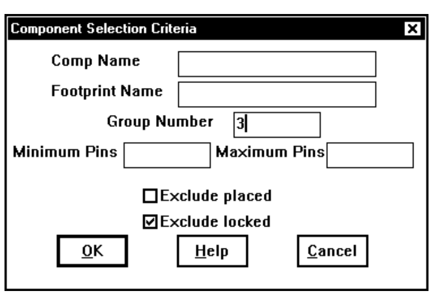
Note On the PCB, you can select any group of parts that were assigned to a component group in the schematic. Clusters and group may also be assigned manually using the Components spreadsheet.

Note If you have not done so, turn off the ratsnest display by choosing the Reconnect/Connect toolbar button. Cluster placement does not work properly with the ratsnest enabled.

- **Select the components to place as a cluster.**

Choose Select from the popup menu or press the S key to display the Component Selection Criteria dialog box.

Enter group number3 in the Group Number text box. Then choose Cluster from the popup menu, or press the C key to place the parts in a cluster.

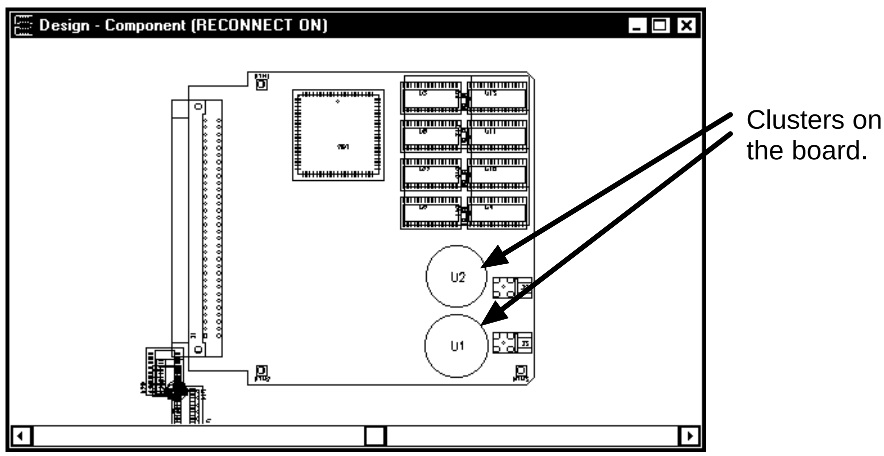


- **Place the cluster.**

  Position the cluster near the J3 component and click the left mouse button to place it.

- **Place a second cluster.**

  Repeat the process using group number 4 and place the second cluster above the first, near J2.



Placing components using the Quick Place command

Once the clusters or groups are placed on the board, Quick Place can unstack and arrange the parts while maintaining minimum separation based on the place outline.

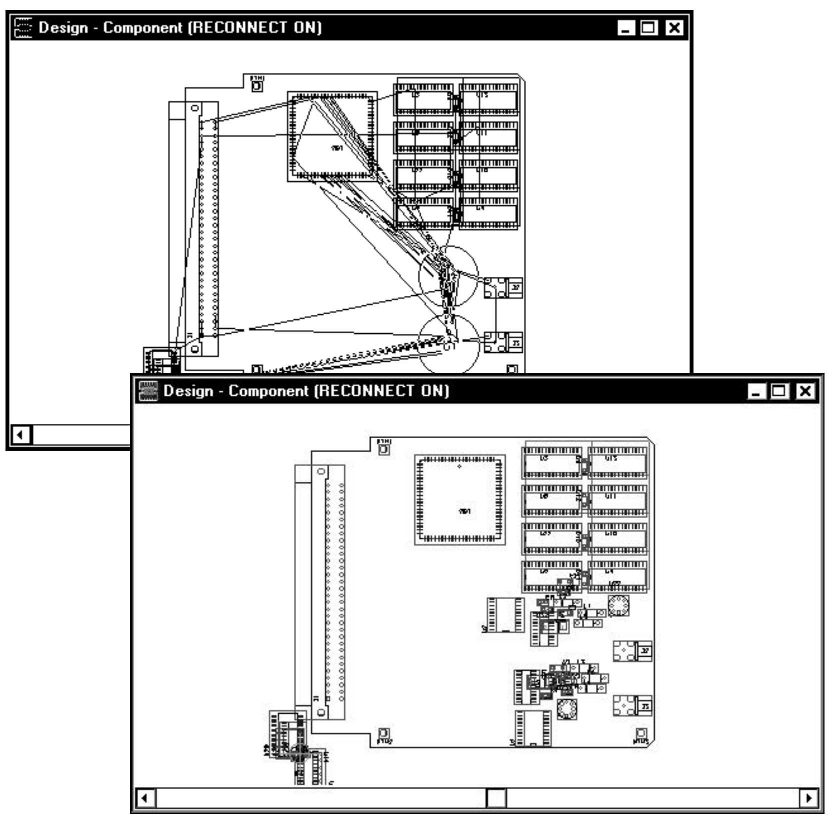
- **Select the clusters.**

Pressing the SHIFT key, select the two clusters, U1 and U2, by clicking on them with the left mouse button.

- **Unpack the clusters.**

Choose Quick Place from the popup menu, or press the P key to unpack the clusters. This results in a fast, general placement.

Note Your components may be placed differently than the components shown here, depending on where you placed the cluster on the board.



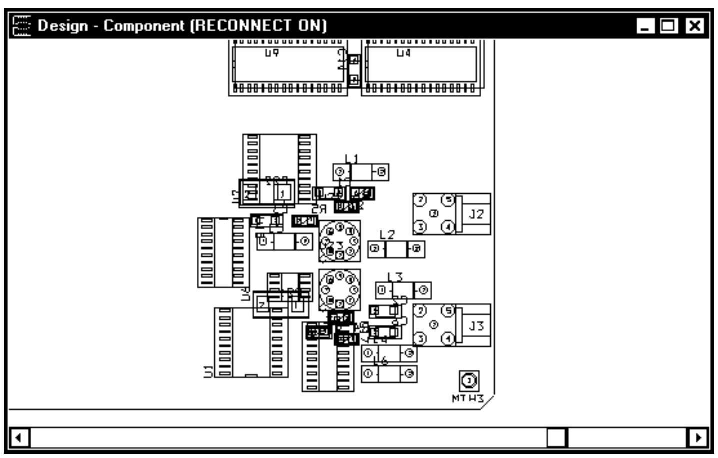
Arranging components on the board

Now that you have placed the clusters, you can arrange them efficiently. As you begin to arrange the components, there are several interactive placement options available for you to use in OrCAD Layout for Windows. Experiment with some of the available commands as described in this section.

Note To perform these commands on a group of components, firs select them by pressing the left mouse button and dragging a windo around them.

- **Magnify the board area.**

Choose Zoom In from the View menu and click on the screen to magnify the lower right corner of the design.



- **Shove the component.**

  One placement option is the Shove Comp (shove component) command. The Shove Comp command moves components out of the way, clearing the place outline you inserted when you created the footprint *in Chapter 2: Footprint Library.*

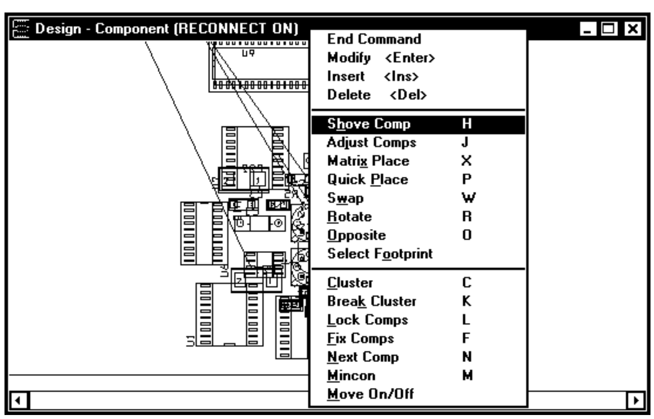
  Select a through-hole discrete component with the left mouse button and move it so that it overlaps another through-hole discrete component. Choose Shove Comp (shove component) from the popup menu, or press the H key.

- **Rotate the component.**

  You can rotate parts individually or as a group. The increment of rotation defaults to 90 degrees, but it can be changed to any value. To set the increment of rotation, choose Grids from the Options menu.

  To rotate the component, select a surface-mount IC and choose Rotate from the popup menu, or press the R key.

Note With group selection, the components will rotate one time for each time you select them. You must reselect them as a group to rotate them again.



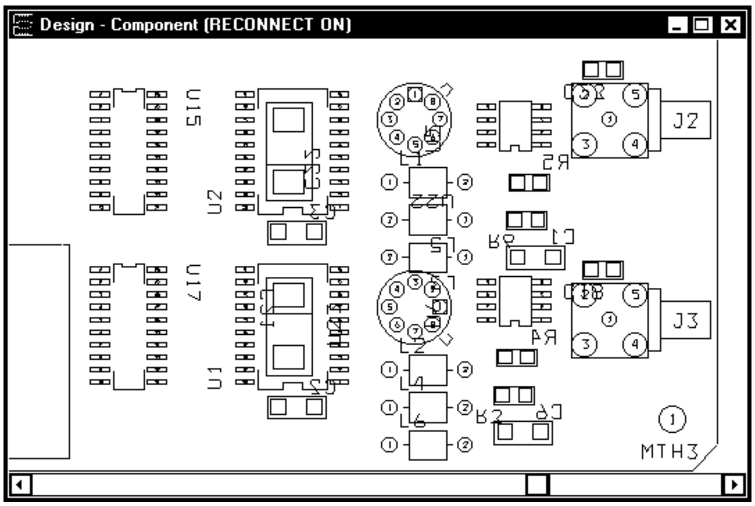
- **Mirror the component.**

  Parts can be mirrored to the opposite side of the board either individually or as a group.

  Choose Opposite from the popup menu, or press the O key to mirror a part to the bottom side of the board.

- **Design the board layout.**

  Study the sample placement of the board shown below. Try to place the components similarly using the commands we have described.



Autoplacement

In OrCAD's Layout Plus for Windows, you can access many options to place components on a board. You used some of the interactive placement options in the previous section. In this section, you will use the autoplacement option which includes advanced placement capabilities such as component push-n-shove and automatic cluster placement.

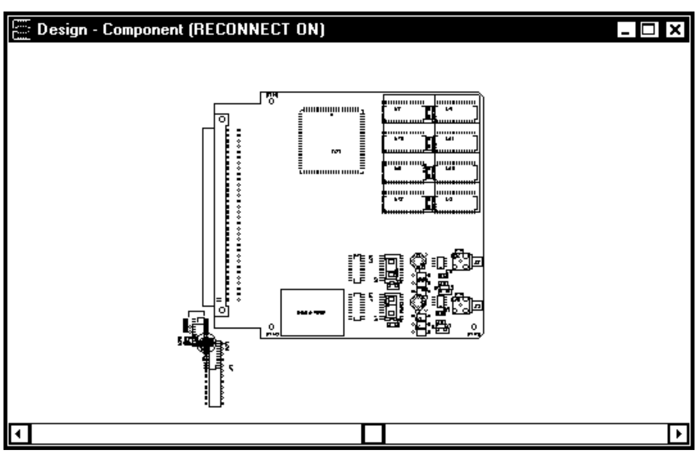
Note The autoplacement capability is only available in the OrCAD Layout Plus for Windows package.

Groups and clusters play an important role in automatic placement. To achieve effective component placement, it is wise to group the components based on their functionality or connectivity. You can do this at the schematic level, or in Layout Plus' Components spreadsheet.

The board you are designing for the demo should resemble the one shown below. If you skipped the interactive placement section, or if your board does not resemble the one shown below, choose Design from the OrCAD frame and open the demo design file DPLACE2.MAX or DPLACE4.MAX.

Note You need a minimum of 16MB of RAM to run DPLACE4.MAX.

Note If the system prompts you to add a new netlist, choose No.



Locking components on the board

When using autoplacement in Layout Plus, you must first secure the components on the board using the Lock and Fix commands.

The Lock command is temporary; the user can easily override the command.

The Fix command, on the other hand, must be disabled in the Edit Component dialog box. The Edit Component dialog box can be accessed by selecting a component and choosing Modify from the popup menu. The Fix command is intended for parts such as connectors and mounting holes that need to be placed permanently in specific locations.

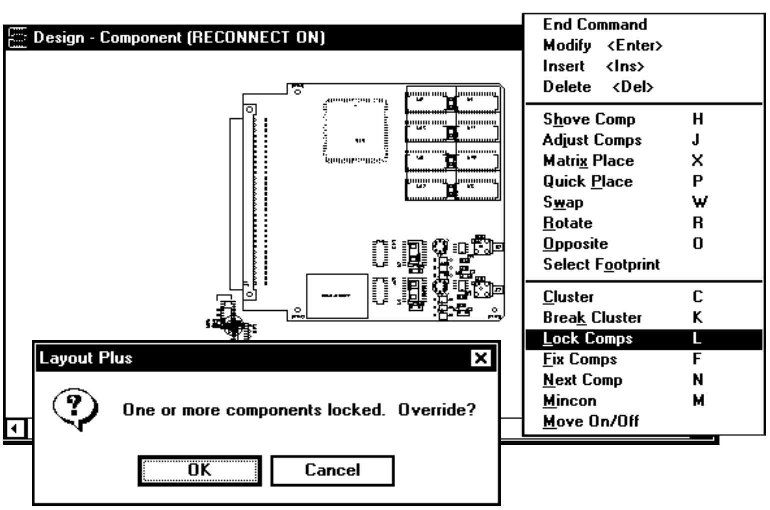
- **Lock the components on the board.**

Select the memory array in the upper right corner of the board and choose Lock Comp from the popup menu or press the L key.

Repeat this procedure for all of the parts in the lower right area of the board.

- **Display the lock override dialog box.**

Select one of the parts in the matrix. A dialog box appears, asking if you want to override the lock. Choose No. The part remains locked.



Loading a strategy file

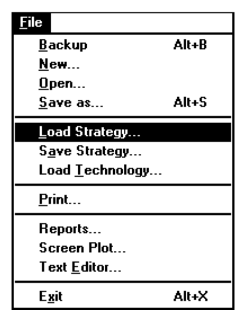
There are several placement strategy files defined for you in Layout Plus. Some of the strategy files feature quick placement to test for adequate board space. Others are designed for extensive component, pin, and gate swapping. When using strategy files, components are placed on the design for you.

For the purposes of this demo, a strategy file has been created using the Layout Plus default settings for autoplacement. These strategies assign components to clusters and place the clusters on the board taking into account any preplaced clusters or fixed components already on the board.

Layout Plus automatically groups components for placement using connectivity, unless the groups are specified by the user in the schematic or at board-level in the Components spreadsheet.

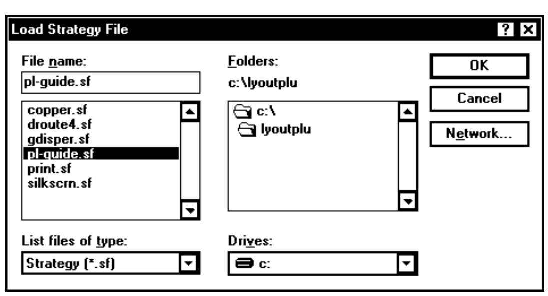
Clusters are then created from the groups to maximize the connectivity between the components. Cluster placement groups heavily interconnected components with the goal of minimizing track length and the number of vias needed to route the connections between the components.

- **Load the strategy file.**



From the File menu, choose Load Strategy to load a placement strategy for this board design.

