

# Interface for SPECCTRA

---

# Copyrights

Software, documentation and related materials:

Copyright © 2002 Altium Limited

This software product is copyrighted and all rights are reserved. The distribution and sale of this product are intended for the use of the original purchaser only per the terms of the License Agreement.

This document may not, in whole or part, be copied, photocopied, reproduced, translated, reduced or transferred to any electronic medium or machine-readable form without prior consent in writing from Altium Limited.

U.S. Government use, duplication or disclosure is subject to RESTRICTED RIGHTS under applicable government regulations pertaining to trade secret, commercial computer software developed at private expense, including FAR 227-14 subparagraph (g)(3)(i), Alternative III and DFAR 252.227-7013 subparagraph (c)(1)(ii).

P-CAD is a registered trademark and P-CAD Schematic, P-CAD Relay, P-CAD PCB, P-CAD ProRoute, P-CAD QuickRoute, P-CAD InterRoute, P-CAD InterRoute Gold, P-CAD Library Manager, P-CAD Library Executive, P-CAD Document Toolbox, P-CAD InterPlace, P-CAD Parametric Constraint Solver, P-CAD Signal Integrity, P-CAD DesignFlow, P-CAD ViewCenter, Master Designer, and Associate Designer are trademarks of Altium Limited. SPECCTRA is a registered trademark of Cadence Design Systems, Inc. All other brand names are trademarks of their respective companies.

Altium Limited  
[www.altium.com](http://www.altium.com)

---

## P-CAD Interface for SPECCTRA®

This manual explains how to use the P-CAD SPECCTRA® autorouter (SP6, SP10, SP4, and SP2) interface. These autorouters work differently than other P-CAD autorouters.

For SPECCTRA, there are no interactive functions as with P-CAD PRO Route. SPECCTRA is driven by a command file called a DO file. You set up a DO file using the Route Autorouters dialog. When you start running SPECCTRA, P-CAD PCB runs the autorouter as a separate Windows process.

### Saving your Design

---

You must save your design in P-CAD ASCII format. P-CAD PCB then translates this format into the native SPECCTRA format. Upon termination of the autorouter, PCB merges the original PCB ASCII file with the routes produced by the autorouter. The resultant file is your routed design.

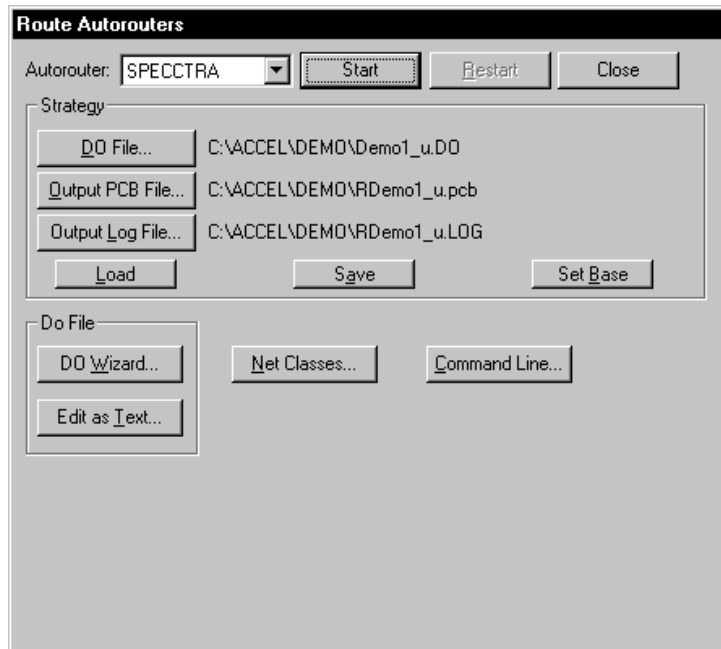
If you start running SPECCTRA and the design is not currently in P-CAD ASCII format, you are given the option to have P-CAD automatically save the design in ASCII format or abort the routing operation.

## Route Autorouters Dialog

---

When you select the SPECCTRA autorouter from the **Autorouters** combobox, the Route Autorouters dialog appears as follows:

*setting up the SPECCTRA autorouter*



This dialog is designed to simplify creating DO files, building net classes, and specifying command line options. SPECCTRA uses a different command file format called a DO file to define the routing strategy. The **Strategy** button in this dialog is replaced with a **DO File** button.

