cadence®

OrCAD X PCB Editor Menu Reference

Product Version 23.1
September 2023

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

Contents

2

Introduction

This document lists OrCAD® PCB Editor menu options available with the following products:

- OrCAD PCB Designer Standard
- OrCAD PCB Designer Professional
- OrCAD PCB Designer Professional w/PSpice

The OrCAD PCB Editor menu have been reorganized to show related commands together to enhance the efficiency and productivity of designers. The menus are arranged on the basis of by tasks and activity types.

To view the standard/legacy PCB Editor menu, set the environment variable `orcad_use_legacy_menu` in the *General* category of *UI* section in *User Preferences Editor*. You can also set this variable using *Use Legacy Menu* option available in the *Display* menu. To view legacy menu options, restart the application after setting the environment variable.

3

File Menu

The *File* menu provides shortcuts for the most frequently used commands. The following table describes the commands available in the *File* menu.

New Menu	Old Menu	Icon	Command Syntax	Description
New	File – New		new	Creates a new document based on the active document. Equivalent to the New command on the File menu.
Open	File – Open		open	Opens an existing document based on the active document. Equivalent to the Open command on the File menu.
Open Project	File – Open Project		open project	Opens a Capture project file (.opj).
Save	File – Save		save	Saves the active design with the current name while keeping the design displayed and active.

Save As	File – Save As		save as	Saves an active design under another name, to another drive, to another directory.
Update Layout	File – Update Layout		design sync	The design sync commands provide an interface to preview the changes between schematic and layout designs in real time before committing.
Update Schematic	File – Update Schematic		design sync	The design sync commands provide an interface to preview the changes between schematic and layout designs in real time before committing.
Viewlog	File – Viewlog		viewlog	Launches a viewer to view log files created by an automatic process.
File Viewer	File – File Viewer			Launches a file browser to find the files and directories.
Capture Canvas Image	File – Capture Canvas Image		capture image	Captures the screen shots of the selected parts of a design canvas and saves in Jpeg format.
Print Setup	File – Plot Setup		plot setup	Sets parameters for printing a design.
Print	File – Plot		plot	Prints the active design pages.
Properties	File – Properties		file property	Sets an optional password-protected database lock from the File Properties dialog box.
Change Editor	File – Change Editor		toolswap	Change the product type (tier) of the tool in which you are working, provided you are licensed for those tool sets.

Script	File – Script		script	Records a series of actions and creates a text file.
Recent Designs	File – Recent Designs			Opens one of the previously opened designs.
Exit	File – Exit		exit	Saves the active design and exits the editor.

4

Import Menu

The *Import* menu provides commands for importing data from other databases into layout. The following table describes the options in the *Import* menu.

New Menu	Sub Menu	Old Menu	Command Syntax	Description
Netlist		File – Import – Logic/Netlist	netin	Imports the logic of a design into its database and establish the operating characteristics for the netrev utility.
MCAD				
	DXF	File – Import – DXF	dxf in	Imports DXF data into a design.
	IDF	File – Import – IDF	idf in	Imports IDF data into a design.
	IDX	File – Import – IDX	idx in	Imports IDX data into a design.
Color/Board Parameters			param in	Imports a database parameter file (.prm) containing customized parameters from one design into another design for reuse.

Techfile		File – Import – Techfile	techfile_in	Imports a tech file into design.
Clipboard		File – Import – Sub-Drawing	clpcopy	Copies design elements to a clipboard file for pasting into other designs and drawings.
Placement		File – Import – Placement	plctxt_in	Places components in a new or existing design, using a text file that specifies component positions and orientations from another existing design.
Translators				
	PADS	File – Import – CAD Translators – PADS	pads_in	Imports PADS data into a design.
	PCAD	File – Import – CAD Translators – PCAD	pcad_in	Imports PCAD, PDIF, and PCB data into design.
	PADS Library		pads_lib_in	Imports PADS library files into symbol drawing databases.
	Altium PCB	File – Import – CAD Translators – Altium PCB		Converts PCB designs created in the Altium to OrCAD® PCB Editor designs.

	Altium Schematic to DE HDL	File – Import – CAD Translators – Altium Schematic to DE HDL		Converts PCB designs created in the Altium schematic to DE HDL.
	Eagle PCB	File – Import – CAD Translators – Eagle PCB		Converts PCB designs created in the Eagle Layout to OrCAD® PCB Editor designs.
More				
	Artwork	File – Import – Artwork	load gerber	Loads Gerber artwork files and creates the appropriate line and pad figure elements in the design database using FPOLYs rather than POLYs.
	IPF	File – Import – IPF	load plot	Displays an intermediate plot file before plotting.
	IPC-2581	File – Import – IPC 2581	ipc2581 in	Translates IPC-2582 data in a design.
	IFF	File – Import – IFF	iff in	Translates FF data in a design.
	Router	File – Import – Router	specctra in	Translates and imports data from a Allegro PCB Router to design file.
	Pin Delay	File – Import – Pin Delay	pin_delay in	Imports pin delay values from another design.

	Annotations	File – Import – Annotations	annotation in	Imports a text file containing the MANUFACTURING layer/ MARKUP subclass information from a design opened in a different version of the tool, for example an PCB Editor design opened in the Free Physical Viewer.

5

Setup Menu

The *Setup* menu provides commands for setting up design parameters.The following table describes the *Setup* menu options.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Application Mode					
	General Edit	Setup – Application Mode – General Edit		generaledit	Provides an environment to perform editing tasks, such as place and route, as well as moving, copying, or mirroring.
	Placement Edit	Setup – Application Mode – Placement Edit		placementedit	Provides an environment to perform tasks relevant during placement.
	Etch Edit	Setup – Application Mode – Etch Edit		etchedit	Provides an environment to perform etch-editing tasks.

