

# INTRODUCTION TO ORCAD

## Hands-On Workshop

### Quick Reference Guide

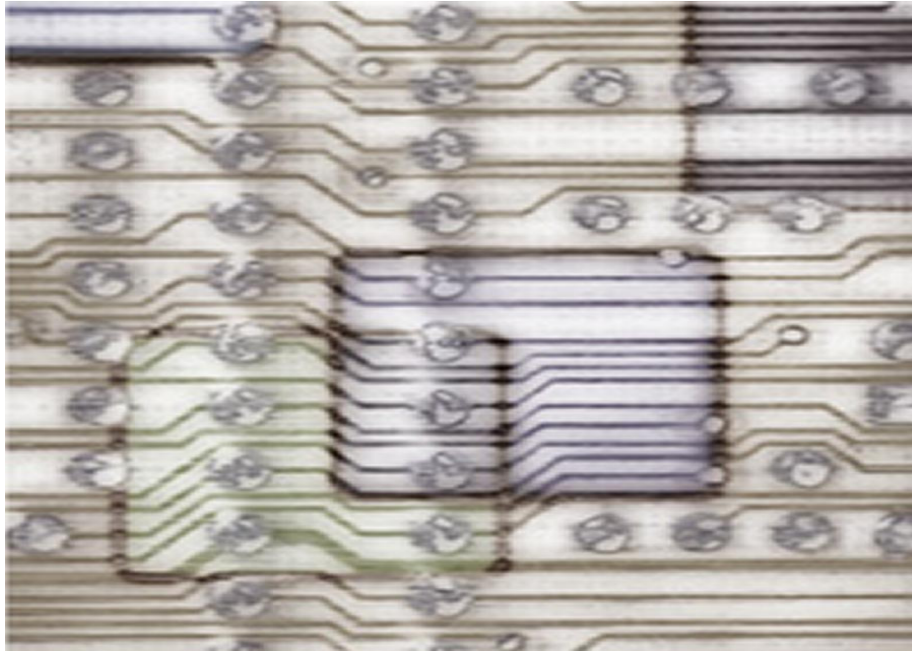


Embedded Solutions



# Embedded Solutions

---



## **Introduction To OrCAD** Quick Reference and Supplementary Material

### **CONTENTS**

- Capture shortcut keys and toolbars
- Layout shortcut keys and toolbars
- Chart of current capabilities of trace widths
- Tips For Successful Schematics and Layout
- Additional sources of OrCAD information

---

Copyright © 2002 Embedded Solutions. All rights reserved.

OrCAD®, OrCAD Capture®, OrCAD Layout® and PSpice® are registered trademarks of OrCAD Corporation.

All other brands and product names mentioned herein are used for identification purposes only, and are trademarks or registered trademarks of their respective holders.

EMBEDDED SOLUTIONS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

EMBEDDED SOLUTIONS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LIST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF EMBEDDED SOLUTIONS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

Contractor/manufacture is:

Kalen Brunham  
Embedded Solutions  
162-2025 Corydon Av.  
Winnipeg, Manitoba  
R3P 0N5 CANADA

General email: [info@embeddedsolutions.net](mailto:info@embeddedsolutions.net)  
Web site: [www.embeddedsolutions.net/training.html](http://www.embeddedsolutions.net/training.html)

This is an unpublished work of Embedded Solutions.


















---

## Useful OrCAD Capture Shortcut Keys

See the Shortcuts topic in Capture online help for a complete list of shortcut keys.




















