

# **Allegro®**

## **Free Physical Viewer**

Series XL and GXL

**Product Version 16.2**  
**November 2008**

© 1991–2008 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

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

Allegro PCB Editor contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. 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 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. 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. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Patents:** Allegro PCB Editor, described in this document, is protected by U.S. Patents 5,481,695; 5,510,998; 5,550,748; 5,590,049; 5,625,565; 5,715,408; 6,516,447; 6,594,799; 6,851,094; 7,017,137; 7,143,341; 7,168,041.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. 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>assign color</u>	8
Options tab for the <u>assign color</u> Command	8
Assigning a Custom Color or Highlighting an Element	9
<u>blank waived drcs</u>	11
Menu Path	11
Procedure	11
<u>color192</u>	12
Procedures	21
<u>colorview create</u>	32
Color Views Dialog Box	32
Procedures	33
<u>colorview restore</u>	36
Applying the Previous Color Visibility View	36
<u>cns show</u>	37
Generating a Constraint Report	38
<u>dehighlight</u>	39
Options Tab for the <u>dehighlight</u> Command	39
Procedure	40
<u>exit</u>	42
<u>grid toggle</u>	43
<u>help</u>	44
<u>highlight</u>	45
Options Tab for the <u>highlight</u> Command	46
<u>layer priority</u>	48
Menu Path	48
Display Priority Dialog Box	48
Assigning a Display Priority To Layers	49
<u>open</u>	50
Menu Path	50
Syntax	50
Examples	50
Dialog Box	50

## Allegro Free Physical Viewer

---

<u>Procedure</u>	51
<u>plot</u>	52
<u>Print Dialog Box</u>	52
<u>Procedures</u>	53
<u>plot setup</u>	56
<u>Plot Setup Dialog Box</u>	56
<u>Procedure</u>	58
<u>prmed</u>	60
<u>Design Parameter Editor Dialog Box</u>	60
<u>Procedures</u>	60
<u>rats all</u>	63
<u>Procedure</u>	63
<u>rats component</u>	64
<u>Procedure</u>	64
<u>rats net</u>	65
<u>Procedure</u>	65
<u>redraw</u>	66
<u>reset dockwindows</u>	67
<u>Menu Path</u>	67
<u>Syntax</u>	67
<u>Displaying All Foldable Window Panes to Default Positions</u>	67
<u>script</u>	68
<u>Scripting Dialog Box</u>	68
<u>Procedures</u>	69
<u>show allpanes</u>	72
<u>Syntax</u>	72
<u>Displaying All Foldable Window Panes</u>	72
<u>showhide find</u>	73
<u>Syntax</u>	73
<u>Controlling the Visibility of the Find Window Pane</u>	73
<u>showhide options</u>	74
<u>Syntax</u>	74
<u>showhide text</u>	75
<u>Syntax</u>	75
<u>showhide view</u>	76
<u>Syntax</u>	76

## Allegro Free Physical Viewer

---

<u>showhide vis</u>	77
<u>Syntax</u>	77
<u>show element</u>	78
<u>Dialog Boxes</u>	78
<u>Procedures</u>	79
<u>show measure</u>	82
<u>Measure Dialog Box</u>	82
<u>Procedure</u>	83
<u>show property</u>	85
<u>Show Property Dialog Box</u>	85
<u>Procedures</u>	86
<u>show waived drcs</u>	89
<u>Menu Path</u>	89
<u>Procedure</u>	89
<u>status</u>	90
<u>unrats all</u>	95
<u>Procedure</u>	95
<u>unrats component</u>	96
<u>Procedure</u>	96
<u>unrats net</u>	97
<u>Procedure</u>	97
<u>viewlog</u>	98
<u>Select File to View Dialog Box</u>	98
<u>Procedures</u>	99
<u>zoom center</u>	100
<u>Procedure</u>	100
<u>zoom fit</u>	101
<u>Procedure</u>	101
<u>zoom in</u>	102
<u>Procedure</u>	102
<u>zoom out</u>	103
<u>Procedure</u>	103
<u>zoom points</u>	104
<u>Procedure</u>	104
<u>zoom previous</u>	105
<u>zoom world</u>	106

**Allegro Free Physical Viewer**

---

Procedure ..... 106

---

# Allegro Free Physical Viewer

---

## assign color

### Procedure

Assigns a color and highlights an element without requiring the use of the *Color* dialog box. Changing the color or highlighting with this command automatically updates the *Nets* section of the *Color* dialog box as well.

This command also functions in a pre-selection use model, in which you choose an element first, then right click and execute the command. Valid elements are:

- Symbols
- Functions
- Nets
- Pins
- DRC Errors

### Menu Path

*Display – Assign Color*

### Toolbar Icon



## Options tab for the assign color Command

The following display only when you choose the *Display – Assign Color* menu item.

*Selected color*                      Indicates the currently chosen color.

*More Colors*                      Displays a palette of 192 modifiable colors, 96 of which display at once. A rainbow gradient scheme initializes the color palette for all undefined colors, meaning at least the first 24 colors are always defined. *Next/Prev* toggle the primary and secondary color palettes. The primary palette is is the Cadence default, organized by a rainbow spectrum of 96 colors. The secondary



## Allegro Free Physical Viewer

### assign color

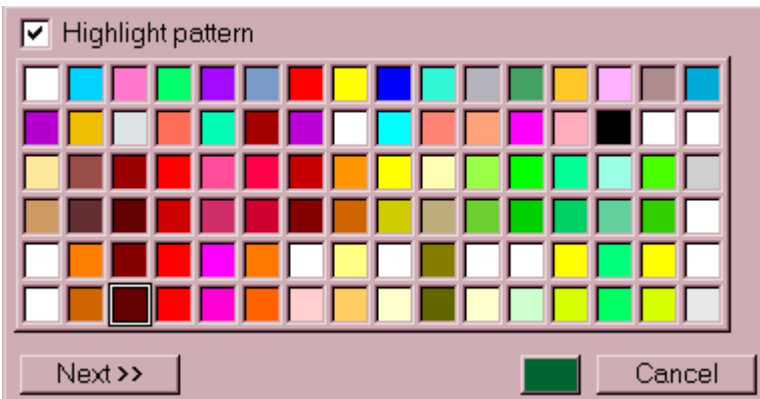
---

palette comprises 96 colors used for customization. The first 24 positions are reserved for colors used in pre-16.0 databases.

#### *Highlight Pattern*

Click to accentuate certain elements with a pattern—or striping—comprising the element's base subclass color and the temporary highlight color defined in the *Display* category of the *Color* dialog box. Once the element becomes highlighted in the design canvas, its name also displays in a bolded font in the *Nets* section of the *Color* dialog box. Striping is only visible when the `display_nohilitefont` variable is disabled.

In the pre-selection mode, after you right-click and choose *Assign Color*, the following palette displays:



## Assigning a Custom Color or Highlighting an Element

1. Hover your cursor over an element.
2. Right-click and choose *Assign Color* from the pop-up menu.

The color palette displays.

Choose *Next* to display the secondary color palette for additional colors.

3. Click the color box of the new color for the element. The selected color displays in the bottom right of the palette, and the element's color changes in the design canvas and in the *Nets* section of the *Color* dialog box.
4. Click *Highlight Pattern* to to accentuate certain elements with a pattern in the selected color if required.

## **Allegro Free Physical Viewer** assign color

---

The element becomes highlighted in the design canvas and its name displays in a bolded font in the *Nets* section of the *Color* dialog box.

## blank waived drcs

The `blank waived drcs` command lets you suppress waived DRC error markers from displaying on the board. This command is the opposite of the `show waived drcs` command.

For more information on waiving DRCs, see [show waived drcs](#), and the *Creating Design Rules* user guide in your documentation set.

### Menu Path

*Display – Waive DRCs – Blank*

### Procedure

#### Concealing Waived DRC Error Markers in the Design

- Run the `blank waived drcs` command.

The waived DRC error markers disappear from the board.

## color192

[Dialog Boxes](#) | [Procedures](#)

Launches the Color dialog box, which supports 192 colors and comprises the Layers and Nets grids.

### ■ Layers Grid

The Layers grid primarily controls the color and visibility settings of classes and subclasses, along with levels of transparency for the design and shapes. Use the *Layers* grid to also control shadow dimming, highlighting, ratsnest display, waived DRCs, and drill holes. You can create your own unique colors or palettes that may be saved to external .col files and then applied to other designs.

### ■ Nets Grid

Database elements may be displayed using either the class or subclass color or a single color assigned to an element, also known as a custom color. To assign a custom color to an entire net or to its pins, vias, clines, shapes, or rats, you use the *Nets* grid to differentiate them from other nets or net elements on a layer. You can also define how the custom color displays the element using a combination of states--none, highlight or custom color state, or highlight plus custom color state--all of which may be set independently.