	Signal Integrity	Setup – Application Mode – Signal Integrity		signalintegrity	Provides an environment to access signal integrity commands.
	Shape Edit	Setup – Application Mode – Shape Edit		shapeedit	Provides an environment to perform shape-editing commands.
	None	Setup – Application Mode – None		noappmode	Exits the current application mode and returns to a menu-driven editing mode.
Design Parameters		Setup – Design Parameters		prmed	Set design parameters that are saved and stored in the database.
Cross-Section		Setup – Cross-section		xsection	Provides layer-specific information. You can add and delete layers and change their values in a stackup.
Lead Editor		Setup – Lead Editor		lead_editor	Adds component lead contact area
DesignTrue DFM Wizard		Setup – Constraints – DesignTrue DFM Wizard		dt_wizard	Creates and assigns constraints for fabrication.

Constraint Modes		Setup – Constraint Modes			Enables design rule checks (DRCs), associated options, and custom measurements in a design.
Constraints		Setup – Constraints		cmgr	Opens Constraint Manager, to create and modify electrical, physical, and spacing constraints.
Add Differential Pairs		Logic – Assign Differential Pair		diff_pairs	Assigns pairs of nets to be routed as differential pairs. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.

Identify DC Nets		Logic – Identify DC Nets		identify_nets	Choose nets to carry a DC voltage in a design. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.
Dummy Net Assignments		Setup – Constraints – Dummy Net Assignments		cns_dummy_net	Assigns dummy nets to Net Classes in either Physical or Spacing domain.
Define B/B Vias		Setup – B/B Via Definitions – Define B/B Via		define_bbvia	Creates, edits, or deletes a blind or buried padstack in order to connect one layer to another.
Auto Define B/B Vias		Setup – B/B Via Definitions – Auto Define B/B Via		auto_define_bbvia	Creates multiple blind or buried vias between a range of etch/conductor layers in a design.
Colors		Display – Color/Visibility		color192	Launches *Color* dialog box with default settings.

Grids		Setup – Grids		define grid	Specifies the X and Y grid values for both etch and non-etch grids and for customizing the grid for each etch layer.
Change Origin		Setup – Change Drawing Origin		chg origin	Specifies an exact point on the canvas as the location for the drawing origin.
STEP Mapping		Setup – Step Package Mapping		step pkg map	Maps package and mechanical symbols to STEP models for more precise representation in 3d viewer
Zones	Zones – Create	Setup – Zones – Create		zone create	Creates special areas in the design with different stackups for rigid, flex, or stiffeners.
	Zones – Manage	Setup – Zones – Manage		zone manager	Manages zones.
Bend	Bend – Create	Setup – Bend – Create		bend area create	Creates area for flex part of the design.
	Bend –Edit	Setup – Bend – Edit		bend area edit	Modify bend area specifications.

Anchor 3D View		Setup – Anchor 3D View		anchor 3d view	Specifies an anchor point to define an area which is not affected bu the bending operations in 3D Canvas.
Datatip Customization		Setup – Datatip Customization		custom datatips	Customizes a context-sensitive datatip that identifies an element.
User Preferences		Setup – User Preferences		enved	Set or unset environment variables (preferences) from a graphical user interface.
More					
	Subclasses	Setup – Subclasses		define subclass	Adds subclasses to those classes that allow user-defined subclasses.
	Materials	Setup – Materials		define materials	Adds, delete and edit the materials used in the layout cross-section.
	Enable On-Line DRC	Setup – Enable On-Line DRC		cns onlinedrc	Sets the on-line DRC on or off in a design.
	Enable Datatips	Setup – Enable Datatips		datatips toggle	Sets the display of datatips.

	Property Definitions	Logic – Property Definitions		define property	Creates and edits property definitions (user-defined properties) in a design.
	IPC2581 Spec Definitions			define ipc spec	Creates IPC2581 spec definitions
	Define Lists	Setup – Define Lists		define list	Creates a list of net names, reference designators, or function designators in a design and save it as a text file.
	SI Design Setup	Setup – SI Design Setup		signal setup	Sets parameters to perform SI simulations in a design.
	SI Design Audit	Setup – SI Design Audit		signal audit	Runs an audit on all or selected nets in a design.
	Customize Toolbar	View – Customize Toolbar		Toolbar Customization	Customize the look of the toolbar on user interface.

6

Display Menu

The *Display* menu offers a quick and easy way to view the design. The following table describes the options in the *Display* top menu.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Zoom					
	In	View – Zoom In		zoom in	Magnifies the design view by a factor of two.
	Out	View – Zoom Out		zoom out	Reduces the magnification of design view by a factor of two.
	Window	View – Zoom By Points		zoom points	Defines an area of design to magnify.
	Fit	View – Zoom Fit		zoom fit	Fits the entire board in the design window.
	World	View – Zoom World		zoom world	Reduces the magnification of a design to view the entire drawing.

	Center	View – Zoom Center		zoom_center	Moves the selected point in the design to the center of the design window.
	Previous	View – Zoom Previous		zoom_previous	Zooms back from the current window extents to the prior view.
View					
	Color View Save	View – Color View Save		colorview_create	Creates or changes a color visibility view.
	Color View Load	View – Color View Load		colorview_load	Loads a specified color visibility view.
	Color View Restore Last	View – Color View Restore Last		colorview_restore	Restores the previous color visibility view used in the current session.
	Refresh	View – Refresh		redraw	Refreshes the work area.
	Split View – Float Split View – Horizontal Split View – Vertical	View – Split View – Float View – Split View – Horizontal View – Split View – Vertical		showhide_views1	Opens a second work area independent of main design window.
	Swap Views	View – Swap Views		zoom_swap_views	Swaps views between main canvas and split view.

3D Canvas		View – 3D Canvas		3d	Launches the 3D Canvas to view and analyze a three-dimensional model of a design.
Flip Design		View – Flip Design		flipdesign	Flips the design along the Y-axis on the design canvas.
Assign Colors		Display – Assign Color		assign_color	Assigns a color and highlights an element without requiring the use of the Color dialog box.
Deassign Color		Display – Deassign Color		deassign_color	The deassign color command removes the color assigned to an element without requiring the use of the Color dialog box.
Highlight		Display – Highlight		hilight	Highlights the selected design elements using settings in the Color dialog box.
Dehighlight		Display – Dehighlight		dehilight	Removes the highlighting pattern from design elements.
Segments Over Voids	Highlight	Display – Segments Over Voids		highlight_sov	Highlights segments of nets that overlaps voids.