### DO File Button

The DO File is an ASCII file that contains SPECCTRA commands that execute in sequence to control autorouting. It includes all data needed by the autorouter to route the board. The DO filename initially appears as the same name as the current design file, but with an .DO extension. This is the default filename.

## Output PCB File Button

The **Output PCB File** button displays a dialog in which you can specify the name and location of your output design file after it has been routed. Here too, a default name has been provided. The letter R (for routed) precedes the current design filename. The last character is dropped if the new name exceeds eight characters. The default file extension is .PCB.

The default name can be overridden by clicking the **Output PCB File** button. The dialog which appears allows you to specify the name and location of your routed PCB board.

## Output Log File Button

In addition to the output file, the SPECCTRA autorouter generates a report file at the end of the routing session, detailing the results of the session. The **Output Log File** button displays the Select Output Log File dialog, in which you can specify the name and location of your report file.

## Load Button

Select the **Load** button to restore saved DO files. Choose the DO filename, then click **Load**. The DO file is then read into memory and may be edited as text or through the DO Wizard.

## Save Button

Anytime after you have selected a DO filename, you can save the file by clicking **Save**. The DO file is also saved automatically when you start the route.

## Set Base Button

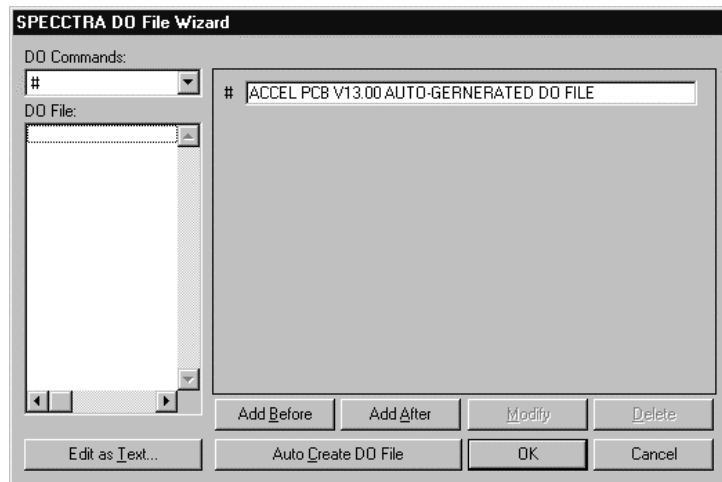
The **Set Base** button returns the DO and output files to their default filenames. This is a simple way to go back and start

over again when assigning filenames. The default names are derived from the design filename, including the full path.

## DO Wizard

The easiest way to create or manipulate a DO file is by clicking the **DO Wizard** button. The SPECCTRA DO File Wizard dialog appears. The SPECCTRA DO File Wizard dialog is an intelligent DO file editor that makes creating and modifying DO files more efficient.

*Setting up a do file*



From this dynamic dialog you can select DO commands, modify or delete commands from the DO file, or add new commands to the DO file. You enter information and add information to the DO file without having to edit DO file text directly. The DO commands chosen closely match those of other P-CAD autorouters. The DO Wizard creates commands with the correct syntax. Refer to your SPECCTRA User's Guide and Reference Manual for a complete discussion on DO file options and use.

The DO Wizard has an **Auto Create DO File** button that allows you to quickly create a DO file complete with current grid, line width and layer settings.

## Edit as Text

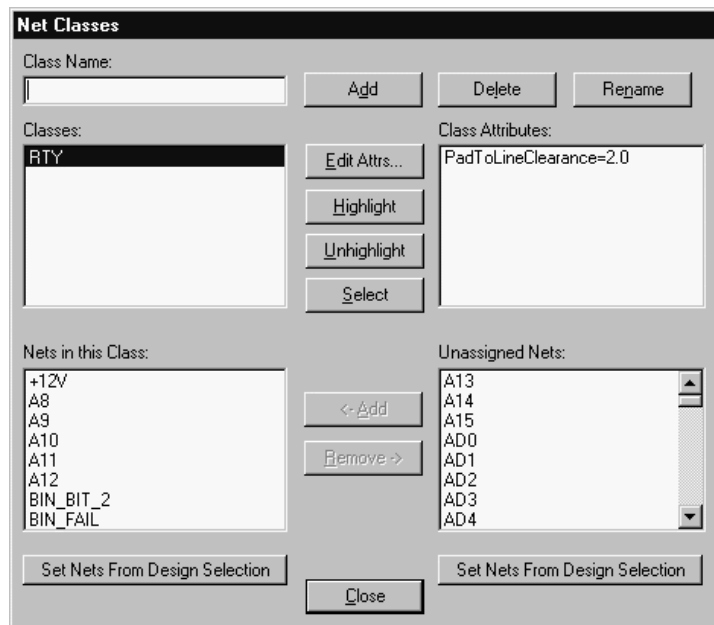
You can click the **Edit as Text** button to bring up the file in a text editor. The DO file created or modified in this manner is used verbatim as input to the SPECCTRA autorouter.