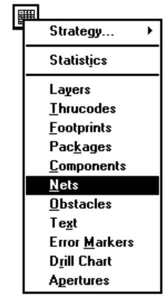
When the popup menu appears, select PL-GUIDE.SF from the list.



Disabling the power and ground signals

- **Display the Nets spreadsheet.**

  Choose the Spreadsheets toolbar button and choose Nets from the list.

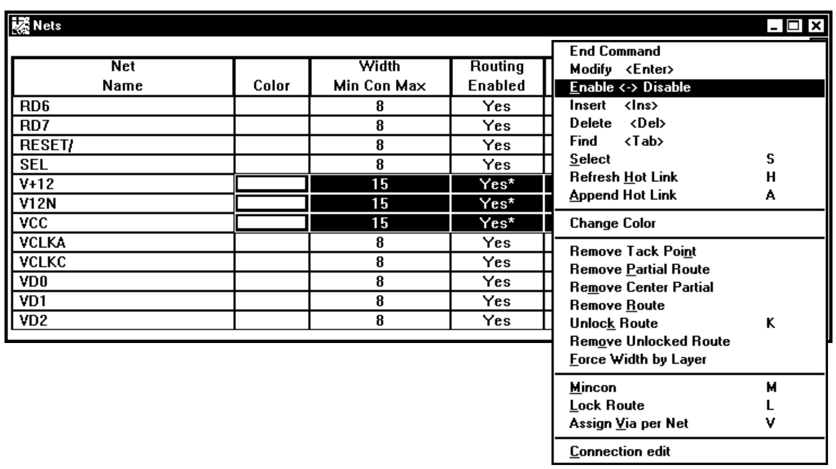


- **Disable the power and ground signals.**

  Because you are placing the components emphasizing circuit functionality, you must disable the power and ground nets. You will reenable these nets before you route the power and ground planes.

  Pressing the SHIFT key, click the left mouse button to select the following nets in the spreadsheet: AGND, GND, GND EARTH, V+12, V12N, and VCC.

  Then, choose Enable <->Disable from the popup menu to disable the selected signals. When the power and ground signals are disabled, *No* displays in the corresponding Routing Enabled column.



OrCAD Layout for Windows—autoplacement

Using autoplacement

OrCAD's Layout Plus for Windows' autoplacement utility attempts five phases of component placement.

Assign clusters. Takes components that are not locked or fixed and places them in clusters based on their interconnectivity.

Place clusters. Places clusters on the board based on their connectivity.

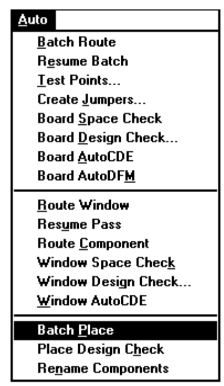
Proximity place. Places components in approximate locations.

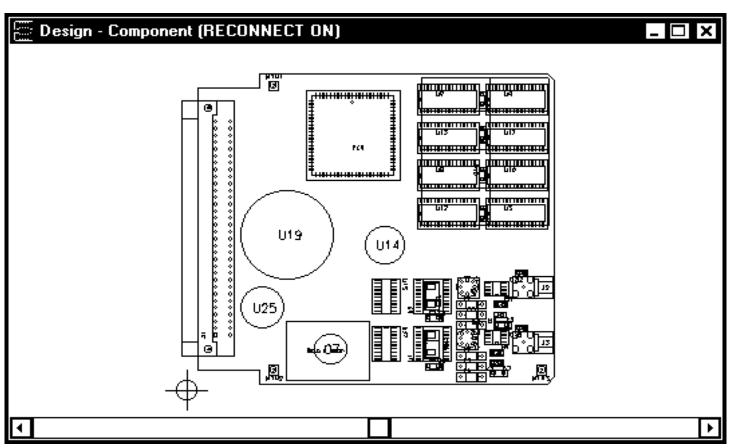
Swap components. Swaps components within a group or with components in neighboring groups in order to improve placement.

Adjust components. Moves components to avoid overlaps.

- **Begin autoplacement.**

  When you are ready to begin autoplacement, choose Batch Place from the Auto menu. The autoplacement routine works through the five phases of component placement.





Using interactive commands to optimize autoplacement

Once you have placed the components on the board using autoplacement, use the interactive placement commands discussed earlier to optimize the design.

- **Edit a component.**

  Select a component and choose Modify from the popup menu to edit the component name, lock or fix components, or change the footprint used for that component.

- **Use the shove component command.**

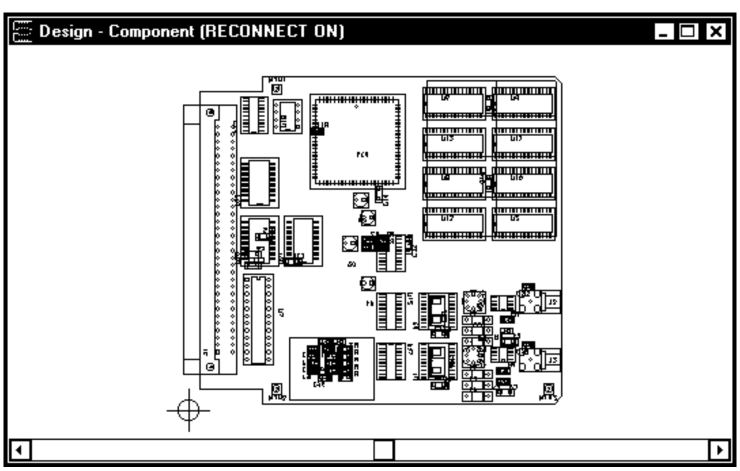
  Select a component and choose Shove Comp from the popup menu to move other components out of the way.

- **Rotate a component.**

  Select a component and choose Rotate from the popup menu to shorten connections or eliminate crossovers.

- **Place a component on the opposite side of the board.**

  Select a component and choose the Opposite command from the popup menu to place components on the opposite side of the board.



Using the density graph

Layout Plus features an innovative density graph, a powerful tool for analyzing the degree of difficulty that will be faced in routing the board using different routing strategies and technologies.

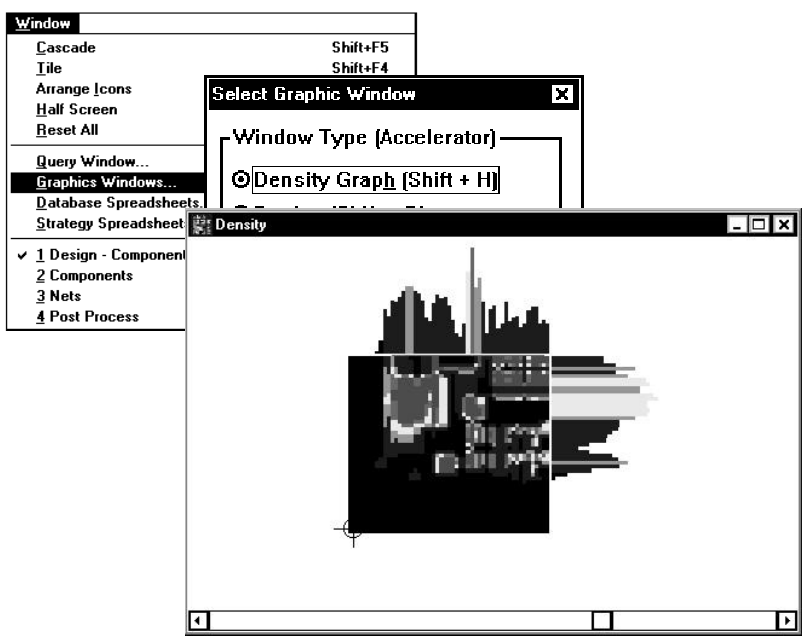
Note The density graph is only available in the OrCAD Layout Plus for Windows package.

The density graph considers a variety of design rules, including the number of routing layers, track widths, DRC settings, and whether surface mount parts are present on both sides of the design, or if the board has predispersed vias for routing.

Three kinds of data are shown on the density graph. The board display shows the board density at each location. (Board density refers to the number of pads and connections in a given area of the board). The bar graphs at the top and right show the channel counts in each direction and their density.

- **Open the Density Graph window.**

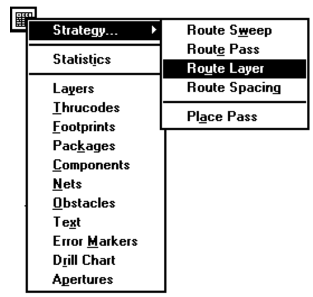
Choose Graphics Windows from the Window menu. Then select the Density Graph radio button.



- **Test routing layer strategies using the density graph.**

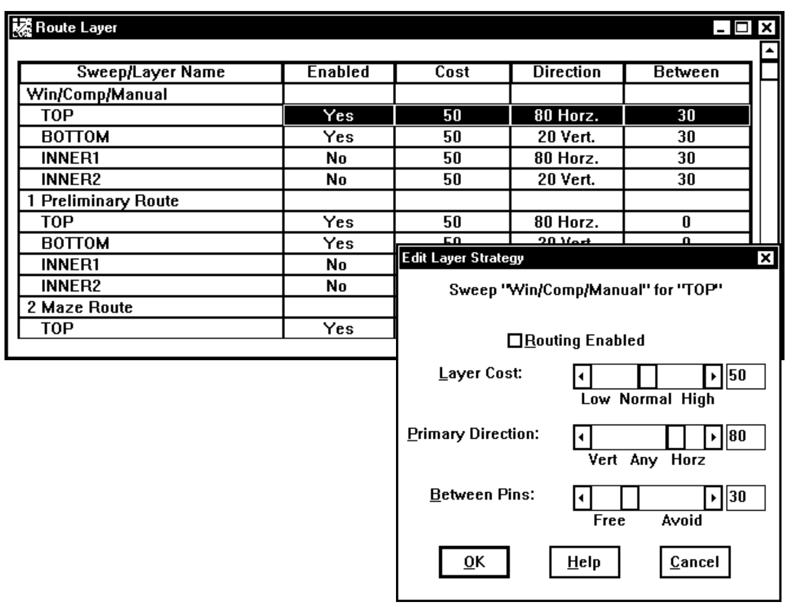
The density graph in OrCAD Layout Plus for Windows analyzes all routing layers, preroutes, widths of routes, and connections when calculating the available routing channels.

By disabling routing layers, you can test the potential results of routing the board with fewer layers. This testing capability can save days or weeks in the design process.

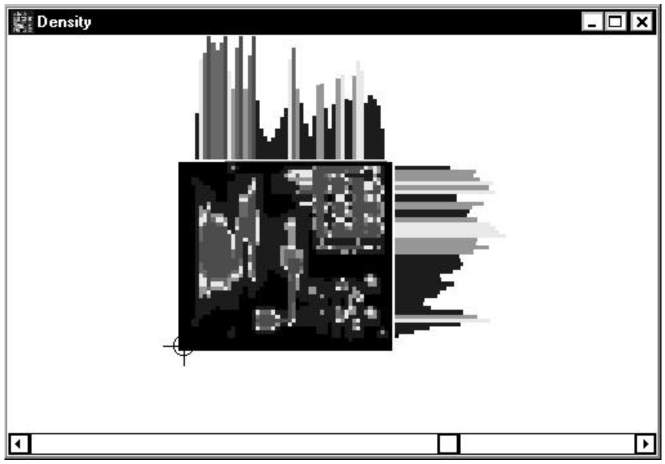


Choose Strategy and Route Layer from the Spreadsheets toolbar button to display the Route Layer spreadsheet.

In the Win/Comp/Manual section, double-click on *TOP* to display the Edit Layer Strategy dialog box. Deselect the Routing Enabled check box, turning off routing on the top layer.

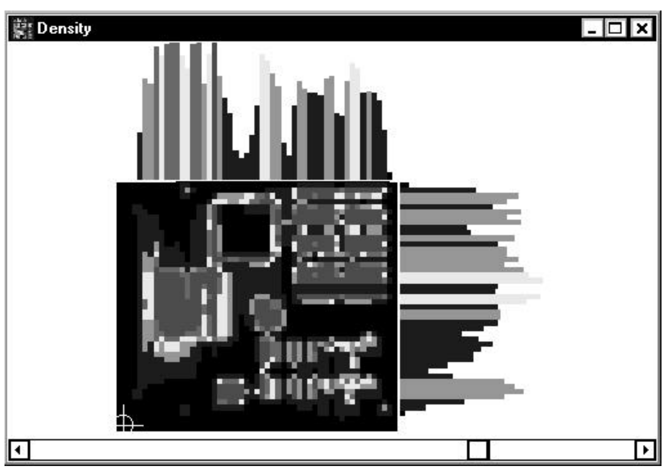


When you close the spreadsheet, the density graph redraws itself. You can see new parameters, resulting from turning off routing on the top layer.



- **Load a routing strategy file.**

Choose Load Strategy from the File menu and select DROUTE2.SF or DROUTE4.SF (depending on which board file you are using). Notice the change in the density graph.



Exiting the placement file

You have finished placing components on the board.

If you want to continue now with *Chapter 4: Interactive Routing and Autorouting* choose Graphics Windows from the Window menu. In the Select Graphic Window dialog box, choose Design.

If you want to stop, choose Exit from the File menu. Keep in mind that you cannot save the board with the current placement. However, the OrCAD Layout for Windows Demo includes a file with a pre-placed board that is ready for routing. When you are ready to continue with the demo, follow the instructions in *Chapter 4: Interactive Routing and Autorouting*.

Go to the next chapter.

Go to the table of contents.

Chapter 4
Interactive routing and autorouting

Go to the table of contents.

Go to the last page of the previous chapter.

Interactive routing

OrCAD Layout for Windows is the most productive tool you will find at any price for PCB editing and routing. Layout uses sweep technology to complete the most difficult portions of the board, and then quickly routes the rest of the board. OrCAD's shove technology minimizes vias and clears paths for tracks. By using Layout's interactive routing commands, you can access all of Layout's automatic power without sacrificing control.

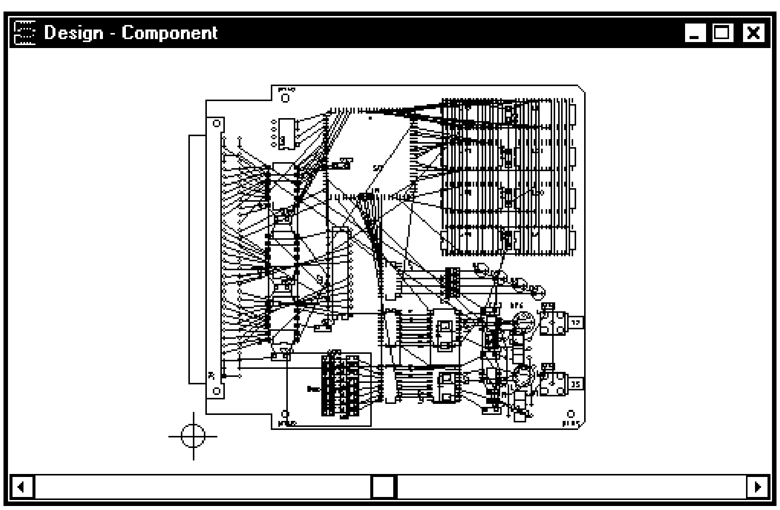
Some of the interactive capabilities you can use in this demo include an AutoPath completion mode, interactive push-n-shove for track input, manual routing with DRC on or off, and batch routing.