	Dehighlight	Display – Segments Over Voids		highlight_sov_clear	Removes the highlighting pattern from the segments of nets that overlaps voids.
Layer Priority		Display – Layer Priority		layer_priority	Assigns a display priority to each layer, and overriding the default display order.
Show Rats					
	All	Display – Show Rats – All		rats_all	Displays existing ratsnest lines in a design.
	Components	Display – Show Rats – Components		rats_component	Displays existing ratsnest lines attached to component pins.
	Nets	Display – Show Rats – Net		rats_net	Displays existing ratsnest lines attached to pins on a net.
	Of Selection	Display – Show Rats – Of Selection		rats_show	Displays the rats associated with one or more selected objects.
	End In View Only	Display – Show Rats – End In View Only		rats_end_inview	Reduces the density of the rats display.
Blank Rats					

	All	Display – Blank Rats – All		unrats all	Hides all ratsnest lines in a design.
	Components	Display – Blank Rats – Components		unrats component	Hides visible ratsnest lines to pins on an individual component or a group of components in a design.
	Nets	Display – Blank Rats – Nets		unrats net	Hides visible ratsnest lines to pins on an individual net or a group of nets in a design.
	Of Selection	Display – Blank Rats – Of Selection		rats blank	Hides the rat display of one or more selected objects associated with the route plan.
Windows					
	Command	View – Windows – Command		showhide text	Toggles the visibility of the Command window pane.
	World View	View – Windows – World View		showhide view	Toggles the visibility of the Worldview window pane.
	Options	View – Windows – Options		showhide options	Toggles the visibility of the Options window pane.
	Find	View – Windows – Find		showhide find	Toggles the visibility of the Find window pane.

	Visibility	View – Windows – Visibility		showhide vis	Toggles the visibility of the Visibility window pane.
	Design Workflow	View – Windows – Design Workflow		showhide_workflow	Toggles the visibility of the Design Workflow window pane.
	Comments			showhide comment	The showhide comment command toggles the visibility of the comment panel.
	Show All	View – Windows – Show All		show allpanes	Restores the last-viewed positions of Options, Worldview, Find, Visibility, and Command foldable window panes.
Vision Manager		View – Vision Manager		vision manager	
UI Settings	Reset UI To All Toolbars	View – UI Settings – Reset UI To All Toolbars			Restores the original positions and display of all toolbars.
	Reset UI To Default	View – UI Settings – Reset UI To Cadence Default		reset dockwindows	Restores the original positions of Options, Worldview, Find, Visibility, and Command foldable window panes.

	Manage Settings	View – UI Settings – Manage Settings		manage settings	Saves and manages pre-defined toolbars and dock panes settings of the layout editor.
	Save Settings	View – UI Settings – Save Settings		save settings	Saves the currently active UI settings of the layout editor with a new name.
Use Legacy Menu (Requires Restart)				enved	Sets or unsets environment variables.

7

Outline Menu

The *Outline* menu offers a quick and easy way to perform common tasks. The following table describes the menu options in the *Outline* top menu.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Design		Setup – Outlines – Design Outline		board outline	Creates, modify, move, or delete a board outline.
Room		Setup – Outlines – Room Outline		room outline	Creates, add name and specify the board layer on which the room is added.
Keepout		Setup – Outlines – Keepout		board keepout	Defines keepout areas to isolate sections within the board outline where component placement is not allowed.
Plane		Setup – Outlines – Plane Outline		board plane	Creates, modify, move, or delete a plane outline.
Package Keepin		Setup – Areas – Package Keepin		keepin package	Adds a package/part keepin area in a design.

Package Keepout		Setup – Areas – Package Keepout		keepout_package	Adds filled package/part keepout areas in a design.
Package Height		Setup – Areas – Package Height		package_height	Attaches properties defining a height restriction to a package/part keepout or place bound rectangle.
Route Keepin		Setup – Areas – Route Keepin		keepin_router	Adds route keepin areas in a design.
Route Keepout		Setup – Areas – Route Keepout		keepout_router	Adds filled route (etch/conductor) keepout areas in a design.
Wire Keepout		Setup – Areas – Wire Keepout		keepout_wire	Adds filled route (etch/conductor) keepout areas in a design.
Via Keepout		Setup – Areas – Via Keepout		keepout_via	Adds via keepout placement areas in a design.
Shape Keepout		Setup – Areas – Shape Keepout		keepout_shape	Adds etch/conductor shape keepout areas in a design.
Z-Copy		Edit – Z-Copy		zcopy_shape	Copy a shape, closed polygon, line, cline, or rectangle and add it to a different class/subclass at the same location in a design.

More					
	Probe Keepout	Setup – Areas – Probe Keepout		keepout probe	Adds probe keepout areas in a design.
	Gloss Keepout	Setup – Areas – Gloss Keepout		keepout gloss	Adds gloss keepout areas in a design.
	Photoplot Outline	Setup – Areas – Photoplot Outline		keepin photo	Creates a photoplot outline that defines the limits of an artwork photoplot data file.

8

Add Menu

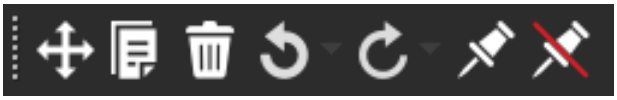
The *Add* menu offers a quick and easy way to perform common tasks in PCB Editor. The following table describes the menu in the *Add* top menu.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Line		Add – Line		add line	Creates non-conductor line segments between two points.
Arc w/ Radius		Add – Arc w/ Radius		add rarc	Creates an arc-shaped element of known radius.
Arc 3pt		Add – Arc 3 pt		add arc	Creates an arc-shaped element of unknown radius.
Circle		Add – Circle		add circle	Creates circles in a design.
Rectangle		Add – Rectangle		add rect	Creates rectangles in a design.