## Menu Path

*Display – Color/Visibility*

## Toolbar Icon



## Dialog Boxes

### Color

<i>File – Load Default Cadence Color Palette</i>	Displays the default color palette, an array of 16 x 6 colors. The first column comprises popular colors typically used in designs.
--	---

## Allegro Free Physical Viewer

color192

---

<i>File – Load Color Palette</i>	Imports your customized color palettes from an external <code>.col</code> file and applies them to the current design. A file browser appears with the filter set to <code>*.col</code> and a list of all <code>.col</code> files available in the current local working directory. You can manually browse to other directories to open a color file.
<i>File – Save Color Palette</i>	Exports the current design's customized color palettes to an external <code>.col</code> file stored in your local working directory. A file browser appears with the filter set to <code>*.col</code> and a list of all <code>.col</code> files in the current local working directory. You can manually browse to other directories to which to save the file.
<i>File – Close</i>	Closes the dialog box without saving any changes.
<i>Global Visibility</i>	Controls whether or not all classes and subclasses are visible.
<i>Left Pane</i>	Displays each class associated with a group. The color and visibility of the subclasses associated with that class display horizontally.
<i>Subclasses</i>	<p>For all but the <i>Display</i> group, which has no associated classes or subclasses, each row lists a subclass. An X indicates the subclass is visible. The color box indicates the color assigned to the subclass element.</p> <p>Clicking the <i>All</i> column or <i>All</i> row enables visibility for the entire row or column.</p> <p>Clicking the intersection of the <i>All</i> row and <i>All</i> column cell (<i>All/All</i> cell) enables visibility globally. By default, subclasses are visible.</p>

## Allegro Free Physical Viewer

color192

Choose this box to enable visibility globally

Choose this box to enable visibility of entire column

Choose this color box to apply color across entire row

Choose this box to enable visibility of entire row

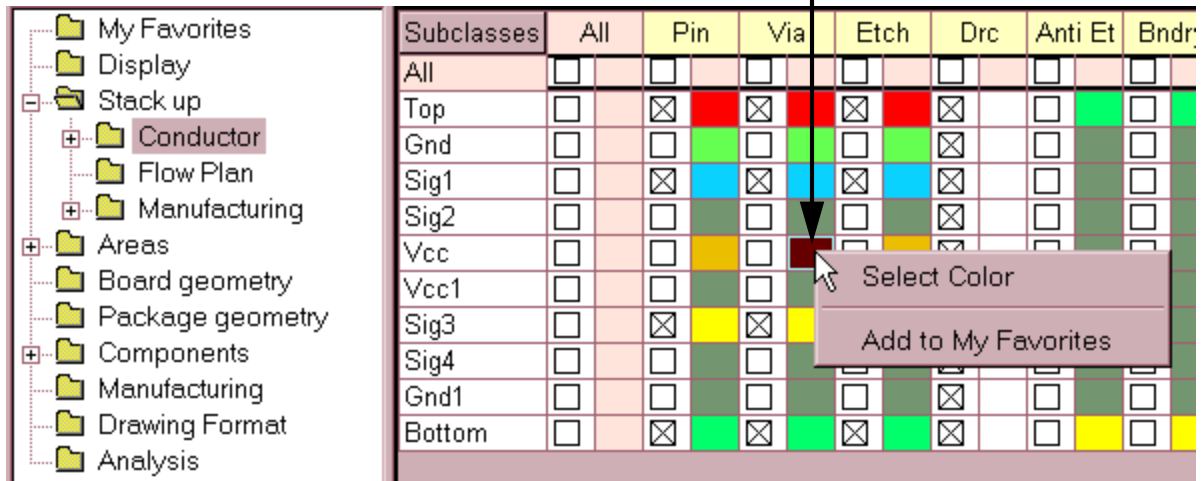
Subclasses	All	Pin	Via	Etch	Drc	Anti Et	Bndry
All	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Top	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Gnd	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Sig1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Sig2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Vcc	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Vcc1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Sig3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Sig4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Gnd1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Bottom	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

**Note:** There may be colors assigned to subclasses you would like to re-use on other subclasses. Similar versions of the color may exist in the color palette, so to source the exact color, hover over the color assigned to a subclass, then right-click and choose *Select Color*. This outlines the color used in the palette, even changing palettes if necessary.

## Allegro Free Physical Viewer

color192

Hover over this color box, right-click and choose Select Color



This color box becomes active



### *My Favorites*

Centralizes a user-defined group of frequently accessed subclasses. Hover your cursor over the color box associated with a subclass, right-click and choose *Add to My Favorites*. The subclasses are copied, rather than moved, to the *My Favoritess* folder. Any changes to the *My Favorites* folder updates the `myfavorites.txt` file in the `pcbenv` directory.

Remove a subclass stored in the *My Favorites* folder by hovering your cursor over the color box, right-clicking, and choosing *Remove from My Favorites*.

### *Display*

Controls the design window's appearance with the following fields. The *Display* group has no associated classes or subclasses.

*Temporary highlight*: Specifies the color of elements that are temporarily highlighted when you run the `highlight` or `assign color` commands or during an interactive command, such as `copy` or `move`. The default setting is white.

*Grids*: Specifies the color of the grids. The default is white.

Differentiates the display of ratsnests using a side-centric coloring scheme.

*Rats top-top*: Specifies the color of ratsnest lines that connect top-side only components (start-end pin on top).

*Rats top-bottom*: Specifies the color of ratsnest lines (one pin on top, other on bottom).

*Rats bottom-bottom*: Specifies the color of ratsnest lines that connect bottom-side only components (start-end pin on bottom). The default is pink.

*Plan DRC*: Specifies the color of floorplan DRC markers.

*Waived DRC*: Specifies the color of waived DRC error markers. The default is yellow.

*Drill holes*: Specifies the color of drill holes.



*Via label:* Specifies the color of via span labels optionally displayed on the pads of blind and buried vias. The default setting is white.

The labels indicate the via hole extents. Pins, through hole vias, and single-layer vias remain unlabeled in the canvas.

The label numbers map to design subclasses in order from top to bottom. Custom subclass names are ignored.

The visibility of via span labels is controlled using the *Via Label* parameter in the *Display* tab of the Design Parameter Editor. For details on the via label nomenclature, see the hover-over description for the *Via Label* parameter in the Design Parameter Editor dialog box.

*Background:* Specifies the design window's background color. The default is black.

### *Shadow mode*

Highlights an individual element without affecting the visibility of that element's entire subclass.

*On/Off:* Activates and deactivates *Shadow mode*, which darkens the colors of objects and elements of your design. Use this with the highlight command.

*Brightness:* Specifies the percentage of brightness applied to colors when *Shadow mode* is set to *On*. The default is 50%.

*Dim active layer:* Applies the *Brightness* percentage to the colors of objects in the active layer, darkening the colors so that the highlighted objects are more prominent.

### *Open GL*

Uses the OpenGL integrated Application Programming Interface to display elements semi-transparently, allowing you to view any elements that lie beneath other elements.

On Windows, OpenGL defaults to off. On Unix, the environment variable `useoglgraphics` enables it. To use the native graphics engine, disable OpenGL with the environment variable `nooglgraphics`.

## Allegro Free Physical Viewer

color192

---

### *Global Transparency*

Assigns varying degrees of transparency to all elements in the entire design.

Sliding the bar completely to the right (100%) represents a pre-16.0 graphics display.

Sliding the bar completely to the left (0%) causes previously filled geometry, such as clines and pads, to display with less intensity.

### *Shapes Transparency*

Assigns varying degrees of transparency to shapes only.

Sliding the bar completely to the right (100%) represents a pre-16.0 graphics display.

Sliding the bar completely to the left (0%) causes shapes to display with less intensity.

### *Net*

Click to display the Nets grid, which alphabetically lists nets hierarchically by bus, differential pair, Xnet, or net class. An X indicates the custom color state is enabled and visible in the design canvas. The color box indicates the custom color assigned to the net or net element.

## Allegro Free Physical Viewer

color192

---

Click to override an inherited custom color

Click to enable custom color state for an element, indicated by an X

Click to assign a custom color to all net elements (e.g., Pins)

Greyed out by design

Type	Nets	Net	Pins	Vias	Clines	Shapes	Rats
	All	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Bus	Ad	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad4	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad7	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad9	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Gnd	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Gnd_Earth	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Vccint	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Vcco_Tahoe	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Vdd_Dis	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Vdd_Reg	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Vddio	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Vddio_Reg	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

**Disable Custom Colors** Choose to display all elements in their Class/Subclass color and disable the display of any highlighting or custom colors throughout the design. However, the highlight, custom color states, and custom colors assignments of elements are retained in the *Nets* grid.