## Net Classes Button

The option lets you define a group of nets that share common rules. Collections of nets sharing the same rules are referred to as a *net class*.

When you click the **Net Classes** button, the Net Classes dialog appears.

*Defining net classes*



This class editor allows you to create named net classes using pre-defined clearance rules or pre-defined SPECCTRA autorouter clearance rules and then assign nets to that class. You can also add user-defined attributes to the net classes for your own use.

PCB Design Rules Checking verifies clearances and the attributes listed below when they have been defined in the net class:

- MaxNetLength
- MaxVias
- MinNetLength
- ViaStyle
- Width

For net clearances the rules can be further refined by specifying clearance rules for pairs of objects, like pad to pad clearances or line to via clearances.

To create named net classes:

1. Enter a class name in the **Classes** box.
2. Click **Add**.
3. To include a net from the **Unassigned Nets** area to the new net class you may use any of the following methods:
  - Select a single net and click the **Add** button.
  - Double click on a net to move it from **Unassigned** to **Nets in this Class** and vice versa.

- Select multiple nets in a block by either 1) holding the *Shift* key while selecting the first and last nets in the block or, 2) select the first net in the list, hold the left mouse button and drag the cursor to the last net in the list and release. Then click **Add**.
  - Select individual nets by holding the *Ctrl* key while clicking on each net, then click **Add**.
4. To remove a net from the **Nets in this Class** area, use the same methods detailed above, but click the **Remove** button instead of the **Add** button.

---

**note:** In addition to the normal selection process you may employ the **Set Nets From Design Selection** buttons to quickly place all currently selected nets in the design into either the **Unassigned Nets** or **Nets in this Class** areas. When no nets are selected in the design, the inactive **Set Nets From Design Selection** buttons are gray. If nets in either area are selected and you click the **Set Nets From Design Selection** button, the other selected nets become unselected.

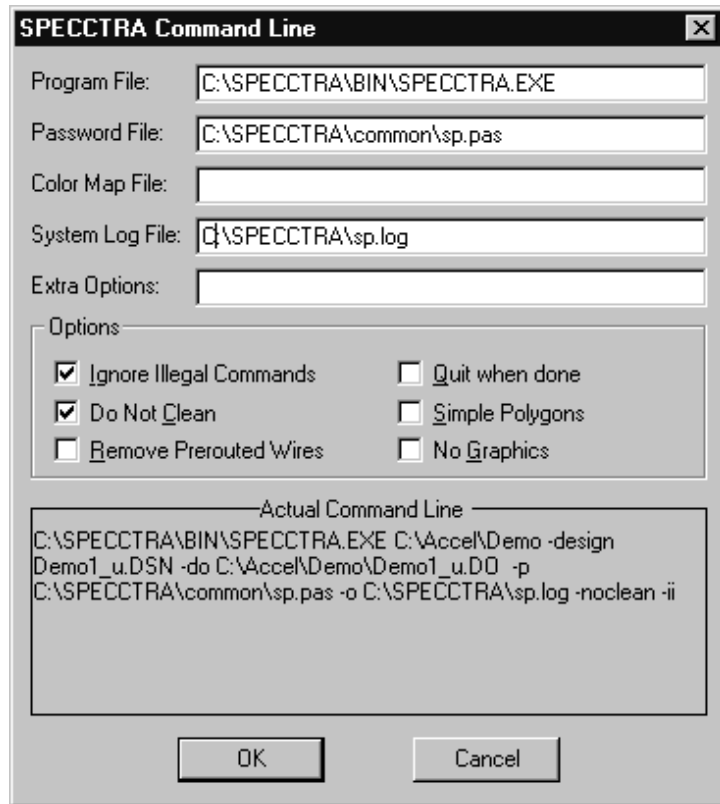
---

5. Use the **Edit Attributes** button to assign one or more attributes to this new net class.

## Command Line Button

When you click the **Command Line** button, the SPECCTRA Command Line dialog appears.