<i>Key</i>	<i>Mode or user interface item</i>	<i>Function or menu command</i>
CTRL+A	View menu	Select All
SHIFT+A	View menu	Ascend Hierarchy
B , SHIFT+B	Place menu	Bus
CTRL+B	View menu	Previous Part
CTRL+C	Edit menu	Copy
SHIFT+D	View menu	Descend Hierarchy
E , SHIFT+E	Place menu	Bus Entry
ESC	Schematic page editor and part editor	End Mode, Unselect any selected object
CTRL+E	Edit menu	Properties
F, SHIFT+F	Schematic page editor	Place Power
CTRL+F	Edit menu	Find
G, SHIFT+G	Schematic page editor	Place Ground
H	Mirror (on the Edit menu)	Horizontally
I	Zoom (on the View menu)	Zoom In
J , SHIFT+J	Place menu	Junction
N , SHIFT+N	Place menu	Net Alias
CTRL+N	View menu	Next Part
O	Zoom (on the View menu)	Zoom Out
P , SHIFT+P	Place menu	Part
CTRL+P	File menu	Print
R , CTRL+R	Edit menu	Rotate
CTRL+S	File menu	Save
T , SHIFT+T	Place menu	Text
CTRL+T	Schematic page editor and part editor	Toggle snap to grid on and off
CTRL+U	Edit menu	Ungroup
V	Mirror (on the Edit menu)	Vertically
CTRL+V	Edit menu	Paste
W , SHIFT+W	Place menu	Wire
CTRL+X	Edit menu	Cut
SHIFT+X	Schematic page editor	No Connect placement mode
Y , SHIFT+Y	Place menu	Polyline
CTRL+Z	Edit menu	Undo

## OrCAD Capture Main Toolbar

<i>Tool</i>	<i>Name</i>	<i>Description</i>
	New document	Creates a new document based on the active document.
	Open document	Opens an existing document based on the active document.
	Save document	Saves the active schematic page or part.
	Print	Prints the active schematic page or part.
	Cut to clipboard	Removes the selected object and places it on the Clipboard.
	Copy to clipboard	Copies the selected object to the Clipboard.
	Paste from clipboard	Pastes the contents of the Clipboard at the cursor.
	Undo	Undoes the last command performed.
	Redo	Redoes the last command performed.
	Most recently used	Displays the most recently placed part name in the drop-down list.
	Zoom in	Zooms in to present a closer, enlarged view
	Zoom out	Zooms out to present more of your document.
	Zoom to region	Specifies an area of the schematic page or part to enlarge to fill the entire window.
	Zoom to all	Views the entire document.
	Annotate	Assigns part references to parts on the selected schematic pages.
	Back annotate	Back annotates the selected schematic pages.
	Design rules check	Checks for design rules violations in the selected schematic pages.
	Create netlist	Creates a netlist from the selected design.
	Cross reference	Creates a cross reference report of the selected schematic pages.
	Bill of materials	Creates a bill of materials report from the selected schematic pages.
	Snap to grid	Turns snap to grid on and off.
	Project manager	Displays a project manager window for the active document, providing an overview of the project contents.











## OrCAD Capture Schematic Page Editor Tool Palette



<i>Tool</i>	<i>Name</i>	<i>Description</i>
	Selection	Selects objects.
	Part	Selects parts from a library for placement.
	Wire	Draws wires. SHIFT allows any angle drawing.
	Net alias	Places aliases on wires and buses.
	Bus	Draws buses. SHIFT allows any angle drawing.
	Junction	Places or deletes junctions.
	Bus entry	Draws bus entries.
	Power	Places power symbols.
	Ground	Places ground symbols.
	Hierarchical block	Places hierarchical blocks.
	Hierarchical port	Places hierarchical ports on schematic pages.
	Hierarchical pin	Places hierarchical pins in the selected hierarchical block.
	Off-page connector	Places off-page connectors.
	No connect	Places no-connect symbols on pins.
	Line	Draws lines.
	Polyline	Draws polylines. SHIFT allows any angle drawing.
	Rectangle	Draws rectangles. SHIFT constrains the shape to a square.
	Ellipse	Draws ellipses. SHIFT constrains the shape to a circle.
	Arc	Draws arcs.
	Text	Places text.