**Clear All Nets** Removes the custom color and state from all nets in the database, as individual color and state boxes are applicable only to nets visible in the *Nets* grid.

**Nets Info**

**Type** Choose to display by nets, buses, differential pairs, or Xnets. *All* displays all nets. *List...* displays nets from an external list.

## Allegro Free Physical Viewer

### color192

---

<i>Filter</i>	Narrow the number of displayed nets using wildcards. The first found net appears in the grid
<i>Show Only Nets with Color Override</i>	Displays only those nets with color overrides. The names of nets with overrides appear in boldface.
<i>Sort</i>	Arranges nets based on an ascending or descending alphabetical order or clusters nets with overrides either first or last in the grid.
<i>Color</i>	Displays a palette of 192 modifiable colors, 96 of which appear at once. A rainbow gradient scheme initializes the color palette for all undefined colors, meaning at least the first 24 colors are always defined.
<i>Next/Prev</i>	Toggles the primary and secondary color palettes. The primary palette is the Cadence default setting, organized by a rainbow spectrum of 96 colors. The secondary palette comprises 96 color used for customization. The first 24 positions are reserved for colors used in pre-Release 16.0 databases.
<i>Customize</i>	Modifies colors, determined by the currently chosen color, using the <a href="#">Color Dialog Box</a> .
<i>Selected</i>	Indicates the current color.
<i>New</i>	Indicates the color your cursor is currently hovering over in the <i>Color</i> section.
<i>Ok</i>	Closes the dialog box.
<i>Cancel</i>	Closes the dialog box and does not save the changes.
<i>Apply</i>	Applies color, transparency, and visibility changes to the current design. The dialog box remains open so you can continue making modifications.
<i>Reset</i>	Resets color modifications to the previous color. This does not affect modifications made to the <i>Display</i> settings.

### Color Dialog Box

Use this dialog box to customize shades and hues of color. After moving the control on the vertical sliding bar for luminosity away from the extremes of white or black, you can move the

crosshair around the spectrum. All the fields in the dialog box reflect the correct number for the color in the crosshair. You can also type values in the fields to choose a color.

<i>Basic Colors</i>	Displays a selection of typical popular colors.
<i>Custom Colors</i>	Displays user-defined colors.
<i>Define Custom Colors &gt;&gt;</i>	A Microsoft Windows feature that is unavailable.
<i>Color   Solid</i>	Displays the color you created with the vertical sliding bar and crosshair.
<i>Hue</i>	Represents the chosen color's hue.
<i>Sat</i>	Represents the chosen color's saturation.
<i>Lum</i>	Represents the chosen color's luminosity.
<i>Red</i>	Represents the amount of red in the chosen color.
<i>Green</i>	Represents the amount of green in the chosen color.
<i>Blue</i>	Represents the amount of blue in the chosen color.
<i>Add to Custom Colors</i>	Moves the color you created with the vertical sliding bar and crosshair to the <i>Custom Color</i> section of available user-defined colors.
<i>Ok</i>	Closes the dialog box and saves your modifications.
<i>Cancel</i>	Closes the dialog box and does not save your modifications.

## Procedures

### Assigning Colors to Subclasses

1. Choose *Display – Color/Visibility*.

The Color dialog box appears.

2. Click *Layers*.
3. Choose a folder from the left pane that contains the subclass whose color you want to change.
4. In the *Color* section of the dialog box, use *Next >>* or *Prev >>* to display the primary or secondary color palettes.

5. Click the new color in the Color section. (This is the first click identified in Figure 1-1).

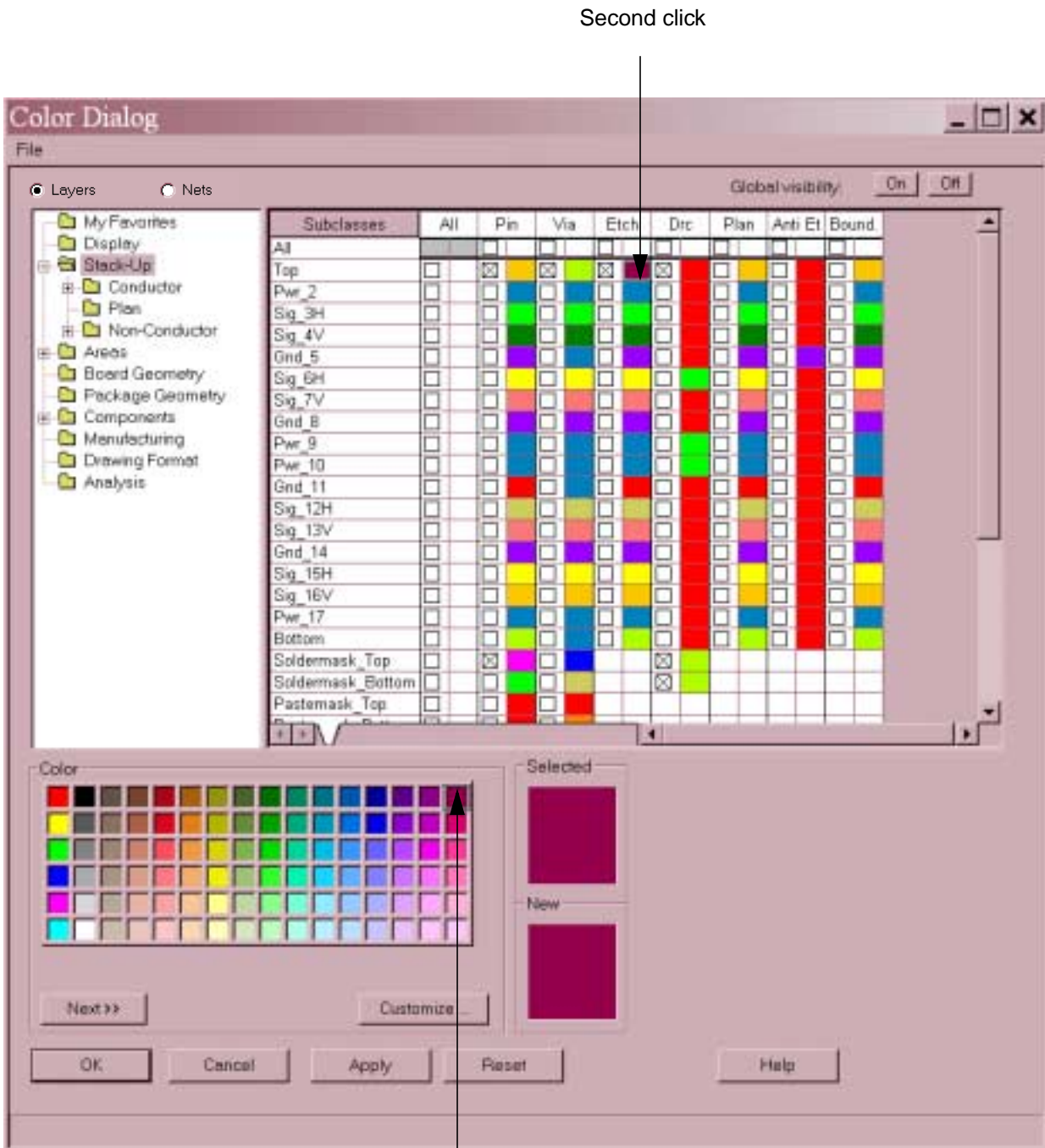
The New color box shows the color you have chosen.

6. Click the color box next to the subclass whose color you want to change. (This is the second click identified in Figure 1-1.) The color box for the subclass changes to the color you chose from the *Color* section. For example, if you choose *Stack-Up*, to change the color of the etch on the top layer, click the color box for *TOP ETCH*.

## Allegro Free Physical Viewer

color192

Figure 1-1 Changing Subclass Colors



7. Click *Apply* to update the drawing and continue changing colors.

- a. To apply the color to an entire row or column, click the color box next to the *All* column or *All* row.
  - b. To apply the color globally, click the color box next to the intersection of the *All* row and *All* column cell (*All/All* cell).
8. Click *OK* to save changes and close the dialog box.

The *Options* window panewindow pane displays the color assigned to a subclass in a color box next to the subclass name.

### Changing Grids, Ratsnest Lines, and Highlighting Colors

1. Choose *Display – Color/Visibility*.

The Color dialog box appears.

