

# **SPECCTRA<sup>®</sup> Tutorial**

**Product Version 15.0**  
**June 2003**

---

© 1994-2003 Cadence Design Systems, Inc. All rights reserved.  
Printed in the United States of America.

Cadence Design Systems, Inc., 555 River Oaks Parkway, San Jose, CA 95134, USA

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 1-800-862-4522.

All other trademarks are the property of their respective holders.

**Restricted Print Permission:** This publication is protected by copyright and any unauthorized use of this publication may violate copyright, trademark, and other laws. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. This statement grants you permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used solely for personal, informational, and noncommercial purposes;
2. The publication may not be modified in any way;
3. Any copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement; and
4. Cadence reserves the right to revoke this authorization at any time, and any such use shall be discontinued immediately upon written notice from Cadence.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

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

---

# Contents

---

<u>About this Tutorial</u> .....	7
<u>Welcome</u> .....	7
<u>Audience</u> .....	7
<u>Conventions</u> .....	7
<u>Special Terms</u> .....	9
<u>Where to Find Additional Information</u> .....	9
<u>How to Contact Technical Support</u> .....	10

<u>Introducing the SPECCTRA Autorouter</u> .....	11
<u>What Your Prior Experience Should Be</u> .....	11
<u>What You Will Learn</u> .....	11
<u>How to Use This Tutorial</u> .....	12
<u>License Considerations</u> .....	13
<u>How SPECCTRA Fits in the PCB Design Process</u> .....	13
<u>Transferring Designs Between the Autorouter and the Layout System</u> .....	14
<u>Understanding the SPECCTRA Design File</u> .....	17
<u>Understanding SPECCTRA's ShapeBased™ Technology</u> .....	18

## 1

<u>Lesson 1: Learning Basic Concepts</u> .....	21
<u>What You Will Learn</u> .....	21
<u>What to do Before You Begin</u> .....	21
<u>Starting SPECCTRA and Loading a Design</u> .....	22
<u>Controlling the Autorouter</u> .....	26
<u>Using the graphical user interface (GUI)</u> .....	27
<u>Controlling the Autorouter with the GUI</u> .....	29
<u>Monitoring Progress and Checking Results</u> .....	33

# SPECCTRA Tutorial

---

<u>Using the Mouse to Pan and Zoom</u>	35
<u>Getting Information and Measuring Distance</u>	37
<u>Saving Your Work</u>	41
<u>Reviewing Session Command History</u>	46
<u>What You Learned</u>	49

## 2

<u>Lesson 2: Placing Components</u>	51
<u>What You Will Learn</u>	51
<u>What to do Before You Begin</u>	51
<u>Understanding the Basic Steps for Placing Components</u>	52
<u>Setting Placement Rules</u>	53
<u>Preplacing Connectors and Critical Components</u>	57
<u>Placing Large Components</u>	66
<u>Placing Small Components</u>	78
<u>Quitting the Autorouter and Saving Placement Results</u>	83
<u>What You Learned</u>	84

## 3

<u>Lesson 3: Routing a PCB Design</u>	85
<u>What You Will Learn</u>	85
<u>What to do Before You Begin</u>	85
<u>Understanding How the Autorouter Works</u>	86
<u>Setting Routing Rules</u>	100
<u>Autorouting With a Do File</u>	104
<u>Monitoring Autorouting Progress</u>	111
<u>What You Learned</u>	116

## 4

<u>Lesson 4: Setting Rules and Constraints</u>	117
<u>What You Will Learn</u>	117
<u>What to do Before You Begin</u>	117
<u>Understanding the Autorouter's Hierarchical Rules</u>	118
<u>Using Commands to Set Rules and Control the Autorouter</u>	121

## SPECCTRA Tutorial

---

<u>Selecting Connections and Autorouting</u> .....	125
<u>Using Routing Keepin Regions</u> .....	128
<u>Controlling Whether Nets are Routed With Starburst or Daisy-Chain Wiring</u> .....	130
<u>Routing Nets</u> .....	132
<u>What You Learned</u> .....	136

## 5

### Lesson 5: Interactive Routing and Editing .....

<u>What You Will Learn</u> .....	137
<u>What to do Before You Begin</u> .....	137
<u>Interactively Routing</u> .....	138
<u>Editing Wires and Vias</u> .....	157
<u>What You Learned</u> .....	168
<u>Congratulations!</u> .....	168

### Index .....

## SPECCTRA Tutorial

---

---

# About this Tutorial

---

## Welcome

SPECCTRA® is a family of printed circuit board (PCB) design automation tools that run on UNIX and Windows platforms. You can use SPECCTRA tools to automatically and interactively place and route dense surface-mount (SMD) and through-pin (PTH) designs.

All SPECCTRA tools include a graphical user interface (GUI) and the same adaptive, ShapeBased technology. SMD pads, through-pins, wires, and other circuit elements are modeled as basic geometric shapes. Each shape can have rules associated with it, which enforce design constraints such as component spacing and orientation, wire width and clearance, timing, noise, and crosstalk.

This tutorial consists of a book and several accompanying files. The tutorial files are copied to the *tutorial* directory when you install the tool on your computer or network. Separate instructions are provided for Windows and UNIX platforms when steps or procedures differ between these platforms. The majority of the screen shots in this tutorial are from the Windows environment unless otherwise noted. For further details on using this tutorial, see [Introducing the SPECCTRA Autorouter](#) on page 11.

## Audience

This tutorial is written for novice users who are familiar with the current methods and practices used to design printed circuit boards.

## Conventions

The following fonts, characters, and styles are used to identify or represent different types of information.

- **Boldface** `courier` type identifies text that you type exactly as shown, such as commands, keywords, and other syntax elements.

For example:

Type `rule pcb (width .008)` in the command entry box.

## SPECCTRA Tutorial

### About this Tutorial

---

Syntax examples and command examples that are not entered by you from the keyboard are not bold. For example:

```
(boundary (rect pcb 0 0 9000 4000))
```

- *Italic* type identifies titles of books and emphasizes portions of text.

For example:

See the *SPECCTRA User Guide* for information about starting SPECCTRA.

Italicized words enclosed in angle brackets (<>) are placeholders for keywords, values, filenames, or other information that you must supply. For example:

<directory\_path\_name>

- *Courier* type identifies prompts, messages, and other output text that appears on your screen. For example, if you misspell the command *define* as *defin*, an error message displays in the output window.

```
'Syntax error in command: token 1 = defin'
```

Courier type also identifies operating system commands and switches.

Courier type enclosed in brackets ([ ]) identifies keys on your keyboard and mouse buttons.

For example, [Shift] means the shift key. The carriage return key is labeled Enter on some keyboards and Return on others. This manual uses [Enter].

- Mouse buttons are identified by two uppercase letters enclosed in brackets.

[LB] left button

[MB] middle button

[RB] right button

If you have a 2-button mouse, press [Alt] and [RB] simultaneously when you see [MB].

- If a task requires a series of steps that are different on different computer platforms, separate procedures are provided.

For example:

Start the tool by using the Startup dialog box (Windows)

- Click Start on the task bar, slide the pointer to Programs, and choose SPECCTRA.

Start the tool by using the Startup dialog box (UNIX)



- Change your current directory to the SPECCTRA tutorial directory.

## Special Terms

The following special terms are used in this manual.

- The word *enter* used with commands means type the command and press [Enter]. For example:
  - “Enter the command **grid wire 1**” means:
    1. Type **grid wire 1**.
    2. Press [Enter].
- *Click* means press and release the left mouse button.
- *Click-middle* means press and release the middle mouse button.
- *Click-right* means press and release the right mouse button.
- *Drag* means press and hold the left mouse button while you move the pointer.
- *Drag* [MB] means press and hold the middle mouse button while you move the pointer.
- *Double-click* means press and release the left mouse button twice in rapid succession.
- *Click twice* means click two times at the same location in the work area.
- *Switch* refers to one or more characters preceded by a dash (-). You can use a switch when you start SPECCTRA.

## Where to Find Additional Information

The following table describes how to access additional technical documentation from the SPECCTRA user interface.

---

Choose . . .	For information about . . .
<i>Help – What’s New</i>	new features and enhancements available in the current release of SPECCTRA.

---

## SPECCTRA Tutorial

### About this Tutorial

---

Choose . . .	For information about . . .
<i>Help – Known Problems and Solutions</i>	known software problems and suggested work-arounds in the current release of SPECCTRA.
<i>Help – Migration Guide</i>	migrating designs from a prior release to the current Cadence PCB and IC Packaging release.
<i>Help – Manuals – Tutorial</i>	learning to route a printed circuit board design using SPECCTRA.
<i>Help – Manuals – User Guide</i>	<ul style="list-style-type: none"><li>■ starting and using SPECCTRA.</li><li>■ the SPECCTRA design flow.</li><li>■ design methodology and autorouting concepts.</li><li>■ menu commands, dialog boxes and related procedures.</li><li>■ console commands, examples and related syntax.</li><li>■ Do file examples.</li></ul>
<i>Help – Manuals – Command Reference</i>	
<i>Help – Manuals – Design Language Reference</i>	<p>the design language that defines printed circuit board (PCB) designs for SPECCTRA.</p> <p>Note: This manual is written for software programmers who develop software to translate design data between SPECCTRA and a PCB layout system.</p>
<i>Help – Manuals – Documentation Roadmap</i>	the PCB Systems design flow, product solutions and related documentation specific to each area of the design flow as well as descriptions of all Cadence PCB and IC Packaging documents.

## How to Contact Technical Support

If you have questions about installing or using SPECCTRA, you can visit the Cadence Support web site at:

<http://sourcelink.cadence.com /supportcontacts.html>

---

# Introducing the SPECCTRA Autorouter

---

This tutorial teaches you how to use the SPECCTRA autorouter to place and route printed circuit board (PCB) designs.

## What Your Prior Experience Should Be

The tutorial is written for printed circuit board designers who understand current design methods and practices but have little or no experience using the SPECCTRA autorouter.

You are expected to know how to:

- start an application.
- use a mouse.
- open and close windows on a Windows or UNIX platform.

## What You Will Learn

Each lesson in this tutorial covers a set of topics that are important to understanding the basic use and operation of the routing and placement tools. The tutorial includes this introductory chapter and five lessons that cover the following areas.

- Basic Concepts
- Placing Components
- Autorouting
- Setting Rules and Controlling the Autorouter
- Interactively Routing and Editing

# SPECCTRA Tutorial

## Introducing the SPECCTRA Autorouter

---

### How to Use This Tutorial

This tutorial is designed as a step-by-step process for learning how to use autorouter. The information you learn in a lesson “builds” upon the previous lessons. However, it is possible to use each lesson as separate tutorial.

---

<b>This lesson</b>	<b>teaches you to . . .</b>
Lesson 1, Learning Basic Components	Use the autorouter’s graphical user interface (GUI)
Lesson 2, Placing Components	Place components automatically and interactively
Lesson 3, Autorouting a PCB Design	Automatically route a PCB design
Lesson 4, Setting Rules and Controlling the Autorouter	Set rules and routing options to control the autorouter
Lesson 5, Interactive Routing and Editing	Route interactively and edit the routing

---

**Note:** If you are only interested in learning about interactive routing, you can skip the first three lessons, but you need to complete Lesson 4. This lesson includes information on setting rules, which is relevant to interactive routing and editing.

This book is accompanied by a series of *Lesson files*. You use the book with these files and the SPECCTRA software to learn by doing.

### Where to find the Accompanying Lesson Files

The Tutorial Lesson files are delivered with your SPECCTRA software on the installation CD. Once SPECCTRA is installed, you can access these files from the following directory on the computer where your SPECCTRA software resides:

On Windows platforms:

`C:\cct_cds\share\specctra\tutorial`

On Unix platforms:

`<install_directory>/share/specctra/tutorial`

## License Considerations

Your SPECCTRA license must have the PlaceBase feature to complete the work in Lesson 2 and the EditRoute feature to complete the work in Lesson 5.

### Important

Depending on your specific license and the product you choose when you start SPECCTRA, you may occasionally encounter a licensing error when working through procedures in some lessons. The Licensing Error dialog box and a sample error message is shown in the following figure.

**Figure 1-1 Licensing Error Dialog Box**



**Note:** These errors will not prevent you from completing a lesson. Simply click the *Ignore Feature for This Session* button to proceed.

### Tip

License error messages warn you of unavailable software features in your license. To avoid these messages, select the *SPECCTRA expert system* product (if you are licensed to do so) when starting SPECCTRA to complete a tutorial lesson. Otherwise, choose the product with the “richest feature set” from the product list that is presented to you at startup.

For further details on SPECCTRA Licensing, see Chapter 1 of the *SPECCTRA User Guide*.

## How SPECCTRA Fits in the PCB Design Process

SPECCTRA extends your PCB CAD system by adding automatic and interactive placement and routing tools. You use SPECCTRA to place components and route your printed circuit board design.

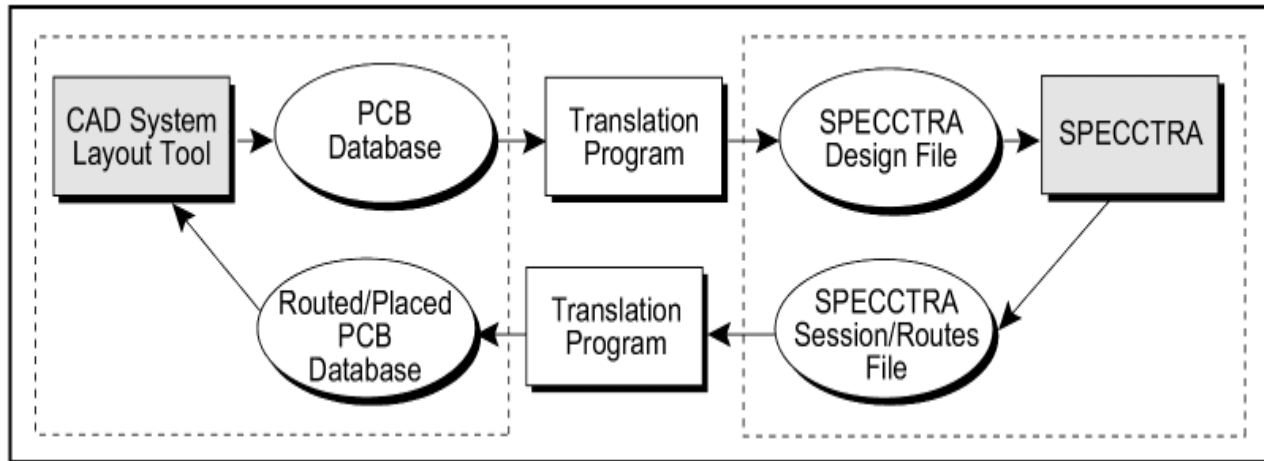
## SPECCTRA Tutorial

### Introducing the SPECCTRA Autorouter

---

After you create a PCB database in your layout system, you translate the design data to a SPECCTRA Design file. You place and route the design in SPECCTRA, save the results, and then merge the placement and routing data with your original PCB database.

#### The Autorouter in the PCB Design Process



#### Transferring Designs Between the Autorouter and the Layout System

Each PCB layout system stores design information in a unique format. However, the files used by SPECCTRA to store design data are the same regardless of the layout system you use.

You transfer your printed circuit board design between SPECCTRA and the layout system by translating the design data from one format to the other. All files that are read and written by SPECCTRA are plain text files. They are described in the following table.

#### SPECCTRA Design Data Files

File Type	Naming Convention	Description
Design	<filename>.dsn	Created by translating design information from the layout system. Contains PCB boundary data, layer definitions, padstack definitions, component data, netlist, preroutes, and design rules.

## SPECCTRA Tutorial

### Introducing the SPECCTRA Autorouter

---

#### SPECCTRA Design Data Files, *continued*

---

File Type	Naming Convention	Description
Session	<filename>.ses	Created by the autorouter. Contains a pointer to the original design file, placement and route data, gate, subgate, pin, and terminator information.
Routes	<filename>.rte	Created by the autorouter. Contains route data that can be translated to your layout system and read by the autorouter.
Wires	<filename>.w	Created by the autorouter. Contains route data that can be read by the autorouter only.

---

Some layout systems require intermediate text files to transfer a PCB design to and from SPECCTRA. Other systems read and write SPECCTRA's files directly without intermediate files. The files that are needed to transfer designs between SPECCTRA and several popular layout systems are described in the following table.

#### Intermediate Design Files

---

This layout system	Uses . . .	That contains . . .
Allegro	<board_name>.brd	All PCB design data, including nets, properties, components, padstacks, preroutes, PCB boundary, and rules
Board Station	tech.tech	Layer definitions and rules
	geoms_ascii	Image definitions, PCB outline, keepout and keepin areas, and rules
	nets.nets	Netlist
	traces.traces	Preroutes, area fill information, and high speed topology specifications
	(Optional) gates.gates	Gate and pin swap information
	(Optional) pins.pins	Properties attached to pins
	(Optional)	

## SPECCTRA Tutorial

### Introducing the SPECCTRA Autorouter

---

#### Intermediate Design Files, *continued*

---

<b>This layout system</b>	<b>Uses . . .</b>	<b>That contains . . .</b>
	testpoints.testpoints  (Optional) mfg/neutral_file  (Optional)	Testpoint information   Pin X,Y coordinates
PADS	ASCII output	All PCB design data, including nets, components, padstacks, preroutes, PCB boundary, and rules
PCAD	PDIF	All PCB design data, including nets, components, padstacks, preroutes, PCB boundary, and rules
Protel	Protel text file	Writes the autorouting design file and reads routes and session files directly

---

Many PCB layout systems have built-in features to transfer designs to and from SPECCTRA. Some layout systems include a choice on a menu or a separate GUI to simplify the transfer process. Refer to the documentation for your layout system or the documentation that was included with your SPECCTRA translator to determine how to transfer designs between your layout system and SPECCTRA.



## Understanding the SPECCTRA Design File

The SPECCTRA Design file is a text file that contains the information needed to represent a printed circuit board in SPECCTRA. The PCB outline, layers, components, padstacks, nets, and preroutes are represented in the Design file in five sections. These five sections are described in the following table.

---

This design file section . . .	Contains this information . . .
Structure	Working units, layer definitions, PCB boundary, power planes, region rules, PCB keepouts, via ids, global rules, grid definitions
Placement	Component instances that consist of image names, reference designators, X,Y locations, PCB side, and rotation
Library	Image definitions that include pin names and pin locations, pin definitions, and padstack definitions
Network	Net list (net names and pin lists), class definitions, class to class definitions, group definitions, differential pair definitions, and net, class, or group rules
Wiring	Preroute information

---

Because the SPECCTRA Design file is a text file, you can view it in SPECCTRA using a report window by choosing *Report – Design*. You can also view it using most any text editor.

**Note:** Do not edit the Design file in a text editor. Most translators use the Design file to merge the routing data with the original layout system database. If you change the Design file and it's no longer synchronized with the layout system database, the translation of route data to your layout system could fail. If you need to make a change to the design data, make the change in your layout system and translate the revised design to the SPECCTRA Design file.

Using a text editor, you can search for keywords. In UNIX, you can use vi, emacs, or textedit. In Windows, you can use Notepad or Write. If the Design file is too large for Notepad, use Write with the *no conversion* option.

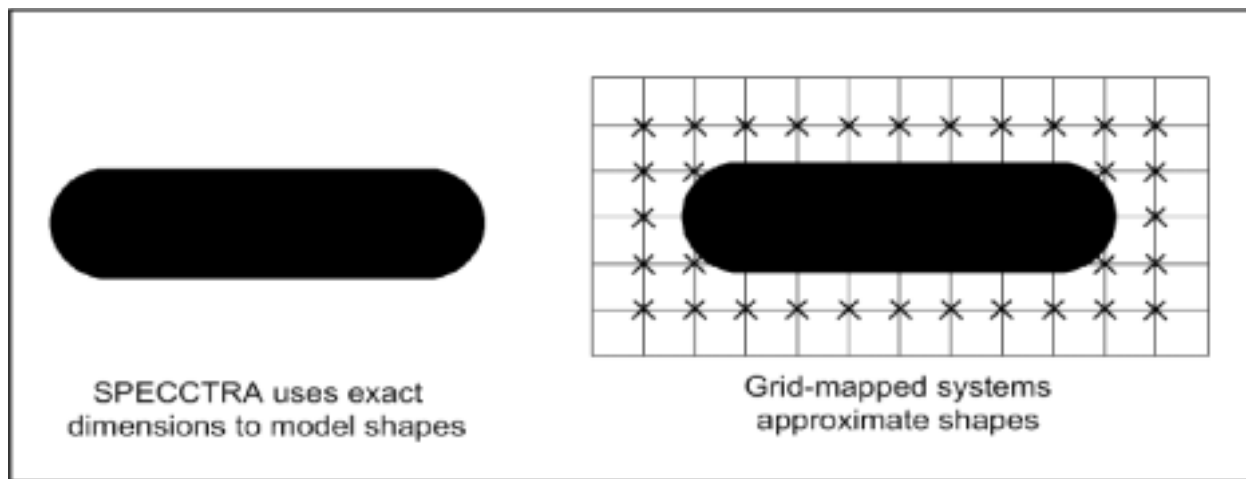
In Windows, you cannot view a Design file in a text editor while the file is loaded in SPECCTRA. However, you can copy and rename the Design file to accomplish this.

## Understanding SPECCTRA's ShapeBased™ Technology

SPECCTRA succeeds in routing large, dense designs because of its ShapeBased technology. The SPECCTRA autorouter differs from traditional grid-mapped systems because it models pins, pads, wires, and vias as true shapes. Grid-mapped systems define these shapes as grid points. Each pin, pad, wire, and via is defined in terms of the grid points it occupies.

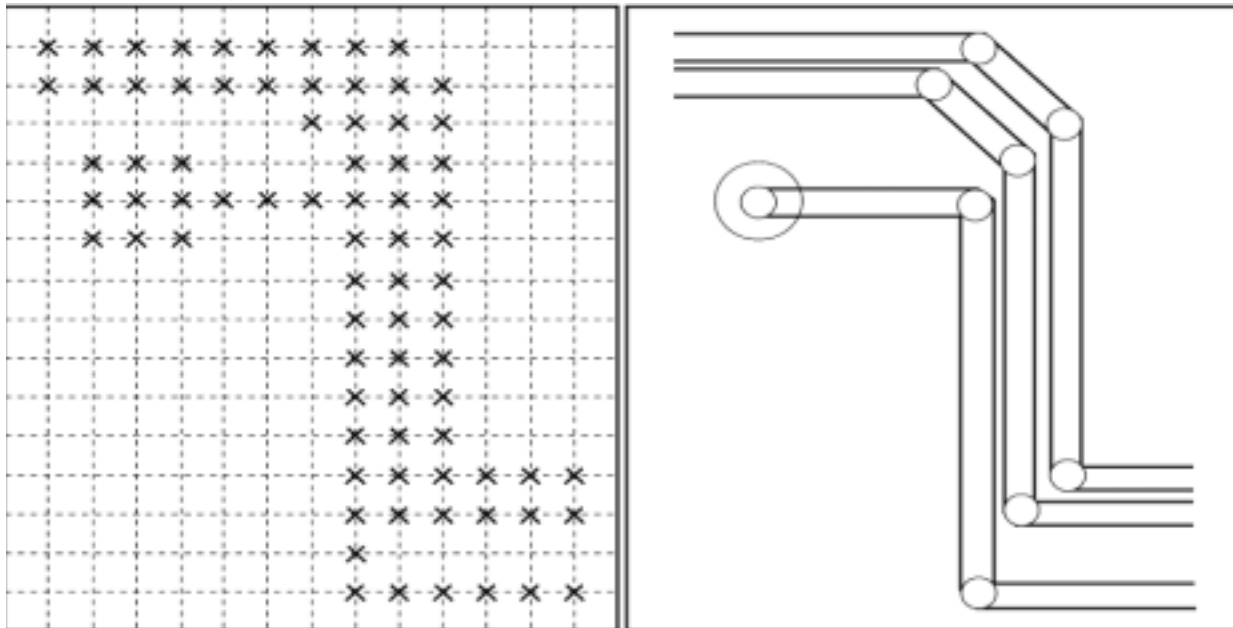
The following figure shows the basic difference between the ShapeBased system and a grid-mapped approach.

**Figure 1-2 ShapeBased vs. Grid-mapped Object Modelling**



While grid-mapped modeling wastes space, its greater weaknesses are its excessive memory and storage requirements. SPECCTRA's ShapeBased approach only requires memory for storing shapes, not grid points. The following figure illustrates the difference between ShapeBased and grid-mapped memory requirements.

### ShapeBased vs. Grid-Mapped Memory Requirements



Grid-mapped system must  
store 78 grid points to  
model 12 shapes

ShapeBased system  
stores 12 shapes

Another advantage of SPECCTRA's ShapeBased technology is its support of complex design rules. Each shape on each layer inherits its own unique set of design rules. This means you can comply with the most complicated design requirements without resorting to tricks and work-arounds during placement and routing.

# **SPECCTRA Tutorial**

## Introducing the SPECCTRA Autorouter

---

---

# Lesson 1: Learning Basic Concepts

---

## What You Will Learn

This lesson explains the autorouter user interface and how to perform basic design tasks.

In this lesson, you will learn how to:

- start the autorouter and load a design.
- use the graphical user interface (GUI) for commands.
- pan and zoom within the Design window.
- measure distance and get information about design objects.
- save your work.
- review a session's command history.

This lesson takes about 45 minutes to complete.

## What to do Before You Begin

Before you begin this lesson, do the following.

- Read [About this Tutorial](#) on page 7 which explains the conventions, terminology, and symbols that are used throughout this tutorial.
- Read [What You Will Learn](#) on page 11 to determine the lessons that should be completed to satisfy your specific learning requirements.
- Make sure you can access the Tutorial Lesson files. For information on the location of these files, see [Where to find the Accompanying Lesson Files](#) on page 12.
- Read [License Considerations](#) on page 13 which notes important license information to keep in mind while working through the lessons in the tutorial.
- Make sure that SPECCTRA has been installed properly on your computer or network.

## Starting SPECCTRA and Loading a Design

When you start the autorouter, you must load a Design file. The Design file is a text file that contains all the PCB design information that is needed by the autorouter.

**Note:** You can also start the autorouter using a Session file, which links the routing and placement data to a Design file. You will learn more about Session files later.

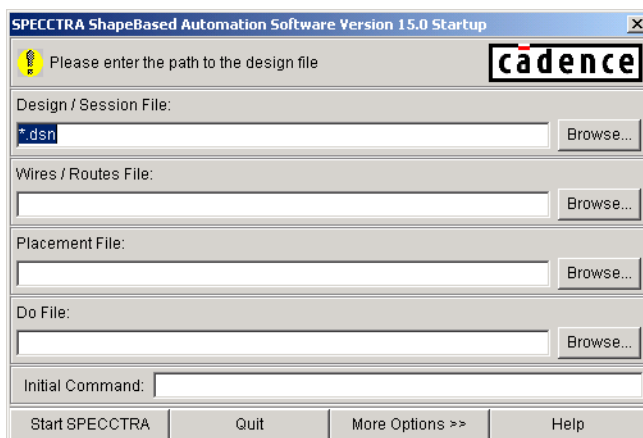
In this lesson, you use the SPECCTRA Startup dialog box to start the autorouter and load a Design file. The procedure varies depending on whether you are using a Windows or UNIX platform. Follow the procedure for your specific platform. You will use these same basic steps frequently throughout this tutorial. If a procedure is not labeled Windows or UNIX, then it is written for both Windows and UNIX.

### Task: Start SPECCTRA using the Startup dialog box (Windows)

#### Procedure

1. Choose *Start – Programs – SPECCTRA V15 – SPECCTRA 15*.

The Startup dialog box appears.



You need to specify a Design file to start the autorouter. You can type the name of the Design file in the *Design/Session File* data entry box or you can use the *Browse* button to search for a file.

2. Click the *Browse* button to the right of the *Design/Session File* data entry box.

The Open dialog box appears.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

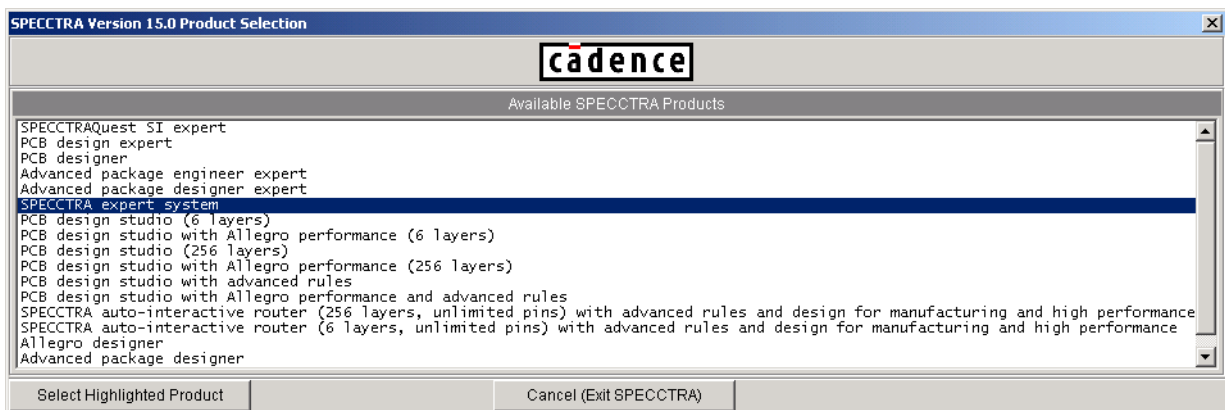
3. Change to the SPECCTRA Tutorial directory. See [Where to find the Accompanying Lesson Files](#) on page 12 for the location of this directory.

4. Open `lesson1.dsn` from the *Files* list.

The path and filename appear in the *Design/Session File* data entry box.

5. Click *Start SPECCTRA*.

The Product Selection dialog box appears.



6. If you are licensed to use the SPECCTRA expert system product, click on it in the product list. Otherwise, click on the the most “feature-rich” product in the list that you are licensed to use. For details on SPECCTRA licensing, products and features, refer to Chapter 1 of the [SPECCTRA User Guide](#).

The product selection highlights.

7. Click the *Select Highlighted Product* button.

SPECCTRA starts and loads the design.

8. Do not exit SPECCTRA. Continue to the next section on [Controlling the Autorouter](#).

You will exit SPECCTRA later in this lesson.

# SPECCTRA Tutorial

## Lesson 1: Learning Basic Concepts

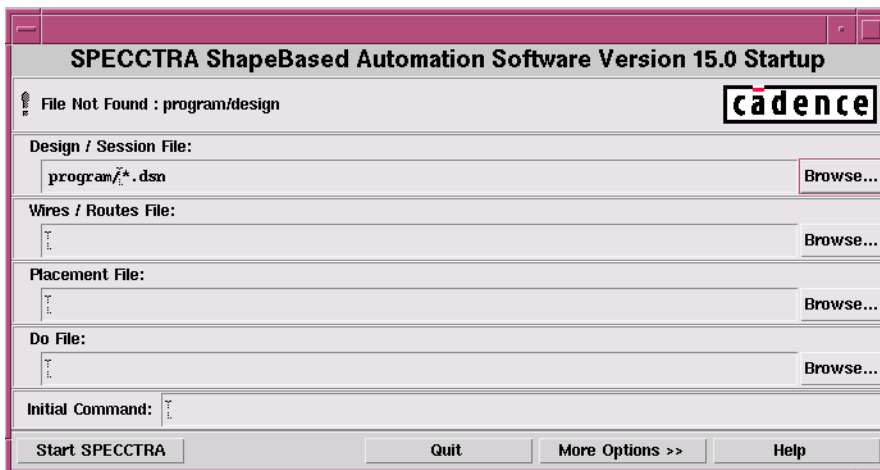
---

### Task: Start the Autorouter using the Startup dialog box (UNIX)

#### Procedure

1. Change to the SPECCTRA Tutorial directory. See [Where to find the Accompanying Lesson Files](#) on page 12 for the location of this directory.
2. Enter **specctra** at the shell prompt.

The Startup dialog box opens.



You need to specify a Design file to start the autorouter. You can type the name of the Design file in the *Design/Session File* data entry box or you can use the *Browse* button to search for a file.

3. Click the *Browse* button to the right of the *Design/Session File* data entry box.
4. The Select File dialog box opens.
5. Select `lesson1.dsn` from the files directory and click *OK*.
6. Click *Start SPECCTRA*.

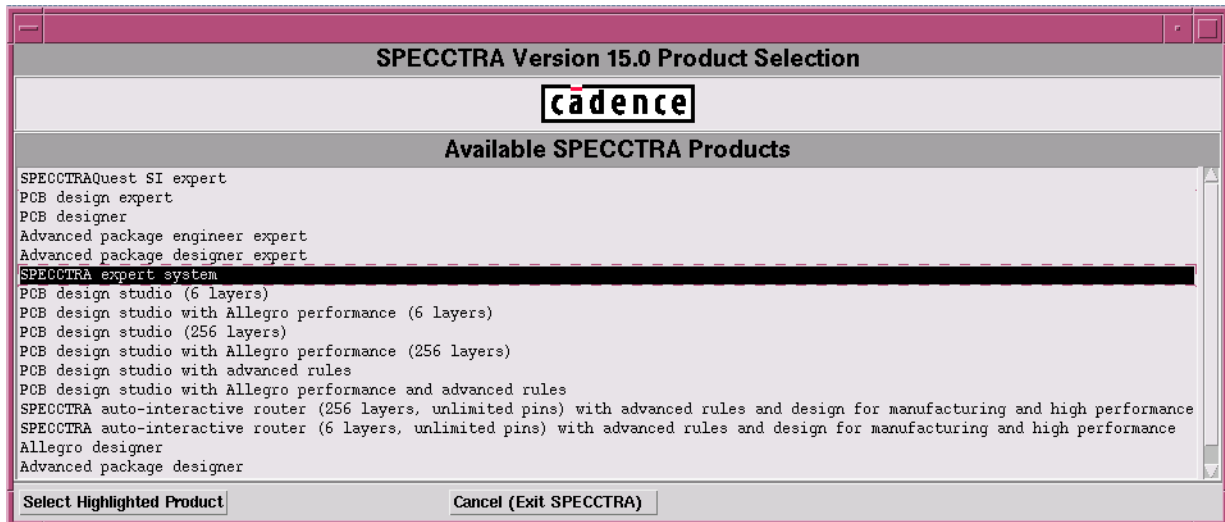


## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

The Product Selection dialog box appears.



7. If you are licensed to use the SPECCTRA expert system product, click on it in the product list. Otherwise, click on the “the most feature-rich” product in the list that you are licensed to use. For details on SPECCTRA Licensing, see Chapter 1 of the [SPECCTRA User Guide](#).

The product selection highlights.

8. Click the *Select Highlighted Product* button.

SPECCTRA starts and loads the design.

9. Do not exit SPECCTRA. Continue to the next section on [Controlling the Autorouter](#).

You will exit SPECCTRA later in this lesson.

## Controlling the Autorouter

In the following sections, you will learn how to use the GUI to control the autorouter, how to autoroute a design, monitor progress, and examine the results.

You control the autorouter by:

- entering console commands.
- running a command file (Do file).
- using the GUI to enter commands.

In a later lesson, you will enter console commands and use a Do file to control the autorouter. In this lesson you focus on the use of the GUI.