If you exited the placed board created in *Chapter 3: Component Placement*, choose Design from the Layout frame and select DROUTE2.MAX or DROUTE4.MAX to open the routing portion of the demo.

Note You need a minimum of 16MB of RAM to run DROUTE4.MAX.

Note If the system prompts you to add a new netlist, choose No.

If you are continuing from *Chapter 3: Component Placement* without exiting, open the Design window.



Loading a routing strategy file

When all of the components are placed, you can begin routing the board. Layout has tools that will route one signal at a time, one component at a time, an area of the board, or the entire board.

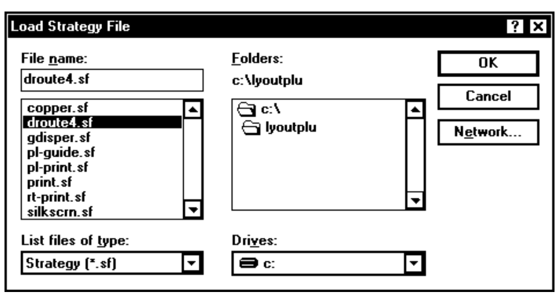
To autoroute an entire board, you must select a routing strategy, set the spacing requirements, and disable for routing all nets on plane layers.

Predefined strategy files are available in the OrCAD Layout for Windows package and as a part of the demo. The files are optimized for specific types of boards based on the type of components on the board, the number of layers enabled for routing, and the preferred track direction on the top layer.

- **Load the strategy file.**

Choose Load Strategy from the file menu.

If you are routing a 2 layer board (using the file DROUTE2.MAX or the equivalent) select DROUTE2.SF from the list. If you are routing a 4 layer board (using the file DROUTE4.MAX or the equivalent), select DROUTE4.SF from the list.



Editing nets

Most net data is established on the schematic level. However, these rules can be enhanced or modified at any time during the design process.

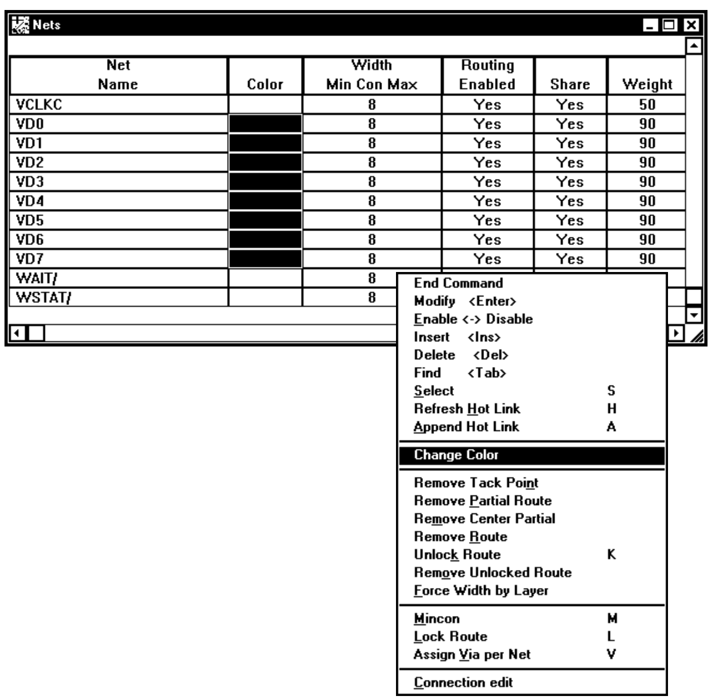
- **Open the Nets spreadsheet.**

Choose the Spreadsheet toolbar button and choose Nets, or press the SHIFT and N keys to display the Nets spreadsheet.

- **Change the color of the ratsnest.**

On the design, the ratsnest is highlighted in yellow. Pressing the SHIFT key, select VD [0-7] and choose Change Color from the popup menu. Select a color from the palette that displays.

It is also possible to highlight the entire net's ratsnest. This can be useful for following a critical line, such as a clock line. This is different than highlighting routed tracks on the design.



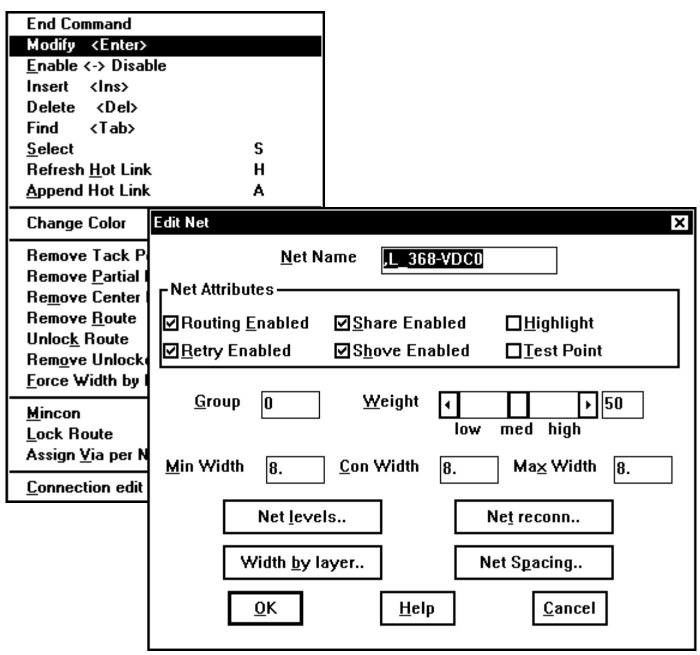
- **Edit the nets.**

  In OrCAD Layout for Windows, you can set net rules on a net-by-net basis.

  Select a net in the Nets spreadsheet and choose Modify from the popup menu to display the Edit Net dialog box.

  In the Edit Net dialog box, you can set the following rules:

  - Net weight is the priority a net is given for routing. The higher the weight, the sooner it will be routed.
  - Share Enabled allows T-connections to be used on the board. Disabling this option forces nets to go to pads only.
  - Shove Enabled allows the selected net to be moved to create space for other tracks.
  - Retry Enabled allows the router to reroute the net to create room for another track. (If Retry and Shove are both disabled, then the tracks for this net are essentially locked in place).

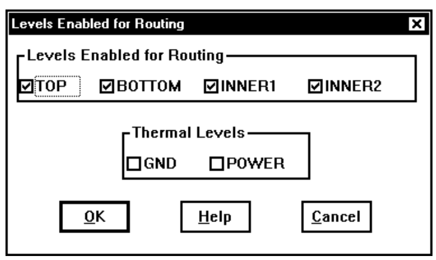


- **Enable layers for routing.**

  In the Levels Enabled for Routing dialog box, you can specify the layer(s) on which you want to route a track. You can also assign a net to a plane layer.

  This option is very valuable for nets that can only be routed on certain layers. If you specify those layers here, then the autorouter will not put a track on a disabled layer. Also, the Design Rule Check will flag any net that is interactively routed on a disabled layer.

  To access the Levels Enabled for Routing dialog box, click on the Net levels button. For the purposes of the demo, close the dialog box without editing the options by clicking on the Cancel button.

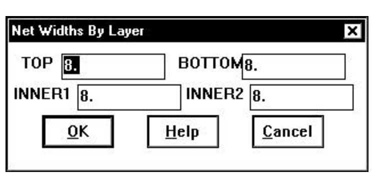


- **Set net widths by layer.**

  In the Net Width by Layer dialog box, you can set a specific track width for each layer for each net. This feature is especially useful for impedance controlled boards.

  If the width of a net differs from this value, the Design Rule Check reports the discrepancy as an error.

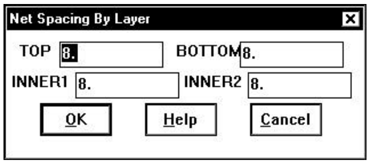
  To access the Net Widths by Layer dialog box, click on the Width by layer button. For the purposes of the demo, close the dialog box without editing the options by clicking on the Cancel button.



- **Set net spacing by layer.**

  Using the Net Spacing By Layer dialog box, you can select a minimum track-to-track setting that is larger than the global DRC parameters. This function is useful for analog boards and boards that must recognize UL-type requirements. The Design Rule Check issues an error message if the specified minimum is violated.

  To access the Net Spacing By Layer dialog box, click on the Nets Spacing button. For the purposes of the demo, close the dialog box without editing the options by clicking the Cancel button.

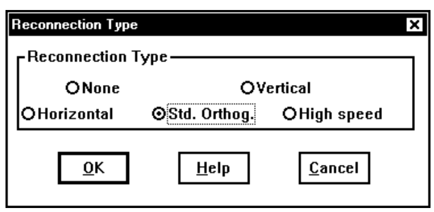


- **Set signal routing order.**

  You control the signal routing order using the Reconnection Type dialog box.

  - Use the None option to maintain the schematic net order.
  - Use the Std. Orthog option for analog boards on which line length is more critical than maintaining a specific direction.
  - Use the High speed option to route a signal from the source through the loads, and finally, to the terminator. Pins must be specified as *source*, *load*, and *terminator* in the schematic.

  To access the Reconnection Type dialog box, click on the Net Reconn button. For the purposes of this demo, close the dialog box without editing the options by clicking the Cancel button.



- **Close the Nets spreadsheet.**

  Once net parameters have been set, individual nets can be routed using one of the several routing tools Layout provides.

Using the Manual Route with Shove command

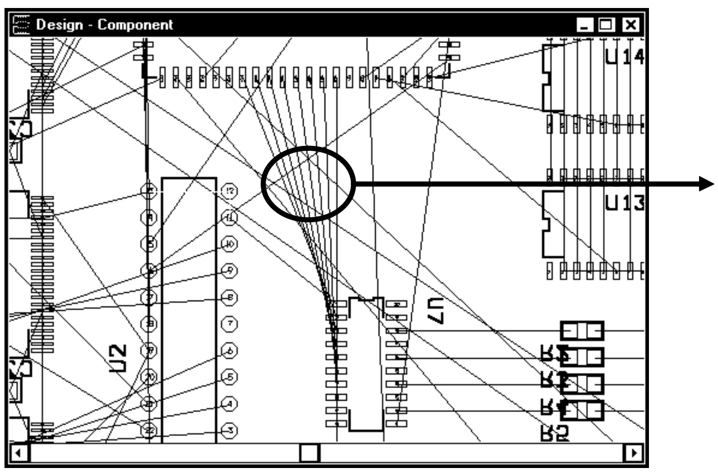
The Manual Route with Shove command uses OrCAD Layout for Window's push-n-shove technology, which simplifies routing by moving tracks to clear the design rules. Since the router will push several tracks at once, tracks are routed more quickly.

- **Locate and magnify the area to route.**

Choose the Manual Route with Shove toolbar button.

Choose Zoom In from the View menu. Using the left mouse button, click on the screen to magnify the components in the center of the board, just below the PLCC.

Note By default, the DRC (Design Rules Check) is always on for routing. If for some reason you need to temporarily disable Design Rules Checking, choose the DRC off toolbar button (it is the first toolbar icon on the left).



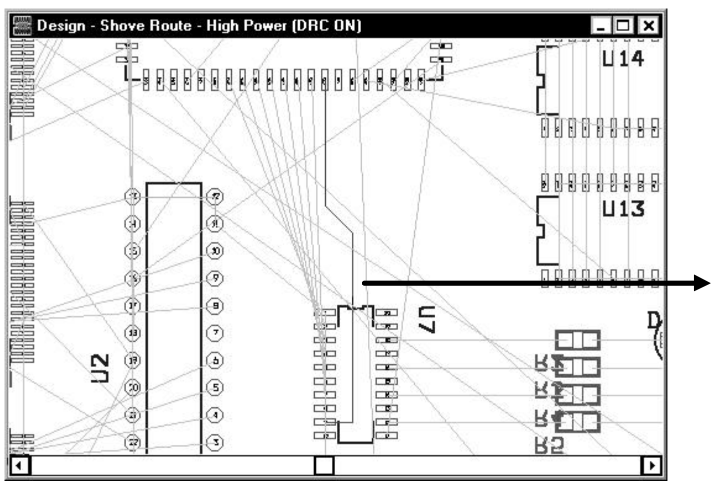
Route the ratnests here using the Manual Route with Shove tool.

- **Route the first track.**

Select a ratsnest with the left mouse button. The ratsnest is attached to the cursor when it is selected. Drag the cursor to draw the track on the board.

Click the left mouse button to create vertices (corners) in the track. Near the last segment for the connection, the tool automatically finishes the connection to the center of the pad. A complete connection is indicated by the cursor changing size (bigger) and the ratsnest disappearing from the cursor.

Tip You can add a vertex by positioning the cursor at the desired position and pressing the SPACEBAR.



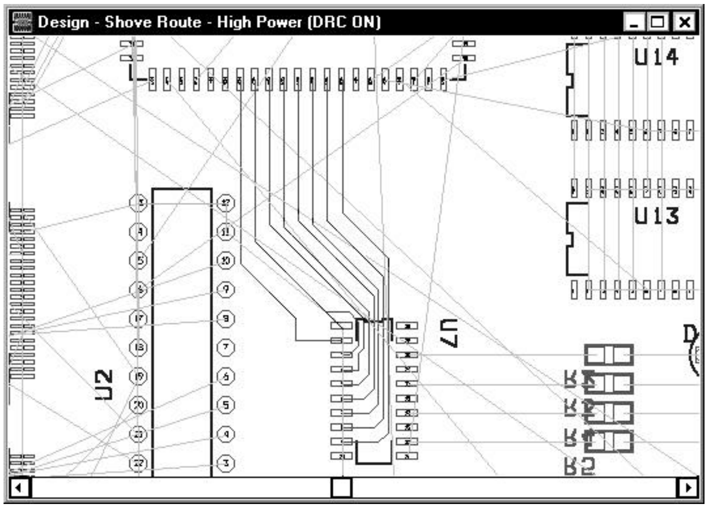
The first track.

- **Route the remaining tracks.**

Input tracks from the PLCC to the surface mount (SMT) part.

As you input the tracks, notice how the push-n-shove utility moves the other tracks out of the way.

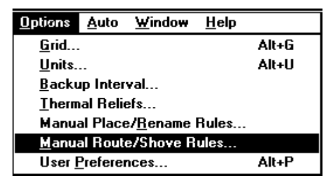
You can delete a routed segment by placing the cursor over a segment of the net and pressing the DELETE key.



- **Experiment with different manual route strategies.**

You can control the amount of shove you wish the router to use when you are routing tracks.

To achieve more control over the shove capability, choose Manual Route/Shove Rules from the Options menu. From the Manual Route Strategy dialog box, select Medium Power or Low Power. Experiment with the settings.

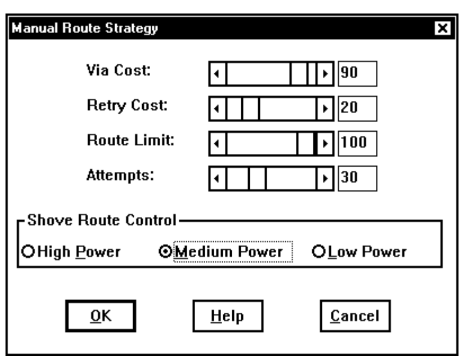


High Power. The router may rip-up, shove, and re-route tracks as you add new tracks.

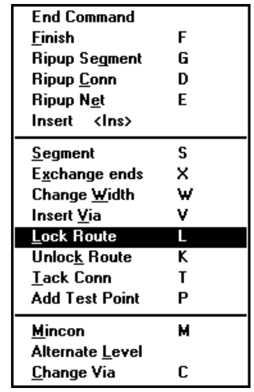
Medium Power. The router shoves tracks and may even push routes over other items such as pads and around other tracks.