*Setting up SPECCTRA files*



The SPECCTRA program is run as a separate Windows process and must be launched with the correct command line arguments. The SPECCTRA Command Line dialog gives you control over the exact manner in which the autorouter is run.

The settings shown here are the recommended defaults. Note that DOS command lines are limited to a maximum of 128 characters. If that limit is exceeded, an error message is displayed in the DOS box or by Windows and the router is not started. The edited command line is saved for the next run. These checkbox options are described in the SPECCTRA manual.

# P-CAD Attributes with the SPECCTRA Autorouter

---

You can specify many SPECCTRA attributes directly in P-CAD PCB. Simply add the attribute to a net or net class. These attributes are transferred to SPECCTRA through the design file (.dsn) automatically when you click **Start** on the Autorouters dialog in PCB. It is recommended that you use the more intelligent, automated DO Wizard to transfer this same information through the .do file to SPECCTRA.

When adding multiple attributes to the same net, they should be added to the net in the order that SPECCTRA requires them to be added to a .do file.

For additional information about the route attributes recognized by the SPECCTRA autorouter, see the SPECCTRA User's Reference. It lists the SPECCTRA design format, numerical attribute value ranges, and available features. It also indicates which attributes are available in each router product.

## Router and Net Categories

The following general P-CAD Router and Net attributes are mapped to SPECCTRA net attributes:

MAXVIAS → LIMIT\_VIAS  
NOAUTOROUTE → Type Fixed  
RIPUP (if False) → Type Protect  
VIASTYLE → USE\_VIA  
WIDTH → WIDTH

## SPECCTRA Router Category

The extensive list of SPECCTRA Router attributes can be summarized into a few groups. This section includes a description of each group and an example of its use.

- **\_GAP attributes:** All attributes ending with the suffix `_gap` are clearance attributes. These attributes require a numerical value which is the clearance length specified in the current units.

PIN\_WIRE\_GAP=25 applies a pin to trace clearance of 25 between items on the selected net and those on another net.

- **\_PIN attributes:** For all attributes ending with the suffix `_pin`, a pin must be specified in the value box (e.g., `U1` or `U1 (pin3)`). Assign to through-hole components only

`EXPOSE_PIN=U1` specifies that any pin on `U1` which is attached to this net will be forced to escape to a via.

---

**note:** `TERMINATOR_PIN` and `SOURCE_PIN` must be used in conjunction with the `REORDER` attribute or with the `REORDER` or `ORDER` commands in the `.do` file.

---

- **Parallel and Tandem attributes:** The `PARAL_NOISE`, `PARAL_SEG`, `TANDEM_NOISE` and `TANDEM_SEG` attributes are specified as follows:

Toggle the `_CHECK` value ON (OFF). This activates (deactivates) checking of the `_THRESH` and `_WEIGHT` attributes. The threshold and weight are assigned an actual numerical value.

- **Other attributes:** For the miscellaneous remaining attributes, you can deduce their appropriate value using the attribute's `.do` file format specified. See your SPECCTRA documentation.

The input value in PCB is the text following the attribute name in the `.do` file minus the outer parentheses. See the following examples:

Attribute	SPECCTRA .DO File	P-CAD PCB Value
REORDER	rule net clk1 (reorder order_type daisy (type balanced))	order_type daisy (type balanced)
VIA_AT_SMD	rule class enable (via_at_smd on (grid on)(fit on))	on (grid on)(fit on)
USE_LAYER	circuit net1 (use_layer top bottom)	top bottom

## The Autorouting Process

---

If you switch back to P-CAD PCB during the route, you see a message telling you that the SPECCTRA autorouter is running and that P-CAD PCB has been suspended until the routing process is completed.

An autorouting menu and toolbar do *not* appear; they are not needed. P-CAD PCB simply waits for the SPECCTRA task to complete. P-CAD PCB knows when the autorouter is done and proceeds with the second part of the translation process.

As part of the autorouting process, P-CAD PCB passes split plane information (placement or routing) to SPECCTRA. This occurs automatically, so you don't need to execute any special commands.

While this feature is helpful for routing complex multilayer designs, SPECCTRA does not handle nested planes correctly in some cases. As an example, suppose you have two planes on a layer called PLANE. One plane, OUTER, is assigned to net NET1. Another plane, INNER, is assigned to net NET2 and resides within the boundaries of plane OUTER. If you have a surface mount device placed over the middle of plane INNER with a pad connected to net NET1, SPECCTRA fans out the via for this pad incorrectly. The router just places the surface mount device in the middle of plane INNER and drops a via through the plane, but does not connect the via to net NET1. In other words, SPECCTRA recognizes the INNER and OUTER planes, but does not properly connect the component on plane INNER to the net assigned to plane OUTER.