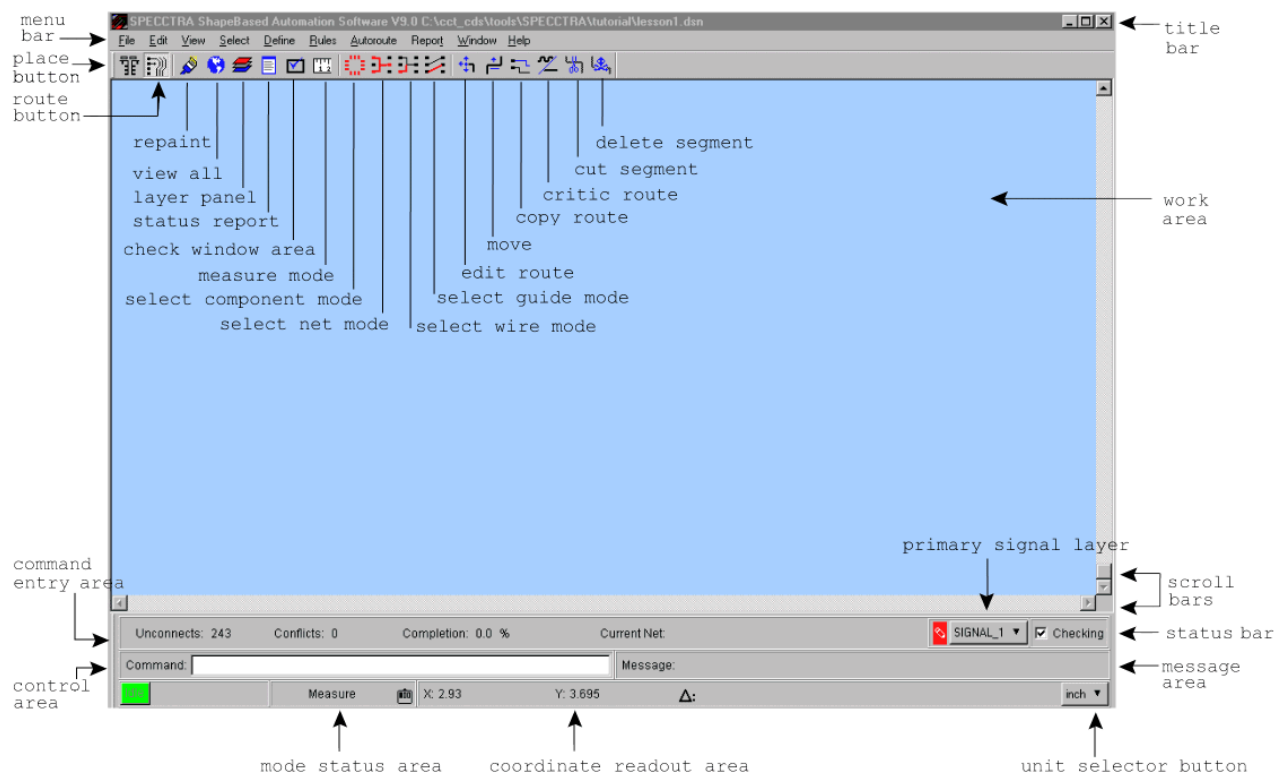
# SPECCTRA Tutorial

## Lesson 1: Learning Basic Concepts

### Using the graphical user interface (GUI)

The GUI consists of graphic elements such as menus, tool bar buttons, dialog boxes, status bar, and message area. These and other important elements are labeled in the following figures.

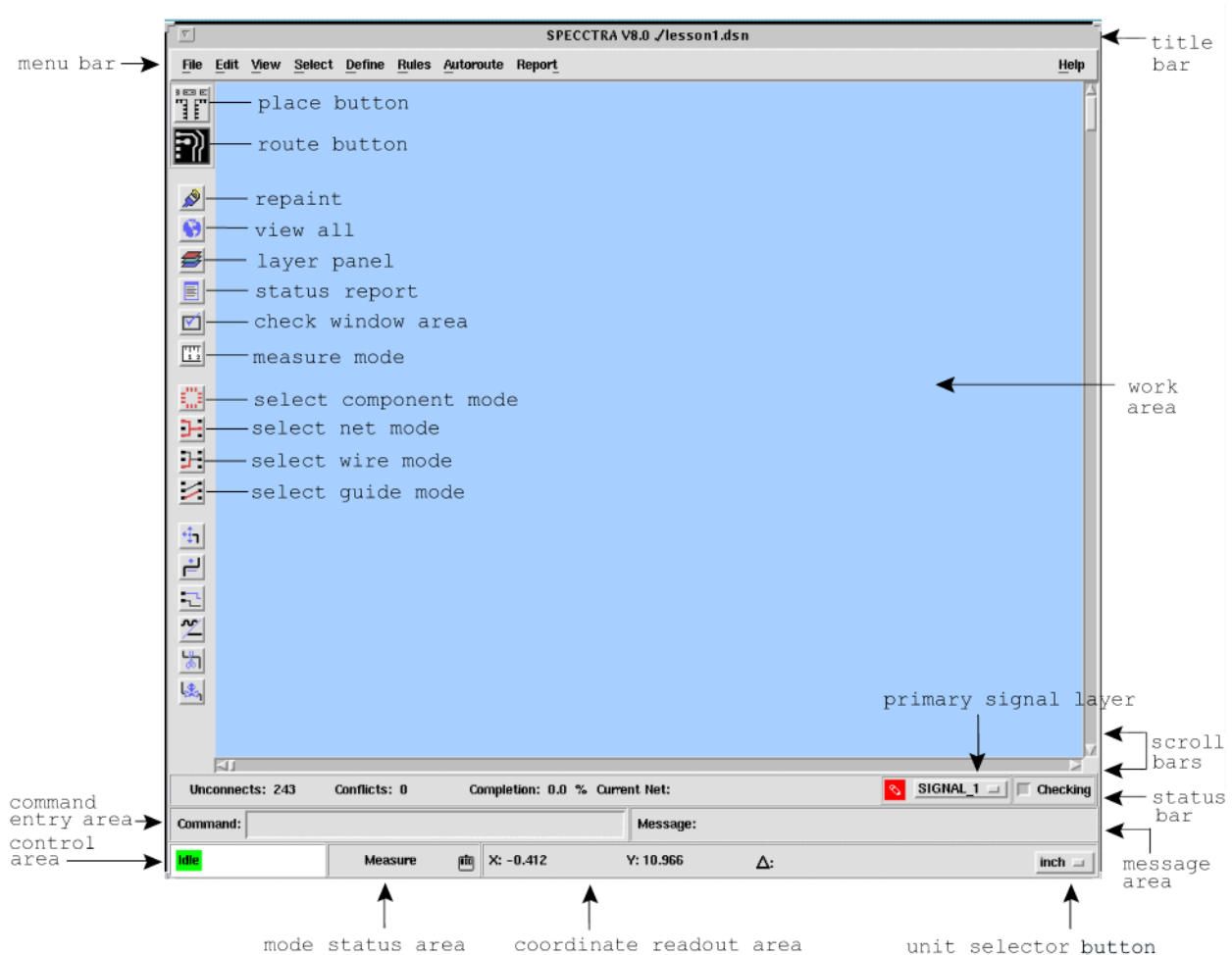
**Figure 1-1 The SPECCTRA GUI (Windows)**



# SPECCTRA Tutorial

## Lesson 1: Learning Basic Concepts

**Figure 1-2 The SPECCTRA GUI (UNIX)**



The last two figures show the GUI for Route mode. A slightly different GUI displays in Place mode. Route is the default mode when you start the SPECCTRA. The Route menu bar and status bar are specific to autorouting. To switch to Place mode, click the *Place* button.

### Task: Switch between Route and Place modes

#### Procedure

1. Click the *Place* button on the tool bar.

The tool bar and status bar change. You will learn about placement in Lesson 2. For now, you will work in Route mode.

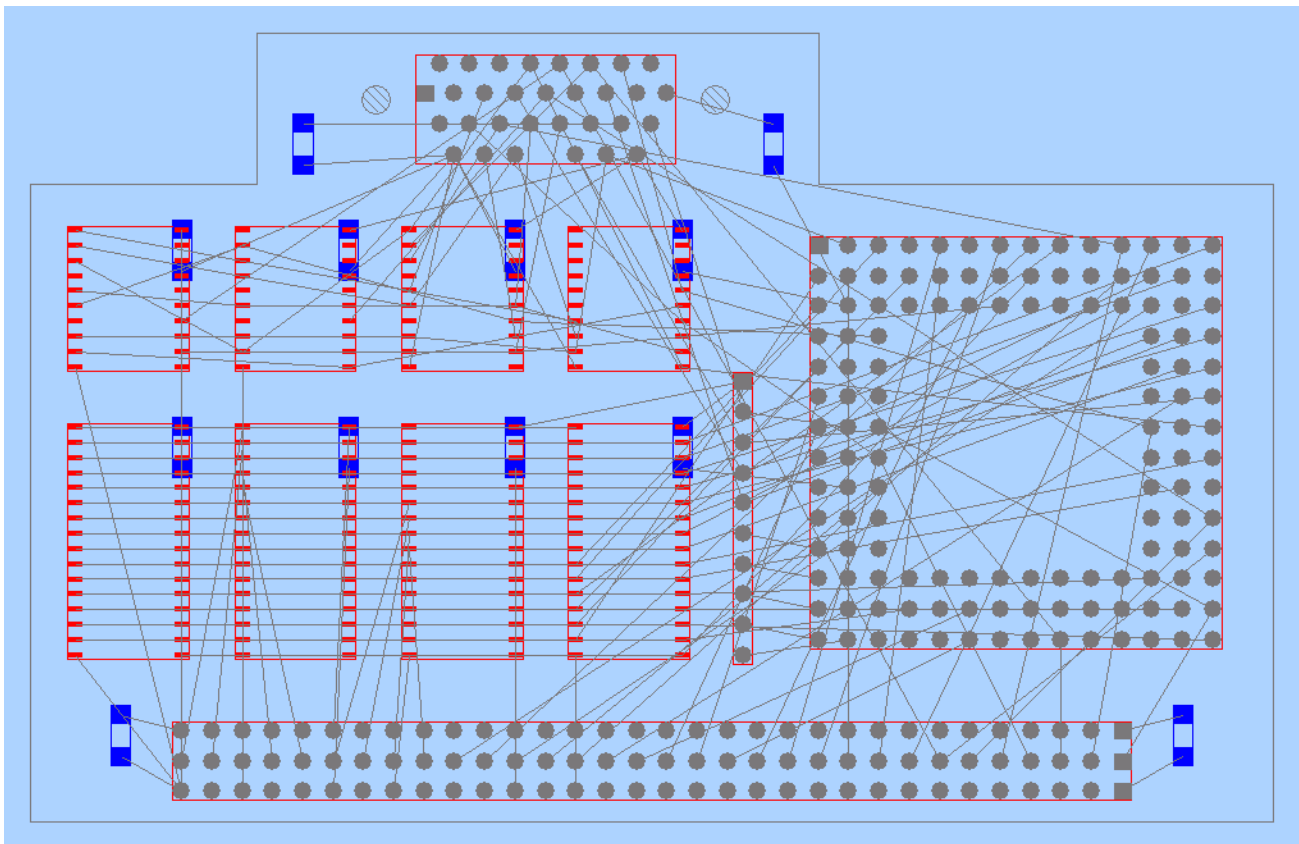
2. Click the *Route* button on the tool bar.

## Controlling the Autorouter with the GUI

The easiest way to control the autorouter is to use the GUI. If you use the GUI, you do not need to know command syntax. You only need to know the task you want to complete.

In the following procedure, you use the GUI to autoroute the printed circuit board shown in the following figure. You will see how easy it is to automatically route a design using the GUI.

**Figure 1-3 lesson1.dsn (unrouted)**



### Task: Use the GUI to autoroute a printed circuit board

#### Procedure

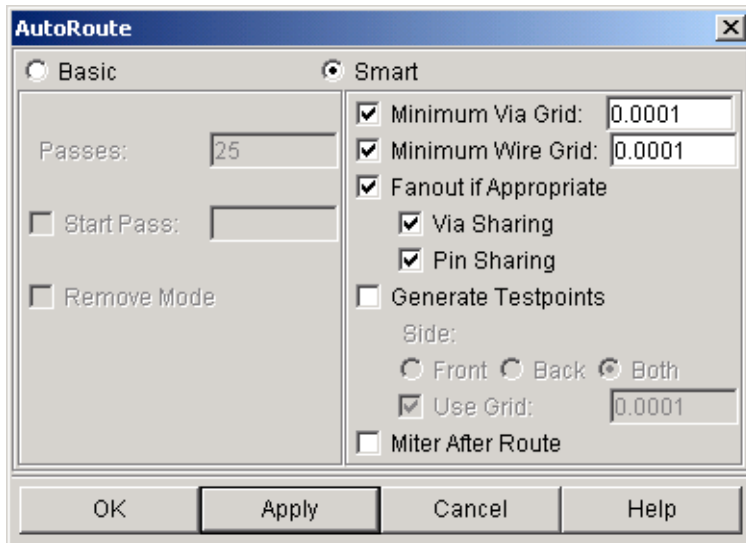
1. Click *Autoroute – Route*.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

The AutoRoute dialog box opens.



2. Select the *Basic* option at the top of the dialog box. Note that 25 route passes are used by default.
3. Click *OK*.

The autorouter begins routing the design.

While the design is routing, note that a red *Pause* button appears in the control area. You can use this button to pause or stop the autorouter. When the autorouter finishes, the *Pause* button in the control area is replaced by a green *Idle* button.

Wait for the autorouter to complete.

**Note:** If you encounter a Licensing Error dialog box, click the *Ignore Feature for This Session* button to continue on. See [License Considerations](#) on page 13 for further details.

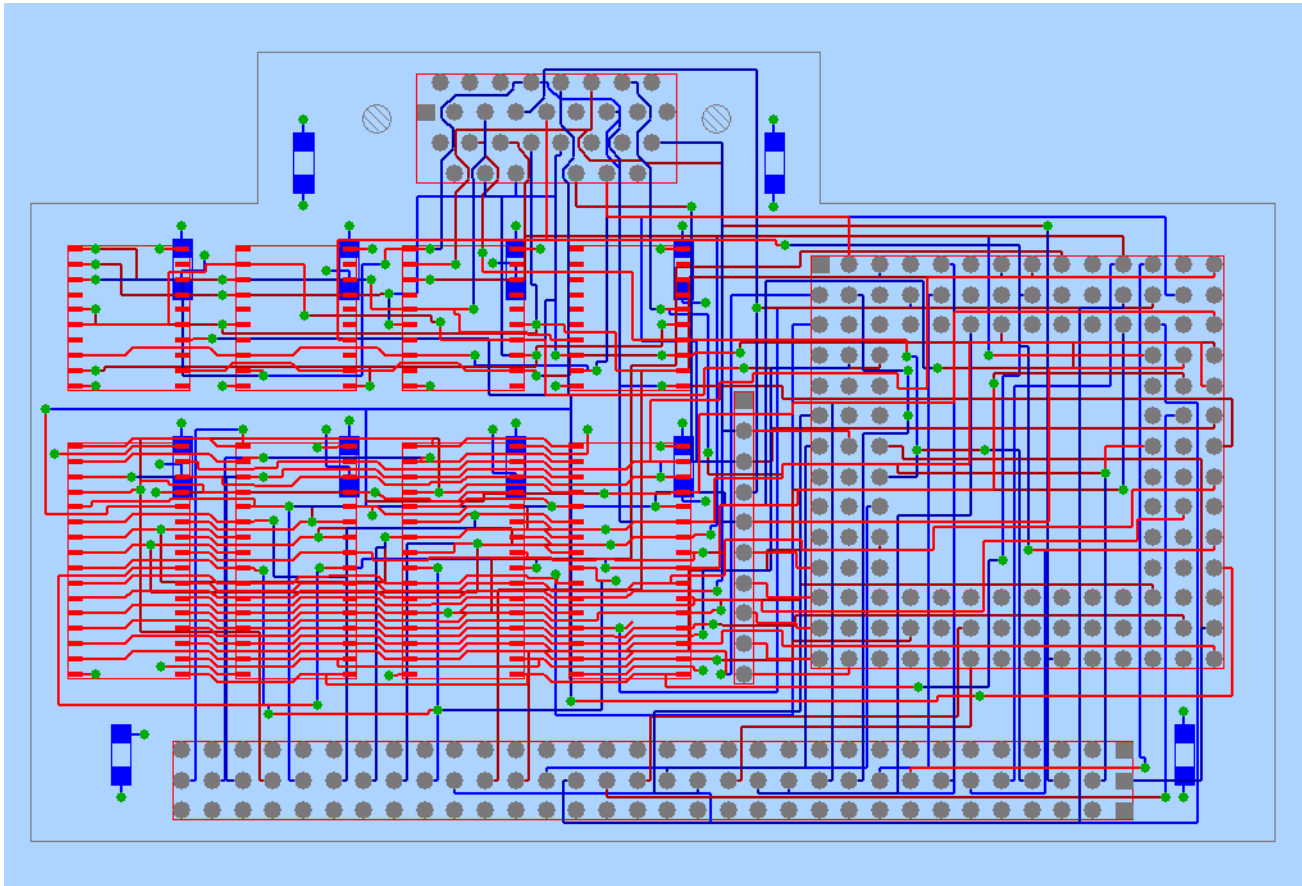
Once fully routed, your design should look like the following figure.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

**Figure 1-4 lesson1.dsn (fully routed)**



4. View the transcript of session information in the Output window. On UNIX platforms, the Output window is the shell in which you started the autorouter.

Use the Output window scroll bars to browse the information.

The Information in the Output window includes the version number and routing results. When warnings or error messages are generated, they also appear in the Output window. A portion of a typical session transcript follows.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

```
# =====
#           SPECCTRA ShapeBased Automation Software
# Copyright 1990-2000 Cadence Design Systems, Inc. All Rights Reserved.
# =====
#
# Software licensed for sale by Cadence Design Systems, Inc.
# Current time = Thu Dec 12 11:02:45 2002
#
# SPECCTRA ShapeBased Automation Software V15.0 made 2002/12/11 at 01:26:55
# Running on: pc-lgiroux2k, OS Version: WindowsNT 5.0.2195, Architecture: Intel Pentium II
# SPECCTRA ShapeBased Automation Software running in Windows 2000 Professional
# startup F:\samples\lesson1.dsn
# Design Name F:\samples\lesson1.dsn
# Did File Name: F:\samples\12121103.did
# Current time = Thu Dec 12 11:03:26 2002
# PCB F:\samples
# Master Unit set up as: mm 100000
# PCB Limits xlo= -0.1750 ylo= 5.9000 xhi= 6.5250 yhi= 9.6500
# Total 2 Images Consolidated.
# Via BB_1-2 z=1, 2 xlo= -0.0125 ylo= -0.0125 xhi= 0.0125 yhi= 0.0125
# Via BB_3-4 z=3, 4 xlo= -0.0125 ylo= -0.0125 xhi= 0.0125 yhi= 0.0125
#
#
#
# Conflicts between polygon wires and fixed objects: 0
# Stub Violations: 0
# Net Order Violations: 0
# Diffpair Uncoupled Length Violations: 0
# Diffpair Phase Tolerance Violations: 0
# Total layerset violations: 0
# Total layerset violations (exclude Fanout/Stagger): 0
# Overall Routing Time: 0:00:27
#
# Wiring Statistics ----- F:\samples\lesson1.dsn
# Nets 72 Connections 243 Unroutes 0
# Signal Layers 4 Power Layers 2
# Wire Junctions 95, at vias 66 Total Vias 185
# Percent Connected 100.00
# Manhattan Length 215.4000 Horizontal 143.6227 Vertical 71.7773
# Routed Length 249.2424 Horizontal 158.7837 Vertical 90.4630
# Ratio Actual / Manhattan 1.1571
# Unconnected Length 0.0000 Horizontal 0.0000 Vertical 0.0000
```

You view the session transcript to monitor messages, command processing, and command results.



## Monitoring Progress and Checking Results

SPECCTRA generates a transcript of your work session in the Output window unless you redirect it to a file. In addition, a status file logs each routing command and the results of each pass. The status file is useful for documenting session progress and monitoring routing results. In the following procedure, you display a Routing Status Report and locate information by searching.

### Task: Locate information in the routing status report

#### Procedure

1. Click the *Status Report* button on the tool bar. See [Figure 1-1](#) on page 27 for assistance.

The Routing Status Report window opens.

You will search for the total number of connections in the design.

2. Enter `connections` in the Search entry box and click on the down arrow symbol.

Connections is highlighted.

The number to the right is the total number of connections in the design.

3. Use the scroll bars to observe the summary at the end of the report. The summary report includes the total number of vias used, total wire length, ratio of the manhattan length to the routed length, and the distribution of routing per signal layer.

4. Close the report.

On the Windows platform, click the x button in the top-right corner of the report window.

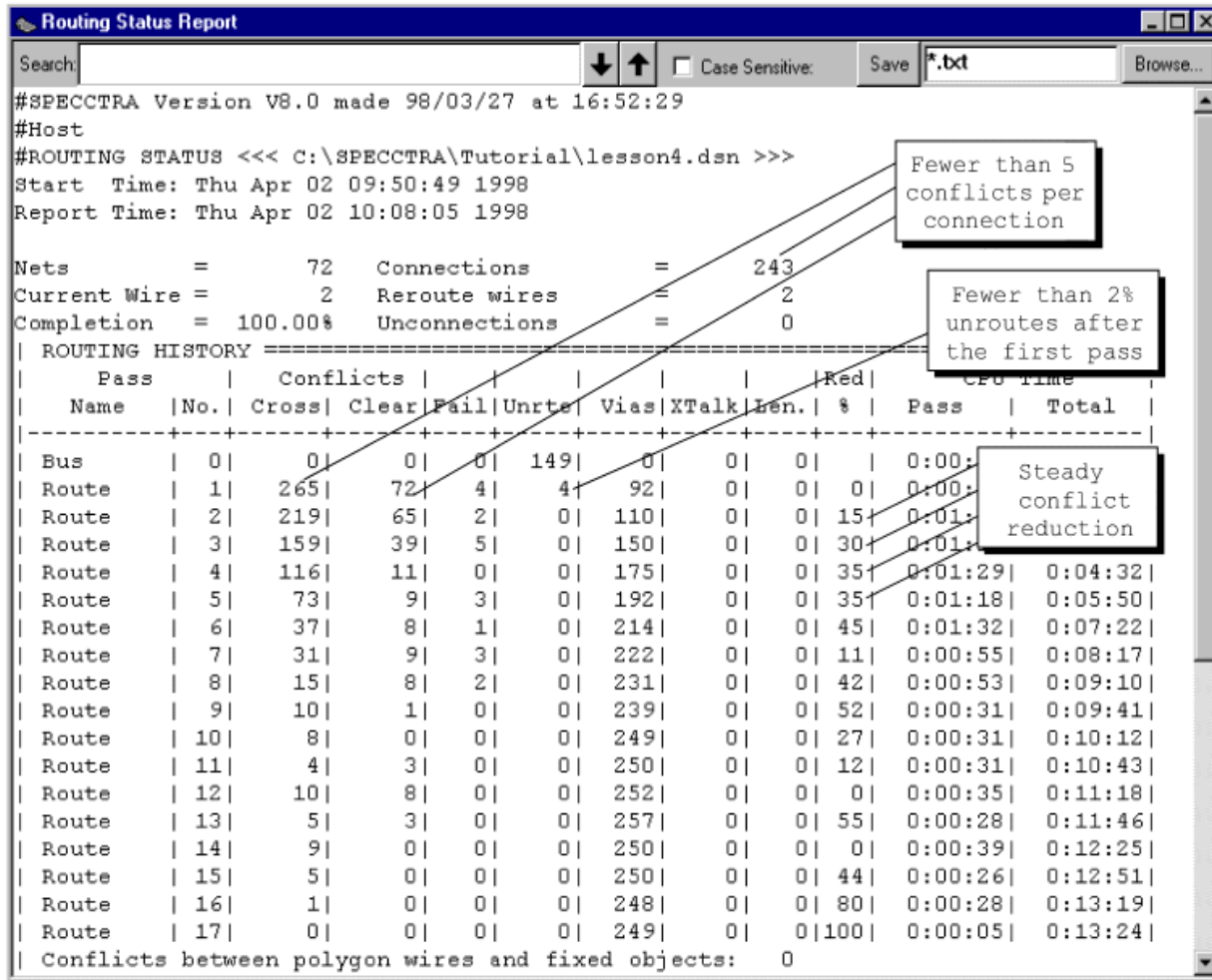
On a UNIX platform, click *C/lose* at the bottom of the report window.

The following figure shows the Routing Status Report.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

**Figure 1-5 Routing Status Report**



You will learn how to read and analyze a Routing Status report in Lesson 3. For now, you need to know how to display the report and locate information.

## Using the Mouse to Pan and Zoom

You zoom in and magnify an area of the design by using the middle mouse button to drag the pointer diagonally from the lower left to the upper right corner of the area you want to magnify. As you drag the pointer, SPECCTRA dynamically changes your view.

You zoom out by dragging the pointer diagonally from upper right to lower left.

If you drag the pointer horizontally, the autorouter fits the whole design in the view.

You pan by clicking the middle mouse button on the point that you want to be the center of your view.

If you have a three button mouse, you use the middle mouse button to zoom and pan your view of the design. If you have a two button mouse, you hold down the [Alt] or [◆] key and use the right mouse button to zoom and pan.

### Task: Zoom and pan using a three button mouse

#### Procedure

1. Identify an area of the design where you want to zoom in.
2. Using the middle mouse button, drag the pointer from lower left to upper right to enclose the area and release the mouse button to zoom in.
3. Repeat the previous step to zoom in again.
4. Move the pointer left, right, above, or below the center of the current view.
5. Click the middle mouse button.

The location of the pointer becomes the new center of view.

6. Repeat the previous step several times.  
Each time you click, the location of the pointer becomes the new center of view.
7. Drag the pointer diagonally from upper right to lower left to zoom out.
8. Click the *View All* button on the tool bar to fit the entire design in the view.

To zoom and pan using a two button mouse, you hold down the [Alt] or [◆] key and use the right mouse button.

**Task: Zoom and pan using a two button mouse**

**Procedure**

1. Identify an area of the design where you want to zoom in.
2. Pressing either [Alt] or [◆] on the keyboard and the right mouse button, drag the pointer from lower left to upper right to enclose the area and release the mouse button.

The enclosed area is magnified to fill the view.

3. Press [Alt] or [◆] and use the right mouse button to zoom in again.
4. Move the pointer to a location that's left, right, above, or below the center of the view.
5. Press [Alt] or [◆], and click the right mouse button.

The view pans and centers at the location under the pointer.

6. Repeat the previous step several times.

Each time you click, the location of the pointer becomes the new center of view.

7. Press either [Alt] or [◆], on the keyboard and the right mouse button, drag the pointer diagonally from upper right to lower left to zoom out.
8. Click the *View All* button on the tool bar to fit the entire design in the view.

**Note:** You can also use *View – Zoom – In* and *View – Zoom – Out* on the menu bar to zoom in and out.

## Getting Information and Measuring Distance

The left mouse button performs a variety of functions. Each function is enabled by setting a *mode*.

The left mouse button mode displays in the mode status area (beside the mouse symbol), as shown in the following figure.

**Figure 1-6 Mode Status Area Identifying Measure as the Current Mode**



**Note:** While using the tool, the left mouse button is always in a mode. If you attempt an operation and it does not work as you expect, check the mode status area to see whether the correct mode is set.

The default left mouse button mode is Measure. When you click a point in the work area, the tool displays the X,Y coordinates in the coordinate readout area, the output window, and in the Measure dialog box. In Measure mode, you can use the left mouse button to:

- report information and properties about design objects.
- report X,Y coordinates when you click at a point in the work area.
- measure distance when you click and drag the pointer in the work area.

Measurements and information about design objects are reported in the Output window and in the Measure dialog box. Coordinates and measured distance are reported in the Output window, the Measure dialog box, and on the status bar beside the ▲ (delta) symbol. Measured distance displays in the current measurement unit.

In the following procedure, you get information about a component pin, a via, and a wire. You also measure the center to center distance between the pins of a component.

**Task: Get information about design objects and measure distance**

**Procedure**

1. Click the *Measure Mode* button on the tool bar. See [Figure 1-1](#) on page 27 for assistance

Measure appears in the mode status area, indicating that the left mouse button is set to measure mode.

2. Click on a component pin (zoom in if necessary).

The Measure dialog box appears.

3. Read the information about the pin in the Measure dialog box and in the Output window.

You can scroll the Output window to view all the pin information.

4. Click on a via.

5. Read the via information in the Measure dialog box and in Output window.

6. Click on a wire.

7. Read the information about the wire in the Measure dialog box and in the Output window.

8. Click and drag the pointer from the center of one pin to the center of another (zoom in or pan if necessary).

9. Read the measured distance on the status bar beside the ▲ symbol.

10. Check the Output window and find the X,Y coordinates of the start and end points, and the delta (measured) distance.

The type of information you can retrieve for different objects from the Measure dialog box and the Output window is listed in the following table.

**Note:** The *Setup* button on the Measure dialog box opens the Interactive Routing Setup dialog box. You can then click on the Measure tab and control whether the measure information appears in the Measure dialog box or in the Output window, or both.

**SPECCTRA Tutorial**  
Lesson 1: Learning Basic Concepts

---

**Table 1-1 Objects and Related Information**

<b>When you click on this object . . .</b>	<b>You get this information</b>
component	Pointer's X,Y location Current measurement units Component outline dimensions (bounding rectangle) Layer location Reference designator Image name X,Y location in working units X,Y location in DBU Degrees of rotation
SMD pin	Pointer's X,Y location Current measurement units Bounding rectangle for the pin and its layer location Component reference designator and image name Component location (in both measurement units and DBU) Component side location and rotation Padstack id for the pin Padstack shape description and layer location Component pin id Name of attached net
through-pin	Pointer's X,Y location Current measurement units Bounding rectangle for the pin Component reference designator and image name Component X,Y location (in both measurement units and DBU) Layer location of component Component X,Y location Pin id Padstack id for the pin Padstack shape descriptions by layer including dimensions Name of attached net

**SPECCTRA Tutorial**  
Lesson 1: Learning Basic Concepts

---

<b>When you click on this object . . .</b>	<b>You get this information</b>
via	Pointer's X,Y location Current measurement units Padstack id for the via Padstack shape descriptions by layer including dimensions Name of attached net Wire path coordinates Width and layer locations of attached wire segments
wire segment	Pointer's X,Y location Current measurement units Path coordinates Width and layer location of the segment Name of attached net



## Saving Your Work

Before you end an autorouting session, you can save your work in a Session file, Placement file, or Routes file. The following table explains the differences between these files.

**Table 1-2 File Differences**

<b>A Session file...</b>	<b>A Placement file...</b>	<b>A Routes file...</b>
Contains placement and route data and a pointer to the original design file. After you create a session file, you can start the autorouter and load the session file instead of the original design file. When the autorouter reads a session file, it loads the original design file and the route and placement data from the session file.	Contains placement data only. When you start the autorouter and load a placement file, you must specify the original design file.	Contains route data only. When you start the autorouter and load a routes file, you must specify the original design file.
Can be loaded only when you start the autorouter.	Can be loaded when you start the autorouter and after it's running.	Can be loaded when you start the autorouter and after it's running.
Can be used in all layout systems to merge the autorouting placement and routing data into the layout.	Can be used in most layout systems to merge the autorouting placement data into the layout.	Can be used in most layout systems to merge the route data into the layout.
Links a design file with specific route data from a particular session. The session file is useful for managing design revisions because, it links the original design file with the route data for a particular session.		

In the following procedure, you save a Routes file and a Session file. You usually need to save only one file. However, both procedures are included so you can become familiar with each procedure.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

#### Task: Save your work and exit the autorouter

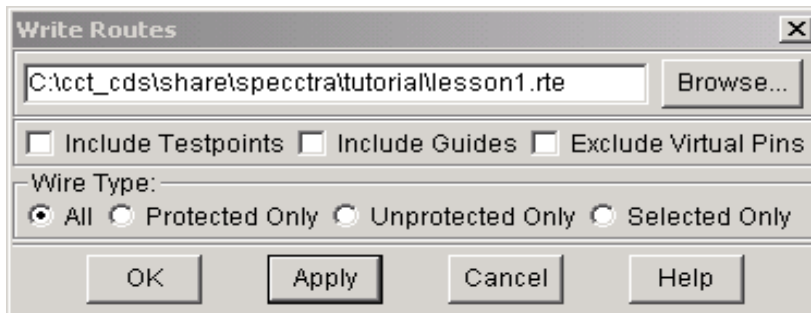
##### Procedure

1. Choose *File – Write – Routes*.

The Write Routes dialog box opens.

On a Windows platform, the path to the `tutorial` directory and the filename `lesson1.rte` appear in the data entry box, as shown in the following figure.

On a UNIX platform, the path and `/lesson1.rte` appear in the data entry box.



2. Click *OK*.

You created a Routes file. You could create a Session file using *File – Write – Session*, but you will create a Session file when you exit the autorouter.

3. Choose *File – Quit*.

The Save And Quit dialog box opens, as shown in the following figure.

On a Windows platform, the path to the tutorial directory and the file `lesson1.ses` appear in the data entry box.

On a UNIX platform, the path and `/lesson1.ses` appear in the data entry box.

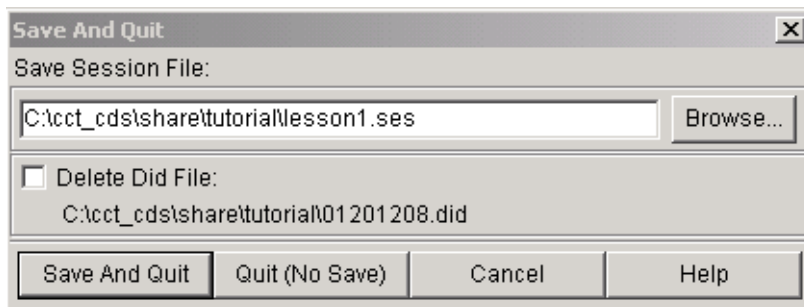
4. Make sure *Delete Did File* is not checked.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

This saves the Did file created during this session. A Did file contains the command history from the SPECCTRA work session. You will look at this Did file later in this lesson.



#### 5. Click *Save And Quit*.

SPECCTRA saves the routing information in a Session file and exits. You will use this Session file to start SPECCTRA in the next section of this lesson.

### Task: Load a Session file instead of a Design file (Windows)

#### Procedure

#### 1. Choose *Start – Programs – SPECCTRA V15 – SPECCTRA V15*.

The Startup dialog box opens.

#### 2. Enter `C:\cct_cds\share\specctra\tutorial\lesson1.ses` in the *Design/Session File* data entry box.

#### 3. Click *Start SPECCTRA*.

The Session file loads the Design file and the Wiring file from the previous session.

#### 4. Choose *Report – File*.

The Report File dialog box opens.

#### 5. Click *Browse*.

The Open dialog box appears.

#### 6. Select the tutorial directory from the Directories list.

#### 7. Select `lesson1.ses` from the File Name list and click *Open*.

The path and filename are added to the Report File dialog box.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

**8.** Click *OK*.

A report window displays the Session file.