## OrCAD Capture Part Editor Tool Palette



<i>Tool</i>	<i>Name</i>	<i>Description</i>
	Selection	Selects objects.
	IEEE symbol	Places IEEE symbols on a part.
	Pin	Places pins on a part.
	Pin array	Places multiple pins on a part.
	Line	Draws lines.
	Polyline	Draws polylines. SHIFT allows any angle drawing.
	Rectangle	Draws rectangles. SHIFT constrains the shape to a square.
	Ellipse	Draws ellipses. SHIFT constrains the shape to a circle.
	Arc	Draws arcs.
	Text	Places text.

## Useful OrCAD Layout Shortcut keys

See the Shortcuts topic in Layout online help for a complete list of shortcut keys.


















<i>Key</i>	<i>Mode or user interface item</i>	<i>Function or menu command</i>
A	With an obstacle selected	Arc
B	Zoom	Zoom DRC/Route Box
SHIFT+C	Spreadsheet	Components
C	Zoom	Zoom Center
D	With a track selected	Unroute
ALT+D	With a track selected	Unroute Net
E	With a track selected	Add Free Via
CTRL+E , ENTER		Properties
F	With a track or obstacle selected	Finish
CTRL+F , TAB		Find/Goto
G	With a track selected	Unroute Segment
CTRL+G		System Settings
SHIFT+G	Spreadsheet	Route Spacing
H		Highlight Net
I	Zoom	Zoom In
CTRL+I		Library manager
J	With a component selected	Shove
CTRL+J	With components selected	Adjust
K	With components selected	Make cluster
CTRL+K	With a component cluster selected	Break cluster
L	With a component or track selected	Lock
CTRL+L	With a track selected	Unlock
M	With a component, track, or pin selected	Minimize Connections
CTRL+M	With a track, obstacle, or text selected	Mirror
N	With a component, track, or error selected	Select Next
O	Zoom	Zoom Out
P	With a track selected	Add Test Point
CTRL+P		Print/Plot
Q		Query window
CTRL+Q	With a component selected	Quick Place
R , CTRL+R	With a component, track, obstacle, pin, or text selected	Rotate

## Useful OrCAD Layout Shortcut keys (continued)

<i>Key</i>	<i>Mode or user interface item</i>	<i>Function or menu command</i>
S	With a track or obstacle selected	Segment
ALT+S		Select Any
CTRL+S		Save file
T	With a component selected	Opposite layer
CTRL+T	With a track selected	Tack
U, CTRL+Z		Undo
V	With a track selected	Add Via
CTRL+V		New or Paste
W	With a track selected	Change Width
CTRL+W	With components, gates, or pins selected	Swap
X	With a track or obstacle selected	Exchange Ends
CTRL+X		Delete
Y	With a track or obstacle selected	Any Angle Corners
Z	Zoom	Zoom Area
ALT+Z	Zoom	Zoom Previous
ESC		End Command
F1		Help Topics
F3		Component Selection Criteria
F5		Redraw
F7		Online DRC
SHIFT+F4		Tile windows
SHIFT+F5		Cascade windows
<	With a track selected	Decrease width
>	With a track selected	Increase width
CTRL+<	In the query window	Query previous object
CTRL+>	In the query window	Query next object
PERIOD		High Contrast
DASH		Visible <> Invisible
BACKSPACE		Clear Screen
HOME		Redraw
SHIFT+HOME	Zoom	Zoom All (Fit)
DELETE		Delete