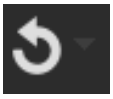
Frectangle		Add – Frectangle		add frect	Creates filled rectangles in a design.
Text		Add – Text		add text	Creates free-form text in a design.
Split Plane Parameters		Edit – Split Plane – Parameters		split plane param	Sets parameters for split planes.
Split Plane		Edit – Split Plane – Create		split plane create	Creates split planes on an ETCH/ CONDUCTOR subclass.
Groups		Edit – Groups		groupedit	Creates groups by randomly selecting database objects and can be referenced as a single object.
Unsupported Prototypes					
	Arc	Add – Unsupported Prototypes - Arc		add frac	
	Help on unsupported utilities	Add – Unsupported Prototypes - Help on unsupported utilities			

9

Edit Menu



The *Edit* menu provides shortcuts for many of the most frequently used editing commands. The following table describes the menu options in the *Edit* top menu.

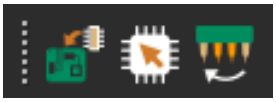
Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Undo		Edit – Undo		undo	Undoes the last command performed.
Redo		Edit – Redo		redo	Redoes the last command performed.
Change Objects		Edit – Change		change	Changes line width, text size and justification, or the subclass to which an element is assigned.
Object Properties		Edit – Properties		property edit	Assigns properties to design elements, or changes or deletes existing property values.
IPC2581 Specs		Edit – IPC2581 Specs		ipc spec edit	Assigns IPC2581 spec to design elements

Net Properties		Edit – Net Properties		net_properties	Launches the Constraint Manager and displays net properties worksheet.
Move		Edit – Move		move	Relocates the position of elements in a design.
Copy		Edit – Copy		copy	Creates copies of elements in a design.
Paste		Edit – Paste		paste	Paste copies of elements at multiple destinations in a design.
Mirror		Edit – Mirror		mirror	Creates mirror image of an element (or a group of elements) around the Y-axis.
Rotate		Edit – Spin		spin	Rotates a graphic element around a selected point.
Delete		Edit – Delete		delete	Removes physical elements from a design without modifying the netlist.
Text		Edit – Text		text_edit	Modifies a text string in a design.
Split Plane		Edit – Split Plane – Create		split_plane_create	Creates split planes on an ETCH/CONDUCTOR subclass.

Net Schedule		Logic – Net Schedule		net_schedule	Interactively schedule or unschedule the order in which pins are routed in a particular net. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.
More					
	Vertex	Edit – Vertex		vertex	Inserts corners into existing connect lines, shape and void boundaries.
	Delete Vertex	Edit – Delete Vertex		delete_vertex	Deletes vertices from cline lines, shape and void boundaries.
	Groups	Edit – Groups		groupedit	Creates groups by randomly selecting database objects and can be referenced as a single object.

10

Place Menu



The *Place* menu provides commands for placing components in a design. The following table describes the commands available in *Place* menu.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Components Manually		Place – Manually		place manual	Places components in a design.
Quickplace		Place – Quickplace		quickplace	Quickly places all the components in a design by placing symbols outside the board outline.
Mechanical Symbols		Place – Manually		place manual	Places mechanical symbols in a design.
Drawing Symbols		Place – Manually		place manual	Places format symbols in a design.
Autoplace					

	Parameters	Place – Autoplace – Parameters		place_param	Sets parameters for automatic placement and runs the command.
	Top Grids	Place – Autoplace – Top Grids		place_set_topgrid	Generates grids on BOARD GEOMETRY/ SUBSTRATE GEOMETRY class, PLACE_GRID_TOP subclass for placing symbols.
	Bottom Grids	Place – Autoplace – Bottom Grids		place_set_bottomgrid	Generates grids on BOARD GEOMETRY/ SUBSTRATE GEOMETRY class, PLACE_GRID_BOTTOM subclass for placing symbols.
	Design	Place – Autoplace – Design		place_area_design	Sets automatic placement mode and place area as package/part keepin for running automatic placement in interactive mode.
	Room	Place – Autoplace – Room		place_area_room	Defines a room for automatic placement of components in a package/part keepin.

	Window	Place – Autoplace – Window		place area window	Defines a window for automatic placement of components in a package/part keepin.
	List	Place – Autoplace – List		place area list	Displays current active area of a design for automatic placement.
Interactive		Place – Interactive		place interactive	Performs automatic placement in interactive mode.
FSP				place fsp	Loads an XML file, containing placement information for all components in the FSP, to place or re-place the analogous components in the board design.
Swap					
	Pins	Place – Swap – Pins		swap pins	Swap pins when they are defined in the same PINSWAP statement of a device file.
	Functions	Place – Swap – Functions		swap functions	Swaps functions or gates in a design window.

	Components	Place – Swap – Components		swap components	Swaps components in a design window.
Autoswap					
	Parameters	Place – Autoswap – Parameters		swap param	Set parameters for automatic swapping and runs the command.
	Design	Place – Autoswap – Design		swap area design	Defines the package/part keepin as the automatic swapping area.
	Room	Place – Autoswap – Room		swap area room	Add names to rooms in a design as an area for automatic swapping.
	Window	Place – Autoswap – Window		swap area window	Defines areas in a design for swapping.
	List	Place – Autoswap – List		swap area list	Displays current active area of a design for automatic swapping.

Via Arrays		Place – Via Array		via array	This set of commands are available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.
Assign RefDes		Logic – Assign RefDes		assign refdes	Assigns reference designators to package symbols.
Update Symbols		Place – Update Symbols		refresh symbol	Replaces new flash symbols in a database with new versions from the disk.

11

Route Menu

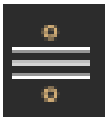
The *Route* menu offers commands for routing. The following table describes the options available in the *Route* top menu.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Connect		Route – Connect		add connect	Routes interactively a single connection as well as differential pairs.
Slide		Route – Slide		slide	Moves cline segments on single nets, differential pairs, or a group of routed connections.
Custom Smooth		Route – Custom Smooth		custom smooth	Smooth out selected clines or cline segments. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.

Delay Tune		Route – Delay Tune		delay tune	Elongates nets that are not meeting minimum timing or length constraints. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.
Phase Tune		Route – Phase Tune		phase tune	Adds phase bumps to either member of the differential pair to eliminate any existing phase-tune DRCs. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.
Create Fanout		Route – Create Fanout		create fanout	Creates clines and vias and connects them to the selected pins or symbols.
Copy Fanout		Route – Copy Fanout		copy fanout	Duplicates a fanout pattern from one component to all other instances of that symbol definition.
Convert Fanout	Mark	Route – Convert Fanout – Mark		mark fanout	Associates clines and vias with their respective component symbol instances.