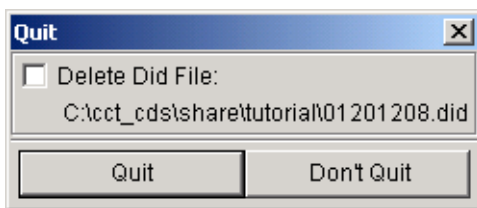
**9.** Browse the Session file and locate the following:

- ☐ Name of the Session file (`lesson1.ses`)
- ☐ Name of the original Design file (`lesson1.dsn`)
- ☐ Date and time the file was created
- ☐ Section of the Session file that contains the route data (located below the `network_out` statement)

**10.** Close the Session file.

**11.** Choose *File – Quit*.

The Quit dialog box opens, as shown in the following figure.



**12.** Make sure *Delete Did File* is not checked.

**13.** Click *Quit*.

### Task: Load a session file instead of a design file (UNIX)

#### Procedure

**1.** Enter `specctra` at the shell prompt.

The Startup dialog box opens.

**2.** Click the *Browse* button to the right of the Design/Session File data entry box.

The Select File dialog box opens.

**3.** Change the filter to `*.ses`.

**4.** Select `lesson1.ses` from the files directory and click *OK*.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

**5.** Click *Start SPECCTRA*.

The Session file is loaded with the wiring from the previous session.

**6.** Choose *Report – File*.

The Report File dialog box opens.

**7.** Click the *Browse* button and select `lesson1.ses` from the Files directory.

**8.** Click *OK* from the Report File dialog box.

A report window opens and displays the Session file.

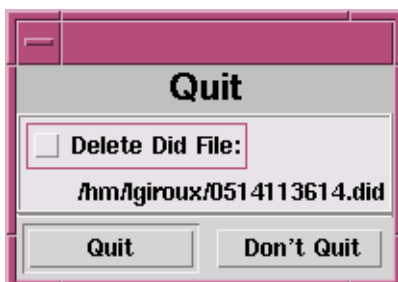
**9.** Browse the Session file and locate the following:

- ☐ Name of the session file (`lesson1.ses`)
- ☐ Name of the original Design file (`lesson1.dsn`)
- ☐ Date and time the file was created
- ☐ Section of the Session file that contains the route data

**10.** Close the Session file.

**11.** Click *File – Quit*.

The Quit dialog box opens, as shown in the following figure.



**12.** Make sure *Delete Did File* is not checked.

**13.** Click *Quit*.

# SPECCTRA Tutorial

## Lesson 1: Learning Basic Concepts

---

### Reviewing Session Command History

SPECCTRA creates a command history file for each session. This command history file is called a Did file.

On Windows platforms, the Did file is created with an eight character, numeric filename that derives from the month, day, hour, and minute when you start an autorouting session. For example, a Did file for a session you started June 15th at 11:45 is named `06151145.did`.

On UNIX platforms, the Did file is created with a 10 character, numeric filename that derives from the month, day, hour, minute, and seconds when you start an autorouting session. For example, a Did file for a session you started June 15th at 11:45:20 is named `0615114520.did`.

The following is an example of a Did file.

```
# Cadence Design Systems, Inc.
# SPECCTRA Automatic Router
# SPECCTRA Version V15.0 made 02/10/24 at 14:08:45
# Running on host 5540alcf
#
#           Command Line Parameters
#           -----
# Design File Name : ./lesson1.ses
# No "-do" or "-docmd" switches specified on command line.
# Colormap File Name : color.std
# FLEXlm License File Name : /usr/local/flexlm/licenses/license.dat
# Status File Name : ./monitor.sts
# Wires File Name : lesson1.w
#
#
report c:\specctra\tutorial\lesson1.ses
quit -c
```

Each time you start SPECCTRA, a new Did file is created. Together, the Status file, Did file, and Session file capture the routing status, command log, and routing data for the autorouting session.

### Task: View the did file from your previous session (Windows)

#### Procedure

1. Start SPECCTRA and load `lesson1.dsn`.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

**2. Choose *Report – File*.**

The Report File dialog box opens.

**3. Click *Browse* in the Report File dialog box.**

The Open dialog box opens.

**4. Scroll the Open dialog box and select the Did file from the previous session. The Did file for the previous session is located just before the Did file for the current session. The last Did file in the list is the one for the current session.**

**5. Open the Did file from the previous session.**

The path and file are added in the Report File dialog box.

**6. Click *OK*.**

A report window displays the Did file.

**7. Browse the Did file and verify the command history from the previous autorouting session.**

**8. Close the report.**

**9. Choose *File – Quit*.**

The Quit dialog box opens.

**10. Click *Delete Did File* to remove the Did file.**

**11. Click *Quit*.**

SPECCTRA exits without saving a Session file or a Did file.

### **Task: View the did file from your previous session (UNIX)**

#### **Procedure**

**1. Start SPECCTRA and load `lesson1.dsn`.**

**2. Click *Report – File*.**

The Report File dialog box opens.

**3. Click *Browse* in the Report File dialog box.**

The Select File dialog box opens.

## SPECCTRA Tutorial

### Lesson 1: Learning Basic Concepts

---

4. Select the Did file from the previous session. The Did file for the previous session is located just before the Did file for the current session. The last Did file in the list is the one for the current session.

5. Click *OK*.

The path and filename are added in the Report File dialog box.

6. Click *OK*.

A report window displays the Did file.

7. Browse the Did file and verify the command history from the previous autorouting session.

8. Close the report.

9. Choose *File – Quit*.

The Quit dialog box opens.

10. Click *Delete Did File* to remove the Did file.

11. Click *Quit*.

SPECCTRA exits without saving a Session file or a Did file.

In Lesson 3, you will learn how to use a Did file to capture GUI commands and create a command file (Do file) that you can run in SPECCTRA.



## **What You Learned**

In this lesson, you learned about the SPECCTRA's user interface and how to perform basic design tasks.

You learned how to:

- start the autorouter and load a design.
- use the GUI to execute commands.
- pan and zoom within in the Design window.
- measure distance and get information about design objects.
- save your work.
- review a session's command history.

In the next lesson, you will learn how to place components interactively and automatically.

# **SPECCTRA Tutorial**

## Lesson 1: Learning Basic Concepts

---

---

## Lesson 2: Placing Components

---

### What You Will Learn

This lesson teaches you how to use SPECCTRA's interactive and automatic placement tools.

In this lesson you will learn:

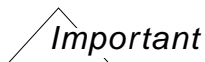
- the basic steps for placing components.
- how to set placement options.
- how to set placement rules.
- how to preplace connectors and other critical components.
- how to place large components.
- how to edit the placement.
- how to place small components.

This lesson takes about 60 minutes to complete.

### What to do Before You Begin

Before you begin this lesson, do the following:

- Complete *Lesson 1: Learning Basic Concepts*.



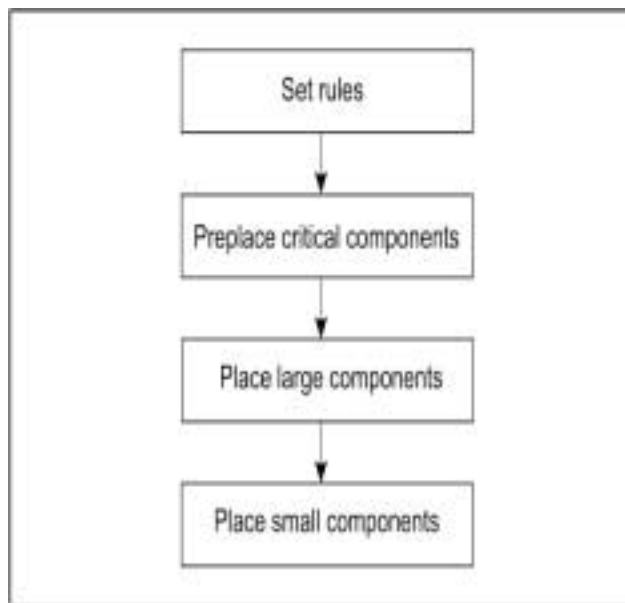
A PlaceBase license is required to complete the tasks in this lesson. For details on SPECCTRA licensing, products and features, refer to Chapter 1 of the *SPECCTRA User Guide*.

## Understanding the Basic Steps for Placing Components

Placing components consists of four basic steps.

- Setting placement rules
- Preplacing critical components
- Placing large components
- Placing small components

**Figure 2-1 Basic Steps of Component Placement**



In the first step, you set placement rules. During both automatic and interactive placement, the design is checked for rule violations.

In the second step, connectors and other position-critical components are preplaced and locked in position using interactive placement tools.

In the third and fourth steps, components are placed and locations are optimized to reduce manhattan lengths and guide crossings. Large components are placed first and then small components.

## Setting Placement Rules

SPECCTRA provides a comprehensive set of placement rules. Basic placement rules control the spacing between components, the orientation of components, and the sides on which the autorouter places components. More advanced rules control floor planning based on power dissipation, power supply, and component height. In this lesson, you will set basic rules. For more information on setting advanced placement rules, see the Placement online help.

## Understanding the Rules Hierarchy

Placement rules can be specified at different levels and, consequently form a hierarchy. Rules at higher levels in the hierarchy override rules at lower levels that are set for the same physical objects. For example, consider what happens if you set a global (PCB) spacing rule of .25 inches for all components and a spacing rule of .8 inches for a specific connector. The autorouter follows the .8 spacing rule only in the area surrounding the connector. The autorouter follows the .25 spacing rule in the areas surrounding other components. The component-level spacing rule overrides the PCB-level spacing rule.

The following table shows the levels of hierarchy in which you can set rules and the order of precedence for all placement rule levels. Global placement rules (PCB rules) have the lowest precedence, and image\_image rules have the highest precedence.

**Table 2-1 Rules Hierarchy**

<b>This rule level . . .</b>	<b>Does this</b>
image_image	Sets rules between images. An image is the footprint definition of a component. This is the highest precedence rule.
family_family	Sets rules between families. A family is a group of images.
room_image_set	Sets rules for an image_set assigned to a room.
room	Sets rules for a room, which is an area of the design in which the autorouter places specified components.
super cluster	Sets rules for a super cluster, which is a group of components that is treated as a single component.
component	Sets rules for a component, which is an instance of an image.
image	Sets rules for an image, which defines a component footprint. An image is the footprint definition of a component.
image_set	Sets rules for images with the same type property. The type properties are large, small, capacitor, or discrete.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

---

This rule level . . .	Does this
pcb	Sets global rules for all components in the design. This is the lowest precedence rule.

---

**Note:** The order of precedence is fixed and cannot be changed.

### Setting a Placement Grid and Spacing Rule

Using the Placement Setup dialog box, you can set a placement grid and a PCB spacing rule.

SPECCTRA does not require that you define a grid for automatic placement, but if design or manufacturing rules dictate a placement grid, you can set the grid for all components (SMD and through-pin) in the Placement Setup dialog box.

**Note:** You can set separate placement grids for SMD and through-pin components by choosing *Rules – PCB – Placement Grids*.

In addition to setting the placement grid, you can use the Placement Setup dialog box to set a PCB spacing rule. A PCB spacing rule sets the minimum spacing allowed between all components in the design. Rules are set at other levels through the *Rules* menu. In the following procedure, you will set a PCB spacing rule of .05 inches.

### Task: Set a placement grid and spacing rule

#### Procedure

1. Start SPECCTRA and load `lesson2.dsn` from the `tutorial` directory. See [Where to find the Accompanying Lesson Files](#) on page 12 for the location of this directory.

When you first start SPECCTRA, you are in Route mode. The menu bar and status bar are specific to autorouting. You need to change to Place mode.

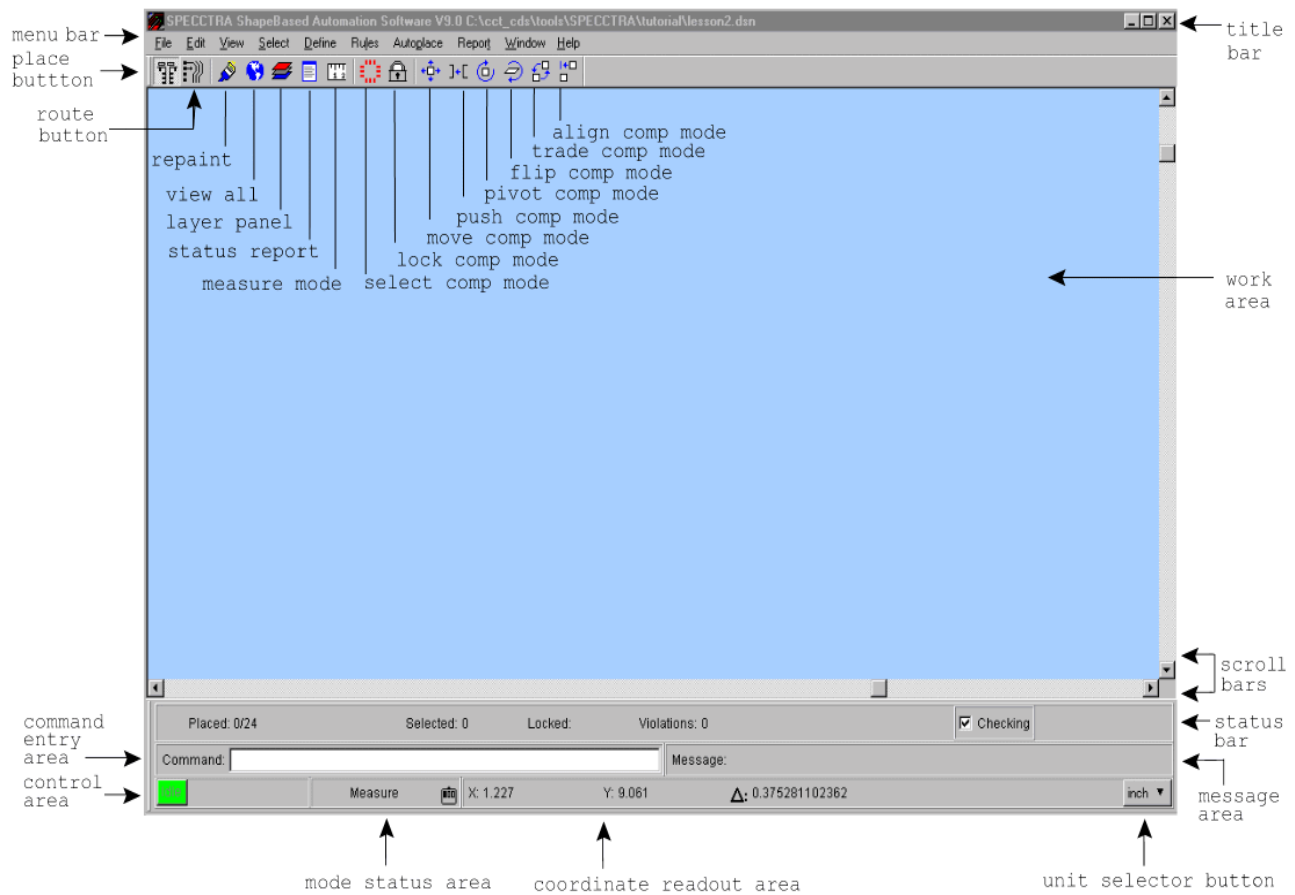
2. Click the *Place* button on the tool bar.

The tool bar changes.

# SPECCTRA Tutorial

## Lesson 2: Placing Components

### Placement GUI (Windows)



### 3. Choose *Autoplace – Setup*.

The Placement Setup dialog box opens.

### 4. Enter **.05** in the *PCB Placement Grid* data entry box.

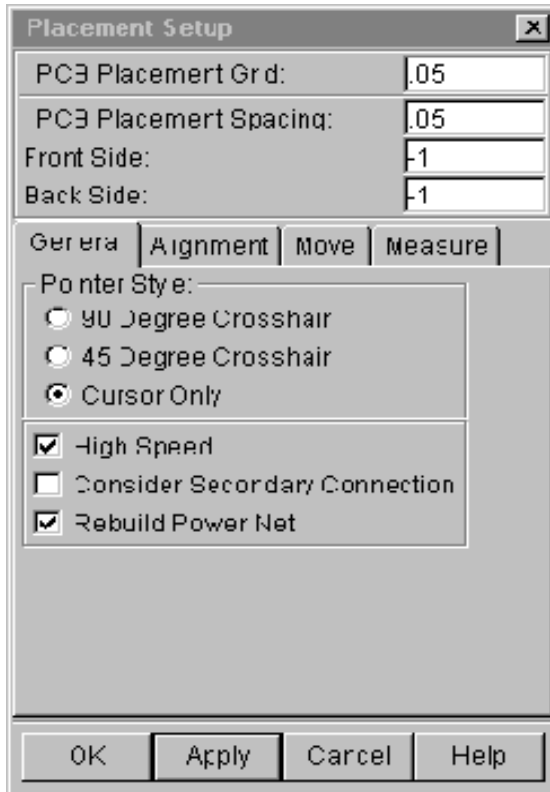
### 5. Enter **.05** in the *PCB Placement Spacing* data entry box.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The Placement Setup dialog box looks like the following.



#### 6. Click OK.

You defined a grid of .05 inches and a spacing rule of .05 inches. Notice that the *Checking* option in the right side of the status area is selected. This option checks for placement rule violations while you interactively place components and prevents placing or moving a component if it violates a rule. If you want to place a component in a location that violates a rule, turn off *Checking*.

**Note:** You can set separate spacing rules for SMD components, through-pin components, and between SMD and through-pin components by choosing *Rules – PCB – Spacing*. You also can use this command to see the current PCB spacing rule.



## Preplacing Connectors and Critical Components

After you set placement rules, you are ready to preplace connectors and other critical components. In this stage of placing components, you will:

- display the component reference designators.
- place components by specifying XY locations.
- place components from a defined list.
- lock the preplaced components.

## Displaying Component Reference Designators

In the following procedure, you turn on the reference designator labels so that you can identify components.

### Task: Display component reference designators

#### Procedure

1. Choose *View – Labels*.

The View Labels dialog box opens.

2. Make sure *Ref Des* is selected.
3. Make sure the *Side* is set to *Both*.
4. Click *OK*.

A reference designator label appears in the center of each component.

**Note:** The router scales the work area. If you do not see all the designator labels, you need to zoom in to adjust the magnification of the work area.

## Placing Components by Specifying X,Y Locations

Through the Interactive Place menu, the autorouter offers a variety of ways to place components interactively. You can place a single component, a list of components, and multiple components. You can place components by specifying X,Y locations, by clicking a point in the work area, or based on connectivity.

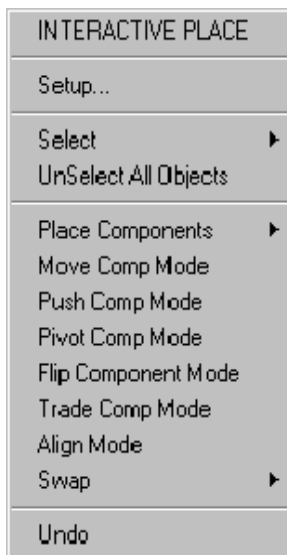
## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The Interactive Place menu appears when you press the right mouse button in the work area as shown in the following figure.

**Figure 2-2 Interactive Place Menu**



To select a command from the Interactive Place menu, you will follow these steps:

1. With Place mode active and your pointer in the Design window, press the right mouse button.

The Interactive Place menu appears.

2. Slide your pointer over the menu to highlight the desired command.
3. Select the command by pressing the left mouse button.

In this section, you will place the origins of components J1, J2, and U9 at exact X,Y locations.

### **Task: Place components by specifying X,Y locations**

#### **Procedure**

1. Press the right mouse button and choose *Place Components – XY Location*.

The Place Component dialog box opens, as shown in the following figure.

2. Scroll the *Components* list to locate J1.

## SPECCTRA Tutorial

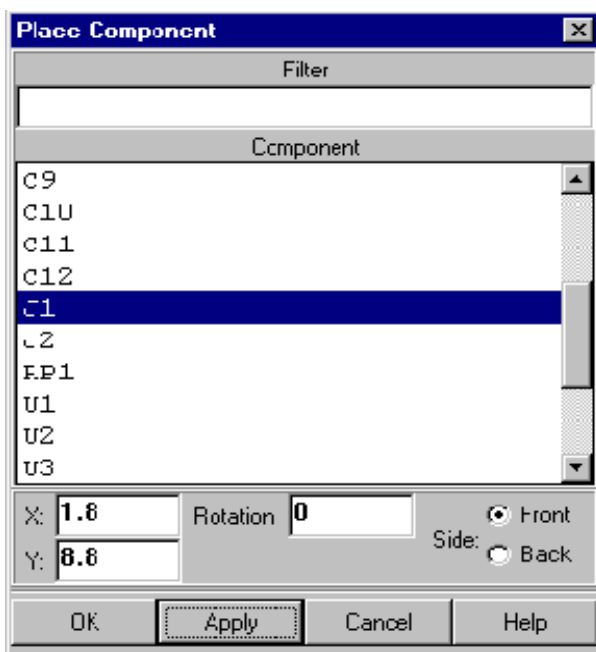
### Lesson 2: Placing Components

---

3. Select J1 in the *Components* list.

J1 is highlighted.

4. Enter **1.8** in the *X* data entry box.
5. Enter **8.8** in the *Y* data entry box.
6. Make sure the *Rotation* is 0.
7. Make sure the *Side* is *Front*.



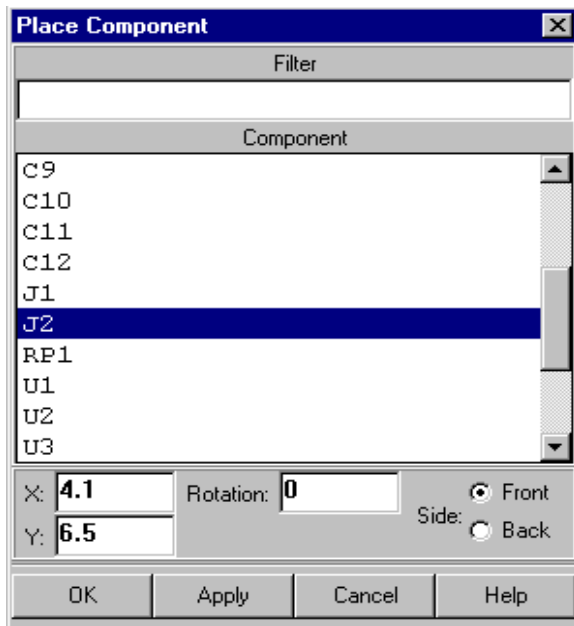
8. Click *Apply* to place the origin of J1 at 1.8, 8.8.  
Next you place J2.
9. Select J2 in the *Components* list.
10. Type **4.1** in the *X* data entry box.
11. Type **6.5** in the *Y* data entry box.  
Keep the *Rotation* at 0 and *Side* set to *Front*.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The dialog box looks like the following.



12. Click *Apply* to place the origin of J2 at 4.1, 6.5.

Next you place the PGA component labeled U9. Like the connectors, U9 has a large number of interconnections with other components in the design, which is why you interactively place it.

13. Click U9 in the *Components* list.
14. Enter **3.1** in the *X* data entry box.
15. Enter **6.9** in the *Y* data entry box.

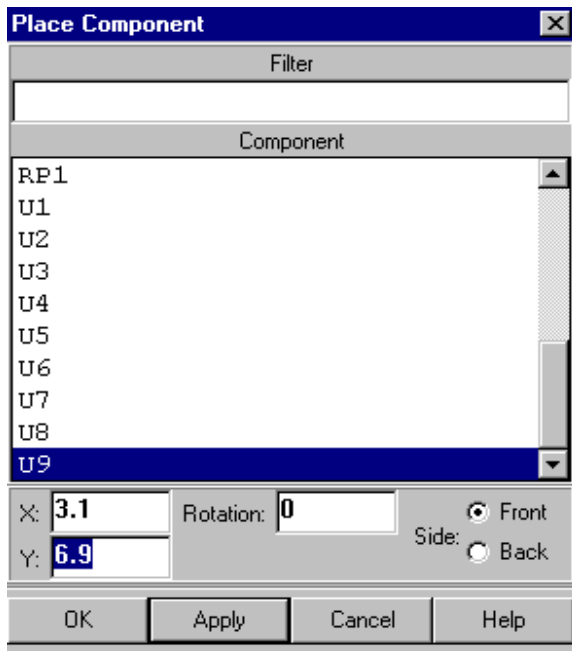
Keep the *Rotation* at 0 and *Side* set to *Front*.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

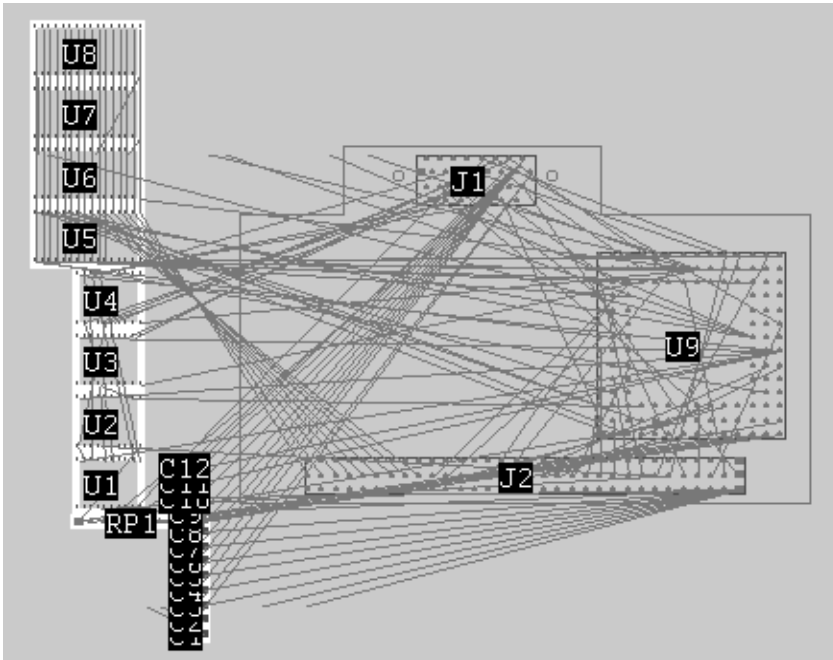
---

The dialog box looks like the following.



16. Click OK.

The design is shown in the following figure.



## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

You just learned how to place components by specifying X,Y coordinates. Next you will learn how to place a list of components.

#### Placing Components From a List

Using Place List mode, you can specify a list of components and place them in the order you specify. The first component in the list attaches to the pointer. Drag the component to the location you want, and click the left mouse button to place it. Immediately, the next component in the list attaches to the pointer so you can drag the component to the location you want and click the left mouse button to place it. You repeat this process until all components in the list are placed.

#### Task: Place components from a list

##### Procedure

1. Press the right mouse button and select *Place Components – Place List Mode*.

The Place Component List dialog box opens.

2. Select *Ordered Components*, which tells the autorouter to place the components in the order that you specify in the *Comp List*.
3. Type the following in the *Comp List* data entry box:

c1 c2 c3 c4

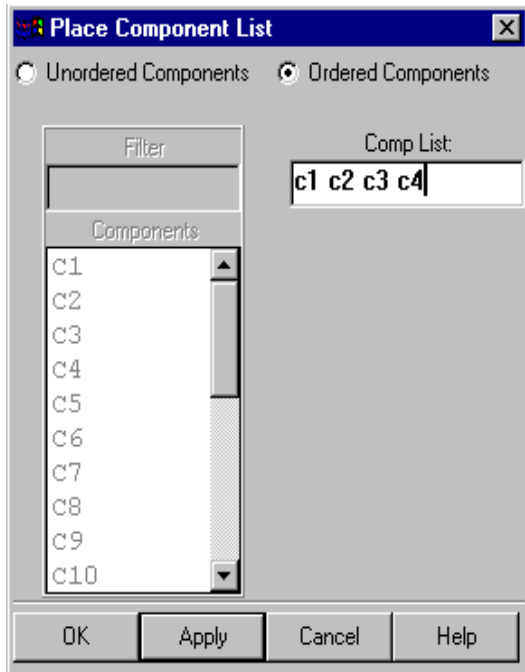
**Note:** Make sure you leave a space between each reference designator.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The Place Component List dialog box looks like the following.



**4. Click OK.**

C1 attaches to the pointer. You will need to rotate it 90 degrees counterclockwise before you place it.

**5. Press the right mouse button and select *Pivot Mode – 90*.**

A pivot arm attaches to the center of the component. This pivot arm looks like a string that pivots the component in the direction you move the pointer.

**6. Move the pointer away from the center of the component in a counterclockwise direction.**

The component rotates 90 degrees counter clockwise. increments as you move the pointer.

*DR:90* appears near the right side of the status bar when the component is 90 degrees from its original position. DR stands for Delta Rotation.

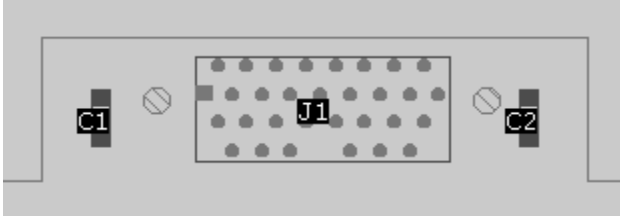
**7. Click the left mouse button when C1 is 90 degrees counterclockwise from its original position.**

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

You are ready to place C1. The following figure shows where to place C1 and C2.



8. Drag C1 to the left of J1 and click to place it.

You can use the middle mouse button for the following operations to pan or zoom to a new location.

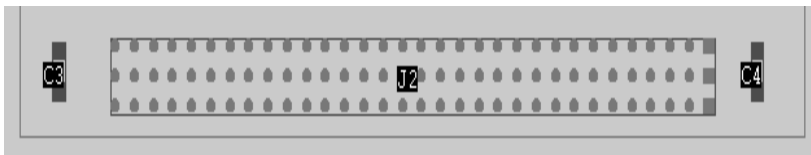
C2 attaches to the pointer after you place C1.

9. Repeat steps 5 through 7 to rotate C2 90 degrees counterclockwise.
10. Drag C2 to the right of J1 and click to place it.

C3 attaches to the pointer.

11. Repeat steps 5 through 7 to rotate C3 90 degrees counterclockwise.

You are ready to place C3. The following figure shows where to place C3 and C4.



12. Drag C3 to the left of J2 and click to place it.
- C4 attaches to the pointer.
13. Repeat steps 5 through 7 to rotate C4 90 degrees counterclockwise.
14. Drag C4 to the right of J2 and click to place it.

**Note:** If you do not like where you placed a component and you want to move it, press the right mouse button and select *Move Component*. Click the component, drag it to the new location, and click to place it.

Next you will lock the preplaced components so the autorouter cannot move them during automatic and interactive placement.



## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

#### Locking Components

After you place critical components, you lock them in position so SPECCTRA does not move them during automatic or interactive placement. If you need to move a component at a later time, you can unlock it, move it, and lock it again.

#### Task: Lock critical components

##### Procedure

1. Click the *Lock* button on the tool bar.

Lock Position appears in the mode status area.

2. Click a point to the left of C1.
3. Drag the pointer diagonally around C1, J1, and C2, and release the left mouse button when the bounding box encloses the components.

The locked components display a magenta border. Magenta is the default *locked* color.

4. Repeat steps 2 and 3 to lock components C3, J2, and C4.

**Note:** You can unlock the components while you are in Lock Position mode. To unlock the components, click on each locked component or draw a bounding box around the components. You also can use *Edit – [Un]Lock Components* and *Edit – [Un]Lock Components Mode* to lock and unlock components.

## Placing Large Components

SPECCTRA assigns the large and small properties to components based on the number of pins. A large component contains four or more pins. A small component contains three or less pins. SPECCTRA usually places large components with the highest connectivity first.

In this section you will:

- define areas where you do not want components placed.
- automatically place large components.
- interchange placed components to reduce manhattan lengths, minimize crossovers, and reduce congestion.

## Defining Areas where SPECCTRA Cannot Place Components

Before you automatically place the large components, you need to provide routing space around the connectors. To do this, you specify an area around each connector where the SPECCTRA cannot place components. These areas are called placement *keepout* areas.

### Task: Define placement keepout areas

#### Procedure

1. Choose *Define – Keepout – Draw Mode*.

Draw Keepout appears in the mode status area. You will draw a bounding box as close as possible to C1, J1, and C2.

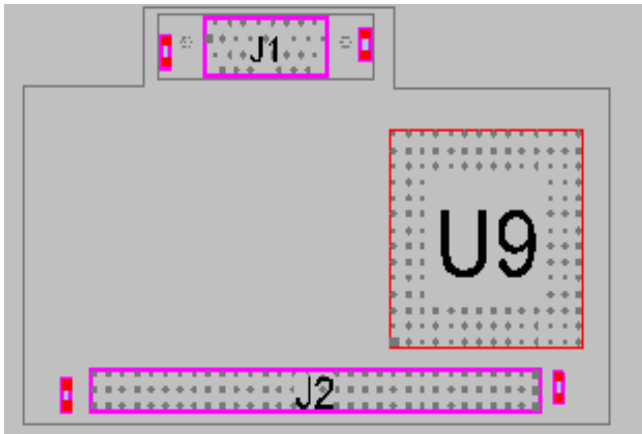
2. Click a point to the lower left of C1.
3. Drag the pointer diagonally around C1, J1, and C2, and release the left mouse button when the bounding box encloses the components. If you do not like how you drew the box, choose *Define – Keepout – Draw Mode* to remove the box and start with step 2 to draw a new bounding box.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

In the following figure, the ratsnest is not displayed. On your screen, the area around C1 and C2 might be difficult to see because the ratsnest is displayed.



The ratsnest lines are called *unroutes*. You will turn off the display of unroutes, so that you can see the keepout area more easily.

4. Choose *View – Guides – Off*.

The keepout area is now easier to see.

5. Press the right mouse button and select *Define Polygon as Keepout*.

The Add Polygon as Keepout dialog box appears, as shown in the following figure.

6. Make sure the Keepout ID is *keepout1*.

The *Keepout ID* is the name the autorouter assigns to the keepout area.

7. Select *Place* as the *Type*.

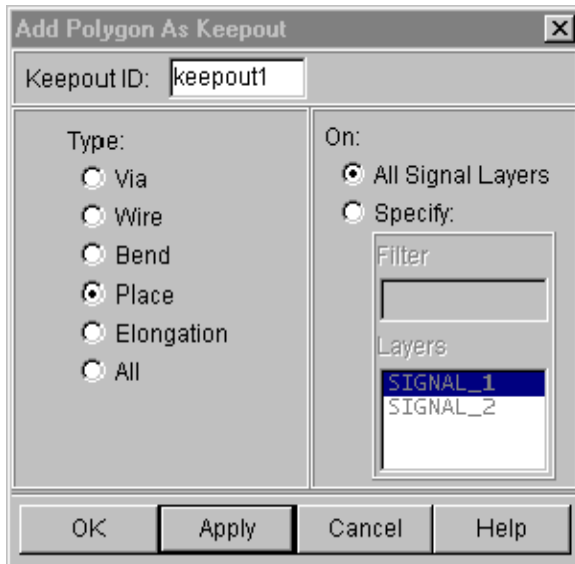
This sets the area as a placement keepout, in which SPECCTRA cannot place components.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The Add Polygon As Keepout dialog box looks like the following.



8. Click **OK**.

The keepout area appears as a bounding box filled with a crosshatched pattern.

**Note:** Information about keepouts will display in the measure box and output window if the object selection button is turned on in the layers panel next to the keepout button.