2. Click *Layers*.
3. Choose the *Display* folder from the left pane.
4. In the *Color* section, click a color box. This is the first click identified in Figure 1-1. It is the new color you want to assign.
5. In the *Display* section, click the color box next to the item (*Grids*, *Ratsnest*, *Temporary highlight*, or *Waived DRC*) whose color you want to change. This is the second click as shown in Figure 1-1.

The color box for this item changes to the color that you chose from the *Color* section.

6. Click *Apply* to update the drawing and continue changing colors.
7. Click *OK* to save changes and close the dialog box.

### Controlling Class and Subclass Visibility

1. Choose *Display – Color/Visibility*.

The Color dialog box appears. .

2. Choose a folder from the left pane.

The classes and subclasses in that folder appear.

3. Do any of the following:

- ☐ To make all classes and subclasses visible or invisible, click *On* or *Off* in the *Global visibility* field. To control visibility for an entire row or column, click the box next to



the *All* column or *All* row. All the subclasses in that class become visible, and an X appears in each box.

- ☐ To control visibility globally, click the box next to the intersection of the *All* row and *All* column cell (*All/All* cell), and an X appears in each box associated with that subclass.
- ☐ To control the visibility of an individual subclass, click its associated box, and an X appears in the box.

4. Click *Apply* to update the drawing.

5. Click *OK* to save changes and close the dialog box.

### **Customizing a Color**

1. Choose *Display – Color/Visibility*.

The Color dialog box appears.

2. In the *Color* section of the dialog box, click the color box for the color you want to change.

3. Click *Customize*. For details, see the [Color Dialog Box](#).

4. Choose a new color from the *Basic Colors* section or the *Custom Colors* section.

5. Move the control on the right hand side vertical sliding bar for luminosity away from the extremes of white or black. The *Hue*, *Sat*, *Lum*, *Red*, *Green*, and *Blue* fields display the numerical color values for the color chosen.

6. Move the crosshair around the spectrum until you have created the color you want.

The *Color / Solid* box displays the color you created with the vertical sliding bar and crosshair.

All the fields in the dialog box reflect the correct number for the color in the crosshair. You can also type values in the fields to choose a color.

7. Click *Add to Custom Colors*. The color box in the *Custom Colors* section dialog box shows the new color.

8. Click *OK* to save the changes and close the dialog box.

9. Click *Apply* in the Color dialog box to update the design with your color changes.

### **Saving a Customized Color Palette**

After you customize a color palette, you can save these settings for use with other designs and for future use with the current design.

1. Choose *Display – Color/Visibility*.

The Color dialog box appears.

2. Click *Apply* after making your color changes.

3. Choose *File – Save Color Palette*.

A file browser appears with the filter set to \*.col in the current local working directory. You can manually browse to other directories to save a color file.

4. Name the customized color palette and click *Save*.

The current design's customized color palette is saved.

**Note:** To revert to the default Cadence color palette, choose *File – Load Default Cadence Color Palette*.

### **Importing a Customized Color Palette**

1. Choose *Display – Color/Visibility*.

The Color dialog box appears.

2. Choose *File – Load Color Palette*.

A file browser appears with the filter set to \*.col and a list of all .col files available in the current local working directory. You can manually browse to other directories to open a color file.

3. Choose a customized color palette from the list and click *Open*.

The customized color palette is applied to the current design.

4. To revert to the default Cadence color palette, choose *File – Load Default Cadence Color Palette*.

### **Setting Transparency Globally**

1. Choose *Display – Color/Visibility* to display the Color dialog box.

2. Click *Layers*.
3. Choose the *Display* folder from the left pane.
4. In the *OpenGL* section, use the *Global transparency* slider bar to vary the level of intensity for the entire drawing.
  - a. Sliding the bar completely to the right (100%) represents a pre-16.0 graphics display.
  - b. Sliding the bar completely to the left (0%) causes previously filled geometry, such as clines and pads, to display with less intensity.

The change takes effect immediately.

### **Setting Transparency for Shapes**

1. Choose *Display – Color/Visibility* to display the Color dialog box.
2. Click *Layers*.
3. Choose the *Display* folder from the left pane.
4. In the *OpenGL* section, use the *Shapes transparency* slider bar to vary the level of intensity.
  - a. Sliding the bar completely to the right (100%) represents a pre-16.0 graphics display.
  - b. Sliding the bar completely to the left (0%) causes previously filled geometry to display with less intensity.

The change takes effect immediately.

### **Adding Subclasses to the My Favorites folder**

1. Run the `color192` command.

The Color dialog box appears.
2. Click *Layers*.
3. Choose a folder from the left pane.
4. Hover your cursor over the color box associated with the subclass you want to add to My Favorites.
5. Right-click and choose *Add to My Favorites* from the pop-up menu.

6. Add as many subclasses as necessary.

The subclasses are copied (not moved) to the *My Favorites* folder.

7. A subclass stored in the *My Favorites* folder can be removed by hovering your cursor over the color box associated with the subclass, right-clicking and choosing *Remove from My Favorites* from the pop-up menu.

### **Assigning Custom Color to Nets and Net Elements**

1. Run the `color192` command.

The Color dialog box appears.

2. Click *Nets*.

3. Choose the type of element to display. *All* displays all nets. *List...* displays nets from an external list.

4. In the *Color* palette of the dialog box, use *Next >>* or *Prev >>* to display the primary or secondary color palettes.

5. Click the new color in the *Color* palette.

The *New* color box shows the color that you have chosen.

6. Click the color box next to the net or net element whose color that you want to change.

The color box changes to the color you chose from the *Color* section. The custom color state is enabled, indicated by the X that automatically appears in the box to the left of the color box.

To assign a custom color to all net elements of a particular type, (all Pins, for example) click the color box in the *All* row. (Note: The *All* row for *Net* is grayed out by design).

7. Click *Apply* to have the color modifications appear in the design canvas.

Any color assigned to these elements applies to lower-level nets and only to those nets without explicit custom color.

### **Assigning Custom Color and Highlighting from Nets and Net Elements**

1. Run the `color192` command.

The Color dialog box appears.

2. Click *Nets*.

## Allegro Free Physical Viewer

color192

3. Hover your cursor over a box with no X in it to the left of the element's color box.
4. Right click, then choose *Set Highlight State*.

Type	Nets	Net	Pins	Vias	Clines	Shapes	Rats
	All	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Bus	Ad	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad4	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Clear Custom Color

Clear Highlight State

Set Highlight State

The element becomes highlighted in the design canvas, and its name displays in boldface in the Nets grid.

### Removing the highlight state from nets and net elements

1. Hover your cursor over an element's color box.
2. Right click, then choose *Clear Highlight State*.

Type	Nets	Net	Pins	Vias	Clines	Shapes	Rats
	All	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Bus	Ad	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<b>Net</b>	<b>Ad2</b>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad4	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Clear Custom Color

Clear Highlight State

Set Highlight State

The highlighting disappears from an element and its name displays in regular typeface in the Nets grid. Its custom color is preserved in the design canvas, and its custom color assignment remains in the Nets grid.

## Removing Custom Color, Highlighting, and States from Nets and Net Elements

1. Run the `color192` command.

The Color dialog box appears.

2. Click *Nets*.

Hover your cursor over a color box, right click and choose *Clear Custom Color*.

Type	Nets	Net	Pins	Vias	Clines	Shapes	Rats
	All	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Bus	Ad	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<b>Net</b>	<b>Ad2</b>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Net	Ad3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Clear Custom Color  
 Clear Highlight State  
 Set Highlight State

The highlight state and the custom color disappear from the element in the design canvas. The custom color assigned to the element in the *Nets* grid is also removed and no longer retained there. (A color box without a custom color assigned to it has no custom color state.) The element then displays using the Class/Subclass color.

Type	Nets	Net	Pins	Vias	Clines	Shapes	Rats
	All	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Bus	Ad	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad10	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
<b>Net</b>	<b>Ad2</b>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad5	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad7	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Net	Ad9	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

## Overriding Custom Colors

1. Run the `color192` command.

The Color dialog box appears.

2. Click *Nets*.
3. Click the new color in the *Color* palette. Use *Next >>* or *Prev >>* to display the secondary color palette if required.

The *New* color box shows the color that you have chosen.

4. Click the color box next to the net or net element whose color you want to override.

The color box changes to the color you chose from the *Color* section. The custom color state is enabled, indicated by the X that automatically appears in the box to the left of the color box.

### **Managing the Display of Nets and Net Elements**

1. Run the `color192` command.

The Color dialog box appears.

2. Click *Nets*. Do one of the following:

- Choose *Filter* to narrow the number of displayed nets. The first found net appears in the grid.
  - Click *Show Only Nets with Color Override* to only display nets with color overrides.
3. Click *Sort* to arrange nets based on an ascending or descending alphabetical order or to cluster net with overrides first or last in the grid.