Low Power. The router moves tracks only slightly.

Whether or not a via is used to make a connection depends on the Via Cost selection you set in the Manual Route Strategy dialog box.



- **Lock the routes.**



The Lock Route command locks all of the corners of the track, from the current segment to the pad, or if the track is complete, the entire track.

A track or net that is locked cannot be moved by the autorouter, but the lock can be overridden during interactive routing. It is especially helpful to lock critical tracks that are routed along a specific path.

On each of the tracks you have inserted, select a segment and choose Lock Route from the popup menu, or press the L key, to lock the routes.

Entire nets can also be locked from the Nets spreadsheet.

Note Other commands are also available for use with Manual Route with Shove. Choose the Manual Route with Shove toolbar button and display the popup menu to view the commands. This menu is also shown above.

Performing manual routing using AutoPath

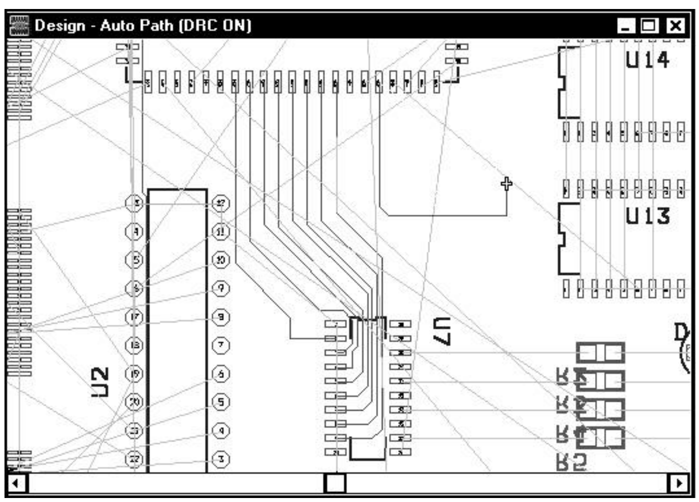
In AutoPath, the router creates the routing path for the selected net. In other words, the router shows you a way to get to the target location rather than you creating the path by drawing it with the cursor.

Using AutoPath, you can focus on the direction of the route while the system automatically decides the best location for corners. This tool is a fast way to route individual tracks.

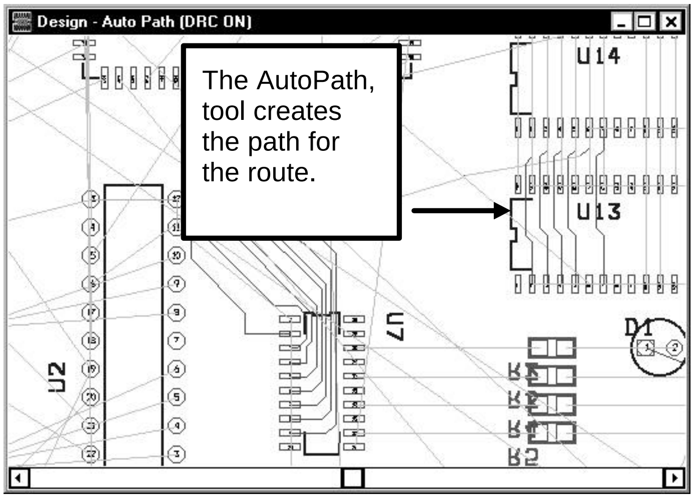
- **Use AutoPath to create a route.**

Choose the AutoPath toolbar button. There are three ways to complete a track using AutoPath.

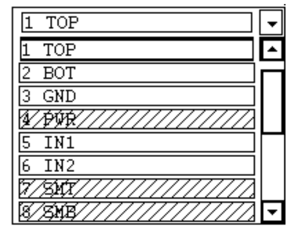
- Select a net and move the cursor on the screen. Click the left mouse button or press the SPACEBAR to add a vertex.



- Using the left mouse button, select a net on the IC that lies just to the upper right of the tracks you have entered. From the popup menu, choose Finish.
- Double-click on the net using the left mouse button. The router works in combination with the push-n-shove algorithm to complete the connection.



Changing layers



As tracks are manually routed, it is important to be able to change layers easily during routing. In Layout, a simple key stroke changes the routing layer, and vias are added automatically.

You can see the current layer in the Layer drop list located in the middle of the toolbar. The numbers that correspond to the layers are keystrokes that you can use to display layers quickly during board layout.

- **Change the routing layer.**

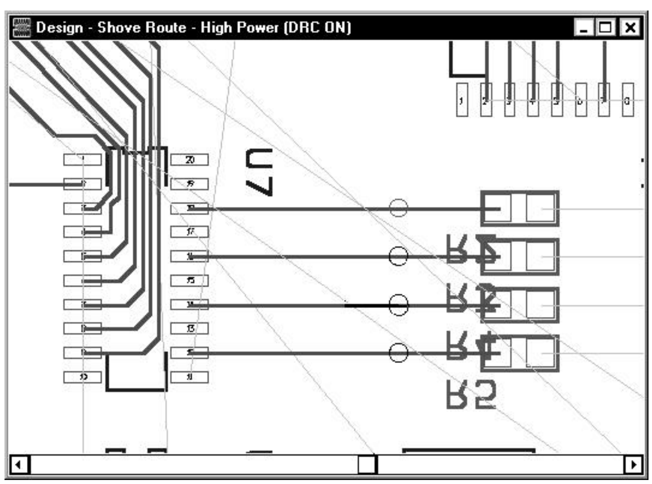
  Change to the Bottom layer by selecting 2 BOT from the layer drop list or by pressing the 2 key. Then, choose the Manual Route with Shove toolbar button.

- **Create vias.**

  To the right of the ICs you routed first, there are four resistors on the solder side, or bottom, of the board. Select a ratsnest track from one of the resistors. Begin routing the track by dragging the cursor to the left.

  Click the left mouse button. Then, press the 1 (one) key to change the routing layer to TOP. Continue dragging the cursor to the left. Click the left mouse button to place the track on the pad.

  Repeat the process for the other three resistors. When you are finished, you will have created four vias.

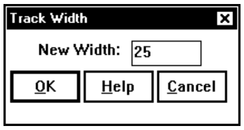


Changing widths

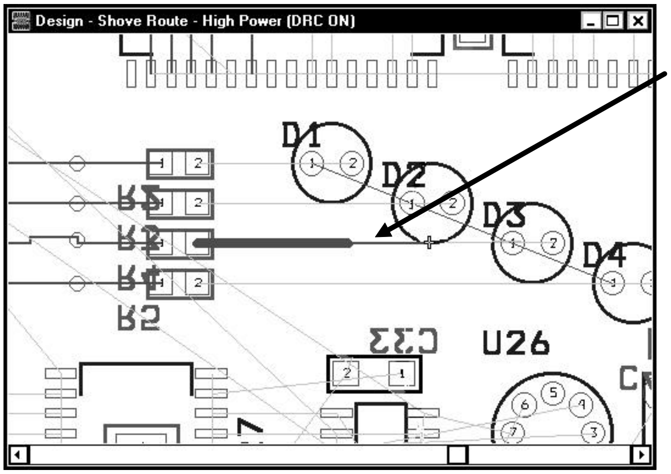
Earlier, you learned how to change the width of nets using the Edit Net dialog box. You can also easily set or change a track width directly from the board, either before or after you input the track.

For a pre-existing track, you have three options: you can change the width of a segment, the width of the connection between two pads, or the width of the entire net.

- **Change the width of a new net.**



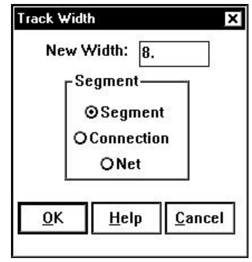
Place the cursor in the design window and press the W key. Enter 25 in the New Width text box. Drag the cursor to begin drawing a track and click the left mouse button to create a vertex. Again, press the W key. This time, enter a value of 8 in the text box. The track will have two segments: one with a width of 8, and one with a width of 25.



A net with two track widths.

- **Change the width of an existing net.**

  Select a net and press the W key. Enter 25 in the New Width text box. In the Segment group box, apply the width to a segment, connection, or the entire net.



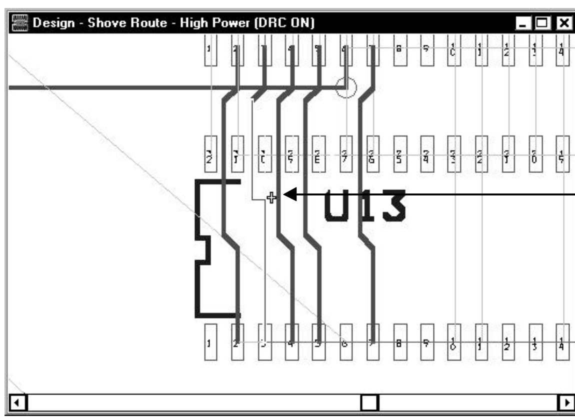
Moving segments

In OrCAD Layout for Windows, there are several ways to edit existing tracks. For example, segments can be moved or added. These options make track editing and board cleanup fast and easy.

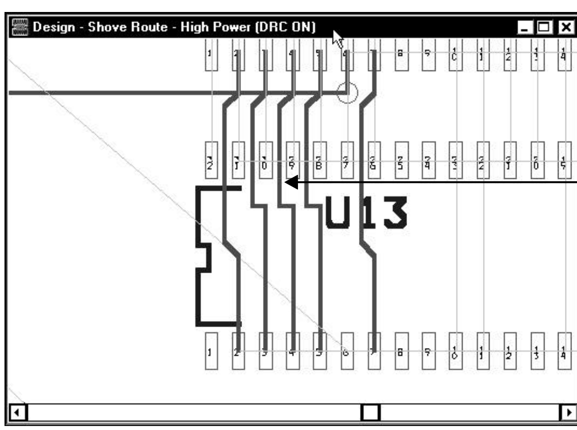
- **Move a segment.**

Choose the Manual Route with Shove toolbar button. Then, select an existing track with the left mouse button.

Choose Segment from the popup menu or press the S key to move the segment. Drag the cursor until you have created the desired shape. Click the mouse button to place the segment. Neighboring tracks are shoved to make room for the new segment.



Draw the segment.



Other segments are shoved out of the way.

Creating duplicate connections

In the Design window you have the ability to insert a duplicate connection from a node (pad), a vertex, or a corner. Using this ability, you can insert guard ring connections for shielding, special routing requirements, or to split nets.

- **Magnify the target pads.**

  Choose Zoom In from the View menu and click on the screen with the left mouse button to magnify the lower right hand corner of the board. Continue to zoom in until you see the pad J2 or J3 as shown below.

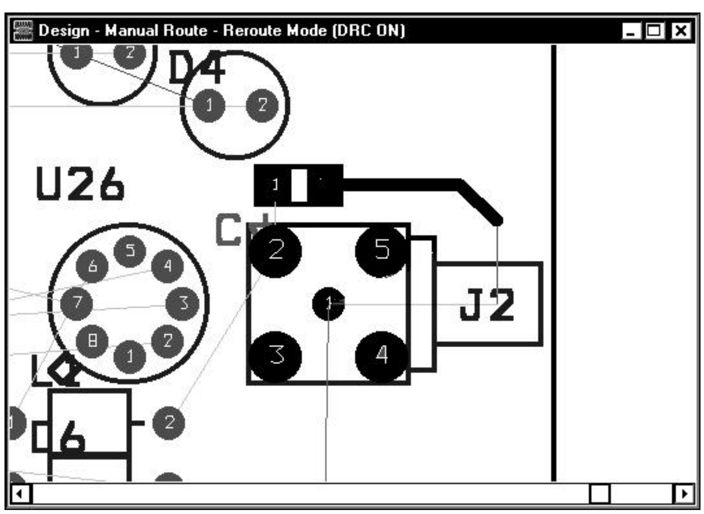
- **Set a net width.**

  Choose the Manual Route with Shove toolbar button.

  Select a connection with the left mouse button and choose Change Width from the popup menu or press the W key. Enter a new width value of 25.

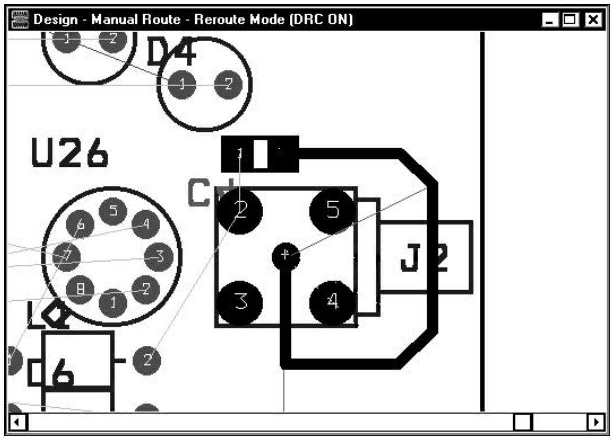
- **Draw the track.**

  Begin inserting a track to the right of the node by dragging the cursor. Click the left mouse button to create vertices.



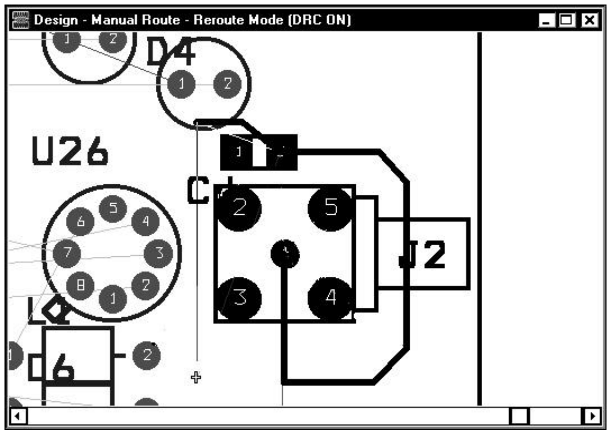
- **Establish a duplicate connection.**

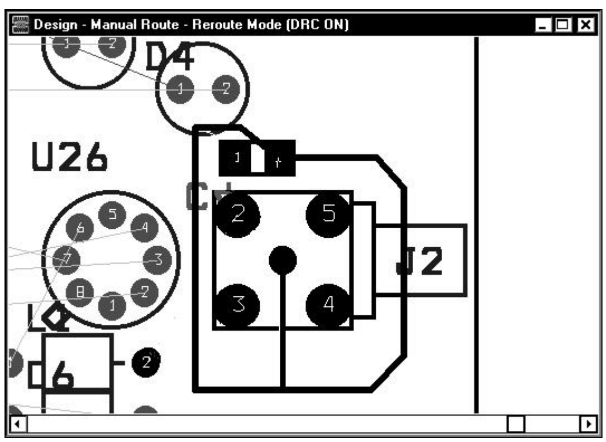
  Choose Insert from the popup menu or press the INSERT key to establish a duplicate connection



- **Create a duplicate track.**

  Continue to input the track. As you now have a duplicate connection displayed from the original pin, you can select that connection and draw another track around the left of the large pads.



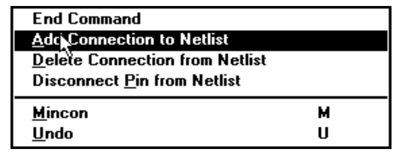


Creating nets interactively

In OrCAD Layout for Windows, you can create nets interactively. This is "true" manual routing.

- **Create a net.**

Choose the Create/Modify Nets toolbar button and choose Add Connection to Netlist from the popup menu.