Next you create a keepout area around C3, J2, and C4.

9. Make sure Draw Keepout mode is still set by checking the mode status area.
10. Click a point to the lower left of C3.
11. Drag the pointer diagonally around C3, J2, and C4, and release the left mouse button when the bounding box encloses the components. Remember to draw the boundary box as close as possible to the components.
12. Press the right mouse button and select *Define Polygon as Keepout*.

The Add Polygon as Keepout dialog box opens.

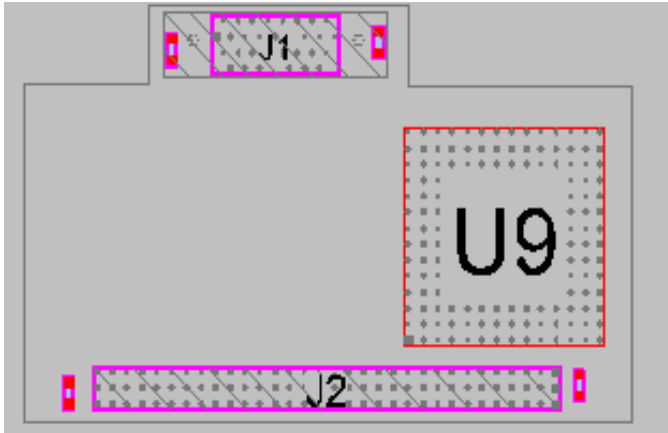
13. Make sure the Keepout ID is *keepout2* and *Place* is selected.
14. Click **OK**.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The keepout areas appear as crosshatched rectangles, as shown in the following figure.



Next you will automatically place the large components. First, display the unrouted so you can see the connectivity while the autorouter places the components.

#### 15. Choose *View – Guides – All*.

The guides display.

**Note:** To create a keepout area by entering coordinates, choose *Define – Keepout – By Coordinates*.

### Automatically Placing Large Components

You initially place ICs and other large components by using the InitPlace Large Components dialog box. You use the dialog box to control the preferred component spacing, placement side, and component orientations.

These preferences are not rules. Preferences set in the InitPlace Large Components dialog box are followed if they do not violate placement rules.

You will place the large SMD components on the front side with a vertical orientation and the PTH component (RP1) on the front side with a horizontal orientation.

### Task: Automatically place large components

#### Procedure

##### 1. Choose *Autoplace – InitPlace Large Components*.

The InitPlace Large Components dialog box opens, as shown in the following figure.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

2. Make sure *All* is selected on the components panel.

This option places all large, unplaced components.

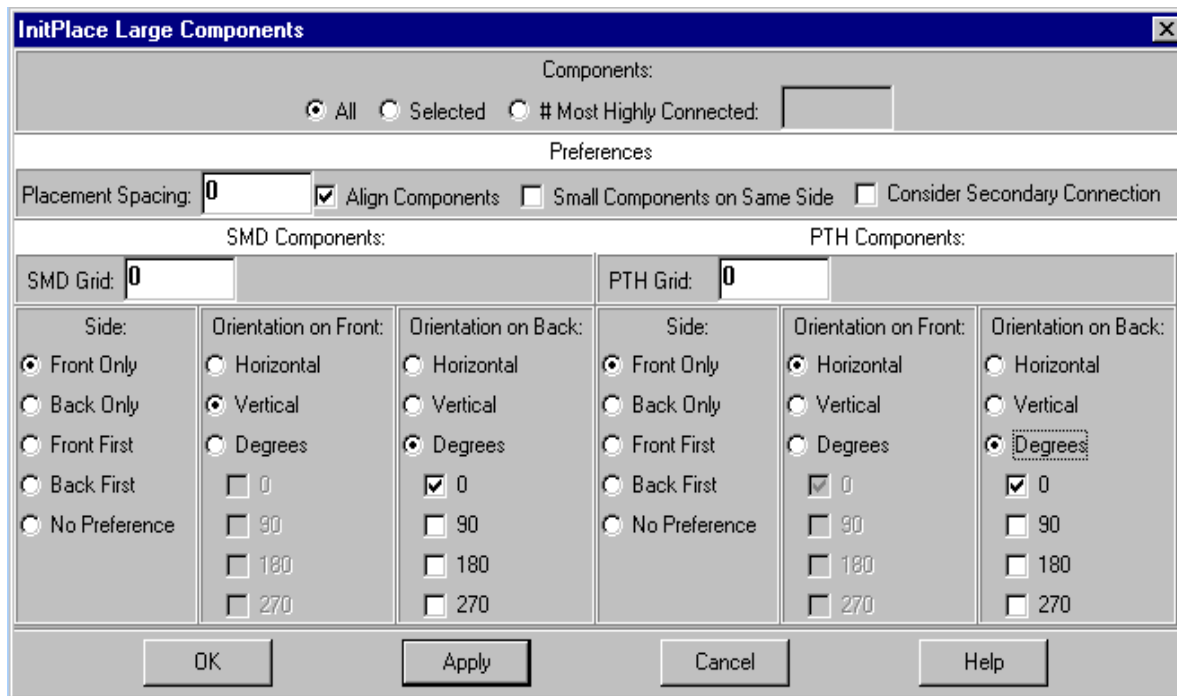
3. Select *Front Only* for the *Side* on the *SMD Components* panel.

This option is on the left side of the dialog box.

4. Select *Vertical* for *Orientation on Front* on the *SMD Components* panel.

5. Select *Front Only* for the *Side* on the *PTH Components* panel.

6. Click *Horizontal* for *Orientation on Front* on the *PTH Components* panel.



The dialog box is titled "InitPlace Large Components". It contains several sections for configuring component placement:

- Components:** Radio buttons for *All* (selected), *Selected*, and *# Most Highly Connected*. There is an empty text field next to the last option.
- Preferences:** A text field for "Placement Spacing" set to *0*, and three checkboxes: *Align Components* (checked), *Small Components on Same Side* (unchecked), and *Consider Secondary Connection* (unchecked).
- SMD Components:**
  - SMD Grid:** Text field set to *0*.
  - Side:** Radio buttons for *Front Only* (selected), *Back Only*, *Front First*, *Back First*, and *No Preference*.
  - Orientation on Front:** Radio buttons for *Horizontal*, *Vertical* (selected), and *Degrees*. Below *Degrees* are checkboxes for *0*, *90*, *180*, and *270*.
  - Orientation on Back:** Radio buttons for *Horizontal*, *Vertical*, and *Degrees* (selected). Below *Degrees* are checkboxes for *0* (checked), *90*, *180*, and *270*.
- PTH Components:**
  - PTH Grid:** Text field set to *0*.
  - Side:** Radio buttons for *Front Only* (selected), *Back Only*, *Front First*, *Back First*, and *No Preference*.
  - Orientation on Front:** Radio buttons for *Horizontal* (selected), *Vertical*, and *Degrees*. Below *Degrees* are checkboxes for *0* (checked), *90*, *180*, and *270*.
  - Orientation on Back:** Radio buttons for *Horizontal*, *Vertical*, and *Degrees* (selected). Below *Degrees* are checkboxes for *0* (checked), *90*, *180*, and *270*.

At the bottom are four buttons: *OK*, *Apply*, *Cancel*, and *Help*.

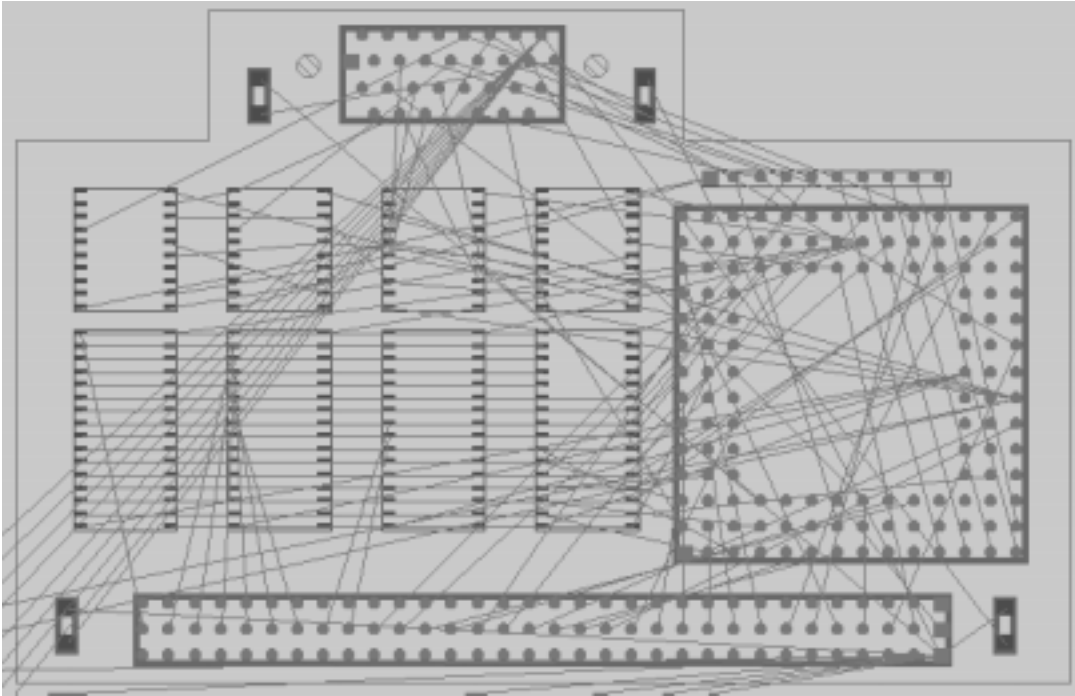
7. Click *OK*.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The autorouter places the SMD and PTH components, as shown in the following figure.



### Interchanging Components

After the initial placement, you interchange component positions to reduce manhattan lengths, minimize crossovers, and reduce congestion.

The interchange operation is sometimes referred to as a pairwise interchange because the operation applies to a single pair of components at a time. The goal is to place interconnected components in close proximity to reduce wire lengths.

Multiple interchange passes usually produce the best results. Use eight or more interchange passes. If an interchange pass does not improve weighted manhattan lengths compared to the previous pass, the interchange operation stops and the remaining passes are skipped. You can use the placement status report to compare manhattan lengths before and after interchange passes.

### Task: Interchange large components

#### Procedure

1. Choose *Autoplace – Interchange Components*.

## SPECCTRA Tutorial

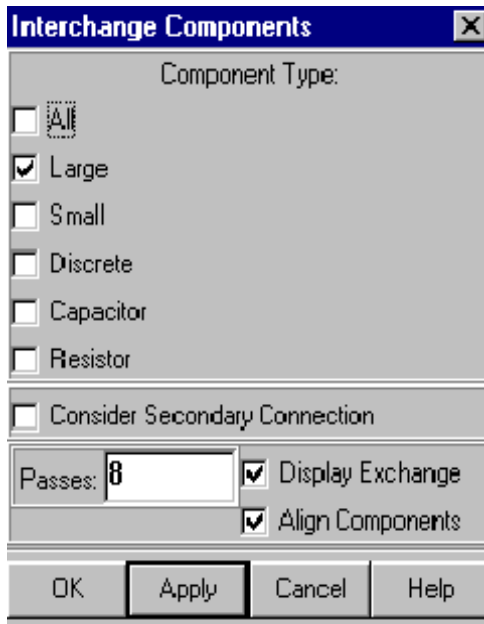
### Lesson 2: Placing Components

---

The Interchange Components dialog box appears.

2. Make sure *Large* is selected.
3. Make sure **8** is in the value for *Passes*.

The Interchange Components dialog box looks like the following.



4. Click *OK*.

After each component interchange, the work area is repainted because *Display Exchange* in the dialog box was selected. This button controls whether graphics are updated with each component interchange. To improve performance with larger designs, you unselect *Display Exchange*.

5. Choose *File – Quit*.

The Save and Quit dialog box opens, as shown in the following figure.

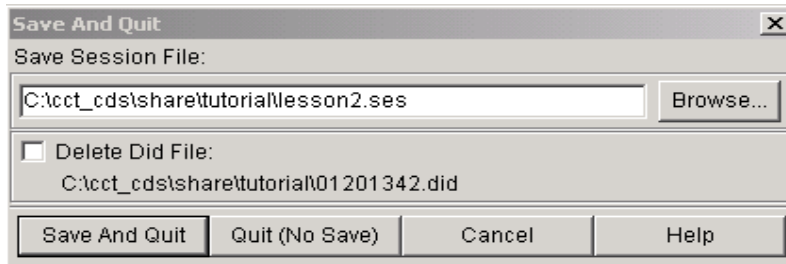


## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

6. Click *Delete Did File* to remove the did file.



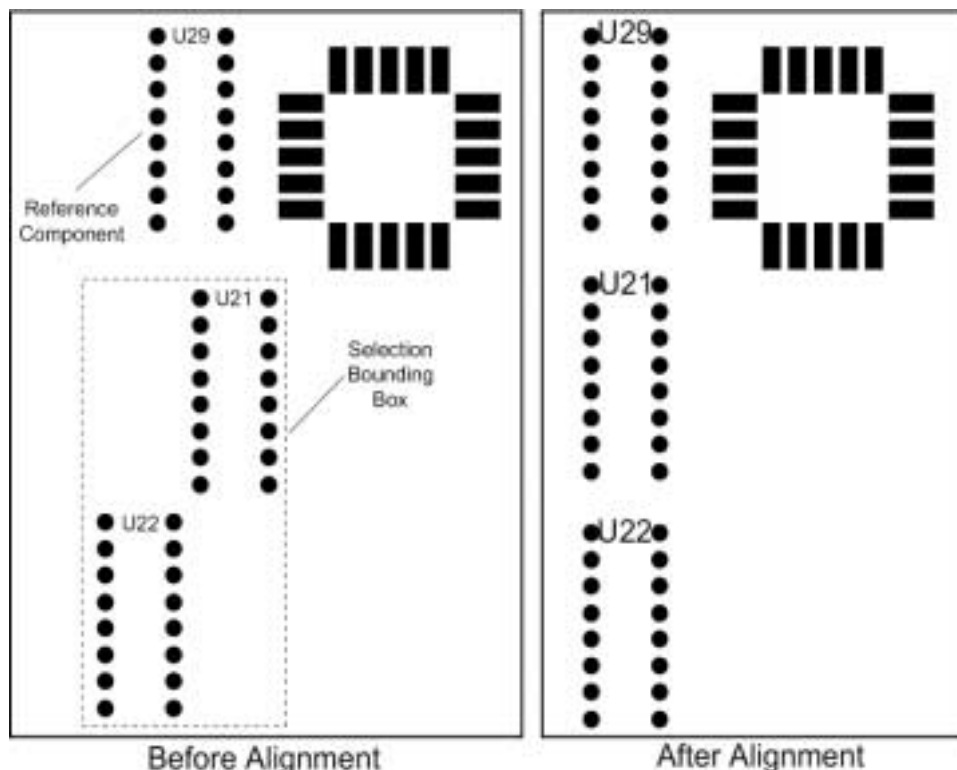
7. Click *Quit (No Save)* to exit without saving a session file and Did file.

Next you will learn to interactively align components.

### Aligning Components

You can align components by using Align Comp mode. In this mode, you select the components you want to move, then you select a reference component to which these components align.

**Figure 2-3 Component Alignment**



## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

If *Checking* is on and the alignment causes a placement violation, SPECCTRA does not align components. If you want to ignore rule violations, turn off *Checking* in the status bar. You'll leave *Checking* on in the following procedure.

#### Task: Align components

##### Procedure

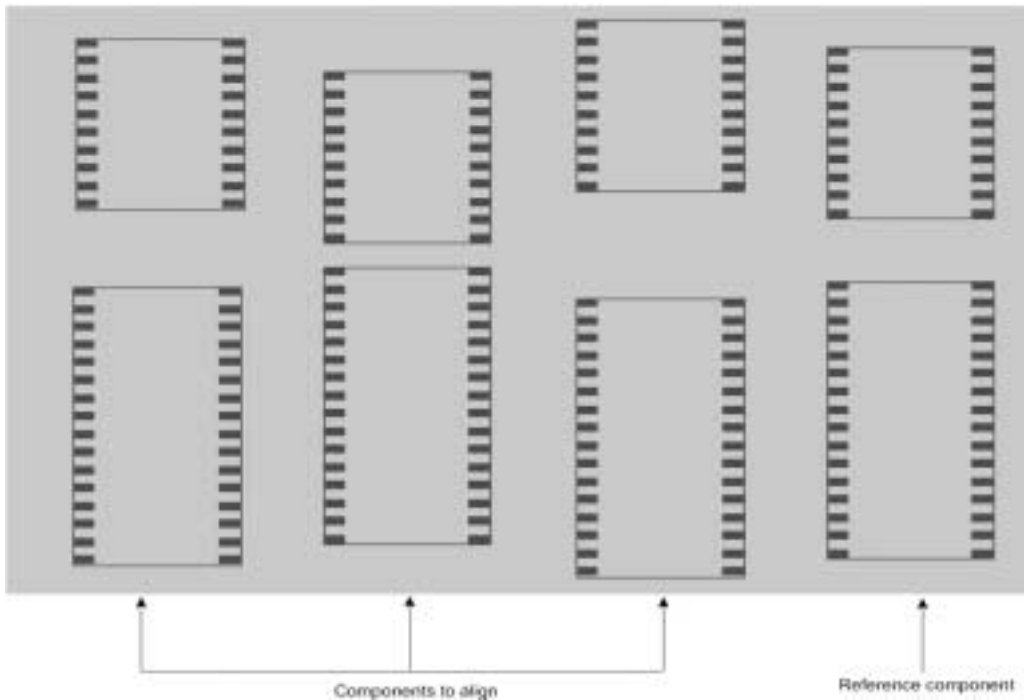
1. Start SPECCTRA and load `lesson2.dsn` from the `tutorial` directory.  
You need to change to Place mode.
2. Click the *Placement* button on the tool bar.  
You will load a placement file.
3. Choose *File – Read – Placement*.  
The Read Placement dialog box opens.
4. Click the *Browse* button.  
The *Open* dialog box appears for Windows platforms.  
The *Select File* dialog box appears for UNIX platforms.
5. Change to the tutorial directory for Windows platforms and open `lesson2.plc`.  
Make sure `lesson2.plc` is selected and click *OK* for UNIX platforms.  
The filename is added in the Read Placement dialog box.
6. Click *OK*.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The Placement file loads. The Small Outline Integrated Circuits (SOICs) are not aligned, as shown in the following figure.



7. Choose *View – Guides – Off* to turn off the guides.

You will align the SOICs.

8. Press the right mouse button and select *Align Mode*.

Align Comp appears in the mode status area.

You will use the right SOIC as the reference component to align the three SOICs.

9. Drag the pointer to enclose the three 32-pin SOICs that are located on the left as shown in the previous figure.

The three SOICs are selected. You will align these SOICs to the 32-pin SOIC that is located on the right.

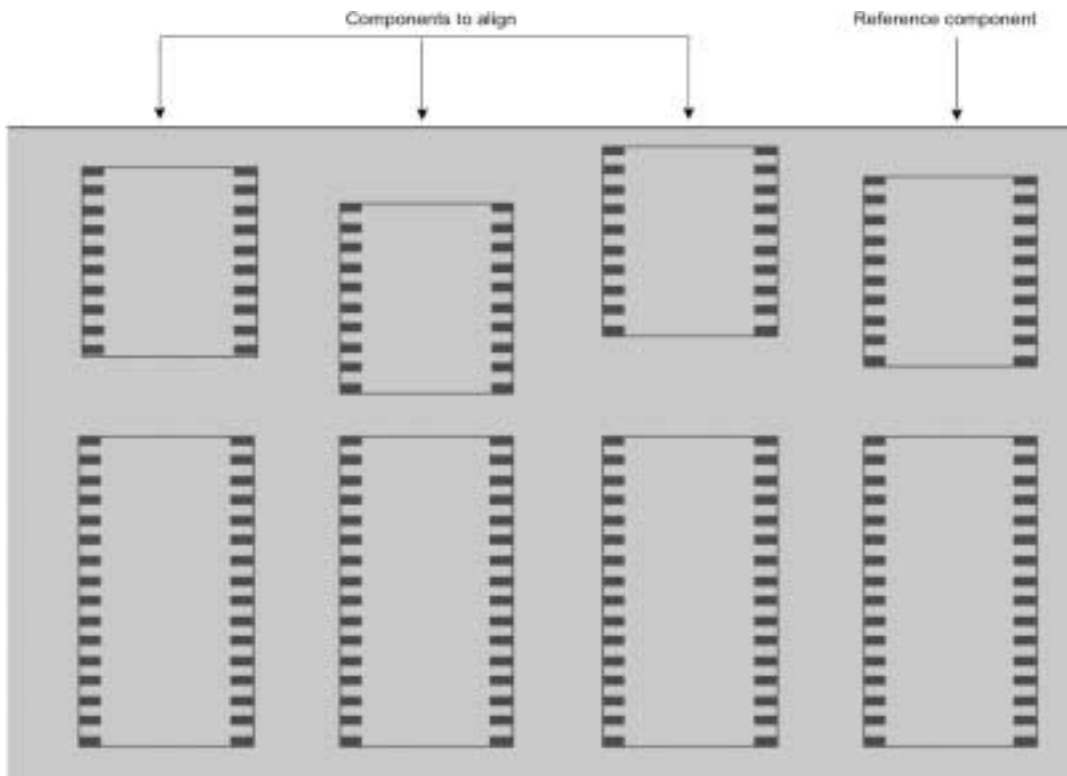
10. Click in the 32-pin SOIC that is located on the right.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The components align, as shown in the following figure.



Next you will align the three 20-pin SOICs that are located above the 32-pin SOICs as shown in the previous figure. Align Comp mode is still active.

11. Drag the pointer to enclose the three 20-pin SOICs that are located on the left.

The three SOICs are selected. You will align these SOICs to the reference 20-pin SOIC component that is located on the right.

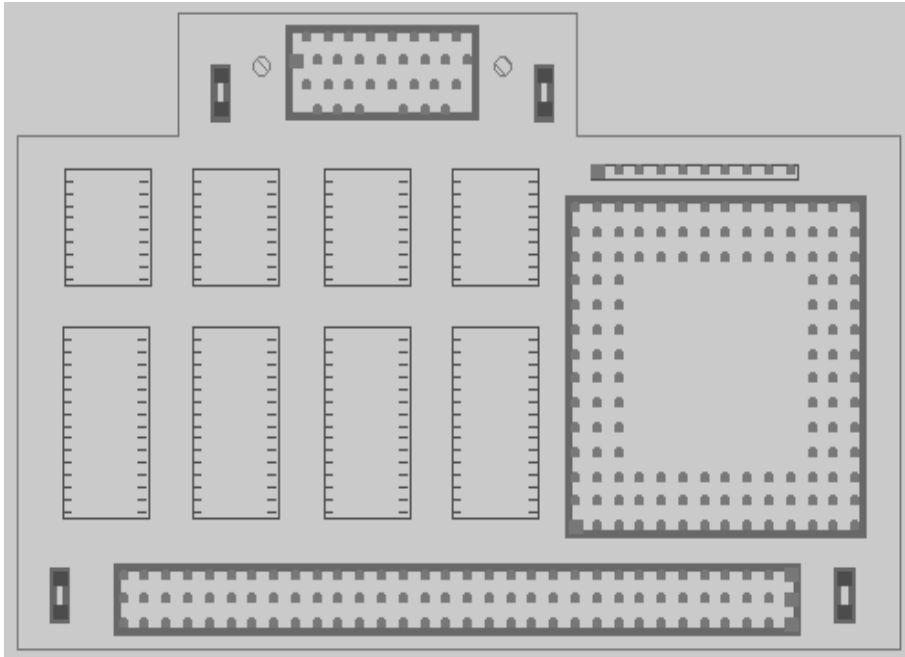
12. Click in the 20-pin SOIC that is located on the right.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

The components align, as shown in the following figure.



**Note:** SPECCTRA uses a reference point on each component to make the alignment. In the previous procedure, the alignment was relative to the upper left pin of the reference component, which is the default. You can change this reference point by using the Placement Setup dialog box.

## Placing Small Components

You can place small signal components by choosing *Autoplace – InitPlace Small Components*. To place decoupling capacitors, you will use another method.

SPECCTRA can *learn* a component pattern, which is the side, location, and orientation of a small component relative to a large component. This component pattern is applied by placing other instances of the small component image in the same pattern to other instances of the large component image.

In this section, you will place the capacitors by having SPECCTRA learn and apply component patterns.

You will complete the following tasks

- Display power pin labels
- Move, flip, and pivot a capacitor
- Learn the component pattern
- Apply the component pattern to other instances of the large component image

### Displaying Power Pin Labels

You will need to display the power pin labels to orient the components. To display the power pins, turn on the power pins layer.

#### Task: Display power pins labels

##### Procedure

1. Click the *Layer* button on the tool bar.

The Layers panel appears. The Layers panel controls layer visibility, layer routing direction, and layer selection.

2. Click *Power pins* on the Layers panel. Power pins is located near the bottom of the Layer panel.

The power pins display.

3. Click *Close* on the Layers.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

Now that you can see the power pins, you will move C5 (on the PCB bottom side) so that the power pin of C5 is under the power pin of U1 (on the PCB top side).

### **Moving, Flipping, and Pivoting a Component**

An easy way to place a component is to use Move Comp mode. When you click on a component in this mode, the component's reference designator displays. As you move the pointer, a ghost image of the component follows. The next click places the component at the location under the pointer.

You will need to display the reference designators, which appear at component centers.

### **Task: Move, flip, and pivot a component**

#### **Procedure**

1. Choose *View – Labels*.

The View Labels dialog box opens.

2. Make sure *Ref Des* is selected.
3. Click *OK*.
4. Press the right mouse button and select *Move Comp Mode*.

Move Comp appears in the mode status area.

5. Click on C5, which is the bottom component in the column of unplaced components. You can use the middle mouse button to zoom in if necessary.

The component attaches to the pointer. You need to flip the component to the back side by using the Move Comp menu.

6. Press the right mouse button and select *Flip* from the Move Comp menu.

This menu contains commands that manipulate the component before you place it.

Next you need to pivot the component so that the power pin is at the bottom of the component, as shown in the following figure.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

7. Press the right mouse button and select *Pivot – 90*.

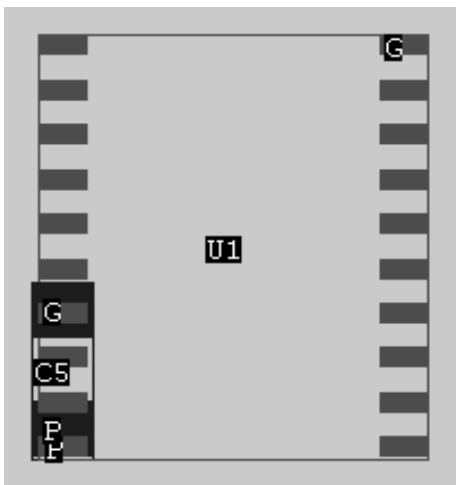


This command pivots C5 in 90 degree increments.

C5 attaches to the pointer.

8. Move C5 to the lower left of U1 (so that the power pin of C5 is under the power pin on U1). You can use the middle mouse button to zoom in if necessary.
9. Click to place the power pin of C5 under the power pin of U1.

The following figure shows the relationship of C5 to U1.



**Note:** After you attach a component to the pointer and move the pointer, you can return the component to the previous location by pressing the right mouse button and selecting *Cancel*.

Next SPECCTRA will learn this component pattern and apply it to the components with the same image ID.

### Learning and Applying the Component Pattern

Now that you placed C1, the autorouter can learn the component pattern of the small component (C5) in relationship to the large component (U1). SPECCTRA learns the



## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

orientation, location, and side of the small component with respect to the large component. It then places other instances of the capacitor image in the same pattern to other instances of the SOIC image.

#### **Task: Learn and apply the component pattern**

##### **Procedure**

1. Click the *Select Comp* button on the tool bar.

Sel Comp appears in the mode status area.

2. Drag the pointer to enclose U1 and C5.

The two components are selected and *Selected:2* appears in the status area.

3. Choose *Autoplace – Small Comp Pattern – Learn*.

SPECCTRA stores orientation, location, and side information about the relationship between C5 and U1.

4. Unselect C5 and U1 by dragging the pointer over them.

5. Choose *Select – Images – Sel Image Mode*.

Sel Image appears in the mode status area. Using this mode, you can select all instances of an image by clicking on one instance.

6. Click on U1.

U1, U2, U3, and U4 are selected.

7. Choose *Autoplace – Small Comp Pattern – Apply to Selected*.

The learned pattern is applied, and the power pins of C6, C7, and C8 are placed under the power pins on U2, U3, and U4.

8. Choose *Select – UnSelect All Placement Objects*.

Next you place the power pins of C9, C10, C11, and C12 under the power pins of U5, U6, U7, and U8 by repeating the steps from the previous section.

9. Press the right mouse button and select *Move Comp Mode*.

You will move the power pin of C9 under the power pin of U8.

10. Click on C9.

You need to flip C9 to the back side.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

11. Press the right mouse button and select *Flip*.

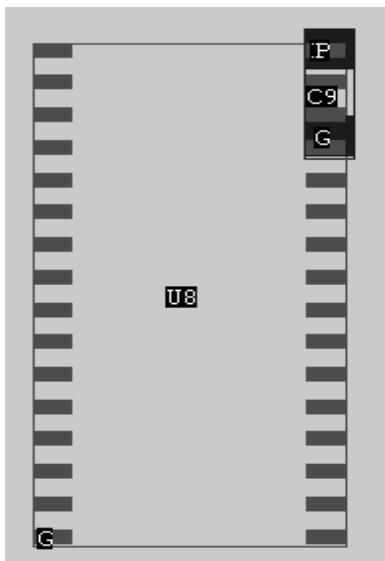
Next you pivot C9 so that the power pin is at the top.

12. Press the right mouse button and select *Pivot – 90*.

13. Move C9 to the upper right of U8 (so that the power pin of C9 is under the power pin on U8).

14. Click to place the power pin of C9 under the power pin of U8.

The following figure shows the relationship of C9 to U8.



Next you will learn and apply this component pattern to the unplaced capacitors.

15. Click the *Select Comp* button on the tool bar.

Sel Comp appears in the mode status area.

16. Select *U8* and *C9* by dragging the pointer over them and releasing the left mouse button.

The two components are selected, and *Selected:2* appears in the status area.

17. Choose *Autoplace – Small Comp Pattern – Learn*.

SPECCTRA stores orientation, location, and side information about the relationship between C9 and U8.

18. Unselect *C9* and *U8* by dragging the pointer over them.

19. Choose *Select – Images – Sel Image Mode*.

**20.** Click on *U8*.

U5, U6, U7, and U8 are selected.

**21.** Choose *Autoplace – Small Comp Pattern – Apply to Selected*.

The learned pattern is applied, and the power pins of C10, C11, and C12 are placed under the power pins of U5, U6, and U7.

**22.** Choose *Select – UnSelect All Placement Objects*.

Congratulations! All components are placed.

## Quitting the Autorouter and Saving Placement Results

When you are satisfied with your placement results, save your work. You can save your work in a Placement file or a Session file. If you save a Placement file, you can reload the file at the start of a session or anytime during the session. The Placement file is useful if you want to perform multiple placement trials and compare the files to choose the best result. You create a placement file by choosing *File – Write – Placement*.

If you create a Session file, you load the Session file only when you start SPECCTRA. A session file contains a reference to the original design filename as well as detailed placement, floorplan, swap, netlist, and route data. You will exit SPECCTRA and create a Session file in the next procedure.

### Task: Quit the Autorouter and save a Session file

#### Procedure

**1.** Choose *File – Quit*.

The *Save And Quit* dialog box appears with *lesson2.ses* in the *Save Session File* data entry box, as shown in the following figure.

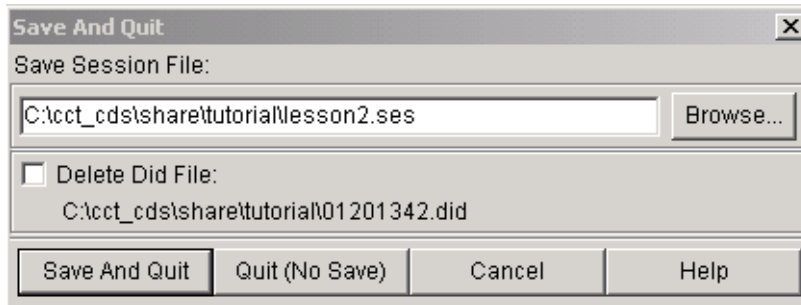
**2.** Click *Delete Did File* to remove the Did file.

## SPECCTRA Tutorial

### Lesson 2: Placing Components

---

In Lesson 1, you learned that a did file contains the command history from a the autorouter session. You do not need this file for the session, so you will delete it.



#### 3. Click *Save and Quit*.

SPECCTRA exits and saves the placement information in the Session file.

## What You Learned

In this lesson, you learned how to place components interactively and automatically.

You learned :

- the basic steps used in placing components.
- how to set placement options.
- how to set placement rules.
- how to preplace connectors and critical components.
- how to place large components.
- how to edit the placement.
- how to place small components.

In the next lesson, you will learn how to autoroute a PCB design.

---

## Lesson 3: Routing a PCB Design

---

### What You Will Learn

This lesson teaches you how to get the best results from the autorouter. In this lesson, you will learn how the autorouter works and how to:

- set routing rules.
- use a Do file.
- monitor autorouting progress.

This lesson takes about 90 minutes to complete.

### What to do Before You Begin

Before you begin this lesson, do the following:

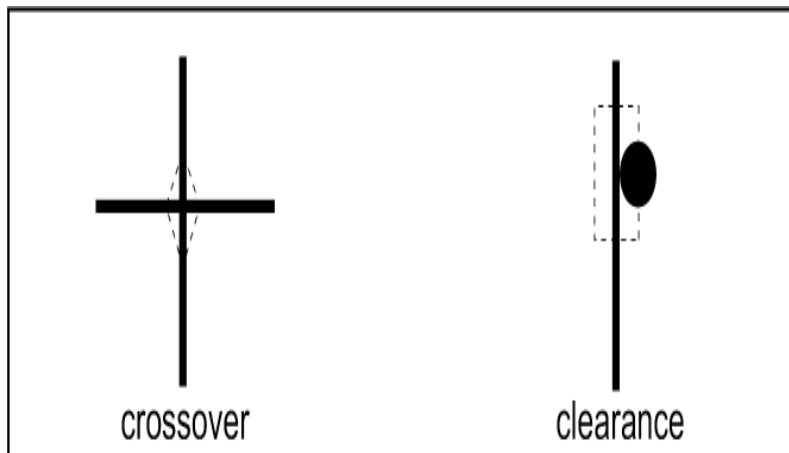
- Complete *Lesson 1: Learning Basic Concepts*.

**Note:** You do not need to complete *Lesson 2: Placing Components* but you should read through the lesson to understand the automatic and interactive placement tools function.

## Understanding How the Autorouter Works

The autorouter works differently from other autorouters. It allows crossing and clearance conflicts during certain routing phases. These conflicts are shown in the following figure.

**Figure 3-1 Crossing and Clearance Conflicts**



During the first routing pass, the autorouter allows conflicts to route every connection. After the first routing pass, the cost for creating these conflicts increases with each pass.

During the first five routing passes, all connections are ripped up and rerouted. After the first five routing passes, the strategy changes, and only wires involved in conflicts are ripped up and rerouted. Wires that are not involved in conflicts are ignored during this phase.