## colorview create

[Dialog Box](#) | [Procedures](#)

Creates or changes a color visibility view, a collection of layer visibility settings that you can apply to subsequent designs using the *View* field on the Visibility form of the control panel. A color view can also display film record visibility settings stored in the current design, unless you suppress the film record names from the list of color views.

You save your settings in a `.color` file that is stored in the current directory.

**Note:** The procedures section for this command includes instructions for deleting a color visibility view.

Related commands are `colorview load` and `colorview restore`.

### Menu Path

*View – Color View Save*

## Color Views Dialog Box

Use this dialog box to create a color visibility view or change an existing one.

<i>Save view</i>	Enter the name of the color visibility view file to which you want to save the current layer visibility settings. The tool automatically appends the <code>.color</code> file extension and stores the file in the current working directory.  Click ... to browse for an existing filename and overwrite its contents.
<i>View Replacement Method</i>	Specifies modes for creating a color visibility view.
<i>Complete</i>	Saves the current layer visibility settings to a color view file. When you load the file later, it completely replaces the design's visibility settings, analogous to how the film option to color views works.



*Partial*

Allows a color view file to be created that stores only *changes* to visibility settings. A partial color view does not replace all of a design's visibility settings when loaded. It only replaces the items you changed when you created the color view file.

For example, if you changed the color for all DRCs to red, when you loaded that color view file into a design, only the DRCs would be affected, all changing to red. All other visibility settings would remain unchanged.

*Partial with toggle*

Functions the same as the *Partial* view replacement method because the color view file only stores changes. However, settings that you change toggle when you load the color view file.

Toggle means that if the visibility for a layer is on in a design, when you load the color view file, it is turned off. If off, it is turned on.

*Close*

Closes the dialog box without creating a new color view file or saving changes to an existing file.

**Note:** To save a new or changed file, you need to click *Save* before clicking *Close*.

## Procedures

### Creating a Color Visibility View

1. Run the `colorview create` command.

The Color Views dialog box appears. For details, see [Color Views Dialog Box](#).

2. In the *Save view* field, enter the name of the color visibility view.
3. For *View Replacement Method*, choose a method.

4. If you selected either of the *Partial* view replacement methods, change visibility settings in the Color dialog box (using the [color192](#) command) or in the Visibility form of the control panel.
5. In the Color Views dialog box, click *Save* and then *Close*.

### Changing a Color Visibility View

1. Run the `colorview create` command.

The Color Views dialog box appears. For details, see [Color Views Dialog Box](#).

2. In the *Save view* field:

Enter the name of the file for the color visibility view you want to edit.

—or—

Click ... and browse for the file.

3. For *View Replacement Method*, choose a method.
4. If you selected either of the *Partial* view replacement methods, change visibility settings in the Color dialog box (using the [color192](#) command) or in the Visibility form of the control panel.
5. In the Color Views dialog box, click *Save* and then *Close*.

### Changing a Color Visibility View

1. Run the `colorview create` command.

The Color Views dialog box appears. For details, see [Color Views Dialog Box](#).

2. In the *Save view* field:

Enter the name of the file for the color visibility view you want to edit.

—or—

Click ... and browse for the file.

3. For *View Replacement Method*, choose a method.
4. If you selected either of the *Partial* view replacement methods, change visibility settings in the Color dialog box (using the [color192](#) command) or in the Visibility form of the control panel.
5. In the Color Views dialog box, click *Save* and then *Close*.

### **Deleting a Color Visibility View**

1. Locate the directory where the file for the color visibility view resides. It has a `.color` extension.
2. Delete the file

## colorview restore

Restores the previous color visibility view you used in the current session. You can also toggle between two color views using this command. A color visibility view stores a collection of layer visibility settings.

Related commands are `colorview create`.

### Menu Path

*View – Color View Restore Last*

### Applying the Previous Color Visibility View

1. Run the `colorview restore` command to apply the color view that preceded the current color view.
2. To toggle back and forth between the two color views, rerun `colorview restore`.

### cns\_show

Allows you to generate a report that provides details about constraints that apply to an object or pair of objects you select. The report appears in a separate window which offers print, search, and save-to-disk functions. The report includes:

- Net owner
- Net Class membership
- Net Class–Net Class membership
- any overrides
- applicable constraint areas

You are prompted to select a single object or a pair of objects. Legal objects are:

- nets
- pins
- vias
- clines
- etch shapes
- ratsnests

For a single object, physical constraints are reported. For a pair of objects, spacing constraints are reported. For nets, all applicable areas are reported. For non-nets (physical objects), the applicable area is determined based on the pick location. All constraints are resolved and reported for physical objects based on their pick location.

### Menu Path

*Display – Constraint*

### Toolbar Icon



## Generating a Constraint Report

1. Run the `cns_show` command.
2. Click on a single object to select it.

-or-

Drag a window around a pair of objects to select them.

The Show Constraints window appears with the constraint information for the object or pair of objects you selected.

3. In the *Constraint Hierarchy* table, click the blue colored text in a *Location* cell to jump to the object in the design window.
4. In the *Constraint Hierarchy* table, click the blue colored text in any of the object cells to open the corresponding worksheets in Constraint Manager.
5. In the *Resolved Spacing Constraints* table, click the blue colored text in any of the cells under the *Source Name* column to open the corresponding worksheets in Constraint Manager.
6. Use the *Save*, *Print*, or *Search* functions in the Show Constraints window to work with the report information.

## dehighlight

[Options Tab](#) | [Procedure](#)

Removes the highlighting pattern from elements, which consists of an alternating checkerboard of the element's color and the temporary highlight color as defined in the *Display* category of the Color dialog box, available by choosing *Display – Color/Visibility* (`color192` command).

This command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command. Valid elements are:

- Symbols
- Functions
- Nets
- Pins
- DRC errors

### Menu Path

*Display – Dehighlight*

### Toolbar Icon



### Options Tab for the dehighlight Command

When you access the command in the pre-selection use model from the right-mouse button pop-up menu, these settings are unavailable.

#### *Active Class and Subclass*

The upper drop-down list box displays the current class of the element that you have chosen for dehighlighting, as well as the color that is assigned to it; the lower drop-down list box, the current subclass with choices for modifying the value.

#### *Retain Objects Custom Color*

If enabled, only removes the highlight state from an element, and preserves the display of its custom color in the design canvas, while retaining the element's custom color assignment in the *Nets* grid of the Color dialog box.

If disabled, removes the highlight state and the custom color from the element in the design canvas. The custom color assigned to the element in the *Nets* grid is also removed and no longer retained there. (A color box without a custom color assigned to it has no custom color state.) The element then displays using the Class/Subclass color.

### *Dehighlight all*

Click *Nets*, *Symbols*, *Functions*, or *Pins* to simultaneously dehighlight all nets, symbols, functions, or pins, respectively.

## **Procedure**

### **Dehighlighting Elements**

Do one of the following:

1. Hover your cursor over an eligible element.
2. Right-click and choose *Dehighlight* from the pop-up menu to automatically launch the command.

The highlighting disappears from the element, and the Command window pane displays the following message:

```
<element type><element name> dehighlighted
```

**Note:** The *Retain Objects Custom Color* option is unavailable when you access the command in the pre-selection use model from the right mouse button pop-up menu.

—or—

3. Choose *Display – Dehighlight* (dehighlight command).

The *Options*, *Find*, and *Visibility* foldable window panes appear depending on whether their visibility was enabled before you ran the command. If these panes were hidden prior to executing the command, they will not appear. Choose *View – Windows* to display the foldable window panes.

4. To remove only the highlight state from an element, click *Retain Objects Custom Color*, which also preserves the display of the element's custom color in the design canvas, while retaining its custom color assignment in the *Nets* grid of the Color dialog box. To remove both the highlight state and the custom color from the element in the



design canvas and from the *Nets* grid, uncheck this option. The element then appears using the Class/Subclass color.

5. Click the element to highlight, or click *Nets*, *Symbols*, *Functions*, or *Pins* to simultaneously dehighlight all nets, symbols, functions, or pins, respectively.

The highlighting disappears from the element, as does the custom color depending on whether you enabled or disabled the *Retain Objects Custom Color* option. The Command window pane displays the following message:

```
<element type><element name> dehighlighted
```

6. Right click and choose *Done* from the pop-up menu.

—or—

7. Click .

The *Options*, *Find*, and *Visibility* foldable window panes appear depending on whether their visibility was enabled before you ran the command. If these panes were hidden prior to executing the command, they will not appear. Choose *View – Windows* to display the foldable window panes.