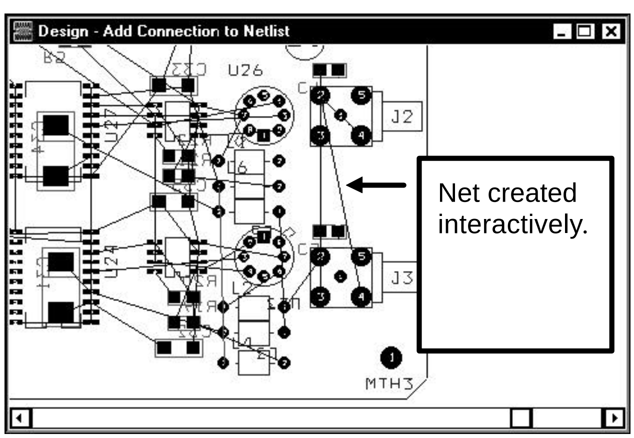


Note When you select this option, Layout reminds you that, although you are adding the net to the board in Layout, the change will not be reflected back to the schematic design.

Select a pin from the J2 component, for example, pin 4. Select a pin that is in the signal GND EARTH, for example, J2, pin 2 or J3, pin 2.

If the nodes are already in a signal, you receive a message asking you to confirm the combined nets. Once you confirm, you are asked to assign a net name to the new nets. You can confirm or cancel at this time.

If you type a net name that already exists, you receive a message: "Net already exists. Do you wish to tie to it?" Choose the Yes button. The system automatically ties the pin to the net.

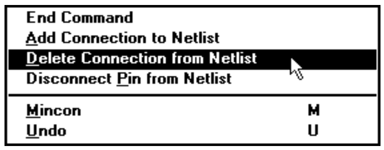


Splitting nets

You can also separate a net into two separate nets interactively.

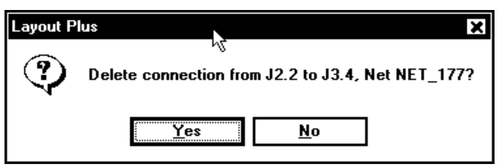
- **Split a net.**

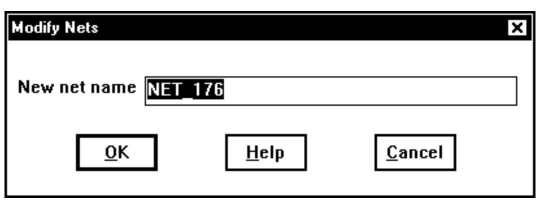
Choose Delete Connection from Netlist from the popup menu.



In the board design, select a net to split into two separate nets. (Do not select a pin at the end of the signal.)

Layout asks you to confirm your decision to delete the connection. If you answer Yes, you are asked to name the new nets individually.

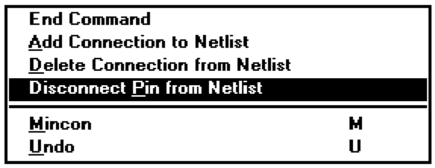




Disconnecting pins

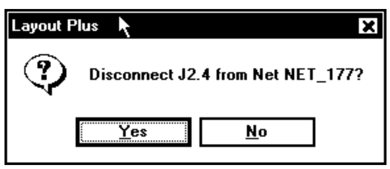
In OrCAD Layout for Windows, you can easily remove pins from a net without splitting the net.

- **Disconnect a pin from the net.**



Choose Disconnect Pin from Netlist from the popup menu. Select the pin you connected to the GND signal, J2, pin 4.

The system asks if you want to disconnect the selected pin from the net. Choose the Yes button to disconnect or delete the pin from the net.



Autorouting

We will now demonstrate autorouting in OrCAD Layout for Windows.

Note If you wish to stop before continuing with autorouting, choose Exit from the file menu. Reload DROUTE2.MAX or DROUTE4.MAX when you are ready to begin autorouting.

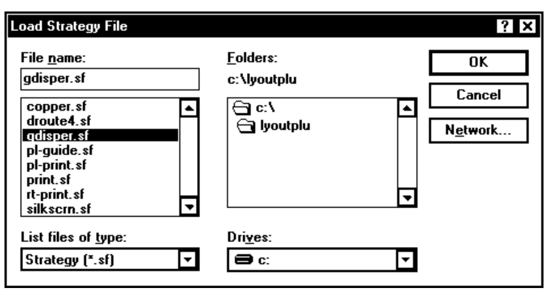
Loading a strategy file

Routing strategies can be used to provide part of the autorouting solution. Layout includes several strategy files that are optimized for specific types of boards based on the types of components used, the number of board layers, and the preferred track direction for the top layer.

- **Load the strategy file.**

Choose Load Strategy from the File menu.

As the demo design has a power and a ground layer, you must first disperse the vias to these planes to complete the necessary connections. Select the strategy file, GDISPER.SF, which has been defined for the dispersion of vias for the power and ground planes.

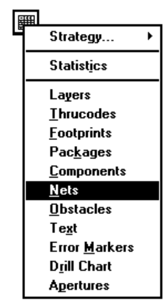


Dispersing vias

Once net parameters are set, individual nets can be routed using one of the several routing tools Layout provides.

During the placement section of the demo, you disabled the power and ground layers for routing. Open the Nets spreadsheet to check the status of these signals. There should be a *No* in the Route Enable column for all of the power and ground signals: AGND, EARTH GND, GND, V+12, V12N, and VCC. If the signals are not disabled, do so as described in *Chapter 3: Component Placement*.

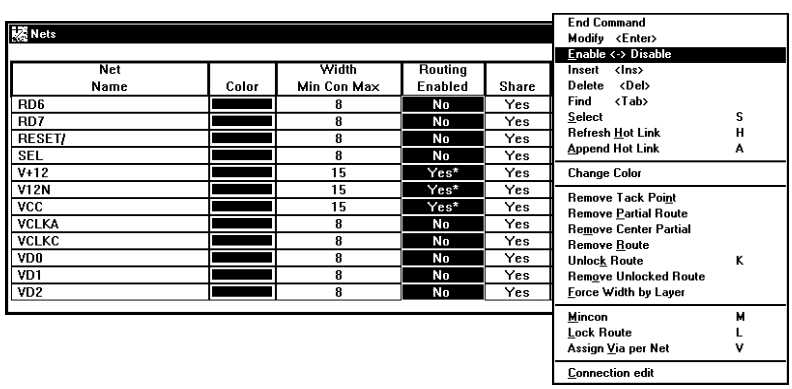
- **Enable routing on only the power and ground nets.**



To disperse the vias, you must switch the nets for routing, enabling the power and ground nets for routing and disabling all others.

In the Nets spreadsheet, select the entire Routing Enabled column. Choose Enable <-> Disable from the popup menu.

Now, the power and ground signals should be the only nets with *YES* in the Routing Enabled column. The only ratsnest of signals you should see on the screen are the power and ground signals



Note The ratsnest for the GND and POWER signals are displayed in the color assigned to that plane layer.

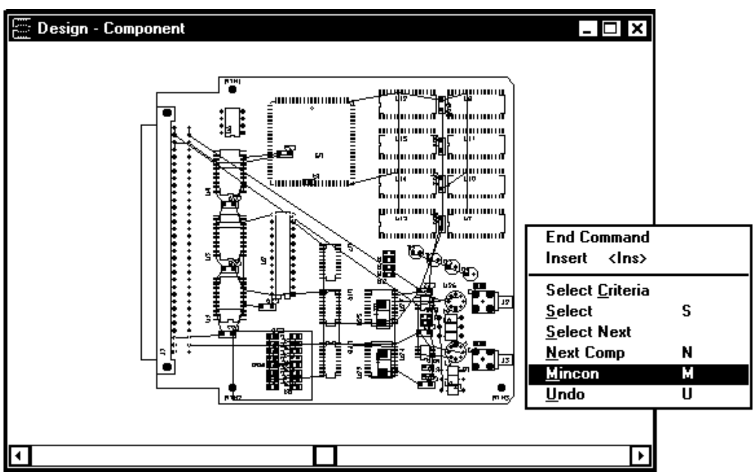
- **Minimize the connections on the power and ground planes.**

  To ensure that the board design has the shortest possible connection within each signal, you can minimize the connections using the Mincon command.

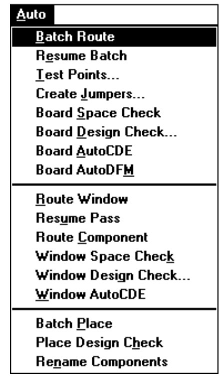
  Mincon evaluates the connections within a signal and finds the shortest route for the signal based on the placement of the pins or components.

  In other words, as the components are now placed, it may be easier to route the track from U1-2 to U16-20 to U3-2 instead of following the order as it appears in the schematic netlist.

  Choose the Component toolbar button. Place the cursor in the Design window and choose Mincon from the popup menu.



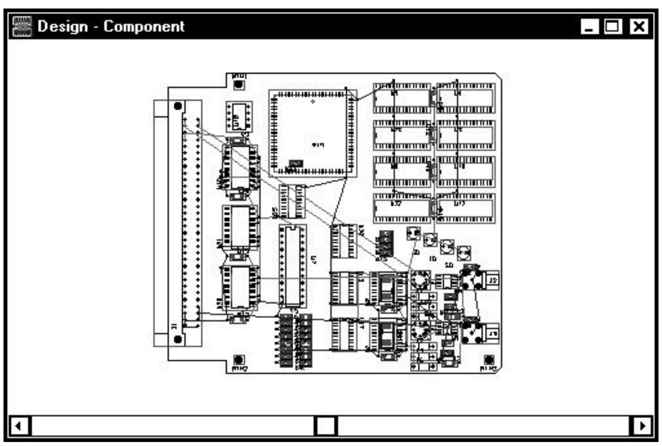
- **Disperse the vias.**



Since you have enabled only the power and ground signals for routing, and the strategy you selected restricts the routing routine to short connections, you can disperse all of the signals, through vias, to the appropriate plane.

Choose Batch Route from the Auto menu.

When the message, “All Sweeps Completed [or disabled]” appears, choose the OK button. Only the power and ground signals should be routed.



Note In some instances, not all of the signals experience successfu via dispersion. The system checks the design rules and the component layout to determine where vias can be placed. Circles indicate areas in which it is not possible to place vias. You can then insert vias interactively, or reset the dispersion strategies and run the Batch process again.

Note Signals can be assigned to planes at the schematic level. Or, you can select power or ground in the Nets spreadsheet and set the appropriate signal using the net layer parameter.

Autorouting the entire board

To autoroute an entire board, you must disable the plane layer nets for routing, set the spacing requirements, and select a routing strategy.

- **Enable the layers for routing.**

  Open the Nets spreadsheet and select the Routing Enabled column. Choose Enable<->Disable from the popup menu. Now the nets to the plane layers are disabled for routing, and the other nets are enabled for routing.

- **Minimize connections.**

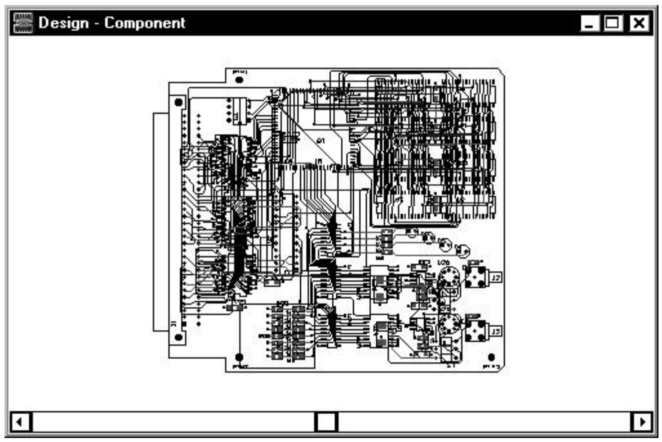
  Position the cursor in the Design window, and choose Mincon from the popup menu.

- **Load a strategy file.**

  Choose Load Strategy from the File menu. Select DROUTE2.SF or DROUTE4.SF.

- **Autoroute the board.**

  Choose Batch Route from the Auto menu to autoroute the rest of the board.



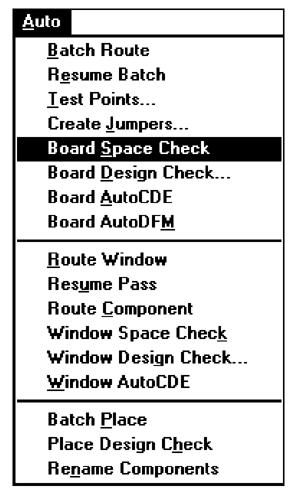
Note If you wish to stop the routing process at any time, press the ESC key.

Running Design for Manufacturability (DFM) tests

Layout uses shove technology to minimize vias for the efficient use of board space. After the board is routed, you can run via reduction passes to reduce the vias even further. Via counts are typically 30% to 50% lower with Layout than with other PCB design programs.

There are also design management utilities that you can run to enhance a design's manufacturability: Board Space Check, Board Design Check, Board AutoCDE, and Board AutoDFM.

- **Run Board Space Check.**



Board Space Check verifies board spacing. Layout does not allow for spacing errors while performing interactive or automatic routing. If you have turned off DRC at any point in the interactive process, you should run Board Space Check. Any problems are marked by a circle and can be queried using the Error tool.

To run Board Space Check, choose Board Space Check from the Auto menu.

- **Run Board Design Check.**

Board Design Check verifies that the design adheres to specific rules specified in the Design Rules dialog box. For example, nets are checked for dispersion to the plane. The utility also checks for proper pad exits for routing.

To run the Board Design Check, choose Board Design Check from the Auto menu. In the Design Rules dialog box, select the type of checks you wish to run.

Via Location Errors. Checks for vias in via-restricted areas.

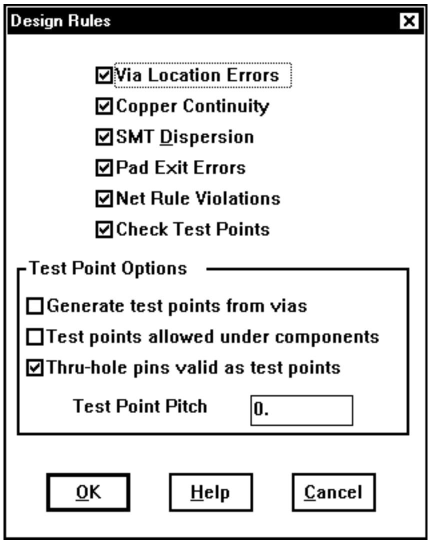
Copper Continuity. Checks for copper areas that are not attached to a net.

SMT Dispersion. Ensures that surface mount pads connected to nets on plane layers have vias to get to the plane.

Pad Exit Errors. Checks for problems such as tracks exiting the sides of surface mount IC pads.

Net Rule Violations. Checks for discrepancies in layers or widths between what exists and what is specified in the Nets spreadsheet.

Check Test Points. Confirms that any net that has been designated to have a test point has a test point.

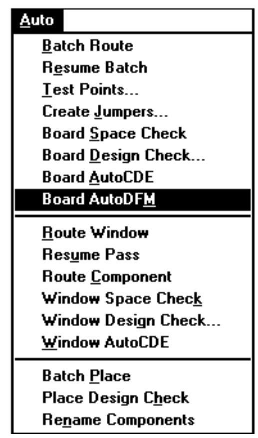


- **Run Board AutoCDE.**

  Board AutoCDE automatically sweeps through the entire design and rips up any routing that violates design rules. By running Board AutoCDE, you can assess the impact of ECO changes and create a clean design for rerouting.