	Unmark	Route – Convert Fanout – Unmark		unmark fanout	Disassociates clines and vias from their respective component symbol instances.
Structure					
	Create	Route – Structure – Create		create structure	Combine patterns of vias and connect lines (clines) into a single design element called a via structure symbol.
	Place	Route – Structure – Place		place structure	Places via structures in a design.
	Replace	Route – Structure – Replace		replace via structure	Replaces some or all instances of an existing via structure with a new via structure.
	Replace Via with Structure	Route – Structure – Replace Via with Via Structure		replace via with structure	Replaces some or all instances of an existing via with a via structure.
	Refresh	Route – Structure – Refresh		refresh via structure	Updates the via structures in a design to agree with the current library definitions of those via structures.
	Redefine	Route – Structure – Redefine		redefine via structure	Selects a via structure and updates definition of all placed instances to match the selected via structure.

	Disband	Route – Structure – Disband		disband_via_structure	Converts via structures to their individual components.
	Export All	Route – Structure – Export All		export_all_structure	Export all structures to a directory.
PCB Router					This set of commands are available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.
	Fanout By Pick	Route – PCB Router – Fanout By Pick		fanout_by_pick	Routes short pin escape wires from pins to vias
	Route Net(s) By Pick	Route – PCB Router – Route Net(s) By Pick		route_by_pick	Routes specific nets and components in a design rather than the entire database.
	Miter By Pick	Route – PCB Router – Miter By Pick		miter_by_pick	Changes 90-degree wire corners to 45 degrees for wires exiting pins and vias.
	Unmiter By Pick	Route – PCB Router – Unmiter By Pick		unmiter_by_pick	Removes 45-degree wire corners and changes them to 90-degree corners.
	Elongation By Pick	Route – PCB Router – Elongation By Pick		elong_by_pick	Increases etch length, usually in inches or mils, to adhere to timing rules.

	Router Checks	Route – PCB Router – Router Checks		specctra_checks	Run router and alignment checks on a current design to identify routing problems prior to running PCB Router.
	Optimize Rat Ts	Route – PCB Router – Optimize Rat Ts		optimize_ts	Optimizes the location of Tpoints in a design.
	Route Automatic	Route – PCB Router – Route Automatic		auto_route	Performs automatic routing for designs that do not require interactive routing.
	Route Custom	Route – PCB Router – Route Custom		custom_route	Categorizes and writes several individual rules files based on the characteristics of a design.
	Route Editor	Route – PCB Router – Route Editor		specctra	Launches the PCB Router.
Resize/ Respace					
	Spread Between Voids	Route – Resize/ Respace – Spread Between Voids		spread_between_voids	Spreads the clines in a specified routing channel.
	Via-Via Line Fattening	Route – Resize/ Respace – Via-Via Line Fattening		line_fattening	Eliminates potential acid traps.

Teardrop/ Tapered Trace					This set of commands are available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.
	Parameters	Route – Teardrop/ Tapered Trace – Parameters		`gloss param fillet`	Access parameters for the glossing applications.
	Add Teardrop	Route – Teardrop/ Tapered Trace – Add Teardrop		`add fillet`	Generates teardrop shapes in a design.
	Delete Teardrop	Route – Teardrop/ Tapered Trace – Delete Teardrop		`delete fillet`	Removes shapes that are designated as teardrops.
	Add Tapered Trace	Route – Teardrop/ Tapered Trace – Add Tapered Trace		`add taper`	Generates teardrops at the junction of two clines of different width.
	Delete Tapered Trace	Route – Teardrop/ Tapered Trace – Delete Tapered Trace		`delete taper`	Removes tapers in a design.

Gloss					This set of commands are available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.
	Parameters	Route – Gloss – Parameters		gloss_param	Access parameters for the glossing applications.
	Design	Route – Gloss – Design		gloss_area_design	Selects an area defined by the route keepin to gloss.
	Room	Route – Gloss – Room		gloss_area_room	Defines a room to gloss.
	Window	Route – Gloss – Window		gloss_area_window	Defines an area to gloss.
	Highlighted	Route – Gloss – Highlighlet		gloss_area_highlight	Selects individual nets or components for glossing.
	List	Route – Gloss – List		gloss_area_list	Displays area selected for automatic glossing.

12

Shape Menu

The *Shape* menu offers a quick and easy ways to create and edit shapes on the canvas. The following table describes the icons on the toolbar.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Polygon		Shape – Polygon		shape add	Adds a multi-sided enclosed polygon and creates a static, dynamic, unfilled, or cross-hatched shape, which may be used for a placebound, route keepout, or a board outline.
Rectangular		Shape – Rectangular		shape add rect	Adds a rectangular shape.
Circular		Shape – Circular		shape add circle	Adds a circular shape.
Select Shape or Isolation/ Cavity		Shape – Select Shape or Void/ Cavity		shape select	Selects a shape, void or filled rectangle for editing or changing parameters at the shape instance level.

Manual Isolation/ Cavity					
	Polygon	Shape – Manual Void/Cavity – Polygon		shape_void_polygon	Creates a non-copper polygon within the copper area.
	Rectangular	Shape – Manual Void/Cavity – Rectangular		shape_void_rectangle	Creates a non-copper rectangle within the copper area.
	Circular	Shape – Manual Void/Cavity – Circular		shape_void_circle	Create a circular element within an etch/conductor shape that is recognized as unfilled during printing and photoplotting.
	Delete	Shape – Manual Void/Cavity – Delete		shape_void_delete	Deletes selected voids in an active shape.
	Element	Shape – Manual Void/Cavity – Element		shape_void_element	Creates automatically an unfilled clearance hole for static (manual) shapes by selecting a pin or via.
	Move	Shape – Manual Void/Cavity – Move		shape_void_move	Moves selected void in an active shape.