8. To remove only the highlight state from an element, click *Retain Objects Custom Color*, which also preserves the display of the element's custom color in the design canvas, while retaining its custom color assignment in the *Nets* grid of the Color dialog box. To remove both the highlight state and the custom color from the element in the design canvas and from the *Nets* grid, uncheck this option. The element then appears using the Class/Subclass color.
9. Click the element to highlight, or click *Nets*, *Symbols*, *Functions*, or *Pins* to simultaneously dehighlight all nets, symbols, functions, or pins, respectively.

The highlighting disappears from the element, as does the custom color depending on whether you enabled or disabled the *Retain Objects Custom Color* option. The Command window pane displays the following message:

```
<element type><element name> dehighlighted
```

10. Right-click and choose *Done*.

## exit

Saves the active layout, exits, and returns to the host operating system. The command displays a browser window asking for a name under which to save the active layout. The default is the name of the active layout. If you do not enter a name but click *OK*, the command displays a dialog box asking whether you want to overwrite the existing layout and exits. If you enter a new name, the command writes the layout to that filename and exits.

## Co-Design Environment

In a co-design environment, the `exit` command checks for unsaved co-design dies and asks you whether to save or discard the changes.

## Menu Path

*File –Exit*

## grid toggle

The `grid toggle` command turns on/turns off the grid display in your user interface.

**Note:** This command is not available in Allegro PCB SI L, XL, or GXL or Allegro Package Signal Integrity.

### Toolbar Icon



## help

Displays information available in the help system.

### Help Menus

The *Help* menu path varies according to the Cadence user interface with which you are working. Most physical verification and IC packaging tools provide access to a command reference, one or more user guides, migration guides, known problems and solutions, and product notes. Some tools may have tutorials and/or flow design documentation.

### Toolbar Icon



### Console Window Prompt Help

You can also access help on a command by typing

```
help <command_name>
```

at the console window prompt.

**Note:** The Cadence product you are using may not feature a command console in the user interface.

### F1 Help

You can access help on any command by highlighting the menu selection in the user interface, then pressing the F1 function key. When a command is already active, you can also get help by pressing the F1 function key.

## highlight

### Procedure

Lets you accentuate certain elements with a pattern—or striping—comprising the element's base subclass color and the temporary highlight color defined in the *Display* category of the Color dialog box, available by choosing *Display – Color/Visibility* (`color192` command). Striping is only visible when the `display_nohilitefont` variable is disabled.

Once the element becomes highlighted in the design canvas, its name also appears in a bolded font in the *Nets* section of the Color dialog box.

Elements highlighted with this command stay highlighted until you choose *Display – Dehighlight* (`dehighlight` command) to disable the highlighting.

This command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command. Valid elements are:

- Symbols
- Functions
- Nets
- Pins
- DRC error markers

A related command is *Display – Assign Color* (`assign color` command), which assigns a color and highlights an element without requiring the use of the Color dialog box or this command. For more information, see *Working with Highlighting and Coloring* in the *Getting Started with Physical Design* section in your documentation set.

### **Cross-Probing in APD and SiP Tools**

When you edit a co-design die in APD or SiP tools, you can highlight a pin on the die or a net that connects to the die to activate cross-probing in I/O Planner. If you highlight a co-design die pin, then the corresponding pin on the die is selected in I/O Planner. If you highlight a net in APD or SiP and there is a co-design die pin connected to the highlighted net, then that pin and the corresponding net are selected in I/O Planner.

### **Menu Path**

*Display – Highlight*

## Options Tab for the highlight Command

When you access the command by clicking the toolbar icon or choosing *Display – Highlight*, the *Options* window pane appears with the current Temporary Highlight color as defined in the *Display* category of the Color dialog box.

### *Default Highlight Color*

Displays the current Temporary Highlight color that will be used to accentuate chosen elements.

## Procedure

### Highlighting Elements

1. Do one of the following:

- a. Hover your cursor over an element.
- b. Right-click and choose *Highlight* from the pop-up menu to automatically launch the command.

The element becomes highlighted with the default highlight color as shown in the *Options* window pane, and the Command window pane displays the following message:

`<element type><element name> highlighted`

—or—

- c. Choose *Display – Highlight* (highlight command).

The the *Options*, *Find*, and *Visibility* foldable window panes appear depending on whether their visibility was enabled before you ran the command. If these panes were hidden prior to executing the command, they will not appear. Choose *View – Windows* to display the foldable window panes.

The *Find* window pane lists the objects that you can highlight for this command. The *Options* window pane shows the default highlight color that you can use to highlight an element.

- d. Click the element to highlight.

The element becomes highlighted with the default highlight color as shown in the *Options* window pane, and the Command window displays the following message:

`<element type><element name> highlighted`

- e. Right-click and choose *Done* from the pop-up menu.

## layer priority

Lets you manage the order in which layers appear, by assigning a display priority to each layer, and overriding the default display order. Elements are drawn based on their assigned layer priority. Your assignments are saved with the board when you click *Apply*. Always-on-top elements include:

- Temporary or transitory objects such as DRCs or ratsnests
- Active subclass

## Menu Path

*Display – Layer Priority*

## Display Priority Dialog Box

Use this dialog box to control the order in which layers are drawn in your design. For example, the default layer at the top of the list appears on top of the layer that appears second in the list.

<i>Default Priority</i>	Shows in a collapsing tree view, the default display priority for all layers in your design. Where a number of layers are listed, the display area shows a folder icon. You can choose all layers by clicking the check box next to the icon or individual layers by clicking the check box next to the layer name.
<i>Prioritized Layers</i>	Layers in this list are drawn before those contained in the <i>Default Priority</i> list.
<i>-&gt;</i>	Moves the chosen layer from the <i>Default Priority</i> list to the <i>Prioritized Layers</i> list.
<i>Up</i>	Swaps the chosen layer with the layer immediately above it in the <i>Prioritized Layers</i> list.
<i>Down</i>	Swaps the chosen layer with the layer immediately below it in the <i>Prioritized Layers</i> list.
<i>Top</i>	Moves the chosen layer to the top of the <i>Prioritized Layers</i> list.
<i>Bottom</i>	Moves the chosen layer to the bottom of the <i>Prioritized Layers</i> list.



## Allegro Free Physical Viewer

### layer priority

---

<-	Removes the chosen layer from the <i>Prioritized Layers</i> list.
<<-	Removes all layers from the <i>Prioritized Layers</i> list.
OK	Saves your changes and closes the dialog box.
Cancel	Exits the dialog box.
Apply	Saves layer priority assignments with the board.

### Assigning a Display Priority To Layers

1. Choose *Display – Layer Priority* (`layer priority` command). The Display Layer Priority dialog box displays.
2. Choose a layer from the *Default Priority* list, and click -> to move it to the *Prioritized Layers* list. Continue to move as many layers as required. Layers in the *Prioritized Layers* list will be drawn before any layers in the *Default Priority* list.
3. Reorder any layers in the *Prioritized Layers* list by choosing layers and doing any of the following:
  - ☐ Click *Up* to swap the chosen layers with the layer immediately above it in the *Prioritized Layers* list.
  - ☐ Click *Down* to swap the chosen layer with the layer immediately below it in the *Prioritized Layers* list.
  - ☐ Click *Top* to move the chosen layers to the top of the *Prioritized Layers* list.
  - ☐ Click *Bottom* to move the chosen layer to the bottom of the *Prioritized Layers* list.
4. Click <- to remove several layers from the *Prioritized Layers* list.
5. Click <<- to remove all layers from the *Prioritized Layers* list.
6. Click *Apply* to save layer priority assignments with the board.

## open

Opens an existing design file in the current directory. You are prompted to save or discard changes in the current open file. A file browser lets you search for the specified design file if you do not provide a file name. A list of your most recently used (MRU) files appears.

### Menu Path

*File – Open*

### Toolbar Icon



### Syntax

You can run the `open` command from the console window prompt. The syntax is:

```
open [<design to open>]
```

If you do not provide the `<design to open>` argument, a browser window opens in the current directory.

### Examples

```
open master.brd
```

The `master.brd` file opens in the current directory.

```
open \boards\master.brd
```

The `master.brd` file, located in the `boards` directory in the current directory, opens.

### Dialog Box

The Open dialog box is a standard file browser. Two buttons appear below the *Help* button. The left button lets you display a text preview of the current design; the right button lets you display the graphics preview of the design. The preview area appears on the right side of the list box.

## Procedure

### Opening an Existing File

1. Run the `open` command.

The file opens in the current directory. If you do not provide a *design to open* argument, the Open dialog box opens in the current directory.

2. Choose a file from the list.

You can also enter the file name in the *File name* field.