  To run Board AutoCDE, choose Board AutoCDE from the Auto menu.

- **Run Board AutoDFM.**

  Board AutoDFM automatically smoothes, miters corners, and checks for aesthetics and manufacturing problems. The problems that are checked for and corrected include off-grid 90 degree angles, acute angles, bad copper share, pad exits, and overlapping vias. Any problem DFM cannot fix is marked with an error circle.

  To run Board AutoDFM, choose Board AutoDFM from the Auto menu.

Exiting the routing file

You have finished routing the board.

You may continue now with *Chapter 5: Thermal Reliefs and Copper Pour Zones* or, if you want to stop, choose Exit from the File menu.

Keep in mind that you cannot save the routed board. However, the OrCAD Layout for Windows Demo includes a file with a fully routed board that you can use for subsequent activities. When you are ready to continue with the demo, follow the instructions in *Chapter 5: Thermal Reliefs and Copper Pour Zones.*

Go to the next chapter.

Go to the table of contents.

Chapter 5
Thermal reliefs and copper pour zones

Go to the table of contents.

Go to the last page of the previous chapter.

Thermal reliefs

In OrCAD Layout for Windows, you control the placement of thermal relief connections. For example, some companies require that ICs receive power directly from capacitors and not from the plane. If a track is routed between a capacitor and the IC power pin, you can choose which pin connects to the plane. You set these rules in the footprint library.

Viewing thermal relief planes

In OrCAD Layout for Windows, thermal reliefs are created automatically when a net is assigned to a plane. You can view the plane connections on the screen without any kind of post processing function, making it easy to confirm them.

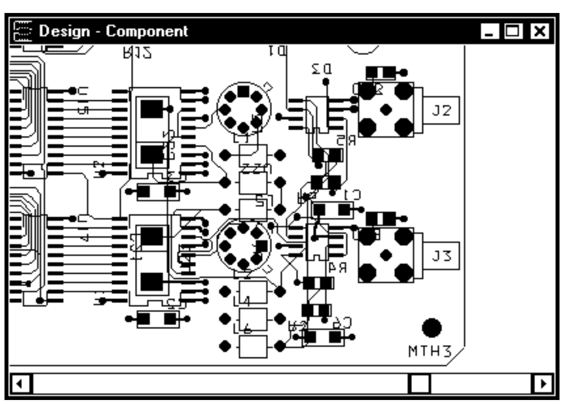
To view the thermal relief plane, you need to make the plane layer visible and the other layers in the design invisible. You can choose to view certain layers by assigning colors to the layers or designating them as visible or invisible.

- **Open a design file.**

Open the file FINISH2.MAX or FINISH4.MAX.

- **Magnify the area to view.**

Choose Zoom In from the View menu and magnify the lower right corner of the board.

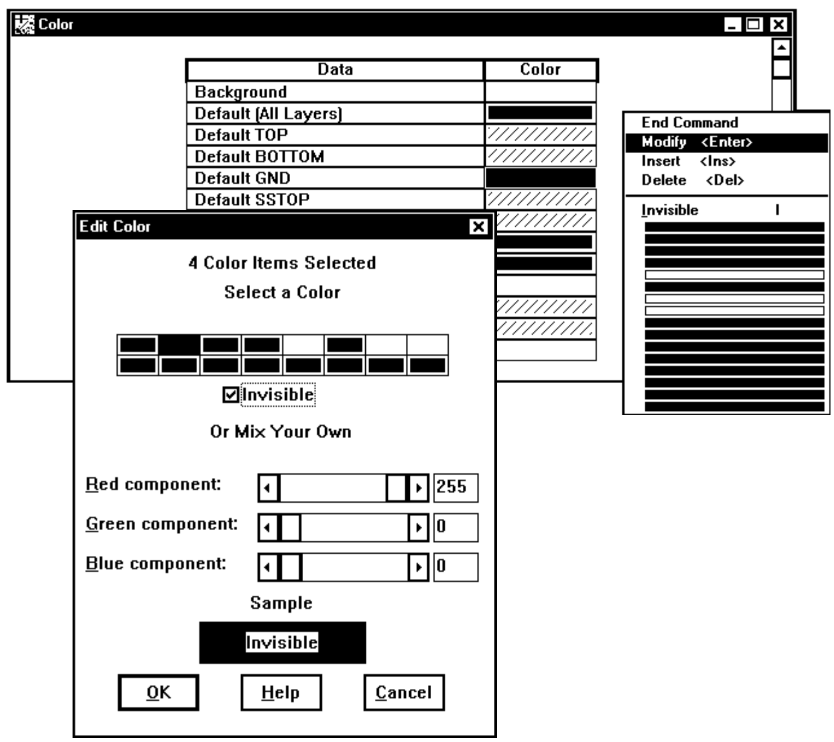


- **Make the top, bottom, and silkscreen layers invisible.**

  Choose the Color toolbar button.

  In the Color spreadsheet, select the top, bottom (inner layers if you are using four routing layers), and silkscreen (SST and SSB) layers. Choose Modify from the popup menu.

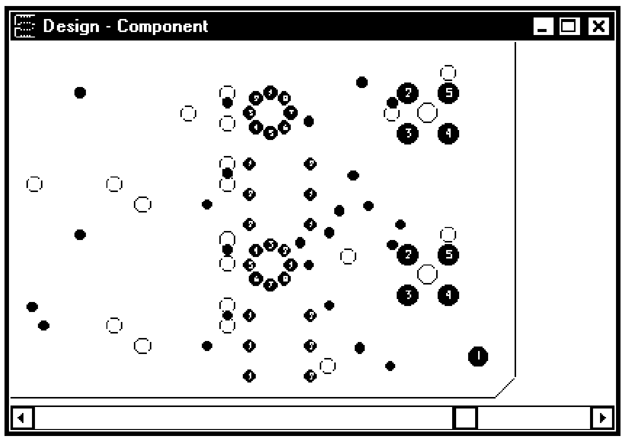
  In the Edit Color dialog box, select the Invisible check box, and from the palette, select a color for the layers.



- **Make the GND plane visible.**

In the Colors spreadsheet, select the GND layer. Again, choose Modify from the popup menu. In the Edit Color dialog box, deselect the Invisible check box and from the palette, select a color for the GND layer.

When you close the Color spreadsheet, the thermal relief connections are displayed on the GND plane.



- **Choose to view the top layer only.**

Next, you will create copper pour zones on the board. It is easiest to view the copper pour zones with only the top layer visible. When you have finished viewing the thermal reliefs, make the top layer visible and the GND layer invisible.

Copper pour zones

A copper pour zone is used to place copper in designated areas. Copper pour also places thermal reliefs on pads and eliminates copper islands. A copper pour zone can be placed on any layer (except the plane layer), can be solid or cross hatched, and can be attached to any net. Copper pour attached to a net assumes the attributes of that net.

There are three types of copper pour zones:

Copper zone. You can use copper zones to create custom pad shapes. Isolation rules do not apply to copper zones.

Copper pour zone. Copper pour zones avoid pins and tracks but attach to pins with a thermal relief.

Anti-copper zone. Use anti-copper zones to create non-copper areas within copper pour zones.

Creating copper pour zones

In OrCAD Layout for Windows, you can create a copper pour zone by drawing and modifying an obstacle.

- **View the top layer.**

Select Top from the layer drop list on the toolbar.

- **Designate a seed point.**

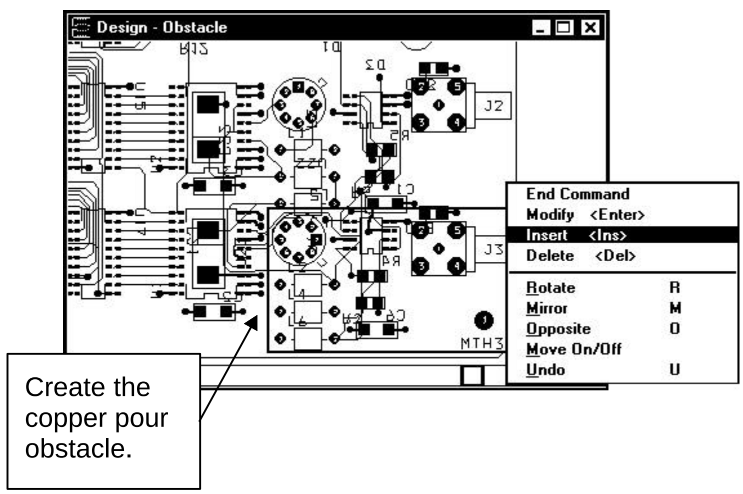
  When you are creating a copper pour zone in OrCAD Layout for Windows, you must designate a seed point. The seed point is the point from which the copper pours.

  Choose the Pin toolbar button. Select a pin attached to the net to which you want to attach the copper pour zone. From the popup menu, choose Toggle Copper Pour Seed to set the copper pour seedpoint.

- **Create the copper pour obstacle.**

  Choose the Obstacle toolbar button. Then, choose Insert from the popup menu.

  Pressing the left mouse button, drag to create a rectangle in the bottom right corner of the board as shown below.

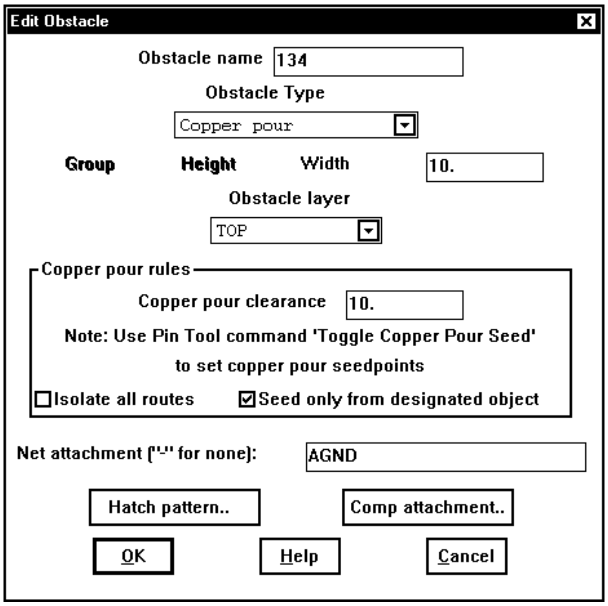


- **Define the copper pour obstacle.**

While pressing the SHIFT key, select the obstacle that you have drawn by clicking on it with the left mouse button. Choose Modify from the popup menu.

In the Edit Obstacle dialog box specify the following parameters for the copper pour zone.

- From the Obstacle Type drop list, select Copper Pour.
- From the Obstacle layer drop list, select Top.
- Change the Obstacle Width to 10.
- In the Net attachment text box, enter AGND. Copper Pour attached to a net assumes the attributes of that net.



- **Set rules for the copper pour zone.**

In OrCAD Layout for Windows, you should set rules to govern copper pour. You set the rules in the Copper pour rules group box in the Edit Obstacle dialog box.

Copper pour clearance. Designates the absolute clearance between a particular piece of copper pour and all other objects.

Isolate all routes. Attaches copper into only pads of a net using thermal reliefs. Tracks remain isolated.

Seed only from designated object. Specifies that the copper pours into as much area as is possible, contiguously from the seed point. Tracks and vias are not used to seed the pour.

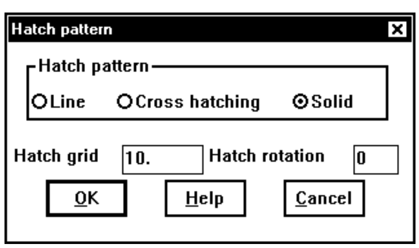
For the purpose of the demo, set the following copper pour zone rules.

- Change the Copper Pour Clearance to 10.
- Select the Seed only from designated object check box.

- **Select a hatch pattern for the copper pour zone.**

Copper and copper pour zones can be solid, a single line, or crossed hatched.

Choose the Hatch pattern button. Select the Solid option and set the Hatch grid to 10.

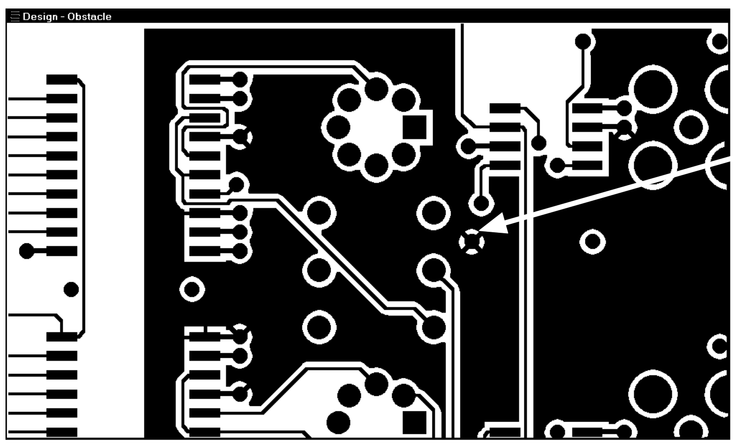


- **Pour the copper.**

  Choose End Command from the popup menu to release the obstacle from the cursor. The copper pour zone forms on the screen.

- **View the thermal reliefs.**

  Choose Zoom In from the View menu to observe an area in which thermal reliefs have been automatically added to the signal pads forAGND.



View the thermal relief.

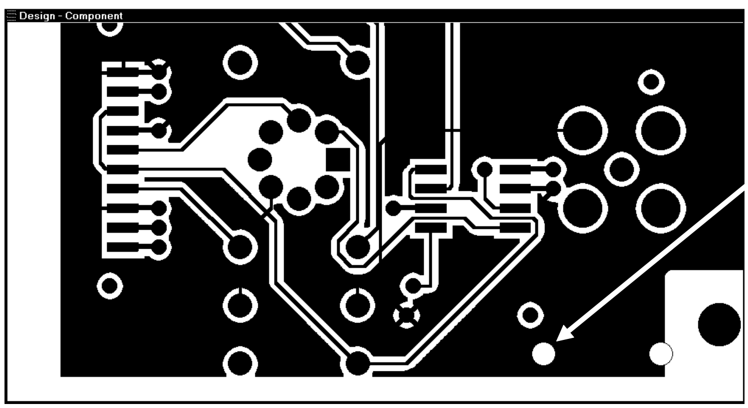
Redrawing the copper pour zone

In OrCAD Layout for Windows, you can edit the board inside a copper pour zone, and then easily repour the copper to accommodate your changes.

- **Insert a part into the copper pour zone.**

Choose the Component toolbar button and select a discrete through-hole part. Press the INSERT key.

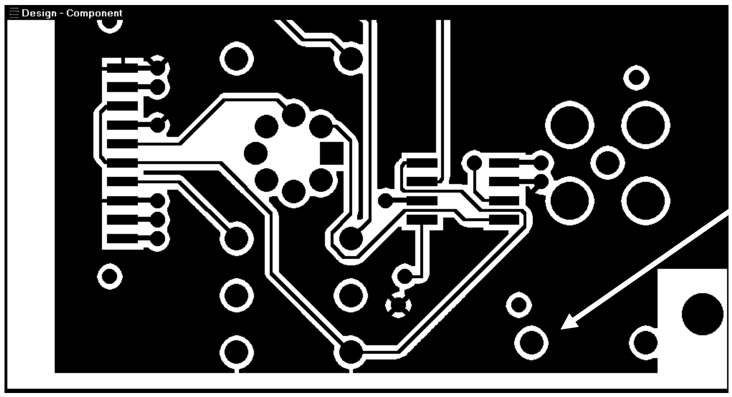
Move the part into the an open area in the copper pour zone. Choose the Manual Route with Shove toolbar button to edit the track as desired.