## OrCAD Layout Main Toolbar



<i>Tool</i>	<i>Name</i>	<i>Description</i>
	Open	Opens an existing board.
	Save	Saves an existing board.
	Library manager	Opens the library manager.
	Delete	Deletes whatever you have selected.
	Find	Displays the Find Coordinate or Reference Designator dialog box, which you use to search for specific coordinates or reference designators.
	Edit	Displays an appropriate editing dialog box, depending on what you have selected.
	Spreadsheet	Displays a list of the available spreadsheets.
	Zoom in	Magnifies selected areas of the board.
	Zoom out	De-magnifies selected areas of the board.
	Zoom all	Zooms so that you can see the entire board.
	Query	Displays the query window, which lists an object's properties.
	Component	Enables you to select, add, move, edit, or delete components.
	Pin	Enables you to select, add, move, edit, or delete pins.
	Obstacle	Enables you to select, add, move, edit, or delete obstacles.
	Text	Enables you to select, add, move, edit, or delete text.
	Connection	Enables you to select, add, combine, or delete net connections.
	Error	Enables you to select error markers related to spacing and design rule violations.

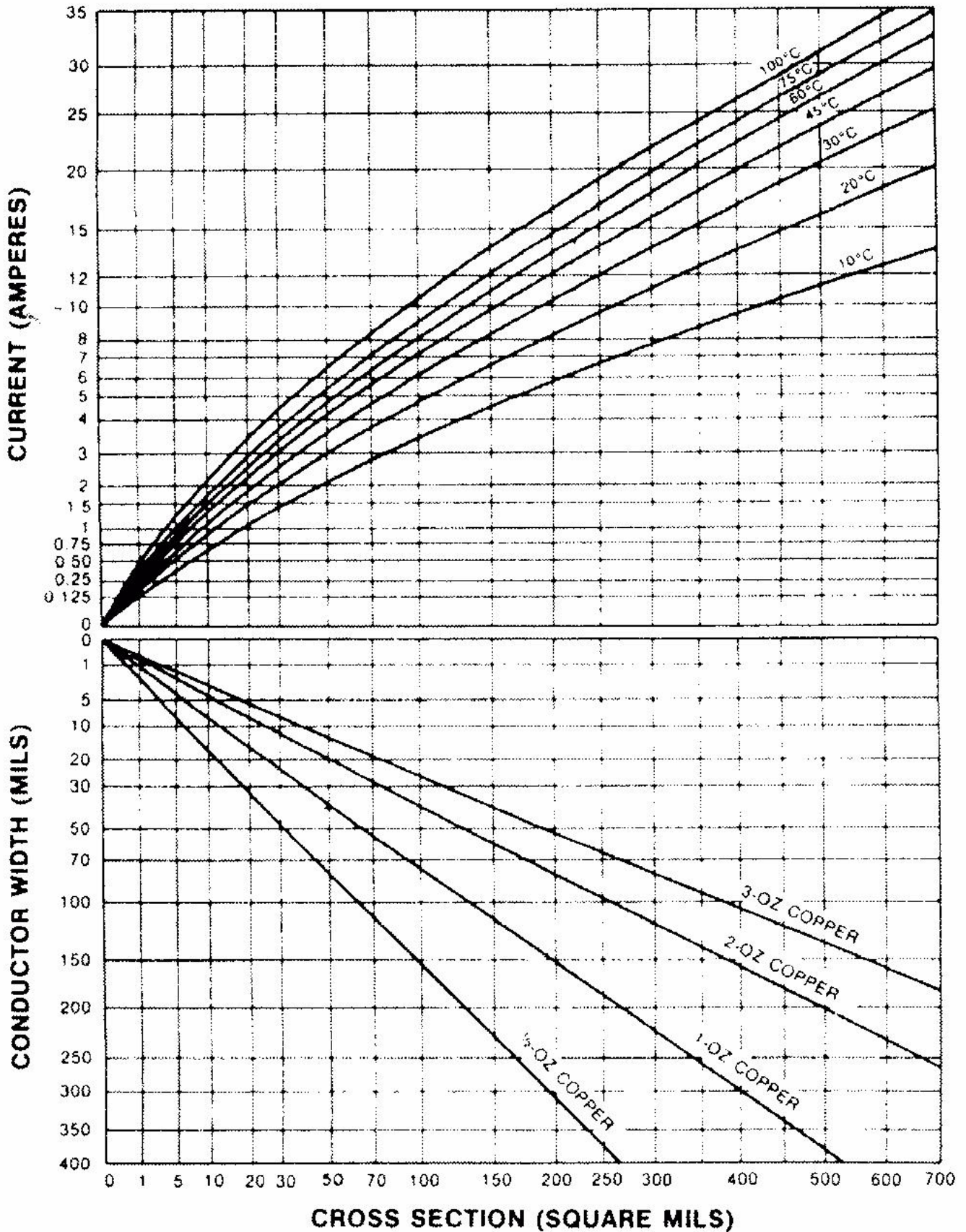
## OrCAD Layout Main Toolbar (continued)