3. Click the left button below the *Help* button to display a text preview of the specified file.

The preview area appears on the right side of the *File name* list.

4. Click the right button below the *Help* button to display the graphics preview of the specified file.

5. Click *Open* to open the file.

## plot

[Dialog Box](#) | [Procedure](#)

The `plot` command lets you preview a plot as it will look when printed. When you choose `plot preview`, the user interface changes to preview the active design as it will plot based on the setup parameters in the *Plot Setup* dialog box and/or the Windows *Print Setup* dialog box.



Windows and Unix operating systems handle plotting differently. See the sections on plotting that are appropriate to the operating system you use.

On Unix operating systems, successful plotting involves correct set-up and the creation of IPF and control files, as well as the `.cdsplotinit` plotter configuration file, which lists available printers/plotters. The `.cdsplotinit` file must reside in `<install_path>/tools/plot`, the current working directory, or your home directory. See *Preparing Manufacturing Data* in the user guide of your documentation set.

On Unix, Allegro PCB Editor and Allegro Package Designer recognize the *Vectorize text* setting on the *Plot Setup* dialog box, available with the `plot setup` command, to permit direct plotting of nonvectorized text with the *File – Plot* command.

### Menu Path

*File – Plot*

### Print Dialog Box

The `plot` command on Windows runs the standard *Windows Print* dialog box.

On Unix, the `plot` command runs the *Plot* dialog box, that contains the following controls:

<i>Print To File</i>	Indicates the plot file is to be sent to the named file.
<i>Printer Name</i>	Indicates the name of the plotter the plot file is to be sent to.
<i>Pen Numbers</i>	Displays the <i>Plot Preference</i> dialog box for assigning colors to pens.
<i>Cancel</i>	Ignores input and closes the dialog box.

**OK**

Creates the plot and closes the dialog box.

## Procedures

### Plotting Prerequisites on a Unix Workstation

To run the `plot` command, a plotter configuration file named `.cdsplotinit` must reside in `<install_path>/tools/plot`, the current working directory, or your home directory.

If a `.cdsplotinit` file resides in multiple locations, the program looks down each path in turn and adds any new information or replace any old information as it is found.

The `.cdsplotinit` file contains information vital to the operation of the `allegro_plot` program such as:

- the name of the output device,
- the output format to be used for the device,
- the paper sizes available for the device,
- the maximum number of pages allowed on the device,
- the Unix commands for spooling jobs to the queue,
- checking the jobs in the queue
- removing jobs from the queue on the device
- other device specific information

The following is a sample `.cdsplotinit` file entry:

```
bos1|Apple LaserWriter II NT/NTX: \  
:manufacturer=Apple Computer: \  
:spool=lpr -Pbos1: \  
:query=lpq -Pbos1: \  
:remove=lprm -Pbos1 $3: \  
:type=postscript1: \  
:maximumPages#30: \  
:resolution#300: \  
:paperSize="A" 2400 3150 75 75: \  
:paperSize="A4" 2332 3360 60 60:
```

For detailed information on setting up the `.cdsplotinit` file, refer to the *Plotter Configuration User Guide*, available on SourceLink.

### Plotting Your Design on Unix

1. Before running `plot`, you must set up your plotting parameters as described in the procedure section of `plot setup`. You must also have created a plotter configuration file, as described in the section above.
2. When setup is complete, run the `plot` command to display the *Plot* dialog box.
3. To direct output to a file, choose *Print to file*. To write the design to the current working directory, enter only a filename. To direct the design file to another location, enter the full path.

—or—

To direct output to a plotter, choose *Printer name* and choose the printer name from the drop-down menu.

4. If necessary, click *Pen Numbers* to make color-to-pen assignments in the Plot Preference dialog box.
5. For each pen assignment you want to change, highlight the pen number and enter a new number.

Each number corresponds to a pen on your plotter. You assign each color in the palette to a corresponding pen number. If there are more colors in your drawing than there are pens in your plotter, assign more than one color to each pen. You should not have a number on your palette higher than the pen numbers in your plotter.

6. Click *OK* to close the dialog box.
7. In the *Plot Setup* dialog box, click *OK* to print or create the design file.

### Plotting Your Design on Windows

You can preview your plot on Windows before producing it.

1. Run the `plot` command to display the *Print* dialog box.
2. Choose the print resolution in the *Print quality* field.

If you want to direct output to a file, check *Print to file*. To write the design to the current working directory, enter only a filename. To direct the design file to another location, enter the full path.

## Allegro Free Physical Viewer

### plot

---

3. If necessary, click *Setup* to set additional printing options in the Windows *Print Manager* dialog box.
4. Click *OK* to print or create the design file.

## plot setup

[Dialog Box](#) | [Procedure](#)

The `plot setup` command lets you set parameters for plotting a design. (See the `plot` command for additional details on plotting.) Although plotting procedures vary according to the operating system you are running, the procedure for `plot setup` is the same for Unix and Windows.

### Menu Path

*File – Plot Setup*

### Plot Setup Dialog Box

**Note:** The `.ini` file retains parameters set in the *Plot Setup* dialog box. Therefore, they remain in effect for every database you open until you change the parameters.

#### General Tab

##### *Plot scaling*

*Fit to page:* Indicates the plot file is to be scaled to fit the entire plotted page.

*Scaling factor:* Indicates the scale of the finished plot.

*Default line weight:* Converts any zero width line to a width proportional to the setting. Aids in displaying very thin lines on high-resolution output.

##### *Plot orientation*

*Auto center:* Centers the design on the plot page. This control automatically invokes when you choose the Fit to page setting.

*Mirror:* Flips the design end-for-end about the Y axis. Useful for viewing top and/or bottom layers.

##### *Plot method*

*Color:* Directs the output to print in color. Color is determined by the method specific to the platform you are using. On Unix, color is read from a user-supplied stipples file (`allegro_plot_param.stipples`); if a stipples file is not found, plotter color defaults are used. On Windows, color selection is determined by setting in the Color and Visibility



## Allegro Free Physical Viewer

### plot setup

---

dialog box.

*Black and white:* Directs the output to print in black and white.

#### *Plot contents*

*Screen contents:* Prints/plots the contents of what is currently displayed in the design area of the user interface.

*Sheet contents:* Prints/plots the extents of all currently-visible graphics within the design (not the drawing extents).

#### *IPF setup*

*Vectorize text:* Specifies that text output to the IPF file is broken into line vectors. The environment variable `PLOT_VECTORIZE_TEXT` determines whether the *Vectorize text* option is enabled or disabled by default.

*Width:* specifies the width for simulating the text characters. The width used is established with the environment variable `PLOT_VECTEXT_WIDTH`. The default is 0.

When *Vectorize text* is enabled, and a negative value is entered in the *Width* field, any other width setting of 0 or greater causes photoplot widths to be ignored, and all text is uniformly stroked with the same specified width. The *Vectorize text* and *Width* settings apply as specified when the `create plot` command executes.

Environment variables are the initial default settings for the Plot Setup dialog box. If you modify the settings in the dialog box, the new settings override any environment variable settings that you may have specified. When you exit, the current dialog box settings are saved in the `.ini` file. These `.ini` file settings are then used in the next session, and again override any specified environment variable settings.

If you modify the environment variable settings after changing settings in the *Plot Setup* dialog box, these new environment variable settings are not used. You must delete the `.ini` file, and then the new environment variable settings take effect.

You can set environment variables using procedures, based on the platform you are running. You can also set these environment variables using *Setup – User Preferences – Plot*.

## Allegro Free Physical Viewer

### plot setup

---

<i>OK</i>	Saves the settings and closes the dialog box.
<i>Cancel</i>	Closes the dialog box without saving the settings.

### Windows Tab

Only appears on the Windows platform. The `.ini` file retains all settings between sessions.

### Non-vectorized Text Control

<i>Non-Vectorized Text</i>	Choose to generate plot files with true font text , which lets you generate PDF-format plot output with searchable text.
<i>Font</i>	Specifies a font to use; defaults to Courier.
<i>Font Height</i>	Enter a percentage scaling factor for the character height to closely match font text with that of the normal vectorized text display/plot.
<i>Font Width</i>	Enter a percentage scaling factor for the character width to closely match font text with that of the normal vectorized text display/plot.
<i>View Available Fonts</i>	Click to review the available text fonts for the plot device.

### Margin Control

<i>Margin Width</i>	Specify the desired margin width in user units. The default equates to 0.25 inches, or 0.0 if the <code>noplotmargins</code> environment variable is set.
<i>OK</i>	Saves the settings and closes the dialog box.
<i>Cancel</i>	Closes the dialog box without saving the settings.