Inserting a new component.

- **Refresh the copper pour zone.**

  Choose the Refresh Copper Pour toolbar button to redraw the copper.



Refresh the copper pour.

Creating odd-shaped copper pour zones

Copper pour can be any shape. It is just as simple to create a round copper pour zone as it is to create a rectangular one.

- **Designate a seed point.**

When you are creating a copper pour zone in OrCAD Layout for Windows, you must designate a seed point. The seed point is the point from which the copper pours.

Choose the Pin toolbar button. Select a pin attached to the net to which you want to attach the copper pour zone. From the popup menu, choose Toggle Copper Pour Seed to set the copper pour seedpoint.

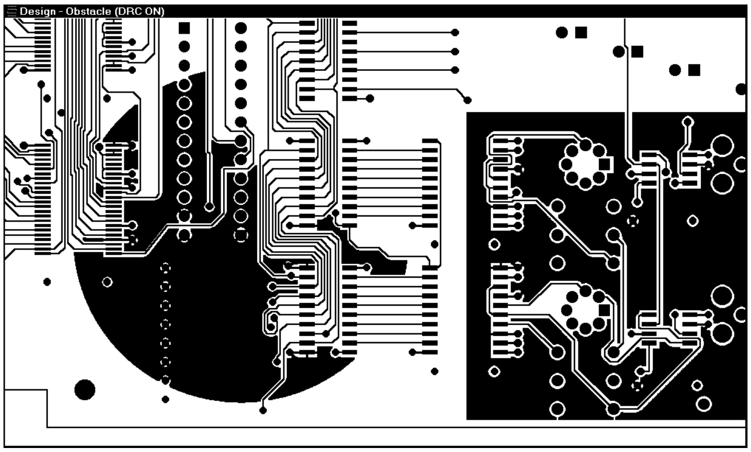
- **Insert and define a new obstacle.**

Choose the Obstacle toolbar button. Press the INSERT key to create a new obstacle.

Choose Modify from the popup menu. Specify the obstacle as copper pour and set the rules and parameters as explained in *Creating a copper pour zones* in this chapter.

- **Create an arc.**

  Click the left mouse button at desired center for your circular copper pour zone. Choose Arc from the popup menu or press the A key. Drag the cursor to begin creating a circle. Click the left mouse button to stop drawing.



Changing the hatch pattern

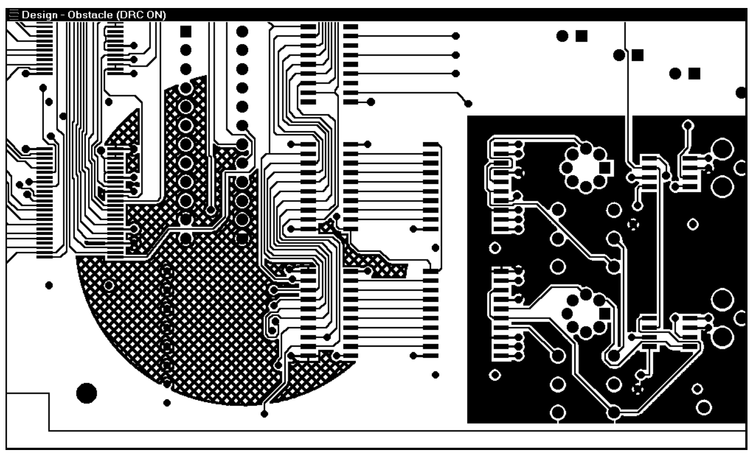
Layout supports any combination of solid and cross hatch pour areas. The cross hatch can be set to any angle.

- **Select a copper pour zone.**

Select the circular copper pour zone you just created and choose Modify from the popup menu.

- **Change the hatch pattern.**

In the Edit Obstacle dialog box, choose the Hatch pattern button. Select the Cross-Hatch option. Change the Hatch grid to 50, and the Hatch rotation to 45.



Exiting the design file

When you have finished with this section, you can either exit the design file by selecting Exit from the File menu, or continue with the*Chapter 6: Post processing*

Go to the next chapter.

Go to the table of contents.

Chapter 6
Post processing

Go to the table of contents.

Go to the last page of the previous chapter.

Post processing

The final task in creating a board is to generate output files.

From Layout, it is possible to extract Gerber files (274D), Extended Gerber files (274X), DXF files, and printer output such as HPGL, HP laser, and dot matrix. Each of these files can be created individually or as a group.

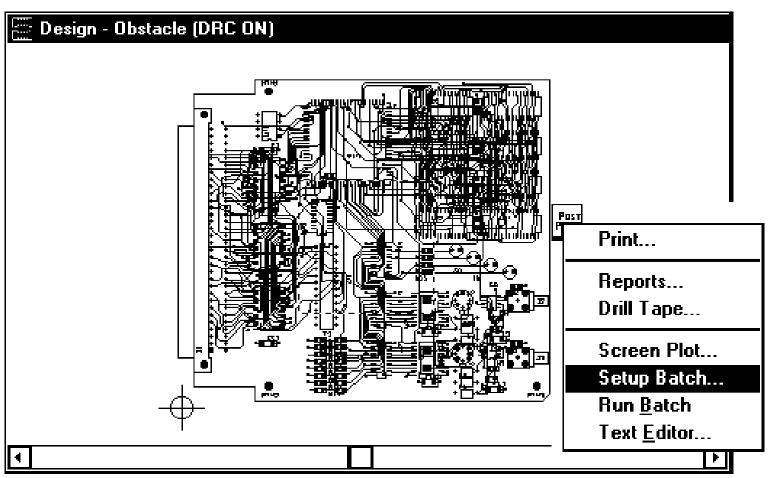
The aperture list for Gerber extraction can be created automatically as the Gerber files are created, or a standard user-defined aperture file can be used.

- **Open the design file.**

Choose Open from the File menu. Select FINISH2.MAX or FINISH4.MAX from the list presented.

- **Display the Post Process spreadsheet.**

Choose the Post Proc. toolbar button. Then choose Setup Batch from the menu that appears. The Post Process spreadsheet displays.



Using the Gerber previewer

Layout includes a Gerber previewer. The previewer is useful for configuring Gerber output. The previewer displays exactly what will be output.

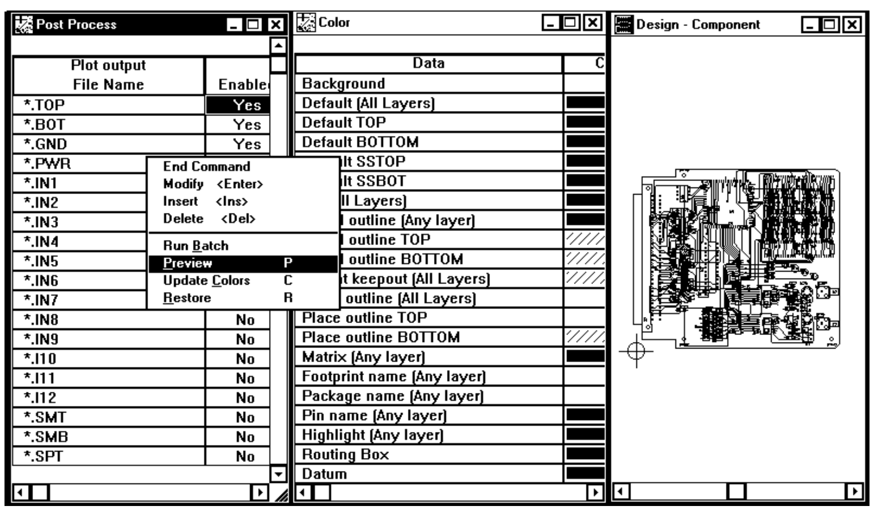
- **Tile the windows.**

Choose the Initialize Color toolbar button. Choose Tile from the Window menu to tile the Post Process spreadsheet, the Design window, and the Color spreadsheet on the screen.

Note You only need to tile the windows if there are objects on the photoplot that do not appear in the preview window.

- **View the Gerber preview.**

Select the *.TOP column in the Post Process spreadsheet and choose Preview from the popup menu or press the P key to preview the Gerber file setup.



Modifying output

To modify a layer setup, you can turn layers on and off in the Color spreadsheet.

- **Make the board outline visible.**

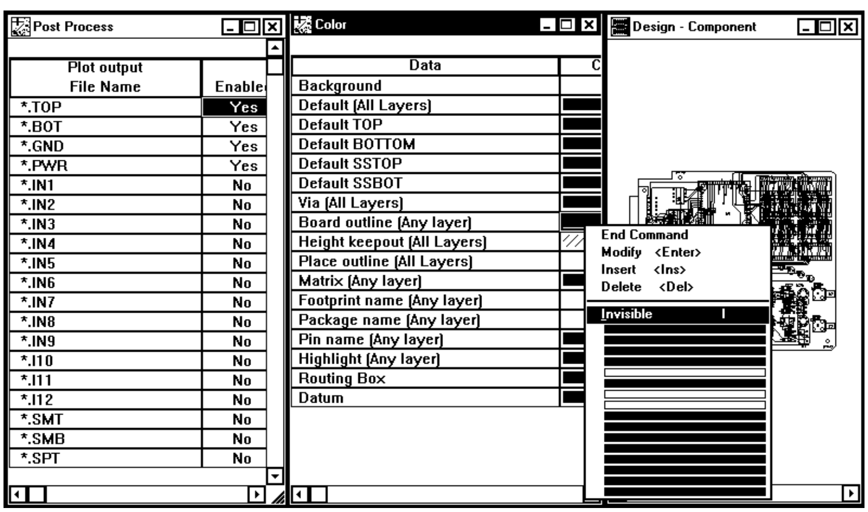
  In the Color spreadsheet, select Board Outline. From the popup menu, choose the toggling Invisible command to make the layer visible.

- **Save the changes.**

  In the Post Process spreadsheet, choose Update Colors from the popup menu to save the changes.

- **View the board outline in the Gerber previewer.**

  Select *.TOP from Post Process spreadsheet and choose Preview from the popup menu. You will now see the board outline.



Creating Gerber output

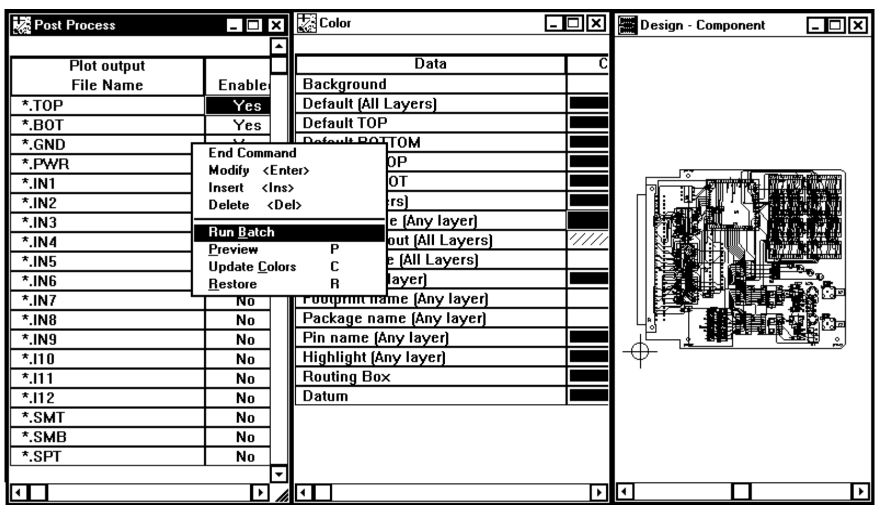
You create Gerber files using the Run Batch command.

When you run the batch process, the system asks you if you want to create the aperture list for the plot(s).

Note As the demo files cannot be saved, you cannot actually run the Gerber files at this time.

- **Create Gerber output.**

Gerber output is not supported by this demo. But, creating Gerber output in Layout is easy. You would simply select the *.TOP column in the Post Process spreadsheet and choose Run Batch from the popup menu.



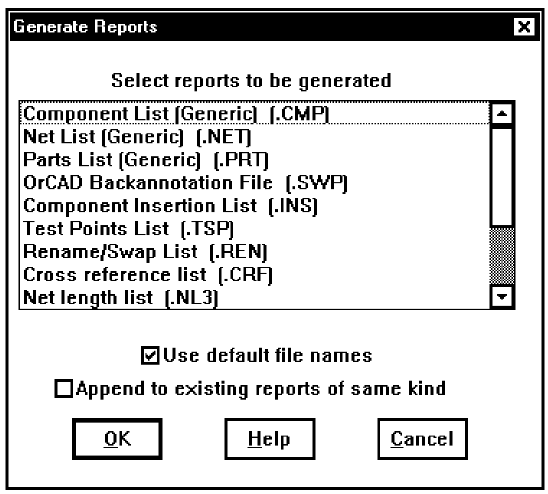
Generating reports

In OrCAD Layout for Windows, a variety of standard reports can be created including netlists, net length reports, insertion lists, drill files, and parts lists.

Custom reports can be created to include attributes passed forward from the schematic, as well as attributes originating within Layout.

- **View the possible reports to generate.**

Choose the Post Proc. toolbar button and choose Reports from the menu that appears. Scan the list of possible reports in the Generate Reports dialog box.



Exiting the design file

When you are finished with post processing, you can exit the design file by choosing Exit from the File menu or you can continue with the *Chapter 7: Using OrCAD Layout for Windows with Capture for Windows*.

Go to the next chapter.

Go to the table of contents.

Chapter 7
Using Layout with OrCAD Capture for Windows

Go to the table of contents.

Go to the last page of the previous chapter.

Annotation and cross probing

OrCAD Layout for Windows has the ability to communicate interactively with OrCAD's Capture for Windows.

Layout's AutoECO utility automatically forward annotates all schematic attributes, component information, and netlist changes to a PCB from Capture for Windows. AutoECO also resolves pin-to-pin conflicts that arise due to pins that are missing from a footprint, or from pins that are named differently in the schematic than they are in Layout. And, when you make changes to your board, AutoECO can back annotate the changes to the Capture netlist.

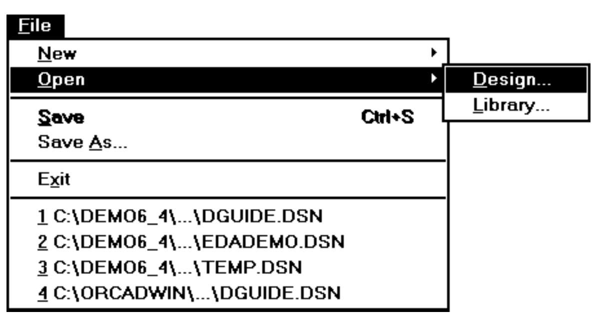
Forward annotating Capture netlist data to your board

This section demonstrates how to bring netlist data into the board design from OrCAD's Capture for Windows.

Note In the demo, we are not actually making changes to the schematic in Capture; therefore, the following process is only a description of the process, and does not actually modify the board file.

- **Open the Capture for Windows schematic design.**

Choose Open and Design from the File menu, and select DGUIDE.DSN from the list presented.

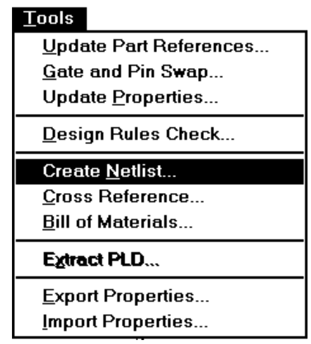


- **Change to physical view.**

Choose Physical from the View menu as this is a hierarchical design.