	Copy	Shape – Manual Void/Cavity – Copy		shape void copy	Copies a user-defined void selected in an active shape.
Edit Boundary		Shape – Edit Boundary		shape edit boundary	Redefines the boundary of the copper area shape or its voids.
Delete Unconnected Copper		Shape – Delete Islands		island delete	Removes islands, which are non-conductive isolated areas of copper.
Merge Shapes		Shape – Merge Shapes		shape merge shapes	Merges overlapped shapes, as well as filled rectangles.
Freeze Shape(s)		Shape – Freeze Shape(s)		shape freeze	
Unfreeze Shape(s)		Shape – Unfreeze Shape(s)		shape unfreeze	
Create Bounding Shape		Shape – Create Bounding Shape		create bounding shape	Creates shapes around the boundary of the selected objects on the specified layers.
Shape Operation	OR	Shape – Shape Operation – OR		shape operations or	Performs logical OR operation on selected shapes and objects (shape or cline).
Shape Operation	AND	Shape – Shape Operation – AND		shape operations and	Performs logical AND operation on selected shapes and objects (shape or cline).

Shape Operation	ANDNOT	Shape – Shape Operation – ANDNOT		shape_operations_andnot	Performs logical ANDNOT operation on overlapping shapes.
Shape Operation	XOR	Shape – Shape Operation – XOR		shape_operations_xor	Performs logical XOR operation on overlapping shapes.
Change Shape Type		Shape – Change Shape Type		shape_change_type	Changes shape fill type from Static Solid to Dynamic Copper or visa versa.
Z-Copy		Edit – Z-Copy		zcopy_shape	Copies a shape, closed polygon, line, cline, or rectangle and adds it to a different class/ subclass at the same location in a design.
Check Shapes		Shape – Check		shape_check	Identifies small or narrow areas that might cause problems during artwork generation.
Create Shape From Lines		Shape – Compose Shape		compose_shape	Converts a group of lines and arcs into a shape.
Create Lines From Shape		Shape – Decompose Shape		decompose_shape	Disconnects lines and arcs connected as a shape.
Global Dynamic Parameters		Shape – Global Dynamic Params		shape_global_param	Sets shape outline parameters to all dynamic copper fill shapes.

Layer Dynamic Parameters		Shape – Layer Dynamic Params		shape_layer_param	The shape layer param command displays the Global Shape Layer Parameters dialog box from which you can apply shape outline parameters to all dynamic copper fill shapes in a specified layer.

13

Check Menu

The *Check* menu provides commands to verify the design integrity. The following table describes the options available in the *Check* menu.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Design Status		Display – Status		status	Verifies the current state of shapes and DRCs in a design.
Elements		Display – Element		show element	Lists the attributes of a graphic element.
Measure		Display – Measure		show measure	Calculates and displays the distance between two user-defined points in a design.
Properties		Display – Property		show property	Displays the properties in a current design in the *Show Property* dialog box.
Constraints		Display – Constraint		cns show	Display constraints information for a selected object or pair of objects.
DRC Update		Tools – Update DRC		drc update	Deletes all DRC markers in a design and re-compute DRC for all constraints that have a DRC mode of either Always or Batch.

DRC Window		Tools – Window DRC		drc window	Deletes all DRC markers in a selected area and re-compute DRC for all constraints that have a DRC mode of either Always or Batch.
Waive DRCs					
	Waive	Display – Waive DRCs – Waive		waive drc	Sets aside design rule violations to meet design requirements.
	Show	Display – Waive DRCs – Show		show waived drcs	Displays all waived DRC error markers in a design.
	Blank	Display – Waive DRCs – Blank		blank waived drcs	Suppresses waived DRC error markers from displaying in a design.
	Restore	Display – Waive DRCs – Restore		restore waived drc	Returns a waived DRC error to active status.
	Restore All	Display – Waive DRCs – Restore All		restore waived DRC errors	Returns all waived DRC errors to active status.
Database Check		Tools – Database Check		dbdoctor	Analyzes a design and fix database problems.

14

Analyze Menu

The *Analyze* menu offers commands to assign and edit models on symbols. The following table describes the icons on the toolbar.

Menu	Old Menu	Icon	Command Syntax	Description
Model Browser	Analyze – Model Browser		`signal library`	Use to perform signal model development tasks.
Model Assignments	Analyze – Model Assignments		`signal model`	Use to assign models to devices and pin.
Model Dump/ Refresh	Analyze – Model Dump/ Refresh		`signal model refresh`	Use to perform verification and source management operations on the device models in a design or library.
Preferences	Analyze – Preferences		`signal prefs`	Use to specify the device and interconnect libraries used by the simulator during signal analysis.
Workflow Manager	Analyze – Workflow Manager			Use to select and run impedance and coupling analysis workflows.

15

Tools Menu

The *Tools* menu offers commands that are applicable on entire design. The following table describes the options available with *Tools* top menu.

Menu	Sub Menu	Old Menu	Command Syntax	Description
Padstack				
	Modify Design Padstack	Tools – Padstack – Modify Design Padstack	padeditdb	Selects and modifies a padstack definition or instances in a design.
	Modify Library Padstack	Tools – Padstack – Modify Library Padstack	padeditlib	Modifies a padstack from the library.
	Replace	Tools – Padstack – Replace	replace padstack	Replaces an existing padstack with a new padstack.
	Group Edit	Tools – Padstack – Group Edit	multpadedit	Modifies individual pad shapes or multiple instances of one pad shape.
	Refresh	Tools – Padstack – Refresh	refresh padstack	Updates the padstacks in a design to agree with the padstacks in the library.
Pad				

	Boundary	Tools – Pad – Boundary	editpad boundary	Changes the geometry for a pad while maintaining a permanent association between the pad and the package/part symbol.
	Restore	Tools – Pad – Restore	editpad restore	Restores derived pads to their original padstacks.
	Restore ALL	Tools – Pad – Restore ALL	editpad restore all	Restores all derived pads to their original padstacks.
Derive Connectivity		Tools – Derive Connectivity	derive connectivity	Sets options to improve accuracy during conversion of Gerber files to the PCB editor.
Metal Usage Report		Tools – Metal Usage Report	metal usage report	Provides an accurate assessment of the percentage of metal in a user-specified region of the design.
Import File Manager		Tools – Import File Manager	import file manager	Provides an interface to set-up the tracking of different types of import files available for update.
MCAD Collaboration		Tools – MCAD Collaboration	ecadmcad	Provides an interface to set up environment for exchanging physical design data between MCAD tools and layout editor whenever any update is available.
Topology Extract			topology template	Provides an interface to extract a topology into Signal Explorer.