---

<i>Tool</i>	<i>Name</i>	<i>Description</i>
	Color	Color Displays the Color spreadsheet, within which you change the color of layers or objects, or their visibility (visible or invisible).
	Online DRC	Enables online design rule checking.
	Reconnect	Enables reconnect mode, which you use to show or hide routes and connections. Note that reconnect mode should only be used during component placement, before any routing is done.
	Auto path route	Enables auto path route mode, which you use to route and place vias interactively using the shove algorithm.
	Shove track	Enables shove track mode, which you use to route manually using the shove algorithm.
	Edit segment	Enables edit segment mode, which you use to select existing tracks and change their positions, while Layout automatically adjusts the angles and sizes of adjacent segments to maintain connectivity.
	Add/edit route	Enables add/edit route mode, which you use to route manually without using the shove algorithm.
	Refresh all	Minimizes connections, repours copper, and recalculates board statistics.
	Design rule check	Runs a design rule check using the options selected in the Check Design Rules dialog box (accessed by choosing Design Rule Check from the Auto menu).

# CONDUCTOR THICKNESS AND WIDTH

(For use in determining current carrying capacity and sizes of etched copper conductors for various temperature rises above ambient)



## Tips for Successful Schematics and PCB Layout

---

These tips are intended to help you draw schematics, ease the PCB layout for you or your PCB design team, and for troubleshooting your circuitry. These tips can be applied to any vendor's PCB design package.

1. Try and only use B size sheets when drawing schematics. This makes it much easier to print and troubleshoot the design.
2. When drawing circuitry, inputs tend to be on the left, outputs tend to be on the right. Using a uniform flow will allow other to easily read your design.
3. All signals going to connectors should be clearly labeled. This will help in debugging your design.
4. Important internal signals should be clearly labeled.
5. Indicate which page off-page connections go to.
6. Give labeled nets meaningful names. However, don't use characters such as ! @ # \$ % ^ & \* \_ as these characters may be interpreted as control characters.
7. Do not draw four way junctions. These can be mistaken for crossovers.
8. Verify pinout of schematic symbols for all new parts that have been created.
9. Add schematic notes where appropriate. Notes identifying high speed nets and high current nets will ease the PCB layout and troubleshooting.
10. Do not change part references or net names on new board revisions. Changing these can result in the layout being completely started over again.

## **Additional Sources for OrCAD Information**

---

- **OrCAD Capture Users Guide**  
(<http://www.orcad.com/files/pdf/r92manuals/capug.pdf>)
- **OrCAD Capture Quick Reference Guide**  
(<http://www.orcad.com/files/pdf/r92manuals/capqrc.pdf>)
- **OrCAD Layout Users Guide**  
(<http://www.orcad.com/files/pdf/r92manuals/layug.pdf>)
- **OrCAD Layout Quick Reference Guide**  
(<http://www.orcad.com/files/pdf/r92manuals/layqrc.pdf>)
- **OrCAD Layout Getting Started Guide**  
(<http://www.orcad.com/files/pdf/r92manuals/layqs.pdf>)
- **OrCAD Technical Library**  
(<http://www.orcad.com/Technical/Library/Library.asp>)
  - Knowledge base
  - Technical articles
  - OrCAD Capture Student Edition
- **OrCAD Help**