This adaptive autorouting uses different routing functions and requires many routing passes. An important part of autorouting strategy involves choosing which routing commands to use and when to use them. You will learn more about these commands in this lesson as well as the following lesson.

### Using the four basic autorouting commands

There are four basic routing commands that you can use individually. These commands are described in the following table.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

---

Command	Description
bus	Routes pins that share the same X or Y coordinate. This command is useful for rerouting memory arrays, backplanes, and other pins that share a common X or Y coordinate.
fanout	Escapes SMD pads and through-pins to a via.
route	Routes with conflicts, and after the first five passes, reroutes only connections involved in conflicts. Escapes SMD pads to vias as needed.
clean	Rips up and reroutes all connections. Adding new conflicts is prohibited.

---

**Task: Use the basic autorouting commands.**

#### Procedure

1. Start SPECCTRA and load `lesson3.dsn` from the `tutorial` directory. See [Where to find the Accompanying Lesson Files](#) on page 12 for the location of this directory.

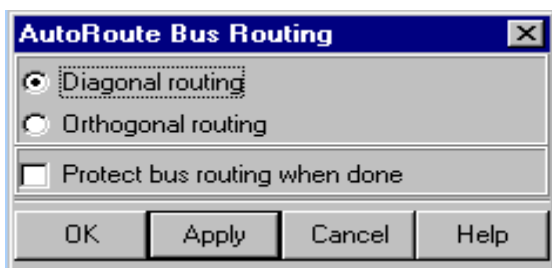
You will route pins that share the same X or Y coordinate.

2. Choose *Autoroute – Pre Route – Bus Routing*.

The AutoRoute Bus Routing dialog box opens.

3. Make sure the *Diagonal routing* option is selected, as shown in the following figure.

This option uses diagonal wire segments, rather than orthogonal wire segments, to connect the pins.



4. Click *OK*.
5. Choose *Autoroute – Pre Route – Fanout*.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

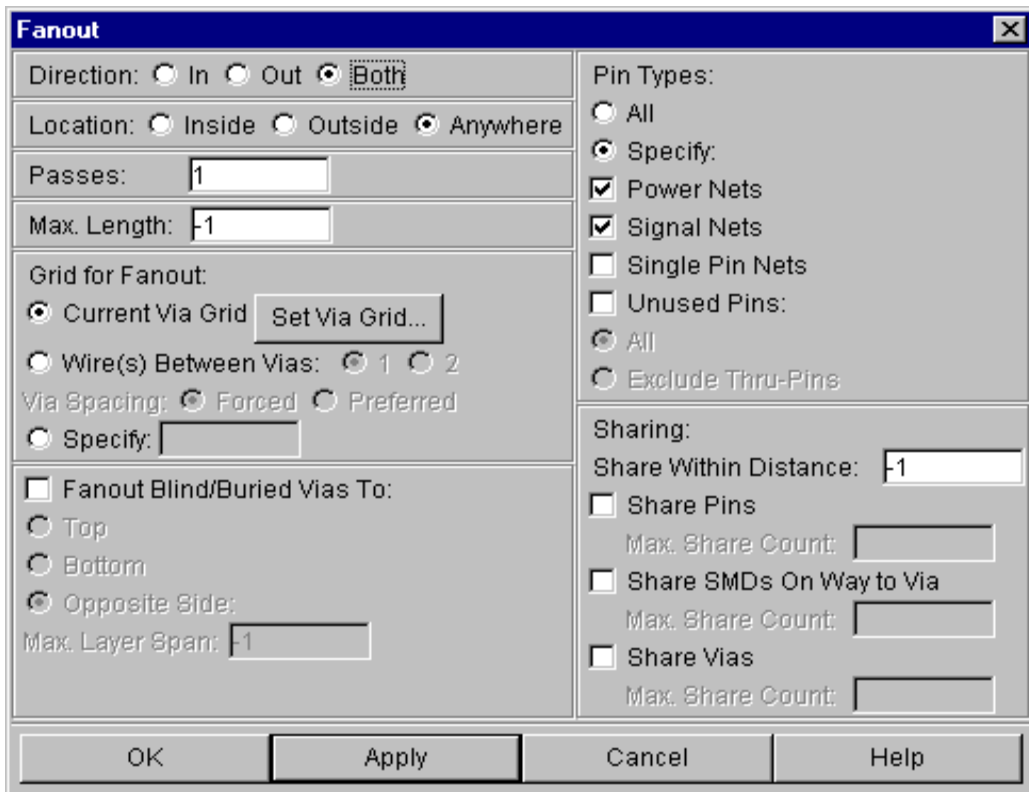
---

The Fanout dialog box opens.

6. Make sure *Specify*, *Power Nets*, and *Signal Nets* are selected under *Pin Types*.

These choices escape all power pins and signal pins that interconnect with one or more pins.

7. Make sure all other options under *Pin Types* are unselected, as shown in the following figure.



8. Click *OK* and wait for fanout to complete.

Next you will automatically route the design.

9. Choose *Autoroute – Route*.

The AutoRoute dialog box opens.

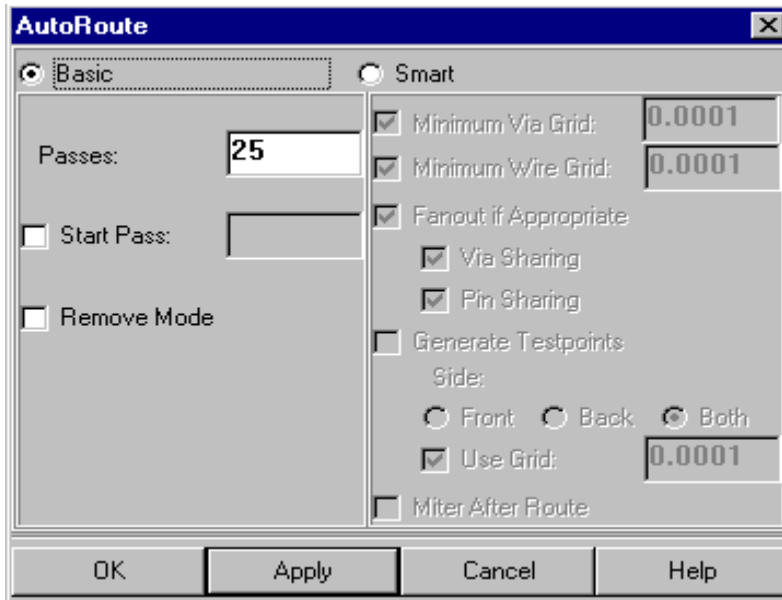


## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

10. Click the *Basic* option in the AutoRoute dialog box, as shown in the following figure.



This option runs basic route passes. Note the default number of passes (25) used for basic routing.

11. Click *OK*.

The `route` command applies 25 routing passes. The routing takes several minutes to complete.

Notice the yellow rectangles and diamond shapes that mark the temporary conflicts during autorouting. The rectangles indicate clearance violations. The diamond shapes indicate crossovers.

While the tool is routing the design, the *Pause* button appears in the control area.

**Note:** Wait for *Idle* to replace the *Pause* button in the control area.

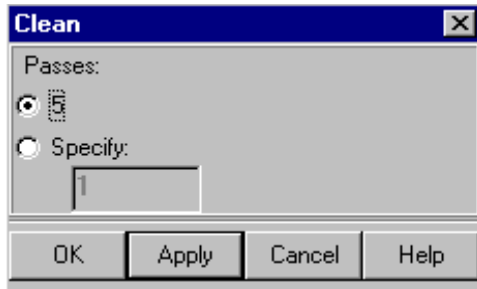
12. Choose *Autoroute – Clean*.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

The Clean dialog box opens, as shown in the following figure.



The `clean` command rips up and reroute all connections. This improves manufacturability by removing unnecessary vias and bends, reducing routed length, and changing SMD entries and exits.

13. Click *OK*.

Wait for *Idle* to replace the *Pause* button in the control area.

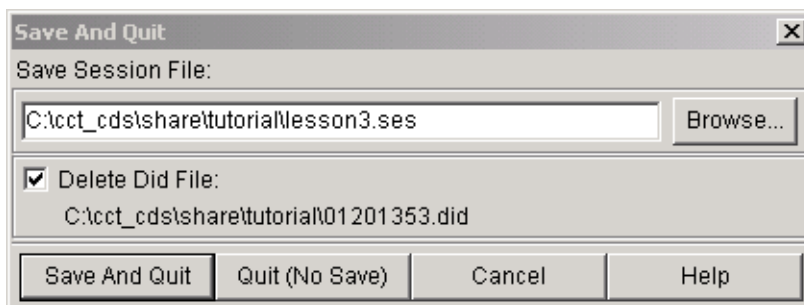
14. Click the *Status Report* button on the tool bar.
15. View the report and determine the actual number of route passes used to route the design.

If routing completes before all route passes are used, the remaining route passes are skipped and the clean passes proceed.

16. Close the Status report.
17. Choose *File – Quit*.

The Save and Quit dialog box opens, as shown in the following figure.

18. Click *Delete Did File* to remove the Did file.



19. Click *Quit (No Save)* to exit the autorouter without saving a Session file and a Did file.

**Note:** Another way to exit the autorouter without saving a Session file is to enter `quit`

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

in the command entry area. The `Did` file is not deleted when you use this method.

Each basic autorouting command serves a special purpose and is used at a specific phase of the autorouting session. The three phases of autorouting are:

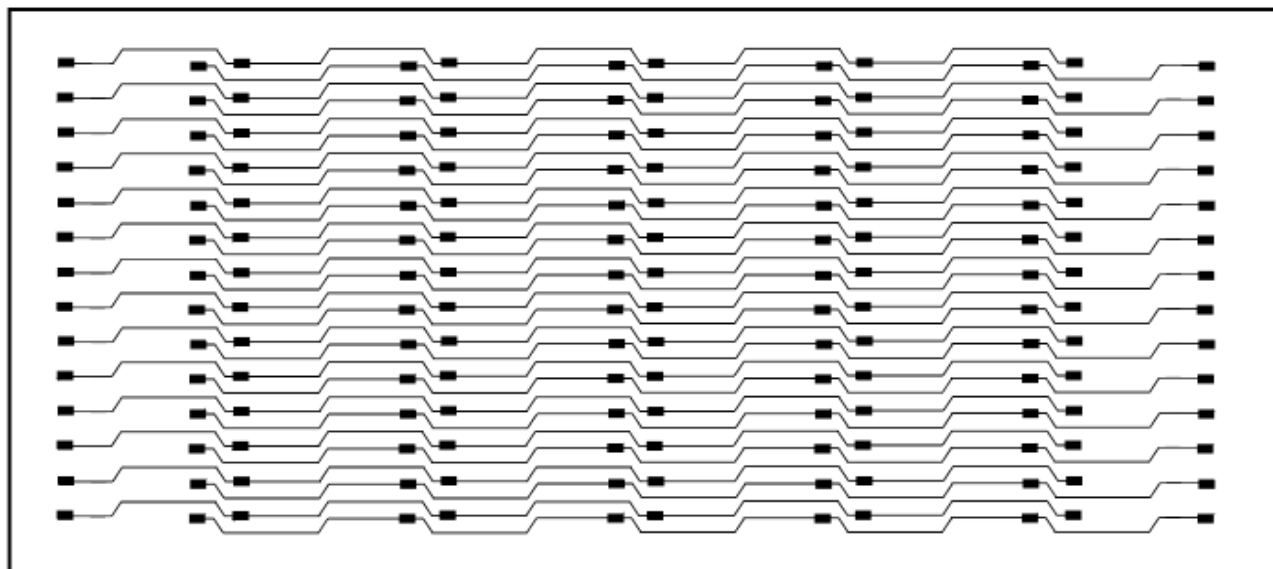
- Prerouting
- General purpose routing
- Post-routing

#### Understanding Prerouting Commands

Prerouting commands, as the name suggests, are executed before route and clean commands. The `bus` and `fanout` commands are prerouting commands.

The `bus` command routes pins that share the same X or Y coordinate and are attached to the same net. Because of its simplicity, connections are routed quickly, and use a minimum of space. The `bus` command is used at the beginning of an autorouting session, before you use other basic autorouting commands. The following figure shows the results of the `bus` command with the diagonal option turned on.

**Figure 3-2 Bus Diagonal Routing Results**



The `fanout` command routes short escape wires and vias from SMD pads and through-pins marked as exposed. This command adds a via that the autorouter can connect to instead of connecting directly to the pad or pin. For power pins that connect to decoupling capacitors,

you can control the fanout order by using the `power_fanout` rule. You can choose whether you want the escape wire to connect to the escape via first or the decoupling capacitor first.

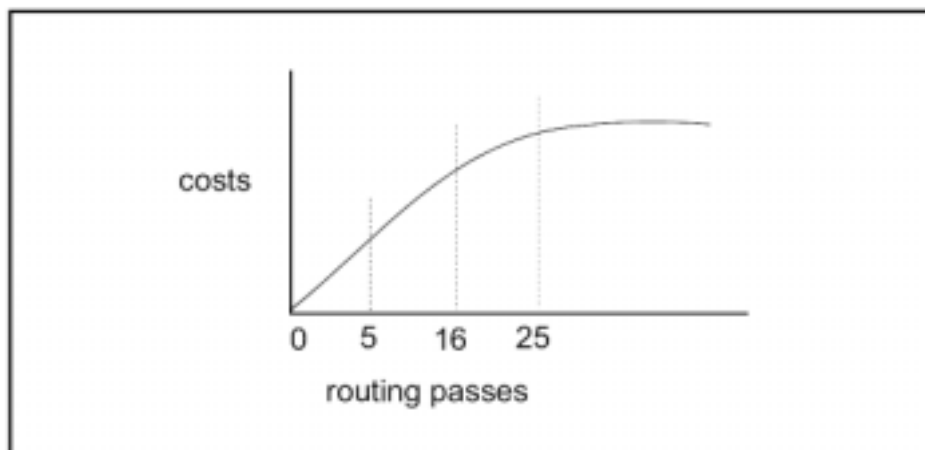
The `fanout` command should follow the `bus` command but precede the `route` command. You can use `fanout` with up to five passes. Use multiple fanout passes with via and pin sharing enabled to optimize SMD escapes.

### Understanding the General Purpose Routing Commands

The general purpose autorouting commands are `route` and `clean`. These commands can route and reroute all connections in the design, except those routed by the `bus` command.

The `route` command uses costs that limit or control resources. These costs control the number of vias per connection, the number of conflicts per connection, and the distance that wires can route on a layer in the wrong direction. The autorouter dynamically changes these costs over a series of route passes based on the routing history and the current wiring conditions. The following figure shows approximately when routing costs change during a series of 25 route passes.

**Figure 3-3 Costs Change During a Series of Route Passes**



Because costs change during a series of route passes, you must specify a sufficient number of passes. After pass five, only connections involved in conflicts are rerouted. If the wiring completes before all passes are used, the extra passes are skipped.

The `clean` command works differently from the `route` command. This command rips up and reroutes all connections (except those routed by the `bus` and `fanout` commands) instead of only the connections involved in conflicts.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

The `clean` command is usually applied over two to four passes and should always follow a series of route passes. Because `clean` rips up and reroutes all connections, tries to reroute them using a different path, and does not add new conflicts, it usually completes more connections, reduces routing length and conflicts.

A typical sequence of basic autorouting commands is:

```
bus diagonal
fanout 5
route 25
clean 2
route 50 16
clean 4
```

During clean passes, the built-in costs for vias and wrong-way routing are highest, thus improving the quality of routing after each clean pass.

**Note:** In the typical sequence of basic autorouting commands, four clean passes are included at the end. Always use at least four clean passes after all routing completes.

### Understanding the Autorouter's `smart_route` Command

The `smart_route` command sets wire and via grids, performs bus and fanout operations, and runs a series of route and clean passes until routing completes.

A single `smart_route` command can replace the four basic autorouting commands, because `smart_route` combines `bus`, `fanout`, `route`, and `clean` commands. The four basic autorouting commands are used instead of `smart_route` if rules must change between bus, fanout, route, or clean passes or if other commands are needed between these different routing functions.

While `smart_route` is running, the autorouter monitors and analyzes routing progress. If it detects design problems, warning or error messages appear. If the autorouter reaches a point where further improvement is unlikely, the autorouter switches to a different method and completes as many connections as possible with zero conflicts.

In the following procedure, you create an artificial design problem by unselecting all signal layers. `Smart_route` detects the problem and generates an error message. If there are not enough signal layers in a design, the same error message is displayed.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

#### Task: Use smart\_route to determine design problems

##### Procedure

1. Start SPECCTRA and load `lesson3.dsn`.

2. Click the *Layer* button on the tool bar.

The Layer panel opens.

3. Click the SIGNAL\_1 layer selection button located near the top of the layers panel.

On a Windows platform, the selection button looks like the following.



On a UNIX platform, the selection button looks like the following.



4. Click the following button:



This symbol means the layer is unselected.

5. Repeat the previous step to unselect SIGNAL\_2, SIGNAL\_3, and SIGNAL\_4 layers.

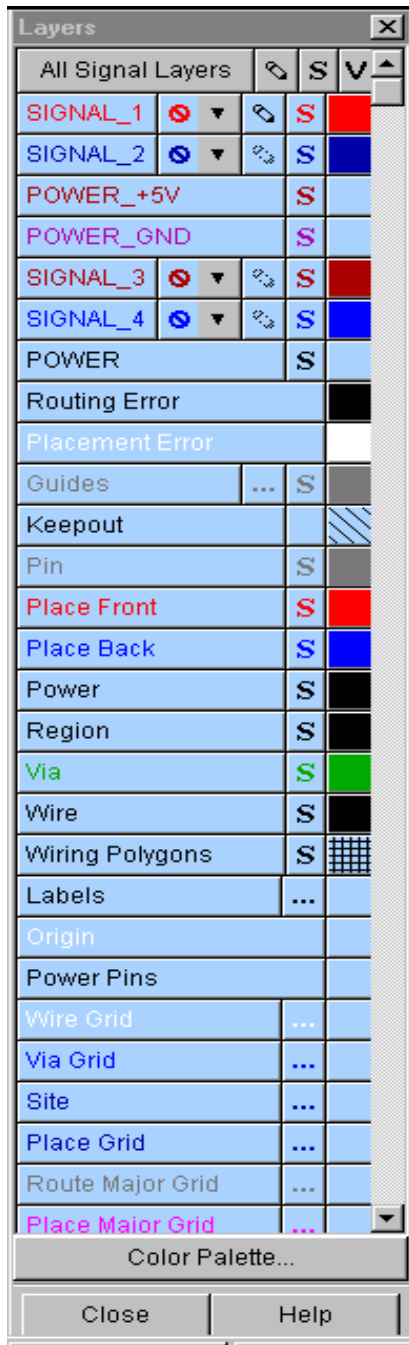
By unselecting all signal layers, you create a design problem.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

The layer panel is shown in the following figure.



#### 6. Choose *Autoroute – Route*.

The AutoRoute dialog box opens.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

7. Make sure the *Smart* option is selected at the top of the dialog box.
8. Click *OK*.

**Note:** If you encounter a Licensing Error dialog box, click the *Ignore Feature for This Session* button to continue on. See [License Considerations](#) on page 13 for further details.

After several routing attempts, an error message displays. Read the error message and click *OK*.

The following figure shows an example of the error message.



Next you will select all four signal layers to make them available for routing. You will enter **smart\_route** from the keyboard instead of using the GUI. Smart\_route analyzes early routing results and determines that you might have more routing layers than necessary to route the design.

#### Use smart\_route to optimize your design

1. Click the SIGNAL\_1 layer selection button and set the routing direction to horizontal.
2. Click the SIGNAL\_2 layer selection button and set the routing direction to vertical.
3. Repeat steps 1 and 2 to set SIGNAL\_3 to horizontal and SIGNAL\_4 to vertical.

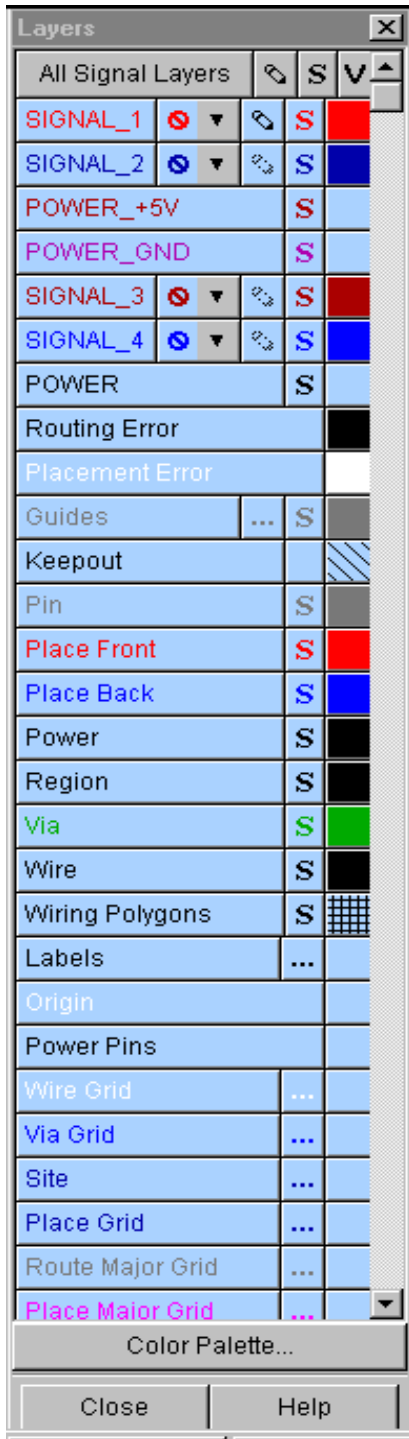


## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

The Layer panel is shown in the following figure.



4. Close the Layer panel.

## SPECCTRA Tutorial

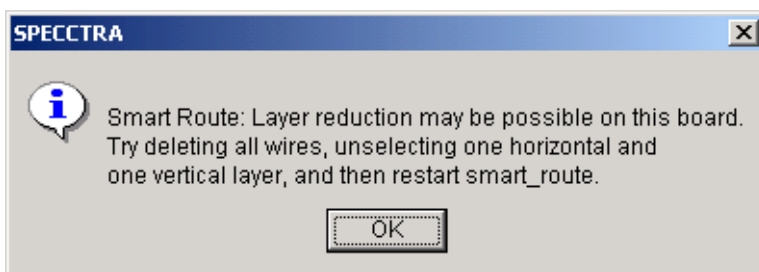
### Lesson 3: Routing a PCB Design

---

5. Enter `smart_route` in the command entry box below the routing status line and press the return key.

**Note:** If you encounter a Licensing Error dialog box, click the *Ignore Feature for This Session* button to continue on. See [License Considerations](#) on page 13 for further details.

After several routing passes, a message popup tells you that you might be able to reduce the number of layers. An example of the message popup is shown in the following figure.



6. Click *OK* to close the message popup.
7. Wait for *Idle* to replace the *Pause* button in the control area.
8. Enter `quit` in the command entry.

This command exits the autorouter without prompting you for a confirmation.

### Understanding the Post-routing Commands

Post-routing commands refine the wiring and add test points after routing is complete. The autorouter's post-routing commands are `spread`, `miter`, `testpoint`, and `recorner`. The following table provides a brief description of the post-routing commands.

**Table 3-1 Post Routing Commands**

Command	Description
<code>spread</code>	Adds extra clearance between wires
<code>testpoint</code>	Adds test points
<code>miter</code>	Changes 90 degree corners to 45 degree diagonals using a range of setback values
<code>recorner</code>	Changes 90 degree corners to 45 degree diagonals

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

All post-routing commands, except `recorner`, are DFM options. An explanation of the DFM post-routing options is beyond the scope of this tutorial. See the *SPECCTRA Command Reference* for more information about using the `spread`, `miter`, and `testpoint` commands.

When you use the `recorner` command, you must include a corner type and a setback value. For example, the following command changes wrong way routes (slant) to 45 degree diagonals, if the slants are one inch or more in length.

```
recorner slant 1
```

The easiest way to change all 90 degree corners to 45 degree diagonals is to use `recorner diagonal` with three setback values. An example of how you use this command is:

```
unit inch
recorner diagonal 1 1 1
```

In this example, the three setback values change 90 degree corners that meet the one inch criteria to 45 degrees.

The `recorner` command is iterative. If at least one corner in the design changes to a diagonal, all remaining corners are attempted again. This iterative process continues until no further changes occur.

You achieve optimum results when you use a series of `recorner` commands with progressively smaller setback values. In the following procedure, you enter two `recorner` commands and use setback values for the second that are half the value of the first.

#### **Task: Change 90 degree corners to 45 degree diagonal corners**

##### **Procedure**

1. Start SPECCTRA and load `lesson3.dsn`.
2. Choose *File – Read – Routes*.

The Read Routes dialog box opens.

3. Click the *Browse* button.

The Open dialog box opens for Windows platforms.

The Select File dialog box opens for UNIX platforms.

4. Change to the SPECCTRA tutorial directory for Windows platforms and open `recorner.rte`.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

Select `recorner.rte` and click *OK* for UNIX platforms.

The filename is added to the Read Routes dialog box.

**5.** Click *OK*.

The Routes file loads.

**Note:** If you have the DFM option, you can use a single `miter` command to replace steps 6 through 8.

**6.** Enter `recorner diagonal 1 1 1` in the command entry box.

**Note:** If you encounter a Licensing Error dialog box, click the *Ignore Feature for This Session* button to continue on. See [License Considerations](#) on page 13 for further details.

All 90 degree corners that meet the one inch setback criteria change to 45 degrees.

**7.** Wait for *Idle* to replace the *Pause* button in the control area.

**8.** Enter `recorner diagonal 0.5 0.5 0.5`.

**9.** Wait for *Idle* to replace the *Pause* button in the control area.

**10.** Examine the wiring changes made by the two previous `recorner` commands.

**11.** Enter `quit` in the command entry box to exit SPECCTRA.

With more complex designs, the following command series is suggested.

```
unit inch
recorner diagonal 1 1 1
recorner diagonal 0.5 0.5 0.5
recorner diagonal 0.25 0.25 0.25
recorner diagonal 0.125 0.125 0.125
```

You can use other units and other values, but for best results always use a series of `recorner` commands. Start with a large setback value, and halve the previous value for the next `recorner` command in the series.

## Setting Routing Rules

SPECCTRA supports a large number of routing rules. Routing rules can be set at many different levels. The routing rule levels are listed from highest to lowest precedence in the following table.

**SPECCTRA Tutorial**  
Lesson 3: Routing a PCB Design

---

**Table 3-2 Routing Rule Levels**

<b>Rule level</b>	<b>Description</b>
region class to class	Sets rules that apply between net classes when the nets are routed in a region.
region net	Sets rules that apply to nets in a region.
region class	Sets rules that apply to a class of nets in a region.
region	Sets rules for an area of the PCB.
padstack	Sets clearance rules for via padstacks.
class to class layer	Sets rules that apply between net classes when the nets are routed on particular layers..
class to class	Sets rules that apply between wires, pins, and vias of one or more net classes.
fromto layer	Sets rules that apply to connections if they are routed on a particular layer.
fromto	Sets rules for pin-to-pin connections.
group layer	Sets rules that apply to a group of fromtos if the fromtos are routed on a particular layer.
group	Sets rules for a group of fromtos.
net layer	Sets rules that apply to nets if they are routed on a particular layer.
net	Sets rules that apply to nets.
group set layer	Sets rules that apply to a set of groups when they are routed on a particular layer.
group set	Sets rules that apply to a set of groups.
class layer	Sets rules that apply to a class of nets if the nets are routed on a particular layer.
class	Sets rules that apply to a class of nets.
layer	Sets rules that apply to wires routed on a particular layer.
pcb	Sets global rules for all nets in the design. This is the lowest precedence rule.

## Understanding SPECCTRA's rule hierarchy

The rule levels described in the previous table form a hierarchy. Higher level rules always override lower level rules that are set for the same physical objects. For example, a wire-to-wire clearance rule for a net overrides a wire-to-wire clearance rule for a class that includes that net.

Rules at the PCB level have the lowest precedence in the hierarchy. Rules at the region class to class level have the highest precedence in the hierarchy. The order of precedence is fixed and cannot be changed.

## Setting Width and Clearance Rules

You can set width and clearance rules at different levels of the hierarchy and for different objects. The object types supported by the autorouter are described in the following table.

**Table 3-3 Object Types**

Object type	What it means
area	Keepout regions and PCB boundary
pad	SMD pad
pin	Through-pin
via	Layer feed-through
wire	Copper path

Two object types separated by the underscore character indicate an object-to-object rule. For example, wire\_smd is used to set clearance rules between wires and SMD pads.

## Task: Set width and clearance rules at the PCB level

### Procedure

1. Start SPECCTRA and load `lesson3.dsn`.
2. Choose *Rules – PCB – Clearance*.

The PCB Clearance Rules dialog box opens, as shown in the following figure.

3. Enter **0.007** in the *Wire Width* data entry box.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

4. Enter **0.007** for *wire\_wire clearance*.

This data entry box is located in the row labeled Wire under the column labeled Wire.

**PCB Clearance Rules**

☐ Same Net Checking

Wire Width:

Taper Wire: ☐ Up to Pin ☒ Down to Pin ☐ Up/Down to Pin ☐ Off

	All	Area	Pin	SMD	Via	Wire	Testpoint
Area		<input type="text" value="0.025"/>					
Pin		<input type="text" value="0.025"/>	<input type="text" value="0.012"/>				
SMD		<input type="text" value="0.025"/>	<input type="text" value="0.025"/>	<input type="text" value="0.025"/>			
Via		<input type="text" value="0.025"/>	<input type="text" value="0.025"/>	<input type="text" value="0.025"/>	<input type="text" value="0.012"/>		
Wire		<input type="text" value="0.025"/>	<input type="text" value="0.008"/>	<input type="text" value="0.008"/>	<input type="text" value="0.008"/>	<input type="text" value="0.007"/>	
Testpoint		<input type="text" value="0.01"/>	<input type="text" value="0.01"/>	<input type="text" value="0.01"/>	<input type="text" value="0.01"/>	<input type="text" value="0.01"/>	<input type="text" value="0.01"/>

SMD-Via Same Net	<input type="text" value="1"/>	Via-Via Same Net	<input type="text" value="1"/>
SMD Escape	<input type="text" value="1"/>	Pad-to-Turn Gap	<input type="text" value="1"/>
Antipad Gap	<input type="text" value="1"/>	SMD-to-Turn Gap	<input type="text" value="1"/>
Buried-Via Gap	<input type="text" value="1"/>	Layer Depth:	<input type="text" value="1"/>

OK Apply Cancel Help

5. Click **OK**.

The width and clearance rules that you changed in the PCB Clearance Rules dialog box are applied.

6. View the session transcript in the Output window and find the following commands:

```
rule pcb (clearance 0.007 (type wire_wire))
rule pcb (width 0.007)
```

7. Enter **quit** in the Command entry box to exit SPECCTRA.

In Lesson 4, you will learn to set additional routing rules.

## Autorouting With a Do File

There are three methods for issuing commands and controlling the autorouter. You can:

- use the GUI.
- enter commands from the keyboard.
- use a Do file.

Until now, you either used the GUI to run commands or you entered console commands using the keyboard. The preferred method for controlling the autorouter is with a Do file. A Do file is a text file that contains one or more autorouting commands. Each command occupies a separate line in the Do file. Commands are organized in the order you want them to run from the start to the end of the file.

The Do file also serves another purpose. It serves as a record of the rules and commands you use during an autorouting session. If you need to apply a design revision to a completed design, you can edit the original Do file and reuse it.

You start with a basic Do file that includes commands that address the following:

- Setup and file management
- Rule setting
- Prerouting
- Routing
- Post-routing

### Using a basic Do file

A basic Do file (`basic.do`) is included with the tutorial files. A listing of the commands in `basic.do` follows. The PCB width and clearance rules are not intended as typical values but are included as examples.

```
bestsave on $\best.wir
status_file $\route.sts
unit mil
rule pcb (clearance 7 (type wire_wire))
rule pcb (width 7)
smart_route
```



## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

```
write routes $\final.rte  
report status $\final.sts
```

**Note:** The `$\` notation represents the path for the file in SPECCTRA. When you want to write a report or data file to the same directory where the design file is located, use the `$\` notation. On UNIX platforms, substitute the `/` character for the `\` character.

Often, you do not need to include PCB clearance and width rules in the Do file, because they are translated from your layout system and embedded in the Design file. If you want to override rules that were set in the layout system or you want to apply other rules, add your rules to the Do file. Make sure these rules precede all routing commands.

#### Task: Use a basic do file to autoroute a PCB design

##### Procedure

1. Start SPECCTRA and enter the path for `lesson3.dsn` in the Design/Session File data entry box.
2. Enter `$\basic.do` in the Do File data entry box located on the dialog box. On a UNIX platform, substitute `/` for `\`.
3. Click *Start SPECCTRA*.

Once you have selected a product, SPECCTRA starts, loads the Design file, and runs the Do file.

**Note:** If you encounter a Licensing Error dialog box, click the *Ignore Feature for This Session* button to continue on. See [License Considerations](#) on page 13 for further details.

A Pause button appears in the control area.

**Note:** After several routing passes, one or more message popups may appear. Click *OK* to dismiss these message popups.

4. Wait for *Idle* to replace the *Pause* button in the control area.
5. View the session transcript in the Output window, and verify that all commands were run.

**Note:** If a Do file contains a syntax error, the Do file stops working and the error is reported in the Output window.

6. Click the *Status Report* button on the tool bar.
7. Browse the status report and check the routing results.

8. Close the report.
9. Choose *Rules – PCB – Clearance*.  
The PCB Clearance Rules dialog box opens.
10. Notice the width and wire-to-wire clearance rules are updated and reflect the 7 mil rules that were set by `basic.do`.
11. Click *Cancel* to close the dialog box.
12. Enter **quit** in the Command entry box to exit SPECCTRA.

### Creating a Do file

In the previous steps, you used the basic do file that is supplied as part of this tutorial. Next, you learn an easy method for creating your own Do files.

After you become more familiar with SPECCTRA, you can construct a Do file from your memory of commands and syntax. While you are learning, you can use SPECCTRA's built-in Rules Did File editor to capture and edit *syntactically correct* commands. The editor records rule setting commands and other commands that you enter through the GUI. You can record these commands to create a syntactically correct Do file. You can control which types of commands are recorded and edit the recorded commands.

### Task: Create a Do file with the Rules Did File Editor

#### Procedure

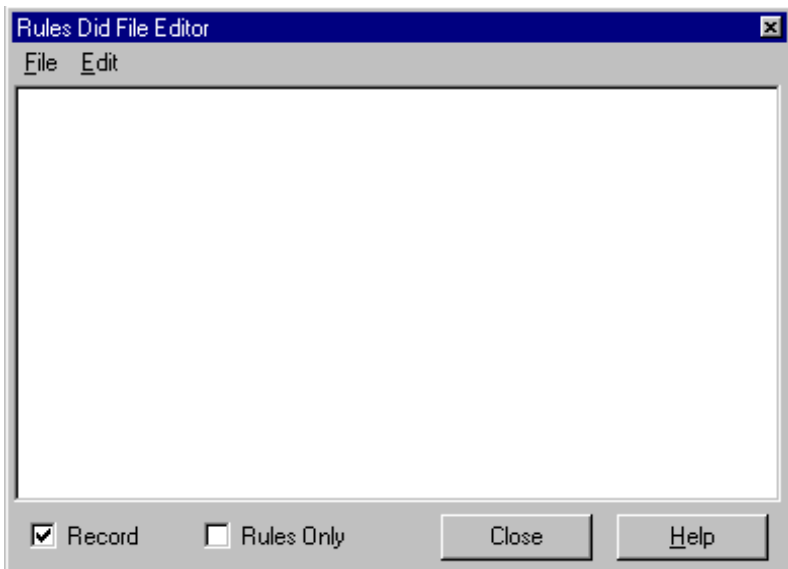
1. Start SPECCTRA and load `lesson3.dsn` from the `tutorial` directory.
2. Choose *Edit – Rules Did File*.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

The Rules Did File Editor opens, as shown in the following figure.