## Log File

---

The log file is a combination of output from the P-CAD-to-SPECCTRA translator, the SPECCTRA autorouter output, and the SPECCTRA-to-P-CAD translator. Postroute processing embeds the status file produced by the autorouter (and specified in the DO file) into the log file only if the System Log File name is provided in the command line.

Thus, the Route View Log command in PCB provides you with postroute information.

## Manual File Translation and Autorouting

---

The P-CAD interface to the SPECCTRA autorouter is fully automatic. You can also invoke the SPECCTRA autorouter outside of P-CAD PCB, for example to run it on an alternate machine.

The steps involved in manually autorouting a PCB design with SPECCTRA are as follows:

1. Create a DO file and set up net classes as described above.
2. Save the unrouted P-CAD PCB design in ASCII format (a .PCB extension is assumed).
3. Open a Windows DOS box and from the DOS prompt change to the P-CAD installation directory:  
`cd\installationdirectory`
4. Run the DOS version of the P-CAD-to-SPECCTRA translator using the following command line arguments:  
`PCAD2SP <DESIGN.PCB> -o <DESIGN.DSN>`
5. The output of step 4, <DESIGN.DSN>, is a SPECCTRA design file. Autoroute it as described in your SPECCTRA documentation, using the DO file created in step 1.
6. The routes output from SPECCTRA must now be merged into the original PCB design. Assuming the routes to be in a file named <DESIGN.RTE>, run the DOS version of the SPECCTRA-to-P-CAD translator using the following command line arguments:  
`SP2PCAD <DESIGN.RTE> -orig <DESIGN.PCB> -o R<DESIGN>.PCB`
7. If no command line arguments are given, the translators prompt for input.
8. The file R<DESIGN>.PCB is the routed design and may be loaded into P-CAD PCB for review.

# Troubleshooting

---

This section contains a description of some problems that you might encounter while using the P-CAD SPECCTRA interface. Solutions are provided for each problem.

## **Router starts but nothing is routed.**

Be sure you have a contiguous board outline in your design. Make sure that all of the lines that form your board outline form a closed polygon and that the endpoints are collocated at each vertex. Other extraneous lines on the board layer can cause routing problems, even if they are outside of the board outline. Use the assembly or drill layers for documentation, dimensions, etc.

## **The P-CAD Translator completes but the router does not start.**

Verify that the SPECCTRA executable name and path in your PCB.INI is correct. For the Windows version, the default name is SPECCTRA.EXE.

Verify that the password file specified in the router command line has the correct name and path. The default name for this file is SP.PAS.

## **Router routes through line keepouts.**

P-CAD line keepouts are not supported by the SPECCTRA autorouter. You must use the fence command in SPECCTRA to define areas to be fenced. Polygon keepouts are supported.

## **“Unable to write to port AUX” error is displayed when starting the autorouter.**

Make sure that a tmp directory exists on your hard disk and that you have the SPECCTRA logfile option specified on the command line.

**“Not Enough Virtual Memory...” message is written into the log file when starting the autorouter.**

Make sure you have your virtual memory for Windows set to be equal to or greater than the amount of physical memory in the system.

**Board does not route. The SPECCTRA Autorouter indicates RouteBase is not enabled.**

The design to be routed exceeds the limitations of the autorouter currently installed. For Route2, the design may have no more than 2 enabled signal layers; for Route4P, the design may have no more than 4 enabled signal layers and must have less than 4000 component pins.

**SPECCTRA router indicates a net has many pins and should be a signal or power net.**

This is an informational message from the autorouter and may be ignored. However, if the net is a power or ground net, you may define a plane in P-CAD for this net. You can also use one of the autorouter options for Nets Ordering to define the net as either a starburst or daisy chain.

**SPECCTRA layer colors do not match P-CAD layer colors.**

Consult your SPECCTRA autorouter documentation for an explanation of how to add a color file that will allow you to set SPECCTRA layer colors to match your P-CAD layer colors.

**The program requires ACTIVATOR to run.**

Verify that the SPECCTRA security key is attached to the port. If this is not the cause of the problem, make sure that the password file is not missing or incorrect.