DRC Browser		Tools – DRC Browser	browse drcs	Provides an interface to locate, view and address DRCs.
Technology File Compare		Tools – Technology File Compare	techfile compare	Compares a tech file to a design.
Markup		Tools – Markup	markup_edit	
More				
	File Manager	Tools – Utilities –File Manager	filemgr	Displays working directory of the active design.
	Design Compare	Tools – Design Compare	design compare	Compares physical netlist data from a variety of sources.
	Database History	Tools – Database Diary	db diary	Tracks changes made to a design.
	Env Variables	Tools – Utilities – Env Variables	set	Temporarily defines or replaces an environment variable settings in a current session.
	Aliases/ Function Keys	Tools – Utilities – Aliases/ Function Keys	alias	Create shortcuts for frequently used commands.
	Keyboard Commands	Tools – Utilities – Keyboard Commands	helpcmd	Displays a list of all commands that can be entered in the command line.

	OpenGL Status	Tools – Utilities – OpenGL Status	opengl_report	Checks system graphic information and creates a report listing vendor card type and version.
	Licenses Used	Tools – Utilities – Licenses Used	license_use	Displays the current licenses.
	Stroke Editor	Tools – Utilities – Stroke Editor	stroke_editor	Creates or edits an existing strokes file.

16

Manufacture Menu

The *Manufacture* menu offers a quick and easy ways to place the symbols on the canvas. The following table describes the options present in the *Manufacture* menu.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
Dimension Environment		Manufacture – Dimension Environment		dimension edit	Enables environment to create or edit dimensions.
Drafting					
	LineFont	Manufacture – Drafting – LineFont		linefont	Specifies a font for a line.
	Chamfer	Manufacture – Drafting – Chamfer		draft chamfer	Generates a chamfer segment between two non-parallel line segments on the same layer.
	Fillet	Manufacture – Drafting – Fillet		draft fillet	Generates an arc segment tangential to two line segments.

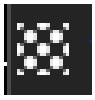
	Create Detail	Manufacture – Drafting – Create Detail		create_detail	Creates an enlarged portion of a selected area in a design.
	Extend Segments	Manufacture – Drafting – Extend Segments		extend_segments	Extends two non-parallel lines or arc segments to a projected intersection point.
	Trim Segments	Manufacture – Drafting – Trim Segments		trim_segments	Removes the unwanted line or arc segments that extend beyond the intersection points.
	Connect Lines	Manufacture – Drafting – Connect Lines		connect_lines	Creates new lines to connect existing line or arc segments.
	Add Parallel Line	Manufacture – Drafting – Add Parallel Line		add_parallel_line	Creates lines parallel to an existing line.
	Add Perpendicular Line	Manufacture – Drafting – Add Perpendicular Line		add_perp_line	Creates a new line that is perpendicular to an existing line.
	Add Tangent Line	Manufacture – Drafting – Add Tangent Line		add_tangent_line	Creates lines tangent to an existing circle or an arc segment.
	Delete By Line	Manufacture – Drafting – Delete By Line		delete_by_line	Removes parts of line or arc segments that exist on one side of a cut line.

	Delete By Rectangle	Manufacture – Drafting – Delete By Rectangle		delete_by_rectangle	Removes parts of line or arc segments, and vias that exist either inside or outside a cut rectangle.
	Offset Copy	Manufacture – Drafting – Offset Copy		offset_copy	Creates multiple copies of design elements (arc, circle, rectangle, frectangle, line, and text) and paste them to a new location that is an offset from the original element.
	Offset Move	Manufacture – Drafting – Offset Move		offset_move	Moves design elements (arc, circle, rectangle, frectangle, line, and text) to a new location that is a an offset from the original location.
	Relative Copy	Manufacture – Drafting – Relative Copy		relative_copy	Creates mirror images of design elements (arc, circle, rectangle, frectangle, line, and text) to a new location that is relative to a line.
	Relative Move	Manufacture – Drafting – Relative Move		relative_move	Moves design elements (arc, circle, rectangle, frectangle, line, and text) to a new location that is relative to a line.

Backdrill		Manufacture – NC – Backdrill Setup and Analysis		backdrill steup	Defines parameters for backdrilling.
Customize Drill Table		Manufacture – NC – Drill Customization		ncdrill_customization	Customizes drill symbol information at the design level.
Create Drill Table		Manufacture – NC – Drill Legend		ncdrill_legend	Creates different types of drill legend tables, which sort hole sizes and map drill figures or text symbols to each drill bit size.
Cross Section Chart		Manufacture – Cross Section Chart		xsection_chart	Generate a cross section chart displaying the drill span, stacked vias, embedded component legend, and layer information.
Auto Rename Refdes					

	Rename	Logic – Auto Rename Refdes – Rename		rename_param	Sets parameters for renaming RefDes information in a design and run the command to automatically renames every component on a design in a single operation. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.
	Design	Logic – Auto Rename Refdes – Design		rename_area_design	Renames automatically every component on a design in a single operation. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.

	Room	Logic – Auto Rename Refdes – Room		rename area room	Assign a room for automatic reference designator renaming. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.
	Window	Logic – Auto Rename Refdes – Window		rename area window	Define an area for automatic reference designator renaming by making two diagonal selections. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.

	List	Logic – Auto Rename Refdes – List		rename_area_list	Displays the LIST AREA dialog box showing the current automatic reference designator rename mode and the areas for renaming. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.
Silkscreen		Manufacture – Silkscreen		silkscreen_param	Defines parameters for the auto silkscreen process.
Thieving		Manufacture – Thieving		thieving	Adds a pattern of non-conductive, single-layer figures to areas on the outer layers of a board that do not contain copper.
Cut Marks		Manufacture – Cut Marks		cut_marks	Defines and adds cut marks at each corner of a board outline.