3. Remove the check from the *Rules Only* box.

**Note:** By default, the SPECCTRA records only the rule setting commands when *Rules Only* is checked. Remove the check from *Rules Only* to record other commands. When you close the dialog box, the settings remain in effect. For this procedure, you will leave the dialog box open.

Next you use the `bestsave` command to save routed wires at the end of a routing pass if routing improved since the previous save.

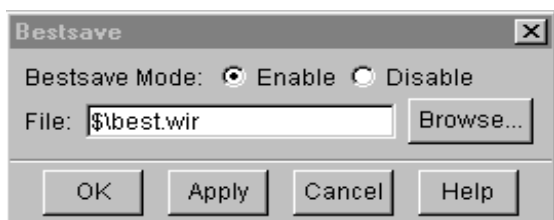
4. Choose *File – Bestsave*.

The Bestsave dialog box opens, as shown in the following figure.

5. Enter **\$\best.wir** in the data entry box.

On a UNIX platform, substitute / for \.

6. Click the *Enable* option on the Bestsave dialog box.



## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

7. Click *OK*.

Notice that the command is recorded in the Rules Did File Editor.

Next you will change the measurement unit to mils.

8. Click the *Unit* button (near the bottom right corner of the window) and choose *mil*.
9. Choose *Rules – PCB – Clearance*.

The PCB Clearance Rules dialog box opens, as shown in the following figure.

10. Enter **7** in the *Wire Width* data entry box.
11. Enter **7** in the *wire\_wire clearance* data entry box.

The *wire\_wire clearance* data entry box is in the row labeled Wire, under the column labeled Wire.

**PCB Clearance Rules**

☐ Same Net Checking

Wire Width:

Taper Wire: ☐ Up to Pin ☒ Down to Pin ☐ Up/Down to Pin ☐ Off

Max Taper Length:

	All	Area	Pin	SMD	Via	Wire	Testpoint
Area		<input type="text" value="25"/>					
Pin		<input type="text" value="25"/>	<input type="text" value="12"/>				
SMD		<input type="text" value="25"/>	<input type="text" value="25"/>	<input type="text" value="25"/>			
Via		<input type="text" value="25"/>	<input type="text" value="25"/>	<input type="text" value="25"/>	<input type="text" value="12"/>		
Wire		<input type="text" value="25"/>	<input type="text" value="8"/>	<input type="text" value="8"/>	<input type="text" value="8"/>	<input type="text" value="7"/>	
Testpoint		<input type="text" value="10"/>	<input type="text" value="10"/>	<input type="text" value="10"/>	<input type="text" value="10"/>	<input type="text" value="10"/>	<input type="text" value="10"/>

SMD-Via Same Net  Via-Via Same Net

SMD Escape  Pad-to-Turn Gap

Antipad Gap  SMD-to-Turn Gap

Buried-Via Gap  Layer Depth:

12. Click *OK*.
13. Choose *Autoroute – Route*.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

The AutoRoute dialog box opens.

14. Make sure the *Smart* option is selected.

15. Click *OK*.

**Note:** If you encounter a Licensing Error dialog box, click the *Ignore Feature for This Session* button to continue on. See [License Considerations](#) on page 13 for further details.

A Pause button appears in the control area.

**Note:** After several routing passes, one or more message popups may appear. Click *OK* to dismiss these message popups.

16. Click the *Pause* button and then click *Stop*.

The autorouter pauses then halts.

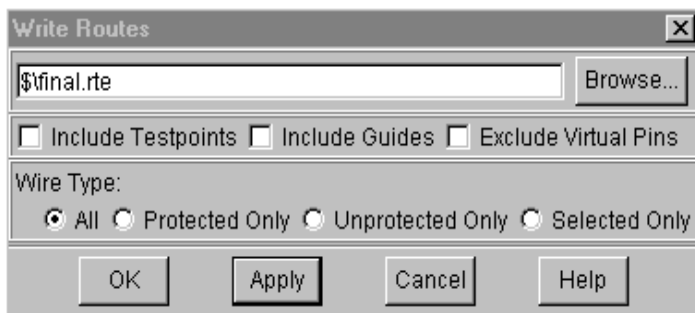
**Note:** You use these buttons to pause and then stop the autorouter because you are creating a Do file (rather than autorouting the design to completion).

17. Choose *File – Write – Routes*.

The Write Routes dialog box opens, as shown in the following figure.

18. Enter **\$\final.rte**

On a UNIX platform, substitute / for \.



19. Click *OK*.

You save the Routes file even though it is incomplete to record the write route command in the Rules Did File editor.

20. Review the recorded commands in the Rules Did File editor.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

#### Edit the Do file with the Rules Did File Editor

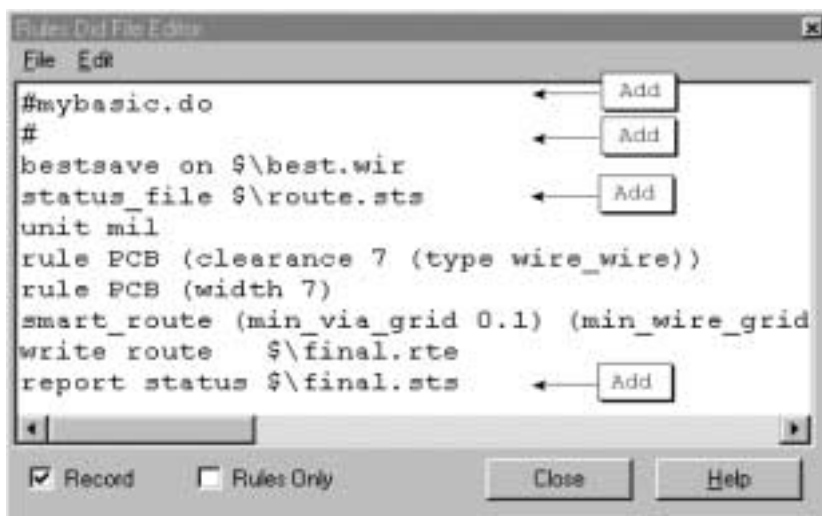
In the previous procedure, you used the Rules Did File editor to record commands that you entered through the GUI. You can record all commands this way. In the following procedure, you will enhance this Do file by entering additional commands and comments into the editor window.

**Note:** A pound sign (#) precedes a comment. SPECCTRA ignores comments in do files.

#### Task: Edit the Do file by adding commands and comments

##### Procedure

1. Add the following commands and comments to the Rules Did File editor. On UNIX platforms, substitute the / character for the \ character.



2. Save the file in the tutorial directory as `mybasic.do`.
3. Close the Rules Did File editor.
4. Enter **quit** in the Command entry box to exit SPECCTRA.

## Monitoring Autorouting Progress

In Lesson 1 you learned about the status report and about the kinds of information it contains. Now you will learn how to use the status report to monitor autorouting progress and determine when and how to adjust your strategy.

The `smart_route` command uses the same information that is recorded in the status report to analyze autorouting progress. Progress is gauged by monitoring *failures*, *unroutes*, and *conflict reduction* from pass to pass. These terms are explained in the following table.

**Table 3-4 Routing Terminology**

Term	What it means
conflict	Conflicts consist of two types. Crossing conflicts occur when two wires cross. Clearance conflicts occur when a wire or via violates a clearance rule.
failure	A failure occurs when the autorouter attempts to route or reroute a connection, fails, and restores the connection to its prior state.
reduction %	Reduction % indicates the percentage of conflict reduction compared to the previous pass.
unroute	An unroute is a connection that is not routed.

### Using the status report

The Routing History table is updated in the status report for each routing pass. This table contains the information you need to analyze routing progress.

When you use `smart_route`, it analyzes the status information for you and alerts you when potential problems are detected or when a design reduction might be possible. If you use basic autorouting commands instead of `smart_route`, *you* must monitor and analyze the status report to determine whether adjustments are needed. To gauge whether the autorouter is making good progress and determine what must be done if progress is poor, you need to understand the indicators of both good and poor routing progress.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

#### Analyzing Conflicts

The number of conflicts after the first route pass is a good indication of whether a design can be routed completely. When the sum of crossing and clearance conflicts is less than five times the total number of connections, routing is likely to complete. The following figure shows a status file with a good ratio between total conflicts and total connections after the first route pass.

Figure 3-4 Status File with a Good Conflict/Connection Ratio

Routing Status Report

Search [ ] [ ] [ ] Case Sensitive: [ ] Save \*.txt Browse...

#SPECCTRA Version V8.0 made 98/03/27 at 16:52:29  
#Host  
#ROUTING STATUS <<< C:\SPECCTRA\Tutorial\lesson4.dsn >>>  
Start Time: Thu Apr 02 09:50:49 1998  
Report Time: Thu Apr 02 10:08:05 1998

Nets = 72 Connections = 243  
Current Wire = 2 Reroute wires = 2  
Completion = 100.00% Unconnections = 0

ROUTING HISTORY

Pass	Conflicts												
Name	No.	Cross	Clear	Fail	Unrte	Vias	XTalk	Len.	%	Pass	Total		
Bus	0	0	0	0	149	0	0	0	0	0:00			
Route	1	265	72	4	4	92	0	0	0	0:00			
Route	2	219	65	2	0	110	0	0	15	0:01			
Route	3	159	39	5	0	150	0	0	30	0:01			
Route	4	116	11	0	0	175	0	0	35	0:01:29	0:04:32		
Route	5	73	9	3	0	192	0	0	35	0:01:18	0:05:50		
Route	6	37	8	1	0	214	0	0	45	0:01:32	0:07:22		
Route	7	31	9	3	0	222	0	0	11	0:00:55	0:08:17		
Route	8	15	8	2	0	231	0	0	42	0:00:53	0:09:10		
Route	9	10	1	0	0	239	0	0	52	0:00:31	0:09:41		
Route	10	8	0	0	0	249	0	0	27	0:00:31	0:10:12		
Route	11	4	3	0	0	250	0	0	12	0:00:31	0:10:43		
Route	12	10	8	0	0	252	0	0	0	0:00:35	0:11:18		
Route	13	5	3	0	0	257	0	0	55	0:00:28	0:11:46		
Route	14	9	0	0	0	250	0	0	0	0:00:39	0:12:25		
Route	15	5	0	0	0	250	0	0	44	0:00:26	0:12:51		
Route	16	1	0	0	0	248	0	0	80	0:00:28	0:13:19		
Route	17	0	0	0	0	249	0	0	100	0:00:05	0:13:24		

Conflicts between polygon wires and fixed objects: 0

Indicators show that the design is routable.

If the total number of conflicts after the first pass is more than five times the total number of connections, this usually indicates the design will not route because of one or more of the reasons listed in the following table.



**SPECCTRA Tutorial**  
Lesson 3: Routing a PCB Design

---

**Table 3-5 Unroutable Design Problems**

<b>Cause</b>	<b>Meaning</b>	<b>Solution</b>
Insufficient routing layers	Layers are unselected or too few signal layers are defined in the layout system.	Use the Layers panel to determine whether signal layers are unselected.
Layer routing directions are incorrect	An insufficient number of vertical or horizontal routing layers are defined. The autorouter eventually resolves this, but it takes longer to route the design.	Use the Layers panel to determine whether signal layer directions are set correctly.
Power and ground nets are being routed	Power and ground nets are not defined as copper planes, and the autorouter is routing power and ground connections.	Use the Layers panel to determine whether power layers are defined correctly. If power layers are defined correctly, you cannot select or unselect the layers and cannot set the routing direction in the Layers panel. If power layers are not defined correctly, fix the problem in the layout system and re-extract the design.
Clearance rules are incorrect	An error was made in setting clearance rule values.	Make sure that the rule values are set correctly for the current unit setting.
Prerouted wires are protected	Protected preroutes prevent the autorouter from ripping up and rerouting connections to eliminate conflicts.	Try autorouting with the preroutes unprotected.

**Task: Use the status report to determine a design problem**

**Procedure**

1. Start SPECCTRA and load `conflict.dsn`.

If a warning dialog box appears, close it.

2. Enter **route 1** in the command entry box.

**Note:** If you encounter a Licensing Error dialog box, click the *Ignore Feature for This Session* button to continue on. See [License Considerations](#) on page 13 for further details.

The *Pause* button appears in the status bar.

Wait for *Idle* to replace the *Pause* button in the control area.

3. Click the *Status Report* button on the tool bar.
4. Observe the number of crossing and clearance conflicts and the failures in the Routing History table.

The number of failures indicates a design problem, which in this case are the result of too few routing layers.

5. Enter **quit** in the command entry box to exit SPECCTRA.

## Analyzing Failures

A failure occurs when the autorouter cannot route or reroute a connection. Routing failures are the same as unroutes during the first route pass and can indicate a difficult design, a design error, or unrealistic design rules.

There are usually no failures during the first route pass, because the cost for allowing conflicts is lower during the first pass than at any other time. If a small percentage of the total connections (less than 2%) fail during the first pass, this could indicate a difficult design. If more than 2% of the total connections fail during the first pass (and the total number of conflicts is high), the cause is usually a design problem.

Failures that occur during and after the second route pass indicate the number of connections that could not be routed with a different path and were restored to their previous state. Failed connections after the first route pass are not necessarily a problem, unless they accompany excessive conflicts or unroutes.

## Analyzing Conflict Reduction

After the first route pass and through pass five, the total number of conflicts should reduce by at least 30% per pass. If conflict reduction is less than 30% per pass during this phase, there can be several causes as listed in the following table.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

**Table 3-6 Conflict Reduction Problems**

Cause	Meaning	Solution
Difficult design	The design is very dense, and a larger than usual number of routing passes is required to route to 100%.	Increase the total number of routing passes. Using several hundred routing passes with difficult designs is not unusual.
Insufficient routing layers	An insufficient number of signal layers is defined or layers are unselected.	Use the Layers panel to determine whether signal layers are unselected.
Autorouter is via starved	The via grid is too coarse, which prevents the autorouter from eliminating conflicts by adding vias.	Set the via grid to zero or to a minimum resolution.  For example, the following commands set both wire and via grids to 1 mil.  <code>unit mil</code> <code>grid smart(vial)(wire 1)</code>

After pass five, conflict reduction proceeds at a slower pace. Although the percentage of reduction is usually much less for each pass of this phase, conflicts should always follow a downward trend over a series of ten passes.

### Analyzing Unroutes

Unroutes should not exist after the fifth routing pass. If unroutes remain after the fifth route pass, the unroutes will probably remain at the conclusion of the autorouting session. The typical causes for unroutes after route pass five are explained in the following table.

**Table 3-7 Unroutes Problems**

Cause	Meaning	Solution
Keepouts block pins	Pins covered by keepouts cannot be routed.	Check the unrouted connections to determine whether they are blocked or covered by keepouts.

## SPECCTRA Tutorial

### Lesson 3: Routing a PCB Design

---

---

Cause	Meaning	Solution
Component pins overlap	If components are placed incorrectly and pins overlap, the pins cannot be routed.	Make sure the pins do not overlap.
Pins are outside the PCB boundary	Pins that are outside or too near the PCB boundary cannot be routed.	Check the unrouted connections to determine whether they are either outside or too near the PCB boundary.

---

In this section, you learned how to detect autorouting problems and what you can do about them. Use this information, to solve routing problems when you are routing your designs.

### What You Learned

In this lesson, you learned how to automatically route a PCB design. You learned how the autorouter works and how to:

- set routing rules.
- use a Do file.
- monitor autorouting progress.

In the next lesson, you will learn how to set rules and control the autorouter.

---

## Lesson 4: Setting Rules and Constraints

---

### What You Will Learn

This lesson teaches you how to set wiring rules and use commands that control the way a design is routed. This lesson differs from the previous lessons because you enter most commands from the keyboard instead of using the GUI.

In this lesson you will learn how to:

- set rules at different levels in the rules hierarchy.
- set wire and via clearance rules for different objects.
- select layers for routing.
- set layer routing direction.
- select vias for routing.
- select connections for routing.
- use routing keepins.
- control whether nets are routed with starburst or daisy-chain wiring.
- define a class of nets and a group of fromtos.
- assign rules to classes and groups.
- assign routing priorities.

This lesson takes about 60 minutes to complete.

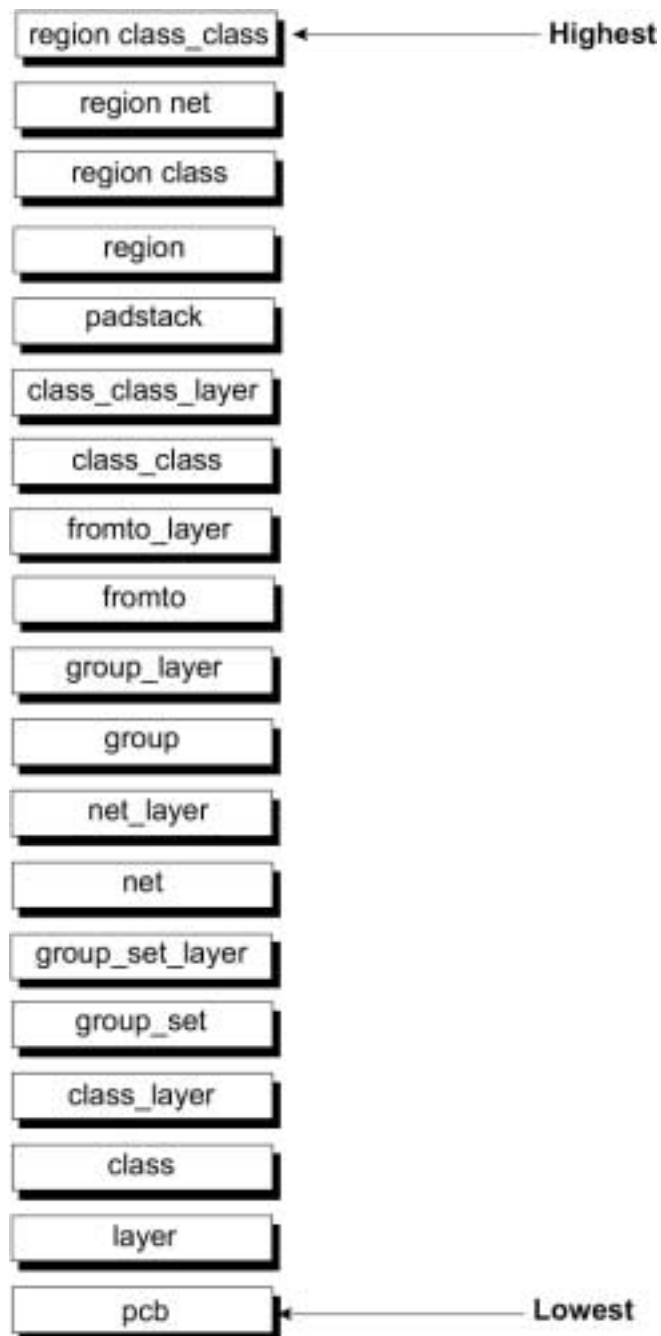
### What to do Before You Begin

Before you begin this lesson, do the following:

- Complete *Lesson 3: Routing a PCB Design*.

## Understanding the Autorouter's Hierarchical Rules

In Lesson 3, you learned that you can set rules at 19 different levels to control how a design is routed. The following figure shows the 19 rule levels.



## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

Rules set at higher levels override those set at lower levels, when they apply to the same object. For example, if you set the following rules:

```
unit mil

rule pcb (clearance 8 (type wire_wire))

rule net (CLK (clearance 10 (type wire_wire)))
```

all nets obey the pcb 8 mil wire-to-wire clearance rule, except net CLK, which obeys the 10 mil clearance rule. Because both rules control wire-to-wire clearance, the higher precedence net rule overrides the pcb rule for the net CLK.

If you set the following rules:

```
unit mil

rule pcb (clearance 8 (type wire_smd))

rule net CLK (clearance 10 (type wire_pin))
```

the CLK net obeys both the 8 mil wire-to-smd PCB clearance rule and the 10 mil wire-to-pin net clearance rule. Both rules are followed, because they are set for different objects and they do not conflict.

In the following procedure, you set wire width rules at the pcb, net, and fromto levels.

#### **Task: Set wire width rules at the pcb, net, and fromto levels**

##### **Procedure**

1. Start SPECCTRA and load `lesson4.dsn` from the tutorial directory. See [Where to find the Accompanying Lesson Files](#) on page 12 for the location of this directory.
2. Type `rule pcb (width .008)` in the command entry box.
3. Type `rule net sig1 (width .012)`.
4. Type `define (net sig1 (fromto U7-1 U8-1 (rule (width .020))))`.
5. Type `select net sig1 sig2 sig3 sig4`.
6. Type `bus diagonal`.
7. Type `unselect all nets`.
8. Examine the wiring that was routed using `bus diagonal`.

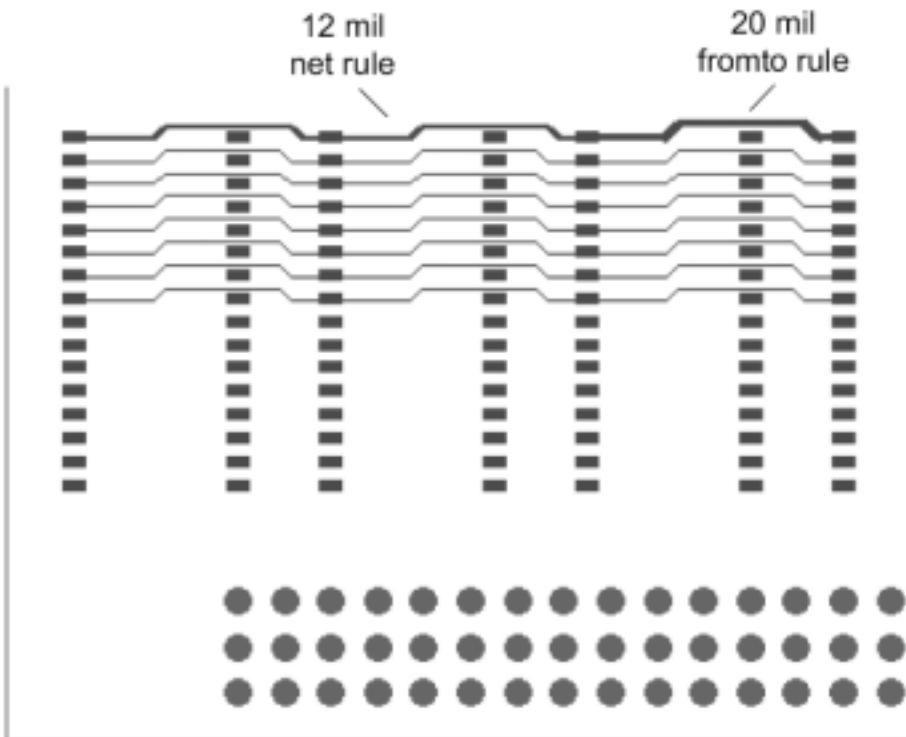
## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

Notice that the top net (sig1) obeys the 12 mil rule except for one fromto in the net that follows the 20 mil rule. The other nets obey the 8 mil pcb rule. The following figure shows an example of the routing.

**Figure 4-1 Fromto Rule Overrides Net Rule Which Overrides PCB Rule**



**Task: Use the measure icon to get information about a design**

#### Procedure

1. Click the *Measure* button on the tool bar.
2. *Measure* appears in the mode status area.
3. Click on each of the different width wire segments.
4. Read the width information in the Measure dialog box.
5. Type `quit` in the command entry box to exit SPECCTRA.



## Using Commands to Set Rules and Control the Autorouter

SPECCTRA provides numerous commands you can use to set rules and control how a design is routed. With a single command or simple combination of commands you can:

- set wire and via clearance rules for different objects.
- define wire and via grids.
- select and unselect layers for routing.
- control layer routing direction.
- control which vias are used for routing.
- control which connections are autorouted.
- control whether nets are routed with a starburst or daisy-chain topology.
- define classes of nets and assign rules.
- define groups of fromtos and assign rules.
- assign routing priorities to nets and classes.

### Setting Wire and Via Clearance Rules for Different Objects

You can set separate clearance rules for wires and vias and other objects. The following table describes how you can set the rules for wire to object and via to object.

**Table 4-1 Clearance Rule Keywords**

Object to Object keyword	What it means
area_via	Clearance rule between PCB boundary or keepouts and vias
area_wire	Clearance rule between PCB boundary or keepouts and wires
pin_via	Clearance rule between through-pins and vias
pin_wire	Clearance rule between through-pins and wires
smd_via	Clearance rule between SMD pads and vias

## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

smd_wire	Clearance rule between SMD pads and wires
via_via	Clearance rule between vias
via_wire	Clearance rule between vias and wires
wire_wire	Clearance rule between wires on different nets

---

In the following procedure, you set several pcb clearance rules, select certain nets, and route them. Usually, you would not select a few nets for routing. The nets are selected in these steps to reduce routing time. The purpose of this procedure is to show how you set wire clearance rules for different objects.

#### Task: Set wire-to-smd, wire-to-pin, and via-to-via clearance rules

##### Procedure

1. Start SPECCTRA and load `lesson4.dsn` from the `tutorial` directory.
2. Type `rule pcb (width .008)` in the command entry box.
3. Type `rule pcb (clearance .008 (type smd_wire))`.
4. Type `rule pcb (clearance .016 (type pin_wire))`.
5. Type `rule pcb (clearance .014 (type via_via))`.
6. Type `select net sig17 sig18 sig19 sig20 sig21 sig22 sig23 sig24 sig25 sig26 sig27`.

If one or more nets are selected, only those connections are routed.

7. Type `bus diagonal`.

Only the selected connections that meet the bus criteria are routed.

8. Type `route 25`.
9. Wait for *Idle* to replace the *Pause* button in the control area.
10. Choose *Select – Unselect All Routing Objects*.
11. Zoom in or pan to view the wiring added by the two previous commands.
12. Click the *Measure* button on the tool bar.
13. Measure the `wire_smd` clearance, which should be at least .008 inches.

**14.** Measure the wire\_pin clearance.

Notice the 0.016 wire-to-pin clearance rule centers wires between through-pins.

**15.** Measure the via\_via clearance.

Vias obey the 0.014 (minimum) via-to-via clearance rule.

**16.** Type `quit` in the command entry box to exit SPECCTRA.

### Selecting Layers for Routing

When a signal layer is *selected*, it is available for routing. If a signal layer is *unselected*, it is not used for routing unless it is an external layer on which SMD components are mounted or certain nets are assigned to route on that layer. You can assign a net to route on a particular layer (if you have the ADV option) by using the `circuit` command with a `use_layer` rule.

The commands that enable or disable a signal layer for routing are `select layer` and `unselect layer`. When you use these commands, include the name of the layer you want to control.

For example:

```
select layer TOP
unselect layer INT1
```

### Setting Layer Routing Directions

If layer routing directions are not set in the layout system, SPECCTRA sets each signal layer's routing direction to horizontal or vertical. The top signal layer is set to horizontal, and the next signal layer below the top layer uses a vertical direction. The autorouter alternates between horizontal and vertical directions for each additional signal layer.

You can override the default layer routing directions by using the Layers panel or by using the `direction` command. If the interconnection between components is biased more in a horizontal or vertical direction, change the routing direction of some signal layers to accommodate the bias. Include the layer name and the *horizontal* or *vertical* keyword with the `direction` command to control layer routing direction.

For example:

```
direction SIGNAL_2 vertical
```

## SPECCTRA Tutorial





### Lesson 4: Setting Rules and Constraints

---

#### Select layers for routing and set layer routing direction

1. Start the autorouter and load *lesson4.dsn*.
2. Click the *Layers* button on the tool bar.

The Layers panel opens. The default signal layers directions display as follows:

SIGNAL_1	
SIGNAL_2	
SIGNAL_3	
SIGNAL_4	

3. Type `unselect layer SIGNAL_2` in the command entry box.

The button in the Layers panel for SIGNAL\_2 changes to



4. Type `direction SIGNAL_1 vertical` in the command entry box.
5. Notice the button in the Layers panel for SIGNAL\_1 changes to



Use the `direction` command to change the routing direction for the other signal layers.

6. Type `direction SIGNAL_2 horizontal` in the command entry box.

Notice in the Layers panel that the direction command changes the layer's routing direction *and* selects the layer.

7. Type `quit` in the command entry box and exit the autorouter.

#### Selecting Vias for Routing

The vias you can use in the autorouter are the vias included by your layout system. Usually, all vias in a design are selected and available for routing, but you can unselect vias that you do not want to use.

The `select` and `unselect` commands control which vias are available for routing. Vias that are defined in the design but are unselected, are not available for routing.

## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

When two or more vias are available for a transition between two layers, the autorouter chooses the one that uses the fewest layers and has the smallest shapes. You can use blind and buried vias if you have the HYB option.

**Note:** You can assign specific vias to nets, classes, and groups of fromtos if you have the ADV option.

#### **Task: Select a single via for routing**

##### **Procedure**

1. Start SPECCTRA and load `lesson4.dsn` from the `tutorial` directory.
2. Choose *Report – Specify*.  
The Report specify opens.
3. Click on Vias.  
The Via Report opens.
4. Browse the report and observe that all vias are selected for routing.
5. Close the report when you finish browsing.
6. Type `unselect all vias` in the command entry box.
7. Type `select via VIA35`.
8. Repeat steps 2 and 3.
9. The report indicates that all vias are unselected for routing except VIA35.
10. Close the report when you finish browsing.
11. Type `quit` in the command entry box to exit SPECCTRA.

#### **Selecting Connections and Autorouting**

If one or more connections are selected, only those connections are routed. Usually you do not route fewer than all connections. You can assign connections a priority if you want to control when certain nets are routed. See *Assigning Routing Priorities to Nets and Classes* later in this lesson.

Occasionally you might want to route fewer than all connections in order to check rule settings or to experiment. You can choose the connections you want to route by selecting them.

## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

You can select connections by component, net, class group and fromto.

The following table describes the commands that implement the selection methods.

**Table 4-2 Commands that Select**

Command	What it does
select component <i>&lt;id&gt;</i>	Selects the connections that are attached to the component identified by <i>&lt;id&gt;</i> , which is the component's reference designator.
select net <i>&lt;id&gt;</i>	Selects all connections on the net identified by <i>&lt;id&gt;</i> .
select class <i>&lt;id&gt;</i>	Selects all the nets that are members of the class identified by <i>&lt;id&gt;</i> .
select group <i>&lt;id&gt;</i>	Selects all the fromtos (connections) that are members of the group identified by <i>&lt;id&gt;</i> .
select fromto	Selects all unrouted connections.

SPECCTRA highlights selected connections in the “select color,” which is usually yellow.

After you select connections, you can apply rules and controls to the connections, then route the connections. Usually, you unselect connections after they are routed.

Use the `unselect` command to return components, nets, classes, groups, and fromtos to normal status. The `unselect all routing` command is used when you want to make certain that all previously selected routing objects are unselected, and that all unrouted connections can be routed.

### Task: Autoroute selected connections

#### Procedure

1. Start SPECCTRA and load `lesson4.dsn` from the `tutorial` directory.
2. Click the *Select Component* button on the tool bar.  
*Sel Comp* appears in the mode status area.
3. Click on a pin or inside the outline of any component.  
All pins on the component and all attached connections are selected.
4. Click on any pin of the same component.

## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

The component is unselected.

5. Click the *Select Net* button on the tool bar.

6. Click on any pin that has a guide attached.

All connections on the net are selected.

7. Drag the pointer across several pins.

The nets attached to the pins are selected.

8. Click the *Select Guide* button on the tool bar.

9. Drag the pointer across several guides.

The connections attached to the guides are selected.

10. Type `unselect all routing` in the command entry box.

All connections are unselected.

11. Type `select net sig1 sig2`.

12. Type `bus diagonal`.

13. Type `select component U1`.

14. Type `route 25`.

Although this command specifies 25 route passes, only two are needed to route the selected connections. The remaining passes are skipped.

15. Type `unselect all routing`.

## Using Routing Keepin Regions

Another method you can use to limit autorouting to fewer than all connections is to create one or more fences. A fence is a keepin boundary for the connections it encloses.

Connections that span a fence's boundaries or that lie completely outside the boundaries, are ignored during routing. You can define one or more individual fences to restrict the routing in several areas. If you create fences that overlap, they merge to form a single, polygonal fence region.

You create fences using one of the following methods.

- Draw a fence.
- Use a dialog box to enter the fence coordinates.
- Use the `fence` command.

The easiest method for creating a fence is to draw it, but often you need to create the fence in your Do file. In a Do file, you use the `fence` command. Before you write the Do file, use the mouse to draw the fence, determine the coordinates from the Did file, and then use the coordinates in your Do file.

### Task: Draw a fence and only route the enclosed connections

#### Procedure

1. Type `delete all wires` in the command entry box.

A message popup prompts you to confirm the command. (The autorouter should be running with `lesson4.dsn` loaded.)

2. Click Yes to delete all wires.

You need to display component reference designators.

3. Choose *View – Labels*.

The View Labels dialog box opens.

Make sure *Ref Des* is selected.

4. Click *OK*.

The component reference designators display.

5. Choose *Define – Fence – Draw Mode*.



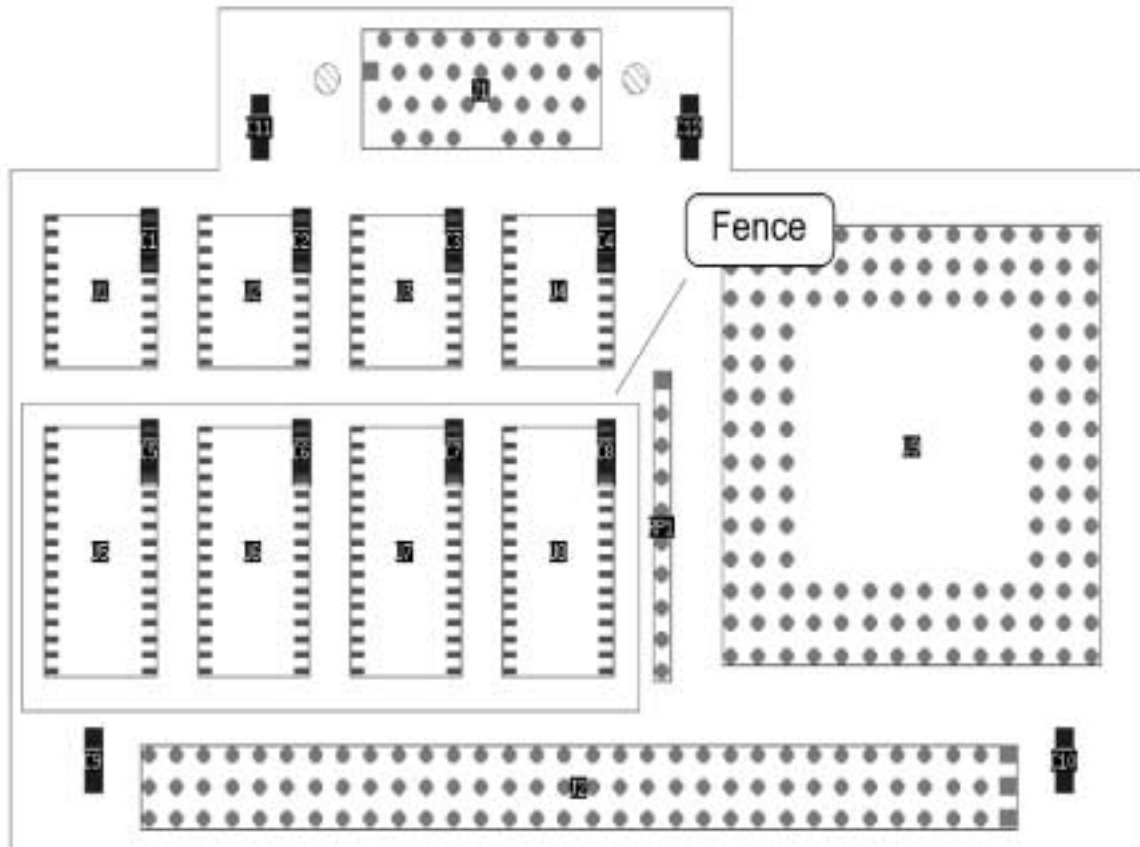
## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

*Draw Fence* appears in the mode status area.