- **Create the netlist.**

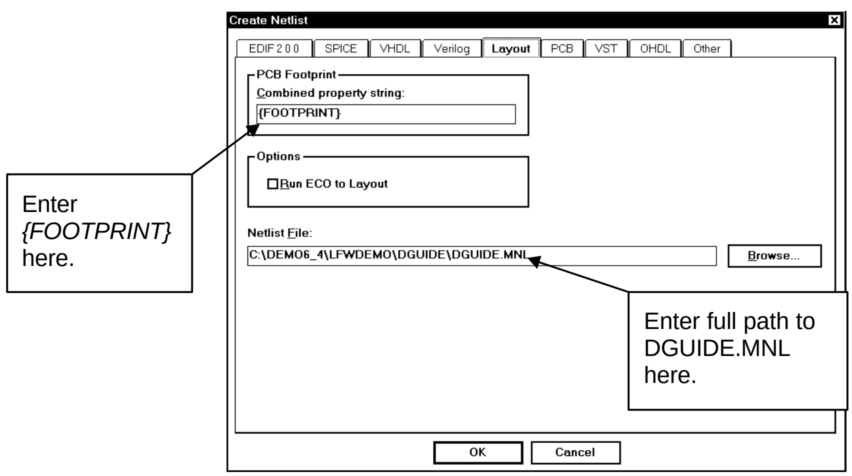
In Capture, choose Create Netlist from the Tools menu. In the Create Netlist dialog box, choose the Layout tab.



In the PCB Footprint group box, enter {FOOTPRINT} in the Combined Property String text box. Type *FOOTPRINT* in capital letters and include the brackets {}.

Ensure that the full path to the netlist file DGUIDE.MNL appears in the Netlist File text box as shown below.

Choose the OK button to process the netlist.

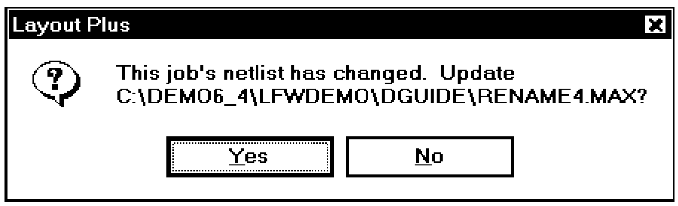


Note You may choose to exit Capture at this time. It is not necessary to run Capture and Layout simultaneously to take advantage of forward annotation; you must have a minimum of 16 MB of RAM to run Capture and Layout at the same time.

- **Open the design in Layout.**

  Choose Design from the OrCAD frame. Select the file RENAME2.MAX or RENAME4.MAX from the list presented.

  As the netlist for the design has changed, Layout asks if you want to load the new netlist for the design. Choose the Yes button. Layout updates the board design to reflect any changes made in the schematic.

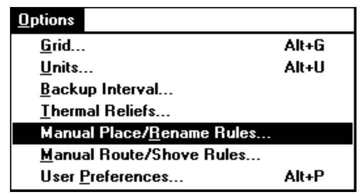


Back annotating information to Capture from Layout

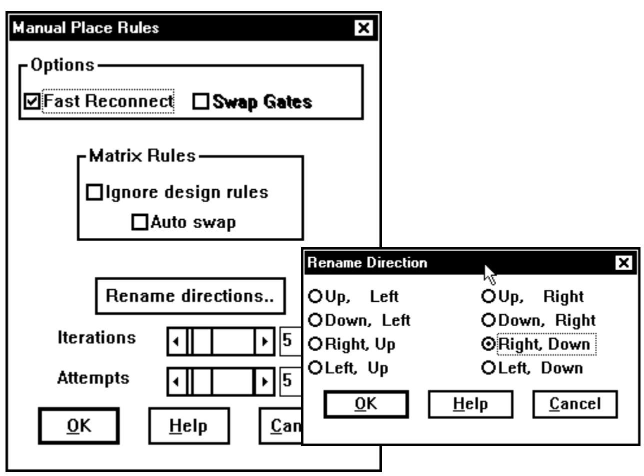
This section demonstrates how to send design data back to Capture from Layout.

Note It is not necessary to run Capture and Layout simultaneously to take advantage of back annotation; you must have 16MB of RAM to run Capture and Layout at the same time.

- **Rename the components on the design.**

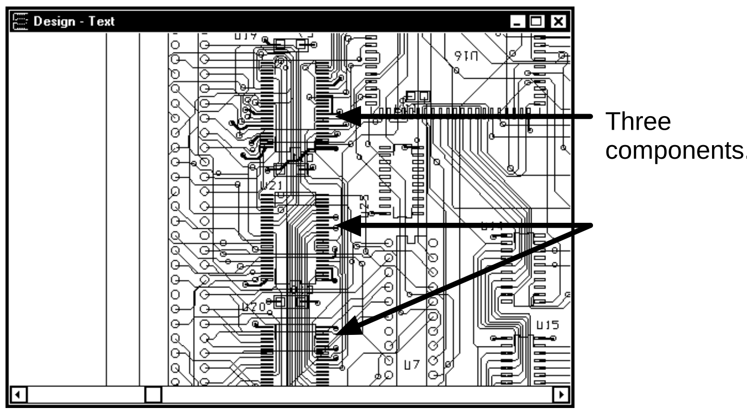


In Layout, choose Manual Place/Rename Rules from the Options menu. In the Manual Place rules dialog box, choose the Rename directions button to display the Rename Directions dialog box. Choose Right, Down to rename the components from the upper left corner of the board to the lower right corner of the board. From the Auto menu, choose Rename Components. Layout renames the components.



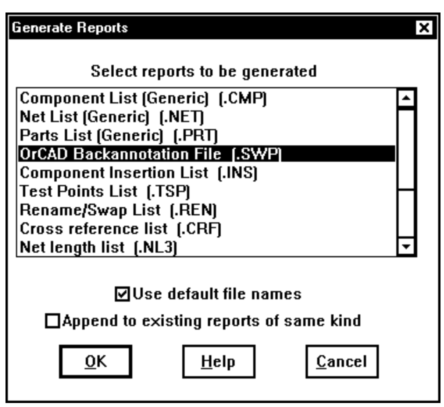
- **View the updated reference designators on the PCB.**

Choose Zoom In from the View menu to magnify the three components on the left side of the board near the connector. The rename components command has updated the component reference designators.



- **Create a swap (SWP) file for back annotation.**

Choose the Post Proc. toolbar button. Then, choose Reports from the pull-down menu. In the Generate Reports dialog box choose the OrCAD Back annotation File (SWP). Choose the OK button to create the file.

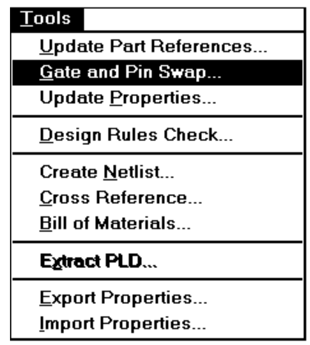


Note When Layout creates the swap (SWP) file, it attempts to save a backup of the design file called BACKANNO.MAX.

- **Open the Capture design.**

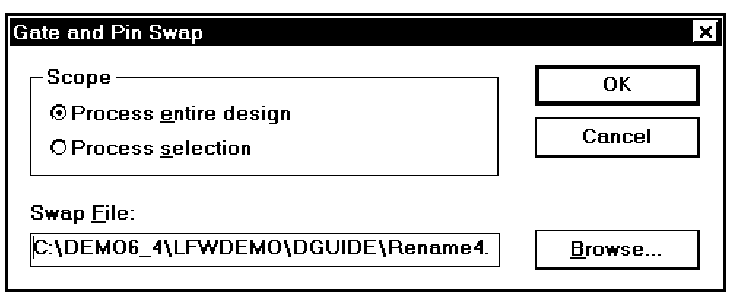
  If Capture is not currently running, choose Open and Design from the file menu and select DGUIDE.DSN from the list presented. Choose Physical from the View menu. If your system resources allow you to run Capture and Layout simultaneously, resize the Capture and Layout application windows to view the schematic and the board side-by-side. It may help to choose Half Screen from the Window menu in Layout.

- **Update the Capture schematic.**



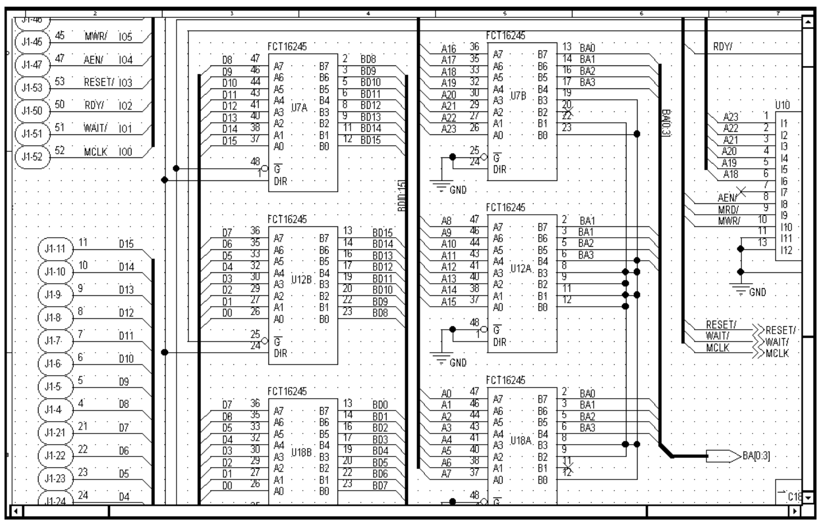
  In Capture, choose Gate and Pin Swap from the Tools menu. Locate and select the swap (.SWP) file you created in Layout, which is probably called RENAME2.SWP or RENAME4.SWP. If the file does not appear by default in the Swap File text box, use the Browse button to search for it. The file should have the same name and should be located in the same directory as your PCB design.

  Choose the OK button. When you restart Capture and reopen the schematic design, it is updated to reflect the changes you made to your board design.



- **View the updated reference designators on the Capture schematic.**

  In the Capture Design Manager, double-click on the Evaluation folder to display the schematic hierarchy. Then double-click on Eval Pg 1 to open page 1 of the Capture schematic. Zoom in to view the reference designators that you modified in Layout. They have also changed on the schematic.



Cross probing between Capture and Layout

In cross probing, selecting a part, component, or net on a Capture schematic or Layout board causes the corresponding component to be highlighted in the other application.

Note It is necessary to run Capture and Layout simultaneously to take advantage of cross probing; you must have a minimum of 16 MB of RAM to perform cross probing using this demo.

- **View the updated reference designators on the PCB.**

  In Layout, choose Zoom In from the View menu to magnify the three components on the left side of the board near the connector.

- **View page 1 of the Capture schematic.**

  In the Capture Design Manager, double-click on the Evaluation folder to display the schematic hierarchy. Then double-click on Eval Pg 1 to open page 1.

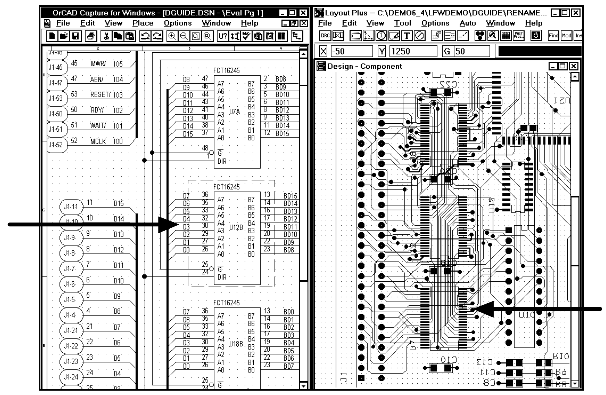
- **View the Capture schematic and the Layout board on the screen.**

  Resize the Capture and Layout windows on the screen.

- **Highlight a component in Layout using cross probing.**

  In Capture, select a part on page 1. The corresponding component is highlighted on the board in Layout.

1. Select this part in Capture.



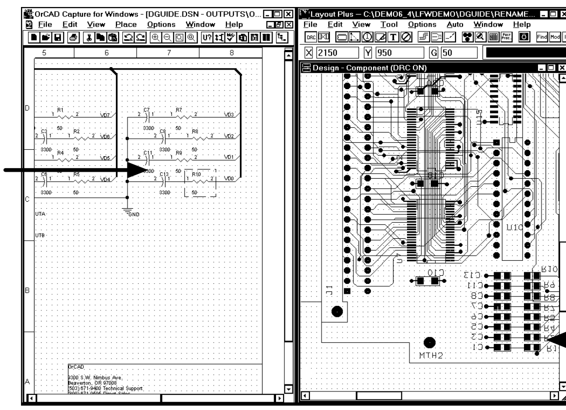
2. To highlight this component in Layout.

- **Locate a resistor in Capture using cross probing.**

In Layout, choose the Component toolbar button. Select one of the terminating resistors in the center of the bottom of the board.

In Capture, the schematic automatically opens and displays the schematic page on which the corresponding symbol is located. Scroll the window until the highlighted symbol is visible.

2. Capture opens the appropriate schematic page.

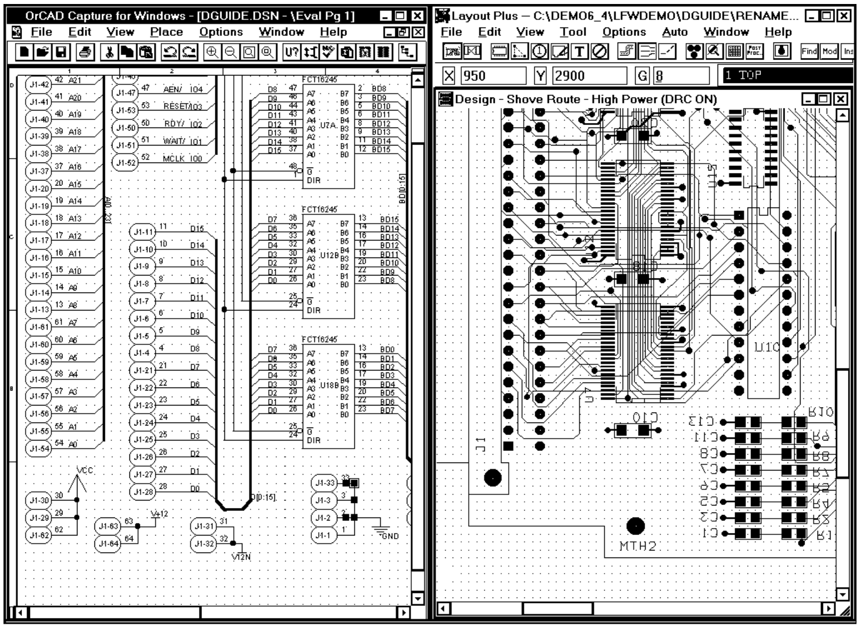


1. Select one of these resistors.

- **Select a signal in Capture to highlight the corresponding net in Layout.**

  You can experiment with highlighting by selecting lines in Capture or tracks in Layout.

  For example, in Capture, select the ground or VCC signal on the schematic. All of the connections of the corresponding net are highlighted on the board in Layout.



Exiting the design file

When you are finished with this section, you can exit the board file by choosing Exit from the File menu. You can run any part of the OrCAD Layout for Windows Demo again by reading the appropriate chapter in the demo guide and opening the corresponding demo board file.

Go to the table of contents.