Shape Degassing		Manufacture – Shape Degassing		degas	Perforates the planes in a design to allow the gas to escape from beneath the metal during manufacturing.
Create Coupons		Manufacture – Create Coupons		create_coupons	Generate Test Coupons
Test Points					
	Automatic	Manufacture – Testprep – Automatic		testprep_automatic	Defines parameters for the automatic testprep process and automatically generate testpoints. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/ PSpice* licenses only.
	Manual	Manufacture – Testprep – Manual		testprep_manual	Manually add, delete, or move testpoints and edit testpoint-related properties on nets and symbols.
	Properties	Manufacture – Testprep – Properties		testprep_properties	Adds testprep-related properties to a single net or symbol.

	Fix/Unfix Test Points	Manufacture – Testprep – Fix/Unfix testpoints		testprep fix	Sets or resets the status on all testpoint locations on the design.
	Create TP Fixture	Manufacture – Testprep – Create Fixture		testprep createfixture	Generates the static subclasses and copies testpoints information to them to graphically compare the differences between the board that represented the fixture and the current design after logic changes.
	Density Check	Manufacture – Testprep – Density check		testprep density	Verifies the testpoint density within user-definable unit areas when *Unit Area Check* is enabled or beneath symbols when *Component Area Check* is enabled.
	Resequence	Manufacture – Testprep – Resequence		testprep resequence	Renames the RefDes text of testpoints sequentially, sorted by X/Y location from left to right and bottom to top on each side, starting with the TOP side first and then the BOTTOM side.

	Parameters	Manufacture – Testprep – Parameters		testprep prmed	Defines the parameters for the testprep process.

17

Export Menu

The *Export* menu provides commands to export design data to other formats. The following table describes the icons on the toolbar.

Menu	Sub Menu	Old Menu	Icon	Command Syntax	Description
IPC-2581		File – Export – IPC 2581		ipc2581_out	Exports physical design data to the IPC2581 data format.
Gerber Parameters		Manufacture – Artwork		artwork	Sets general artwork parameters.
Gerber		Manufacture – Artwork		artwork	Creates photoplot film files in an ASCII format.
NC Parameters		Manufacture – NC – NC Parameters		ncdrill_param	Defines parameters for numerically controlled routing and NC Route output files in a text file, which specifies the drill coordinate data format.
NC Drill		Manufacture – NC – NC Drill		nctape_full	Generates customized NC drill output files based on parameters defined by the `ncdrill_param` command.

NC Route		Manufacture – NC – NC Route		ncroute	Generates output for an NC route based on the parameters defined by the `ncdrill param` command.
ODB++ Inside		File – Export – ODB++ inside		odb out	Exports physical design data to a Valor ODB++ database.
IPC-356		File – Export – IPC 356		ipc356 out	Exports physical design data to the IPC-D-356 format.
Pick/Place Data		File – Export – Placement		plctxt out	Exports positions and orientations of placed components from an existing design into an ASCII text file.
FABMaster		File – Export – Fabmaster out		fabmaster out	Exports fabmaster output in a text file.
Test Point NC Drill		Manufacture – Testprep – Create NC drill data		testprep ncdrill	Outputs testpoint locations marked as valid to NC files used to drill testpoints in fixtures based on the parameters set by the `testprep prmed` command.
PDF		File – Export – PDF		pdf out	Exports physical design as a PDF file.
MCAD					
	DXF	File – Export – DXF		dxf out	Exports mechanical design data from a design to a DXF file in ASCII format

	IDF	File – Export – IDF		idf out	Exports data from a design for input to IDF format,
	IDX	File – Export – IDX		idx out	Exports incremental physical design data to the IDX data format.
	STEP	File – Export – STEP		step out	Exports a layout design as a STEP model.
	Creo View	File – Export – Creo View		export creoview	Exports physical design data into a PTC's Creo View compatible database.
Variants					
	Create Assembly Drawing	Manufacture – Variants – Create Assembly Drawing		variant assembly	Set options to generate an assembly drawing layer for components belongs to a variant of the current design.
	Create Bill of Materials	Manufacture – Variants – Create Bill of Materials		variant bom	Sets options for generation of a bill of materials report for components belongs to a variant of the current design.
Fab Panelization Tool		Manufacture – Fab Panelization Tool		tbx panelize	
Quick Reports		Tools – Quick Reports		reports	Displays a list of available reports.

Reports		Tools – Reports		reports	Generates reports to provide information about a design.
Back Annotation Netlist		File – Export – Logic/ Netlist		feedback	Exports logic information from a design to another file or program.
Clipboard		File – Export – Sub-Drawing		clpcopy	Copies design elements to a clipboard file for pasting into other designs and drawings.
More					
	Color/ Board Parameters	File – Export – Parameters		param out	Creates a database parameter file containing customized parameter records from a design for dynamic fill; grid settings; artwork format; and Xhatch style, linewidth, spacing, and angle, and so on.
	Techfile	File – Export – Techfile		techfile out	Exports constraints information into a technology file.
	Libraries	File – Export – Libraries		dlib	Writes library elements from an existing design file to the current directory.
	Netlist w/ Properties	File – Export – Netlist w/ Properties		netout	Generates a netlist output file that contains pin and net properties for the current design.

	Annotations	File – Export – Annotations		annotation out	Transfers drawing data from one design to another, or from one version of tool, for example, Allegro Free Physical Viewer, to a full version of Allegro PCB Editor.
	Router	File – Export – Router		specctra out	Generates routing files from design and launches PCB Router to autoroute the design.
	IPF	File – Export – IPF		create plot	Creates intermediate plot (IPF) and control files from a current design.
	Symbol Spreadsheet	File – Export – Symbol Spreadsheet		symbol to spreadsheet	Exports information about a placed component to a standard spreadsheet tool such as Microsoft Excel. **Note:** This command is available with the *OrCAD PCB Designer Professional* and the *OrCAD PCB Designer Professional w/PSpice* licenses only.
	Pin Delay	File – Export – Pin Delay		pin_delay out	Create a file containing pin-delay values,
	Down Rev Design	File – Export – Downrev design		downrev	Revises a design database containing new functionality, so that it can be opened in previous release.

	Strip Design	File – Export – Strip Design		strip design	Creates an output database for sharing by removing intellectual property from it.