6. Drag the pointer and draw a fence around components U5, U6, U7, and U8, as shown in the following figure. If you do not like the fence you drew, choose *Define – Fence – Draw Mode* to remove the fence and start with step 6 to draw a new fence.



7. Press the right mouse button and choose *Define Polygon As Fence* from the menu.
8. View the session transcript in the Output window, and locate the mode edit fence, area init\_pt, and area add\_pt commands.

The X,Y coordinates beside the `area_init_pt` and `area_add_pt` commands represent diagonally opposite corners of the fence.

9. Read the coordinate information beside the `area_init_pt` and `area_add_pt` commands.

You could use this coordinate information with the `fence` command in a Do file.

For example,

## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

```
fence 0.6 6.85 2.75 7.75.
```

10. Type **bus diagonal** in the command entry box.

11. Type **route 25**.

The autorouter routes only the connections that are *completely inside* the fence region.

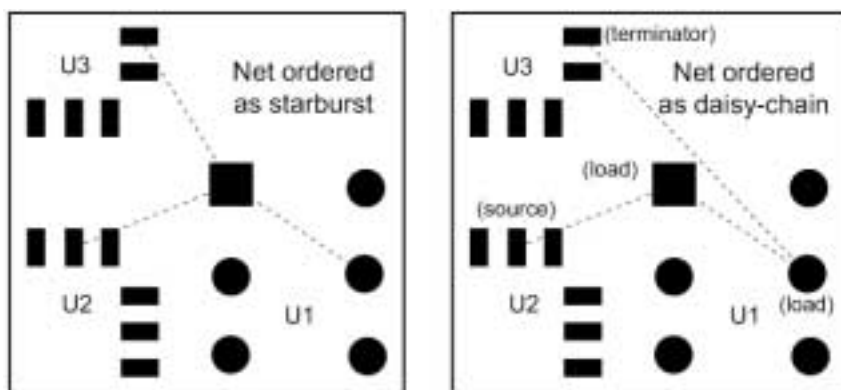
12. Wait for *Idle* to replace the *Pause* button in the control area and type **quit** in the command entry box to exit SPECCTRA.

**Note:** You can also use soft fences if you have the FST option. Soft fences are useful for maintaining separation between analog and digital circuits. See the `set` command in the *SPECCTRA Command Reference* for additional information about soft fences.

## Controlling Whether Nets are Routed With Starburst or Daisy-Chain Wiring

Nets are usually ordered for starburst routing because it produces the most efficient wiring. Ordering determines how the pins on a net are wired. Starburst is the most efficient because the autorouter orders and routes starburst nets with minimum wire lengths. The following figure shows the basic difference between starburst and daisy-chain wiring.

**Figure 4-2 Comparison Between Starburst and Daisy-Chain Wiring**



Nets are ordered for daisy-chain routing in the layout system and translated to the Design file, or they can be ordered in SPECCTRA. All nets default to starburst ordering unless they are explicitly ordered as daisy-chain.

Use the `order` command to define net ordering. For example, `order daisy net <net id>` orders a single net for daisy-chain routing. Replace `<net id>` in the command with the name of the net you want to order.

## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

When you order a net for daisy-chain routing, you often want to control which pins are source, load, and terminator. If you do not assign these properties, SPECCTRA treats all pins on a daisy-chain net as loads and chains them together in an optimum configuration.

You use the `assign_pin` command to assign source, load, or terminator properties to pins. If a net has only one pin on a given component, you can assign a property to that pin by referencing the component and then ordering the net.

For example:

```
assign_pin source U23
assign_pin terminator U29
order daisy net S01
```

If a net has several pins on a single component, you assign pin properties by adding the pin IDs to the command.

For example:

```
assign_pin source U33 (pins 2)
assign_pin terminator U33 (pins 11)
order daisy net PRX
```

**Note:** You assign source, load, or terminator properties to pins before you use the `order` command. If you assign pin properties but do not use the `order` command, the properties are ignored by the autorouter.

### Task: Assign pin properties and order a net for daisy-chain routing

#### Procedure

1. Start SPECCTRA and load `lesson4.dsn` from the `tutorial` directory.
2. Click the *Layer* button on the tool bar.
3. Click on *Labels* in the Layer panel.
4. Type `select net sig1` in the command entry box.
5. Type `assign_pin source J2 (pins 32B)`.
6. Type `assign_pin source U5 (pins 1)`.
7. Type `order daisy net sig1`.

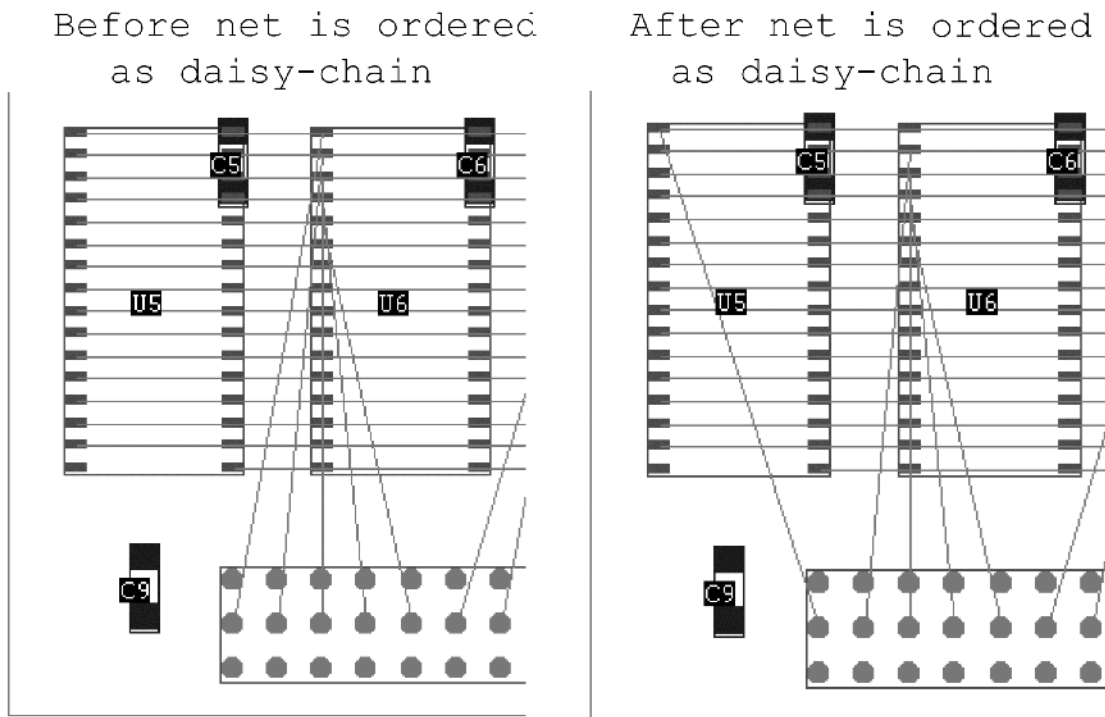
## SPECCTRA Tutorial

### Lesson 4: Setting Rules and Constraints

---

The net is ordered for daisy-chain routing. When two or more pins on a net are assigned the same property, they are chained together. Source pins then chain to loads, and load pins chain to terminators (if any are defined).

8. Notice that the guide has moved from U6-1 to U5-1 per step 6.



9. Type `route 25`.

Net sig1 is routed in daisy-chain fashion.

10. Type `unselect all routing`.

## Routing Nets

In this section, you will learn how to route a set of nets or fromtos with the same rules.

You will:

- define a class of nets.
- define a group of fromtos.
- assign rules to classes and groups.

- assign routing priorities to nets and classes.

### Defining a Class of Nets

If you have a number of nets that you want to reference as a set, you can assign them a class name. You use the `define` command to create a class id and assign the net ids. The command is:

```
define (class <class id> <net id> ... <net id>)
```

Use any combination of alpha and numeric characters for *<class id>* and substitute any number of net names you want in place of *<net id>*.

After defining a class, you can apply rules and reference all nets in the class by using the class name.

For example, the command to select a class is

```
select class <class id>
```

Substitute a class name for *<class id>*.

### Task: Define a net class, select the class, and route it

#### Procedure

1. Type `delete all wires` in the command entry box.  
A message popup prompts you to confirm.
2. Click Yes to delete all wires.
3. Type `define (class C1 sig17 sig18 sig19 sig20 sig21 sig22 sig23 sig24)`.  
You created a class named C1 that consists of the eight nets.
4. Type `select class C1`.
5. Type `bus diagonal`.  
Only the selected nets are routed.
6. Type `unselect all nets`.
7. Type `quit` in the command entry box to exit SPECCTRA.

## Assigning Rules to Classes and Groups

A common reason for defining a class of nets or a group of fromtos is so you can easily assign rules. (A fromto is a single pin-to-pin connection on a net.) If several nets or fromtos must follow the same rules or be routed in the same way, define a class or group and apply the rules to all members with a single reference.

Class and group rules are followed according to their rankings in the rule hierarchy. Net rules override class rules, group rules override net rules, and fromto rules override group rules. Refer to the Rule Hierarchy figure in the beginning of this chapter to review the rule hierarchy.

### Task: Define a class and a group, assign rules, and route

#### Procedure

1. Start SPECCTRA and load `lesson4.dsn` from the `tutorial` directory.
2. Type `define (class C1 CLK sig34 sig35 sig47)` in the command entry box.  
You defined a class named C1 that consists of the nets CLK, sig34, sig35, and sig47.
3. Type `define (group G1 (fromto J1-8 U3-13)(fromto J1-12 U4-8)(fromto J1-10 U3-3)(fromto J1-14 U3-15))`.

**Note:** If you define a group and omit a fromto in error, you must use `forget group <group id>` before you can redefine the group.

4. Type `rule class C1 (width .008)`.
5. Type `rule group G1 (width .020)`.
6. Type `select group G1`.
7. Type `route 25`.
8. Type `select class C1`.
9. Type `route 25`.
10. Zoom in and examine how the wiring for group G1 between connector J1, U3, and U4 differs from the other wiring.

The fromtos in group G1 follow a 20 mil width rule. The other connections follow the 8 mil width rule that is set for class C1.

11. Type `quit` in the command entry box to exit SPECCTRA.

## Assigning Routing Priorities to Nets and Classes

You can schedule when nets and classes are routed by assigning them priorities. Assigning priorities is preferred to selecting nets to route them. You assign priorities using the `circuit` command.

Routing priorities are assigned with numeric values in the range from 1 to 255. A value of 255 assigns the highest priority, and a value of 1 assigns the lowest priority. Nets that are assigned higher priorities are routed before those assigned with lower priorities. All nets are assigned a default priority value of 10.

If you must assign several priority values to different nets or classes of nets, assign values that differ by at least ten. Priority values that differ by less than ten might be ignored due to other factors that also control routing priority. For example, the autorouter usually routes larger width wires before smaller width wires.

### Task: Assign priorities to nets and route them

#### Procedure

1. Start SPECCTRA and load `lesson4.dsn` from the `tutorial` directory.
2. Type `define (class C1 sig1 sig3 sig5 sig7)` in the command entry box.
3. Type `circuit class C1 (priority 100)`.
4. Type `circuit net sig8 (priority 255)`.
5. Type `select net sig1 sig2 sig3 sig4 sig5 sig6 sig7 sig8`.
6. Type `route 25`.

Net `sig8` routes first because it is assigned the highest routing priority. Next, the nets in class `C1` are routed. Then, the nets `sig2`, `sig4`, and `sig6` are routed.

7. Wait for *Idle* to replace the *Pause* button in the control area and type `quit` in the command entry box to exit SPECCTRA.

## What You Learned

In this lesson, you learned how to set rules and control the autorouter.

You learned how to:

- set rules at different levels in the autorouter's hierarchy.
- set wire and via clearance rules for different objects.
- select layers for routing.
- set layer routing direction.
- select vias for routing.
- select connections for routing.
- use routing keepins.
- control whether nets are routed with starburst or daisy-chain wiring.
- define a class of nets and a group of fromtos.
- assign rules to classes and groups.
- assign routing priorities.

Study the commands that set rules and perform specialized routing functions. In particular, study the `circuit`, `define`, `rule`, `fanout`, `bus`, `smart_route`, `route`, and `clean` commands. These commands are fundamental to using the routing tools.

In the next lesson, you will learn how to interactively route a PCB design.



---

## Lesson 5: Interactive Routing and Editing

---

### What You Will Learn

This lesson teaches you how to use EditRoute, which is the toolset for interactively routing and editing PCB wiring.

In this lesson, you will learn how to:

- set up your interactive routing environment.
- use the routing tools.
- control the routing layer.
- add vias as you route.
- move wiring.
- copy wiring.
- eliminate extra angles from a wire.
- replace a via with another type.
- change the width of a wire segment.

This lesson takes about 60 minutes to complete.

### What to do Before You Begin

Before you begin this lesson, do the following:

- Complete [Lesson 4: Setting Rules and Constraints](#).
- Read [About this Tutorial](#) on page 7, which explains conventions, terminology, and symbols that are used throughout this tutorial.
- Read [License Considerations](#) on page 13 which notes important licensing information to keep in mind as you work through the tutorial.

## Interactively Routing

EditRoute is the interactive routing and editing toolset within SPECCTRA. These interactive tools use the same ShapeBased technology and gridless environment as the AutoRoute tools. You can use the interactive tools to preroute critical connections before you autoroute. You can also use them to change or reroute connections after autorouting.

### Setting your Interactive Routing Environment

You set the interactive routing and editing environment by choosing *Setup* from the Interactive Routing menu. The Interactive Routing menu appears when you press the right mouse button within the work area.

**Note:** If another Windows application uses the right mouse button, you might need to hold down the [Ctrl] key when you press the right mouse button to display the Interactive Routing menu.

The Interactive Routing Setup dialog box contains five tabs.

- General
- Measure
- Bus
- Style
- Move/Copy.

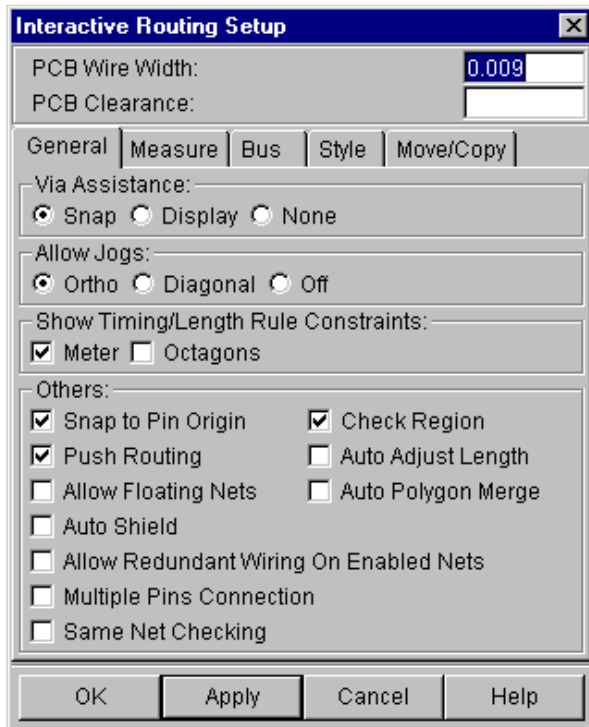
You can use these tabs to set Measure mode options, set move and copy options, and to set interactive routing controls in three categories: General, Bus, and Style. You can also use the Interactive Routing Setup dialog box to set global (PCB) clearance and wire width rules.

The Interactive Routing Setup dialog box opens with the General tab displayed. To change to another category, click on the corresponding tab. The General tab displays controls that you can use to:

- set via assistance, wiring jog controls, and timing/length rule indicators.
- enable or disable wire snap to pin origin, push routing, region rule checking, and redundant wiring on enabled nets.
- allow or disallow floating nets, automatic polygon merging, automatic shielding, and automatic length adjustments.

The following figure shows the General (default) tab.

**Figure 5-1 The Interactive Routing Setup Dialog Box (General Tab)**

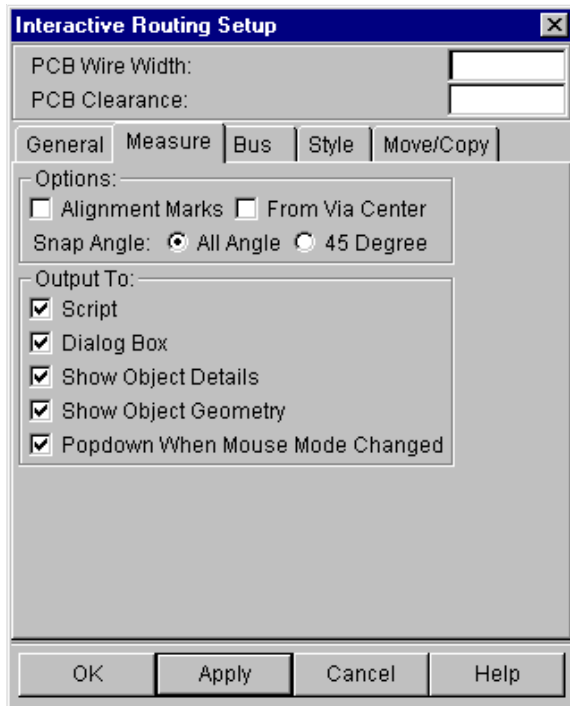


The Measure tab displays controls you can use to:

- control whether horizontal and vertical tick marks display.
- control whether the tool measures blind and buried via gap clearances from via centers or from via edges.
- control how the pointer snaps to the location you are measuring.
- control whether measurement information is displayed in the Output window or in the Measure dialog box, or both when the Script or Dialog Box is checked.
- control whether additional information is displayed in the Measure dialog box for objects you measure.
- control whether the information displayed in the Measure dialog box for an object includes its shape and its location in the design.
- control whether the Measure dialog box automatically opens when you click or drag the pointer in the work area.

The following figure shows the Measure tab.

**Figure 5-2 The Interactive Routing Setup Dialog Box (Measure Tab)**

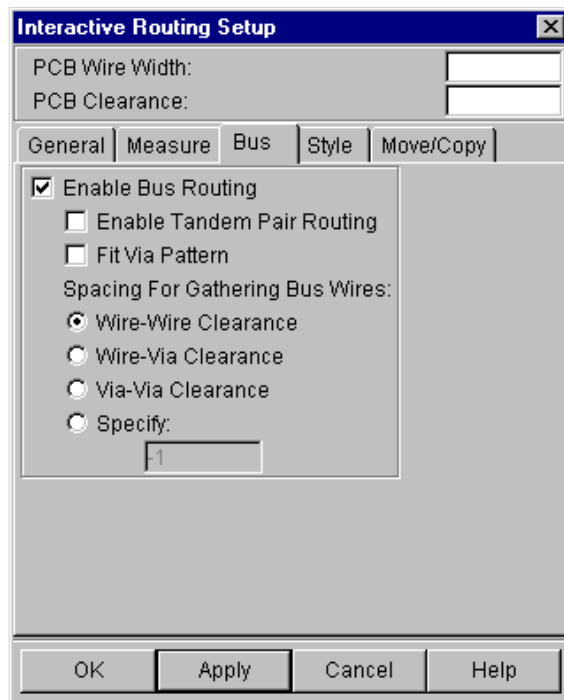


The Bus tab displays controls that you can use to:

- enable or disable bus routing.
- control tandem layer pair routing and via pattern fitting.
- set the bus wire clearance.

The following figure shows the Bus tab.

**Figure 5-3 The Interactive Routing Setup Dialog Box (Bus Tab)**

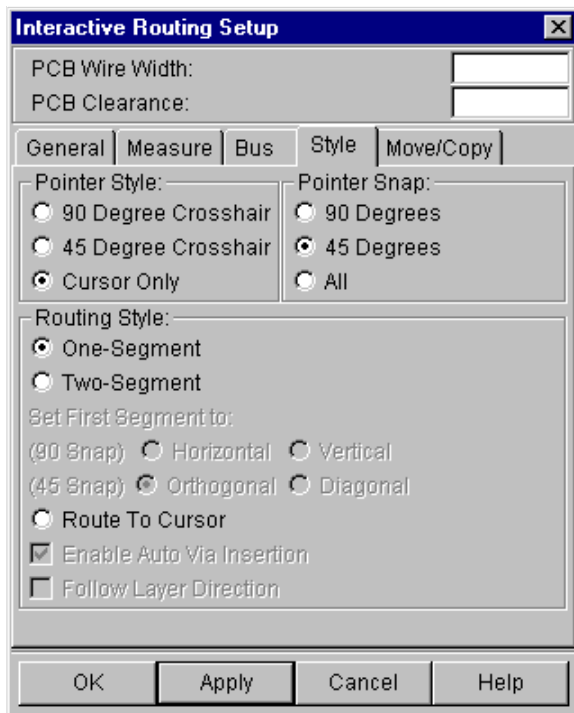


The Style tab displays controls that you can use to:

- set the pointer style to 90 degree crosshairs, 45 and 90 degree crosshairs, or arrow only pointer style.
- control whether the pointer snaps to any angle, or snaps to a 45 or 90 degree when routing.
- control the one or two segment routing style.

The following figure shows the Style tab

**Figure 5-4 The Interactive Routing Setup Dialog Box (Style Tab)**

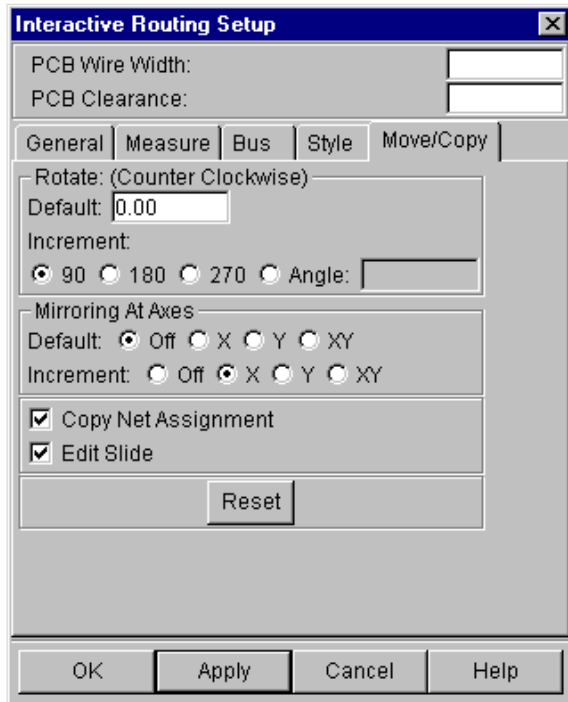


The Move/Copy tab displays controls that you can use to:

- set the default rotation angle and mirroring axis for automatic object rotation and mirroring during move or copy operations.
- change the incremental rotation angle for incremental object rotation.
- change the incremental mirroring axis for incremental object mirroring.
- control whether copies of wiring polygons have the same net assignment as the original or are unassigned (for Copy Polygon mode only).
- control whether the tool slides the objects, attempting to push obstacles out of the way, or moves them over obstacles and drops them in the new location (for Move mode only).

The following figure shows the Move/Copy tab.

**Figure 5-5 The Interactive Routing Setup Dialog Box (Move/Copy Tab)**



In this section, you will load a Wires file, which contains routed wires. You will then use the Interactive Routing Setup dialog box to change the pointer style and change the pointer snap angle, which will affect the interactive routing environment. Next, you will route a connection to see how EditRoute pushes wires aside as you are routing.

### **Task: Load a wires file**

#### **Procedure**

1. Start SPECCTRA and load `lesson5a.dsn` from the `tutorial` directory. See [Where to find the Accompanying Lesson Files](#) on page 12 for the location of this directory.
2. Choose *File – Read – Wires*.  
The Read Wires dialog box opens.
3. Click the *Browse* button.  
The Open dialog box opens for Windows platforms.  
The Select File dialog box opens for UNIX platforms.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

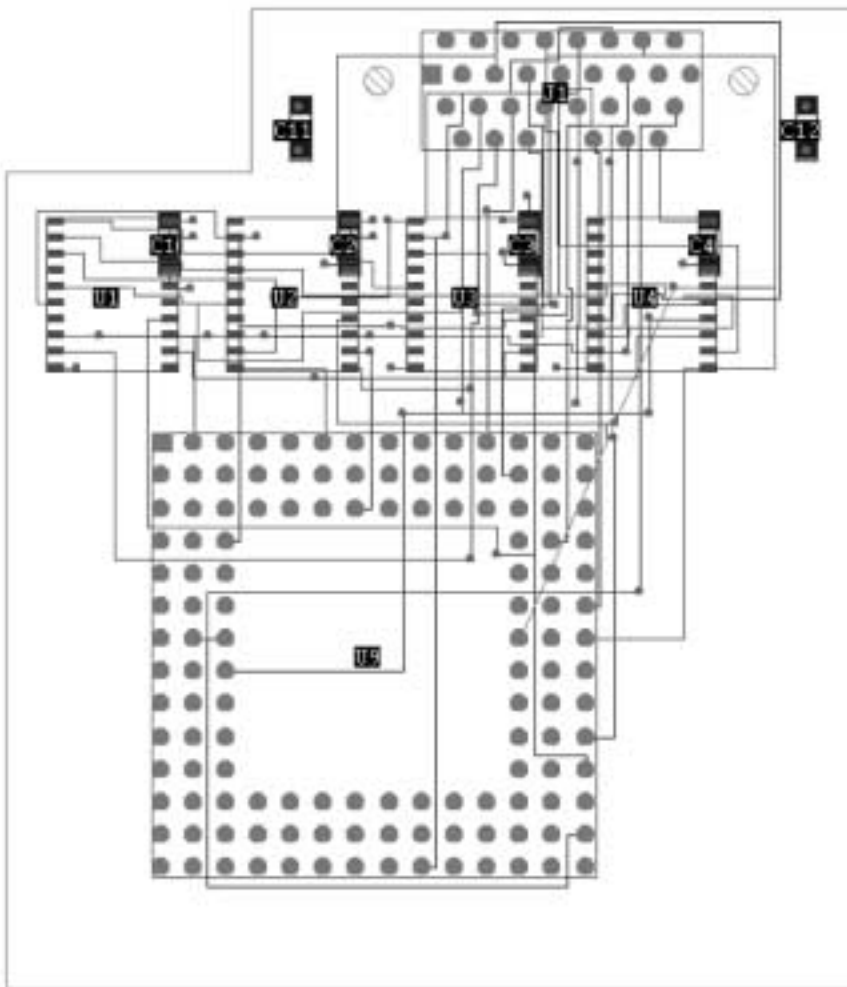
4. Change to the `tutorial` directory for Windows platforms and open `lesson 5a.w`.

Select `lesson 5a.w` and click *OK* for UNIX platforms.

The filename is added to the dialog box.

5. Click *OK* on the Read Wires dialog box.

The Wires file loads. All connections except one are prerouted as shown in the following figure.





### **Task: Set up your interactive routing and editing environment**

#### **Procedure**

1. Move the pointer into the work area and press the right mouse button.  
The Interactive Routing Menu appears.
2. Choose *Setup* from the Interactive Routing Menu.  
The Interactive Routing Setup dialog box opens.
3. Click on the *Style* tab.
4. Set the pointer style to a 90 degree crosshair and the pointer snap angle to 90 degrees by doing the following:
  - a. Select *90 Degree Crosshair* for the *Pointer Style*.
  - b. Select *90 Degrees* for the *Pointer Snap*.
  - c. Click *OK*.
5. Display reference designators and pin ID's by doing the following:
  - a. Click *View – Labels*.  
The View Labels dialog box opens.
  - b. Select *Ref Des and Pin ID's*.
  - c. Click *OK*.  
The Reference designators display with the components.

### **Task: Route a connection using Edit Route**

#### **Procedure**

1. Zoom in to view the unrouted connection of U4-16 to U5.  
You will route the unrouted connection of U4-16 to U5 using Edit Route mode.
2. Press the right mouse button and choose *Edit Route Mode*.  
A pointer appears in the work area and Edit Route appears in the mode status area.
3. Click pin 16 on U4.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

4. The pointer changes to 90 degree crosshairs.

5. Press the right mouse button and choose *Cancel*.

The routing operation stops. Edit Route still appears in the mode status area.

The crosshairs do not display unless you are routing a connection.

6. Click on U4-16 and route the connection out of the right side of the pin and down half the distance to the target pin on U5. Zoom in if necessary.

The route adds unnecessary angles to the surrounding wires. You will learn how to remove unnecessary angles from wires by using Critic Route later in this lesson. You will ignore these angles for now.

**Note:** If you make a mistake, you can press the right mouse button and choose *Undo* to undo the last action or *Cancel* to cancel the last command.

7. Press the right mouse button and choose *Setup*.

8. Click *45 Degree Crosshair* for the *Pointer Style*.

9. Click *45 Degrees* for the *Pointer Snap*.

10. Click *OK*.

The pointer appears as both 45 and 90 degree crosshairs.

11. Use a 45 degree corner to finish routing the connection to U5.

When you click the target pin on U5, the connection completes, and the wire disconnects from the pointer.

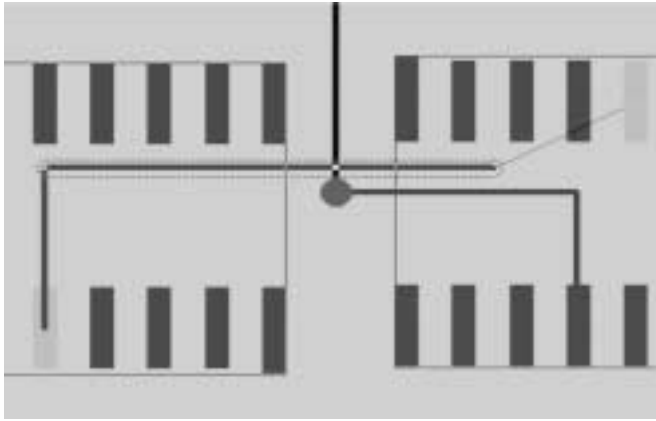
12. Type `quit` in the command entry area to exit SPECCTRA.

You do not need to save the Session file or the Did file.

### Routing and Replacing Wires

You use the Edit Route mode to route connections and to replace existing wires and vias. While you are routing or replacing a wire, an envelope surrounds the wire. This clearance envelope corresponds to the clearance rule for the wire you are routing. The envelope helps you see the clearance from the wire to other objects. The following figure shows the clearance envelope that surrounds a wire during an interactive routing session.

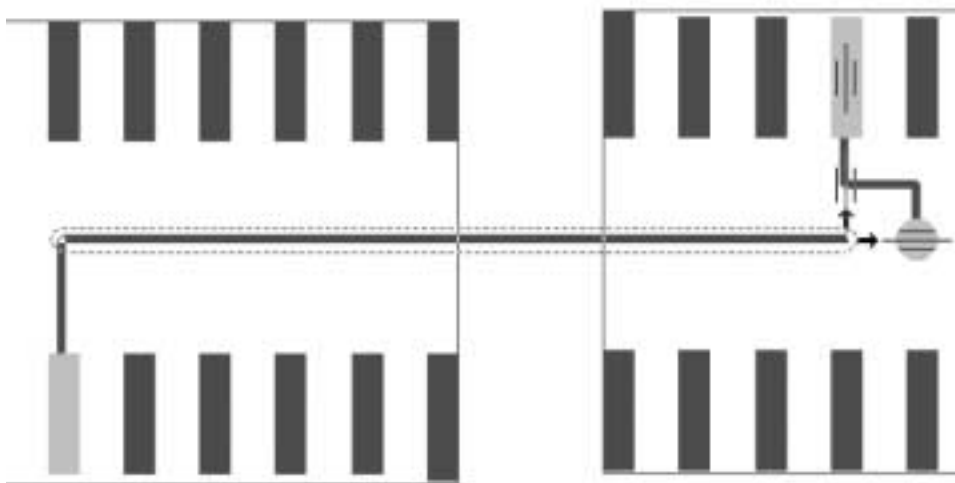
**Figure 5-6 Clearance Envelope Surrounding a Wire Being Routed**



The clearance envelope changes if, for example, you add a via and change to a layer with a different clearance rule.

As you route a connection, alignment marks and arrows display to help you align the end of the wire with a target wire, pin, or via. When the end of the wire aligns with a target, one or more arrows appear at the wire end, and alignment marks appear at the target object. The following figure shows how the alignment marks and arrows appear during a routing session.

**Figure 5-7 Alignment Marks and Arrows on a Wire Being Routed**



## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

You will use Edit Route mode to route several connections. As you route each connection, use the clearance envelope to help you route the wires as close together as possible without creating clearance violations. Use the alignment marks and arrows to make straight connections to the target pins.

#### **Task: Interactively route several connections**

##### **Procedure**

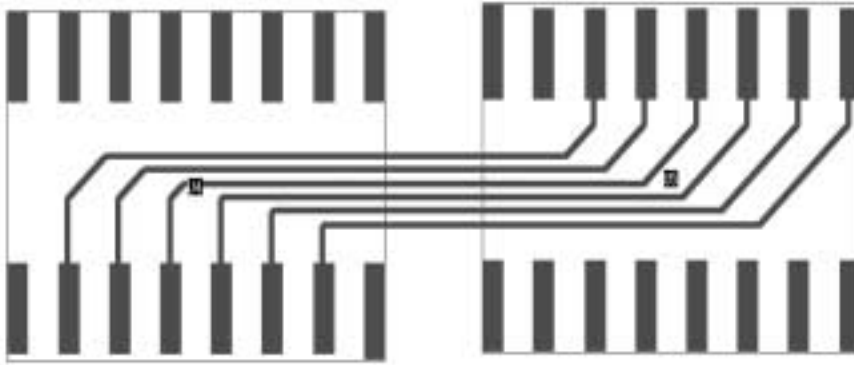
1. Start SPECCTRA and load `lesson5b.dsn` from the `tutorial` directory.
2. Display the reference designators and pin ID's by doing the following:
  - a. Click *View – Labels*.  
The View Labels dialog box displays.
  - b. Select *Ref Des and Pin IDs*.
  - c. Click *OK*.  
The Reference designators display with the components.
3. Move the pointer into the work area and press the right mouse button.  
The Interactive Route dialog box opens.
4. Choose *Edit Route Mode* from the Interactive Routing menu.  
Edit Route appears in the mode status area.
5. Zoom in to view U4 and U5.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

6. Route the six connections between U4 and U5, as shown in the following figure.



7. Type **quit** in the command entry area to exit SPECCTRA.

You immediately exit SPECCTRA without saving a Session file. The Did file remains in the tutorial directory. You will not need the Did file, so you can delete it.

Next you will learn how to finish an unrouted or partially routed connection.

### Finishing the Routing

While you are routing in Edit Route mode, you can finish an unrouted or partially routed connection by choosing *Finish Route* from the right button menu. *Finish Route* adds wire segments and vias, if required, to complete a connection.

*Finish Route* searches for a path within a limited area around the connection you are routing. If a path cannot be found in the search area, the connection cannot be finished automatically and the operation is aborted.

In the following procedure, you finish several connections using the *Finish Route* option in Edit Route mode.

### Task: Finish routing connections

#### Procedure

1. Start SPECCTRA and load `lesson5b.dsn` from the `tutorial` directory.
2. Display the reference designators and pin IDs.

3. Click the *Edit Route* button on the tool bar.

Edit Route appears in the mode status area.

4. Zoom in to U4, U5, and the lower half of U6.
5. Click on one of the pins in the upper row of U4.
6. Digitize a portion of the connection above U4.
7. Press the right mouse button and choose *Finish Route*.

The connection is completed for you.

8. Click on another unrouted pin in the upper row of U4.
9. Press the right mouse button and choose *Finish Route*.

The connection routes automatically.

You will route the remaining unrouted pins on the upper row of U4. You could do this by selecting each pin and repeating the previous step, but you will learn a faster way to route these connections by completing the following steps.

10. Click on another unrouted pin in the upper row of U4.
11. Press [Ctrl-A].

Using these two keys, you can repeat the last menu command. In this step, you repeated step 9.

12. Repeat steps 10 and 11 to route the remaining unrouted pins on the upper row of U4.
13. Exit the autorouter by choosing *File – Quit* or by typing `quit` in the command entry area.

You do not need to save a Session file or the Did file.

## Reversing and Restoring Interactive Operations

If you make a mistake, SPECCTRA lets you undo a previous interactive routing function. If you did not intend to undo something, you can restore it using the `redo` command. The following table shows how to use the undo and redo commands.

**SPECCTRA Tutorial**  
Lesson 5: Interactive Routing and Editing

---

**Table 5-1 Undo and Redo Functions for Interactive Routing**

Function	Description	Execute by . . .
undo	Reverses the last interactive function.	Pressing the F3 function key - or - Choosing <i>Undo</i> from the right button menu. - or - Choosing <i>Edit – Undo</i> from the menu bar
redo	Restores the last interactive function reversed with <code>undo</code> .	Holding down the [Shift] key and pressing F3. - or - Choosing <i>Edit – Redo</i> from the menu bar.

**Note:** You can undo an individual or a series of interactive functions. However, if you interrupt an interactive routing, editing, or placement session with a select, report, autorouting or autoplacement function; undo and redo memory is cleared and you cannot undo or redo prior functions.

The following procedure uses undo and redo to reverse and then restore interactive routing functions.

**Task: Use undo and redo to reverse and restore interactive routing**

**Procedure**

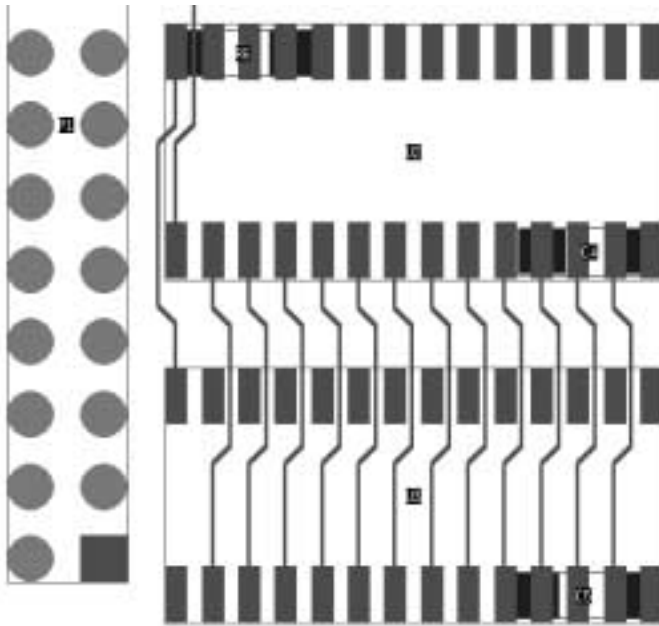
1. Start SPECCTRA and load `lesson5c.dsn` from the `tutorial` directory.
2. Press the right button menu and choose *Setup* from the Interactive Routing menu.  
The Interactive Routing Setup dialog box opens.
3. Click on the the *Style* tab and do the following.
  - a. Click *90 Degree Crosshair* for the *Pointer Style*.
  - b. Click *OK*.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

4. Click the *Edit Route* button on the tool bar.  
Edit Route appears in the mode status area.
5. Display the reference designators.
6. Route several connections between U2 and U3 as shown in the following figure.



7. Press `F3` to undo the last function.
8. Press `[Shift]F3` to redo the function you reversed with `undo`.
9. Complete all the connections between U2 and U3
10. Use `F3` to undo all previous routing steps.
11. Use `[Shift]F3` to restore all the routing you removed with `undo`.
12. Exit SPECCTRA without saving a Session file or a Did file.

### Adding Vias and Changing Layers

While you are interactively routing, the layer you are routing on is the *primary layer*. The layers that you can change to when you add a via are the *secondary layers*. The primary layer is always named on the status bar in the area beside the pencil button.



## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

If you add a via and change to a different routing layer, the layer you change to becomes the primary layer and the previous routing layer becomes a secondary layer. You control the initial primary and secondary routing layers by using the Layers panel. In the Layer panel, you can disable layers if you do not want to use them during interactive routing.

A pencil button in the Layer panel identifies whether a signal layer is the primary layer or a secondary layer. The bold pencil identifies the primary layer. A dim pencil identifies a secondary layer. If neither the dim nor bold pencil appears for a signal layer, the layer is *disabled* for interactive routing.

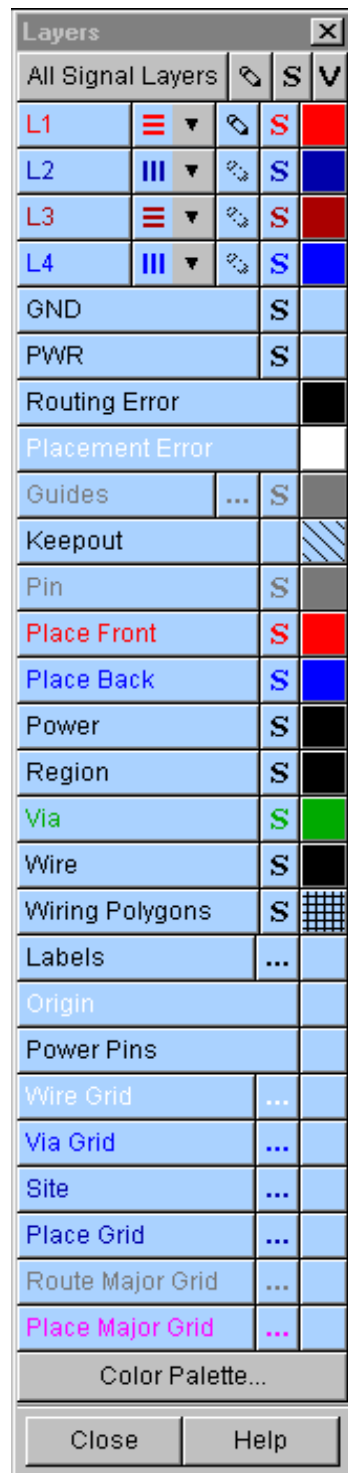
The following figure shows the Layer panel with L1 as the primary layer and L2, L3, and L4 as secondary layers.

# SPECCTRA Tutorial

## Lesson 5: Interactive Routing and Editing

---

**Figure 5-8 The Layers Panel**



## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

You set a layer as primary, secondary, or disabled by clicking on the pencil. If you click on a dimmed pencil, the layer becomes primary. If you click a second time, the pencil box is emptied and the layer is disabled for interactive routing.

After you choose which layer is primary, which layers are secondary, and whether layers are disabled, you use the mouse or function keys to add vias and change the routing layer. The layer you change to when you add a via depends on how you set up layers as primary, secondary, or disabled.

When you digitize twice at the end of a wire segment, the autorouter adds a via, determines an optimum secondary layer, and changes to that layer. Instead of allowing the autorouter to choose the layer, you can choose the layer change by using the F5 and F6 function keys.

The F5 key adds a via and changes routing to the next available layer *below* the current routing layer. Each time you press F5, the next active signal layer becomes the current routing layer. If you press F5 and no layer is available below the current routing layer, the system beeps and the message “*No enabled layer below <layer id>*” displays in the message area.

The F6 key adds a via and changes routing to the next available layer *above* the current routing layer. Each time you press F6, you move up to the next available signal layer. If no layer is available above the current routing layer, the system beeps and a warning message appears in the message area.

In the following procedure, you set up the primary and secondary routing layers, disable two signal layers for interactive routing, and use the F5 and F6 function keys to add a via.

#### **Task: Set up the routing layers and use function keys to change layers and add a via**

##### **Procedure**

1. Start SPECCTRA and load `lesson5d.dsn` from the `tutorial` directory.
2. Click the *Layer* button on the tool bar.

The Layer panel opens, as shown in the following figure. You will disable layers L2 and L3.

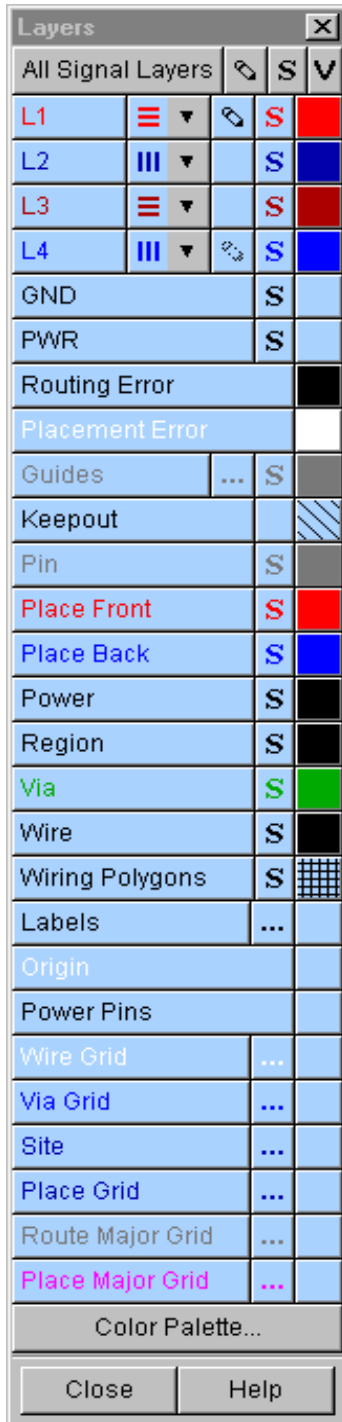
3. Click twice in each pencil box to disable layers L2 and L3.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

The pencil boxes for layers L2 and L3 are empty, which means these layers are disabled.



4. Click *Close*.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

5. Display the reference designators.

6. Click the *Edit Route* button on the tool bar.

*Edit Route* appears in the mode status area and L1 appears on the status bar as the primary layer.

7. Click on pin 1 which is the square pin of P1.

**Note:** You will need to zoom in to see pin 1.

8. Move the crosshairs a small distance to the right and away from the pin.

9. Press F5.

The wire changes to L4 which is the next available layer below L1 (remember that you disabled L2 and L3). L4 appears in the status area as the primary layer.

10. Press F5 again.

The message area indicates that no additional layers are available below the current layer.

11. Press F6.

The wire changes to L1 which is the available layer above L4.

12. Press F5 to set the primary layer to L4.

13. Digitize a wire segment that ends just above the target pin on U3.

14. Press F6.

A via is added. The primary layer changes to L1.

15. Finish routing the connection to U3.

16. Exit SPECCTRA.

You do not need to save a Session file or the Did file.

## Editing Wires and Vias

You edit wires and vias by setting one of the following editing modes for the left mouse button.

- Move
- Copy Route
- Critic Route

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

- Change Via
- Change Wire Width

You set an editing mode by clicking the right mouse button and choosing the mode from the Interactive Routing menu or by clicking a button on the tool bar.

When you edit wires and vias, SPECCTRA observes clearance rules if *Checking* is enabled on the status bar. If *Checking* is enabled, and you attempt an edit that violates a clearance rule, the function is ignored.

### Moving Wires and Vias

You use Move Route mode to move wire segments and vias. As you move a wire segment, the segment stretches at its corners and at the points where it attaches to pins and vias.

If you move a wire segment against a component pin, and *Allow Jogs* is enabled on the General tab in the Interactive Routing Setup dialog box, the segment “jogs” around the pin if possible. When you move a wire against another, or against a via (and *Push Routing* is enabled on the General tab in the Interactive Routing Setup dialog box), the stationary wire or via is pushed to avoid a clearance violation.

If you move a via, the attached wires stretch to follow. You cannot move a via to a location that creates a clearance violation if *Checking* is enabled in the status bar.

If you move a via against a wire and *Push Routing* is enabled on the General tab in the Interactive Routing Setup dialog box), the wire “deforms” around the via to avoid a clearance violation. If you move a via against a second via, the second via is pushed to avoid a violation.

Next you will use Move mode to move wires and vias. You will see how Move mode works with *Push Routing* enabled and disabled.

### Task: Move wires and vias with Push Routing enabled and disabled

#### Procedure

1. Start SPECCTRA and load `lesson5e.dsn` from the `tutorial` directory.
2. Display the reference designators.
3. Zoom in on DS1, U4, U5, and on the via above U4.
4. Click the *Move* button on the tool bar.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

*Move* appears in the mode status area.

5. Click on the horizontal segment of wire that connects U4 and U5.

The pointer changes to an arrow (Windows) or a four-sided arrow (UNIX) to indicate you can move the wire.

6. Move the pointer up until the wire is stopped by the upper row of pins.

**Note:** The wire cannot move beyond the upper row of pins because the vertical segment creates a short circuit with the top left pin of U4.

7. Move the pointer down until the wire is stopped by the lower row of pins.

8. Press the right mouse button and choose *Cancel*.

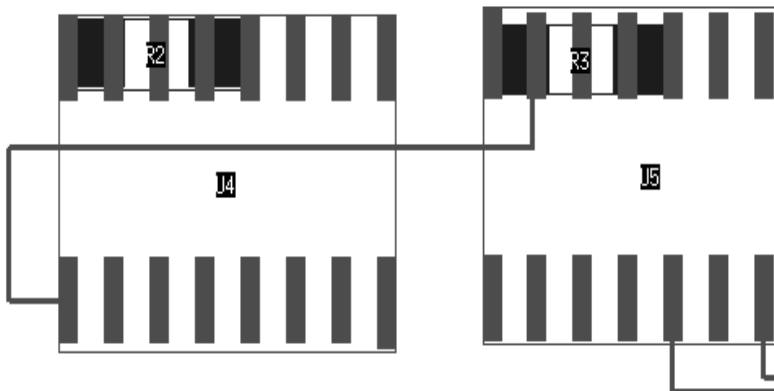
9. Click on the vertical segment of wire in U4.

Next you will move the vertical segment of the wire in U4 so that the horizontal of the wire can jog around the pins.

10. Move the pointer to the left and outside the outline of U4.

11. Click to drop the segment between DS1 and U4.

The following figure shows the wire path after you move the vertical segment to the left.



12. Click on the horizontal segment of the wire that connects U4 and U5.

13. Move the pointer up to the via above the upper row of pins in U4.

The segment now jogs over the upper row of U4 pins.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

14. Move the pointer back down until the wire is stopped by the lower row pins in U4.

Notice how the segment jogs around each pin in the row.

15. Press the right mouse button and choose *Cancel*.

16. Press the right mouse button and choose *Setup* from the Interactive Routing menu.

The Interactive Routing Setup dialog box appears.

17. Unselect *Push Routing* on the General tab to disable *Push Routing*.

18. Click *Apply*.

19. Drag the Interactive Routing Setup dialog box to the side of the screen and out of the way.

20. Click on the horizontal segment of the wire that connects U4 and U5.

21. Move the pointer up.

**Note:** The wire is stopped by the upper row of pins because *Push Routing* is disabled.

22. Press the right mouse button and notice that *Allow Jog Diagonal* is disabled.

23. Choose *Cancel*.

24. Select the *Push Routing* option in the Interactive Routing Setup dialog box on the General tab and click *OK* to enable *Push Routing*.

25. Click on the horizontal segment of the wire that connects U4 and U5 and move the pointer up to the via above U4.

The wire segment attached to the pointer pushes the via.

26. Press the right mouse button and choose *Cancel*.

27. Exit SPECCTRA.

You do not need to save a Session file or the Did file.

### Copying Wires

You copy wires by using Copy Route mode, which you set by clicking the right mouse button and choosing *Copy Route Mode* from the Interactive Routing Menu or by clicking the Copy Route button on the tool bar.

You set an editing mode by choosing the mode from the Interactive Routing Menu or by clicking a button on the tool bar.



## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

With Copy Route mode set, click on the wire or on a pin attached to the wire you want to copy. Then click on the target pin you want to route with the copied wire. The target connections must be unrouted and you must be able to wire target connections with the same wire path as the connection you copy. If the paths of the copied connection and the target connection are different, the copy function fails and the message *"Path cannot be copied"* appears in the Message area.

You will copy a wire path and click on an adjacent pin to replicate the path. You can also drag the pointer across several pins to replicate the wire path to multiple connections in a single operation.

#### **Task: Copy a wire path to several unrouted connections**

##### **Procedure**

1. Start SPECCTRA and load `lesson5f.dsn` from the `tutorial` directory.
2. Display the reference designators.
3. Zoom in on U4 and U5.
4. Click the *Copy Route* button on the tool bar.
5. Click on the wire that connects U4 and U5.

The wire is copied to memory and is highlighted.

6. Click on the bottom pin in U4 that is next to the pin that is highlighted.

The wire you copied in the previous step is replicated.

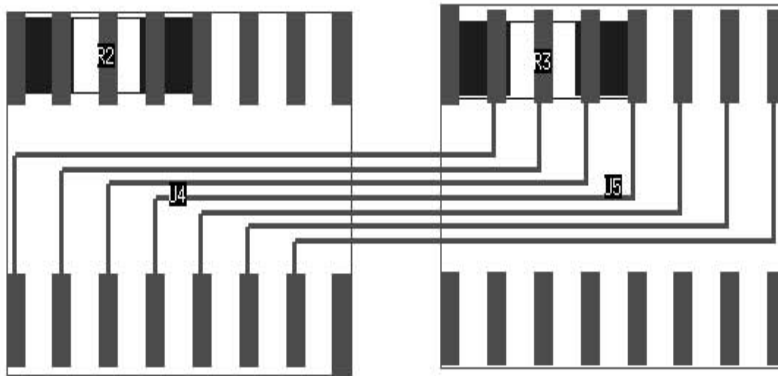
7. Drag the pointer across the five adjacent pins to the right.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

The original wire is copied to the five connections. The following figure shows the copy results.



#### 8. Exit SPECCTRA.

You do not need to save a Session file or the Did file.

### Removing Unnecessary Angles from Wires

You remove unnecessary angles from wires by using Critic Route mode. You set this mode by clicking the right mouse button and choosing *Critic Route Mode* from the Interactive Routing Menu or by clicking the *Critic Route* button on the tool bar. Critic Route mode removes unnecessary angles by moving existing wire segments.

Critic Route mode cannot remove unnecessary angles if one or more new wire segments are needed to perform a function or if the function creates a violation. You remove unnecessary angles from a single wire by clicking on the wire, or you can remove unnecessary angles from multiple wires by dragging the pointer over them.

In the following procedure, you remove the extra angles from a single wire by clicking on the wire. You remove the extra angles from multiple wires by dragging the pointer across them.

### Task: Remove unnecessary angles from wires

#### Procedure

1. Start SPECCTRA and load `lesson5g.dsn` from the `tutorial` directory.
2. Choose *File – Read – Wires*.

The Read Wires dialog box opens.

3. Click the *Browse* button.
4. Choose `lesson5g.w` from the tutorial directory.
5. Click *OK*.

The file `lesson5g.w` appears in the Read Wires data entry box.

6. Click *OK*.
7. Display the reference designators.
8. Zoom in on U4 and U5.
9. Click the *Critic Route* button on the tool bar.
10. Click on the wire attached to the lower left pin of U4.

The extra angles are removed from the wire.

11. Drag the pointer across all wires that connect between U4 and U5.

The extra angles are removed from all the wires in a single operation.

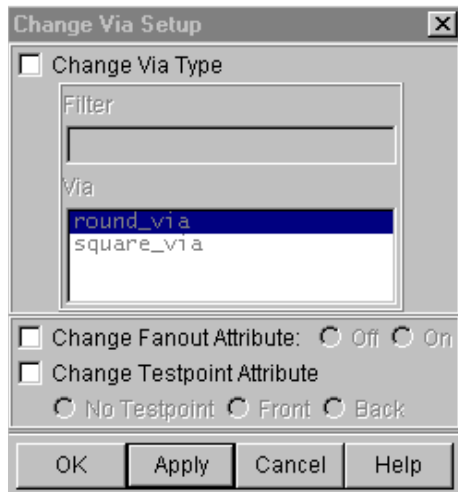
12. Exit SPECCTRA.

You do not need to save a Session file or the Did file.

### **Replacing Vias with Another Type**

You replace an existing via with a different type by using Change Via Mode. You set this mode by clicking the right button menu and choosing *Change – Change Via Mode*. When you set Change Via mode, a dialog box appears and you must choose which via you want to use to replace existing vias. The Change Via Setup dialog box, which is shown in the following figure, determines the replacement via.

**Figure 5-9 Change Via Setup Dialog Box**



After initially choosing a replacement via, you can choose a different replacement via by pressing the right mouse button and choosing *Setup Change Via*.

In the following procedure, you will set Change Via mode, choose a replacement via, replace several existing vias, and then choose a different replacement via.

**Task: Set Change Via mode, choose a replacement via, and replace existing vias**

**Procedure**

1. Start SPECCTRA and load `lesson5h.dsn` from the tutorial directory.
2. Choose *File – Read – Wires*.
3. Click the *Browse* button.
4. Choose `lesson5h.w` from the tutorial directory.
5. Click *OK*.  
The file `lesson5h.w` appears in the Read Wires data entry box.
6. Click *OK*.
7. Display the reference designators.
8. Zoom in on the wiring in the area between U4 and U6.
9. Press the right mouse button and choose *Change – Change Via Mode*.

The Change Via Setup dialog box opens.

10. Choose *square\_via* from the Via list.

11. Click *OK*.

*Change Via* appears in the mode status area.

12. Click on a via that is attached to one of the connections between U4 and U6.

A square via replaces the round via.

13. Press the right mouse button and choose *Setup Change Via*.

14. Choose *round\_via* from the Vias list.

15. Click *OK*.

16. Click the square via.

The square via changes to a round via.

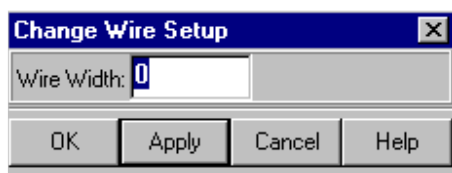
17. Exit SPECCTRA.

You do not need to save a Session file or the Did file.

### Changing the Width of an Existing Wire Segment

When you route a connection in SPECCTRA, whether you are autorouting or interactively routing, wire width is determined by a rule setting. You can change the width of an existing wire segment without setting a rule by using Change Wire mode. You choose this mode by clicking the right mouse button and choosing *Change – Change Wire Width Mode* from the Interactive Routing menu. The Change Wire Setup dialog box is initially displayed as shown in the following figure.

**Figure 5-10 Change Wire Setup Dialog Box**



After setting the Edit Wire Width value, you can set a different value by clicking right and choosing *Setup Change Wire* from the menu.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

Change Wire mode changes the width of a wire segment if the action does not create a violation. If it is determined that changing the width of a wire segment will result in a violation, the action is ignored.

**Note:** Because Change Wire mode does not set or change a wire width rule, changed widths are not retained if you reroute the segments. If you want to retain the new widths during an autorouting or rerouting operation, use the `protect` command to prevent the autorouter from altering the wires.

In the following procedure, you will set Change Wire mode and set the edit wire width. You will then protect the edited wires to prevent them from changing during subsequent autorouting or rerouting operations.

#### **Task: Set Change Wire mode, change the width and protect the wires**

##### **Procedure**

1. Start SPECCTRA and load `lesson5i.dsn` from the `tutorial` directory.
2. Display the reference designators.
3. Zoom in on U4 and U5.
4. Click the *Edit Route* button on the tool bar.
5. Route a short escape wire from the top left pin of U4.
6. Double-click at the end of the escape wire to add a via.
7. Press the right mouse button and choose *Done*.

A short escape wire and via is routed from the top left pin of U4.

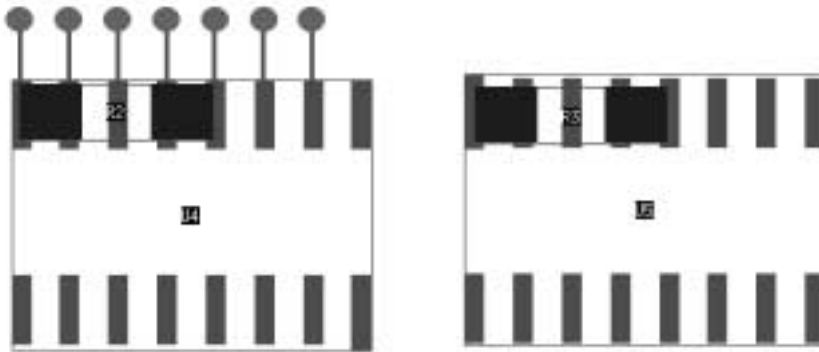
8. Repeat steps 6 and 7 to route the remaining six pins in the top row of U4.

## SPECCTRA Tutorial

### Lesson 5: Interactive Routing and Editing

---

The wiring result is shown in the following figure.



You will change the wire width of these segments to 10 mils.

9. Press the right mouse button and choose *Change – Change Wire Width Mode*.

The Change Wire Setup dialog box opens.

10. Type 10 in the *Wire Width* data entry box.
11. Click *OK*.
12. Click on each escape wire attached to U4.

The wire width changes to 10 mils.

13. Click the *Select Wire* button on the tool bar.
14. Drag the pointer across all seven escape wires.

The wires change from red to yellow, which is the select color.

15. Type **protect selected** in the command entry area.

A message popup prompts “*Protect All Selected Wires?*”

16. Click *Yes*.
17. Press the right mouse button and choose *UnSelect All Objects*.

The escape wires have a thin white line through their centers and the pins and vias have a small white circle to indicate that they are protected.

18. Exit SPECCTRA.

You do not need to save a Session file or the Did file.

## What You Learned

In this lesson, you learned how to:

- set up your interactive routing environment.
- use the routing tools.
- control the routing layer.
- add vias as you route.
- move wiring.
- copy wiring.
- eliminate extra angles from a wire.
- replace a via with another type.
- change the width of a wire segment.

## Congratulations!

You have completed the *SPECCTRA Tutorial*.

If you worked through each lesson in this tutorial, you learned how:

- the autorouter works.
- to use the autorouter's GUI.
- to place components interactively and automatically.
- set up the autorouter and analyze early results.
- set rules and control the autorouter.
- to route wires automatically.
- how to use the interactive routing tools.

With the information and procedures you learned, you are ready to begin your own routing projects.

When you are using SPECCTRA, remember there is a wealth of information available to you online. Browse the SPECCTRA Help menu and explore the manuals. SPECCTRA documentation is available on a broad range of topics from general information to step-by-step procedures for accomplishing your design tasks.



---

# Index

---

## A

- aligning components [73](#)
- angles, removing [162](#)
- autorouting
  - adaptive method [86](#)
  - analyzing unroutes [115](#)
  - costs [86](#), [92](#)
  - daisy-chain [130](#)
  - failures
    - analyzing [114](#)
    - definition of [111](#)
  - monitoring progress [111](#)
  - phases [91](#)
  - post-routing [98](#)
  - prerouting [91](#)
  - prioritizing nets [135](#)
  - selecting connections [125](#)
  - smart\_route [93](#)
  - starburst [130](#)
  - unroute, definition of [111](#)
  - using keepins [128](#)

## B

- bus command [87](#), [91](#)
- bus routing [87](#)

## C

- capacitors, placing [78](#)
- classes of nets [133](#)
- clean command [87](#), [92](#)
- clearance rules [122](#)
- command file [104](#)
- commands
  - autorouting [86](#)
  - bus [87](#), [91](#)
  - clean [87](#), [92](#)
  - fanout [87](#), [91](#)
  - fence [128](#)
  - in do file [104](#)
  - post-routing [98](#)
  - prerouting [91](#)

- protect [166](#)
- recorner [99](#)
- route [87](#), [92](#)
- smart\_route [86](#), [93](#)
- unselect [126](#)
- components
  - aligning [73](#)
  - displaying information about [39](#)
  - interchanging [71](#)
  - learning and applying patterns [80](#)
  - locking in place [65](#)
  - moving [79](#)
  - placing
    - at X,Y locations [57](#)
    - capacitors [78](#)
    - from a list [62](#)
    - large components [66](#)
  - rotating [63](#)
  - selecting [81](#)
- conflicts
  - analyzing [112](#), [114](#)
  - causes of [112](#)
  - poor reduction [114](#)
  - types of [86](#), [111](#)
- corners, changing to 45 degrees [99](#)

## D

- daisy-chain routing [130](#)
- design file
  - definition of [14](#)
  - sections of [17](#)
- displaying
  - power pin labels [78](#)
  - reference designators [57](#)
- do file [104](#)

## E

- editing
  - changing wire width [165](#)
  - copying wires and vias [160](#)
  - moving wires and vias [158](#)
  - removing unnecessary angles [162](#)

- replacing vias [163](#)
- rules did file [106](#), [110](#)
- editor
  - rules did file [106](#)
- escaping SMD pins [87](#), [91](#)
- exiting SPECCTRA [90](#)

## F

- fanout command [87](#), [91](#)
- fence command [128](#)
- files
  - command file [104](#)
  - design file [17](#)
  - do file [104](#)
  - for transferring designs [15](#)
  - placement [83](#)
  - session [83](#)
  - types of [14](#)
  - wires [143](#)
- fromtos
  - definition of [134](#)
  - selecting [126](#)
- function keys, using
  - for undo and redo [150](#)
  - to change layers [155](#)

## G

- graphical user interface
  - placement mode [54](#)
- grids
  - placement [54](#)
  - routing [93](#)
- groups of fromtos [134](#)
- GUI
  - placement mode [55](#)

## I

- images
  - definition of [53](#)
  - selecting [81](#)
- Interactive Place menu [58](#)
- interactive routing [138](#)
- interchanging components [71](#)

## K

- keepout areas [66](#)

## L

- layers
  - changing primary layer [152](#)
  - selecting for routing [123](#)
  - setting routing direction [123](#)
- Layers panel [78](#), [97](#), [153](#)
- license considerations [13](#)
- load pins, assigning [131](#)
- locking components [65](#)

## M

- moving
  - components [79](#)
  - wires and vias [158](#)

## N

- net classes
  - assigning routing priority to [135](#)
  - assigning rules to [134](#)
  - defining [133](#)
- nets
  - prioritizing for routing [135](#)
  - scheduling for routing [135](#)

## P

- pairwise interchange operation [71](#)
- pin sharing [92](#)
- pins, displaying information about [39](#)
- placement
  - grid [54](#)
  - keepout areas [66](#)
  - learning and applying patterns [80](#)
  - mode [54](#)
  - rules [53](#)
  - steps of [52](#)
  - violations [56](#)
- placement file
  - creating [83](#)

- definition of [41](#)
- loading [74](#)
- placing components
  - at X,Y locations [57](#)
  - capacitors [78](#)
  - critical components [57](#)
  - from a list [62](#)
  - large components [66](#)
  - small components [78](#)
- power pins, displaying [78](#)
- protect command [166](#)
- protecting wires [166](#)

## R

- recorner command [99](#)
- reference designators, displaying [57](#)
- reports, routing status [111](#)
- rotating components [63](#)
- route command [87](#), [92](#)
- routes file
  - definition of [15](#), [41](#)
- routing
  - assigning properties [131](#)
  - busses [91](#)
  - classes of nets [133](#)
  - conflict types [86](#)
  - daisy-chain [130](#)
  - failures
    - analyzing [114](#)
    - definition of [111](#)
  - in batch mode [104](#)
  - interactively [138](#)
  - keepin regions [128](#)
  - one or more connections [125](#)
  - preventing rerouting [166](#)
  - prioritizing nets [135](#)
  - restricting to area [128](#)
  - rules [100](#), [118](#)
  - running basic passes [88](#)
  - selecting layers [123](#)
  - selecting vias [124](#)
  - setting layer direction [123](#)
  - starburst [130](#)
- routing interactively
  - adding vias [152](#)
  - aligning with pins and vias [147](#)
  - finishing automatically [149](#)
  - pushing wires [158](#)
  - redoing an operation [150](#)
  - setting your environment [138](#)
  - undoing an operation [150](#)
- rules
  - assigning to
    - groups of fromtos [134](#)
    - net classes [134](#)
  - clearance [122](#)
  - hierarchy [53](#)
  - placement [53](#)
  - precedence of [53](#), [100](#)
  - routing [100](#), [118](#)
  - setting width and clearance [102](#)
  - setting wire and via clearance [121](#)
  - spacing [54](#)
  - width [119](#)
- rules did file [106](#), [110](#)
- rules did file editor
  - creating a do file [106](#)

## S

- saving
  - placement file [83](#)
- selecting
  - components [81](#)
  - connections for routing [125](#)
  - images [81](#)
  - layers for routing [123](#)
  - vias for routing [124](#)
- session file
  - creating [83](#)
  - definition of [15](#), [41](#)
- smart\_route command [86](#), [93](#)
- SMD escape [91](#)
- source pin, assigning [131](#)
- starburst routing [130](#)
- status report, routing [111](#)

## T

- terminator pin, assigning [131](#)
- transferring designs [15](#)

## U

- unroute, definition of [111](#)
- unselect command [126](#)
- unselecting

objects [81](#)  
vias for routing [124](#)

## V

### vias

- adding [152](#)
- changing [163](#)
- copying [160](#)
- displaying information about [40](#)
- moving [158](#)
- replacing [163](#)
- selecting for routing [124](#)
- setting clearance rules [122](#)
- sharing [92](#)
- unselecting for routing [124](#)
- using blind and buried [125](#)

## W

### wires

- changing width [165](#)
- copying [160](#)
- displaying information about [40](#)
- editing [146](#)
- moving [158](#)
- removing unnecessary angles [162](#)
- setting clearance rules [122](#)

wires file [15](#), [143](#)