## Procedure

### Setting Parameters for Plotting a Design

1. Adjust the visibility of the display layer and the view (zoom) level.
2. Run `plot setup` to display the Plot Setup dialog box.

## Allegro Free Physical Viewer

### plot setup

---

3. Set plot parameters as described in the section above. Parameters that you set in Plot Setup are retained in the `.ini` file. Therefore, they remain in effect for every database you open until you change the parameters.
4. Click *OK* to save the settings.

## prmed

[Dialog Boxes](#) | [Procedures](#)

The `prmed` command displays the Design Parameter Editor, which provides a convenient, centralized location for editing parameters that are saved and stored in the database. In the Design Parameter Editor, select tabs for *Display*, *Design*, and *Text* and edit the specific parameters in each of these categories.

**Note:** With Allegro Free Viewer and Allegro Viewer Plus, only the *Display*, *Design*, and *Text* tabs are available.

### Menu Path

*Setup – Design Parameters*

### Toolbar Icon



## Design Parameter Editor Dialog Box

Use this dialog box to edit the parameters you want to apply to the design. Hover your cursor over each parameter and a description of its functionality displays in the Parameter Description area of the dialog box. The parameters are grouped under the following tabs:

<i>Display</i>	Lists parameters that control the display of the design.
<i>Design</i>	Lists parameters that control the drawing size and extents, line lock and text controls.
<i>Text</i>	Lists parameters that control the display of text.

## Procedures

### Changing Display Parameters

1. Click the *Display* tab.



**Tip**

When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.

2. In the *Display* group box, enter new values for the parameters you want to change.
3. In the *Enhanced Display Modes* group box, enable or disable the check boxes to either display or hide particular objects.
4. In the *Grids* group box, enable *Grids on* to display the grids. Click *Setup Grids* to display the Define Grid dialog box and specify the grid spacings you want to use for different layers.
5. Click *Apply* to apply the changes.

### Changing Design Parameters

1. Click the *Design* tab.



**Tip**

When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.

2. In the *Size*, *Extents*, *Move Origin*, *Symbol* and *Drawing Type* group boxes, enter new values for the general design parameters you want to change.
3. In the *Line Lock* group box, enter new values for *Lock direction*, *Lock mode* and *Minimum radius*.
4. Click *Apply* to apply the changes.

### Changing Text Parameters

1. Click the *Text* tab.



**Tip**

When you hover your cursor over a parameter, the full description of that parameter is shown in the *Parameter description* group box.

2. In the *Size* group box, enter new values for *Justification*, *Parameter block* and *Text marker size*.

3. Click *Setup Text Sizes* to display the Text Setup dialog box and specify new parameters for the text blocks.
4. Click *Apply* to apply the changes.

## **rats all**

The `rats all` command displays all existing ratsnest lines in your design.

To control the color of ratsnest lines, use the [`color192`](#) command. To display ratsnest lines as straight or jogged lines, run the `prmed` command to display the *Design Parameter Editor*, click the *Display* tab and set *Ratsnest Geometry*.

### **Menu Path**

*Display – Show Rats– All*

### **Toolbar Icon**



## **Procedure**

### **Displaying All Existing Ratsnest Lines in Your Design**

- Run `rats all`.

All ratsnest lines in the design display.

## **rats component**

Displays existing ratsnest lines attached to component pins. To control the color of ratsnest lines, use the color192 command.

To display ratsnest lines as straight or jogged lines, use the *Design Parameter Editor*, click the *Display* tab and set *Ratsnest Geometry*.

### **Menu Path**

*Display – Show Rats– Component*

### **Procedure**

#### **Displaying Existing Ratsnest Lines Attached to Component Pins**

1. Run `rats component`.
2. Choose a component.

Ratsnest lines to pins on the components that you choose are displayed.



## rats net

Displays existing ratsnest lines attached to pins on a net. To control the color of ratsnest lines, use the color192 command.

To display ratsnest lines as straight or jogged lines, use the *Design Parameter Editor*, click the *Display* tab and set *Ratsnest Geometry*.

### Menu Path

*Display – Show Rats– Net*

## Procedure

### Displaying Existing Ratsnest Lines Attached to Pins on a Net

1. Run `rats net`.
2. Choose a net.

Ratsnest lines to pins on the nets that you choose are displayed.

## **redraw**

Refreshes the work area.

### **Menu Path**

*View – Refresh*

### **Toolbar Icon**



## reset\_dockwindows

Restores the *Options*, *Worldview*, *Find*, *Visibility*, and *Command* foldable window panes to display in their original positions.

To show all window panes in the positions in which you last viewed them, use *View – Windows – Show All* (`show_allpanes` command).

### Menu Path

*View – Windows – Reset to Default*

### Syntax

`reset_dockwindows`

## Displaying All Foldable Window Panes to Default Positions

1. Choose *View – Windows – Reset to Default*.

The *Options*, *View*, *Find*, *Visibility*, and *Command* foldable window panes display in their original positions.

## script

[Dialog Box](#) | [Procedures](#)

The `script` command records a series of actions. It creates a text file containing the commands that you execute and adds a `.scr` extension to the file name. You can use scripts to perform global tasks such as setting up dialog box options, adding elements to multiple databases at the same location, and duplicating drawings. Using the interactive version of the `script` command that displays the Scripting dialog box, you can also replay the script.

A macro is a script that lets you automate a series of point selections and replay them, starting at another coordinate. When you replay a macro, Allegro PCB Editor prompts you for a starting point (origin). The macro places the point selections you recorded relative to this starting point. This is useful in performing operations that you need to repeat on a board/design drawing, such as repeating complex geometric operations.

The current settings in your design are recorded in the script or macro. To display the script with different settings, you must change them as part of the script.

### Menu Path

*File – Script*

## Scripting Dialog Box

### *Script File*

<i>Name</i>	Specifies the name of the file in which you record your actions. Allegro PCB Editor adds the <code>.scr</code> extension to the file name.
<i>Browse</i>	Displays the script file data browser that lets you choose a script file to replay.
<i>Library</i>	Displays the script file data browser that lets you choose a script file to replay. Opens to your script path location.
<i>Generate</i>	Displays a file browser from which you can choose a <code>.jrl</code> file to convert into a script without having to leave the current environment. To process the journal file and reconstruct the appropriate script outside of Allegro PCB Editor, run:

```
j2script <source_jrl_file> <target_allegro_script>
```

## Allegro Free Physical Viewer

### script

---

#### *Record/Replay*

<i>Macro record mode</i>	Specifies whether or not you record as a macro. When replaying, a macro requires a starting point.
<i>Record</i>	Starts recording your actions.
<i>Stop</i>	Stops recording your actions or replaying a script.
<i>Replay</i>	Starts replaying a macro or script.
<i>Cancel</i>	Closes the dialog box.
<i>Help</i>	Displays the Help window.

## Procedures

### Creating a Script

1. Run the `script` command.  
The Scripting dialog box appears.
2. In the *Name* text box, enter a name for the script.
3. Click *Record*.  
The Scripting dialog box disappears.
4. Perform the tasks that you want the script to run.  
The name of the file and the *Rec* status appears in the Status window.
5. Run `script` again, then click *Stop* in the Scripting dialog box.

### Creating a Macro

1. Run the `script` command.  
The Scripting dialog box appears.
2. In the *Name text* box, enter a name for the macro.
3. Click *Macro Record Mode*.
4. Click *Record*.  
The Scripting dialog box disappears.

5. Perform the tasks that you want the macro to run.

The name of the file and the *Rec* status appears in the Status window.

6. Run `script` again, then click *Stop* in the Scripting dialog box.

### Replaying a Script

1. Run the `script` command.

The Scripting dialog box appears.

2. In the *Name* text box, enter the name of the script that you want to replay.

If necessary, use the *Browse* button to locate the correct file.

3. Click *Replay*.

The script replays.

### Replaying a Macro

1. Run the `script` command.

The Scripting dialog box appears.

2. In the *Name* text box, enter the name of the macro that you want to replay.

If necessary, use the *Browse* button to locate the correct file.

3. Click *Replay*.

The script replays.

### Converting a .jrl File to a Script

1. Run the `script` command.

The Scripting dialog box appears.

2. Click *Generate*.

A file browser appears.

3. Choose a journal file to convert, which then creates a file of the same name with `.scr` appended to it in the same directory as the source journal file. Once Allegro PCB Editor generates the file, its name populates the *Name* text box.

4. Repeat for as many journal files as you want to convert.

### **Recording/Replaying Padstack Scripts**

You can automate the process of entering padstack data by creating a script that lets you record the entries that you make in the Padstack Designer dialog box. To define new padstacks that share similar padstack specifications, you can replay the script file and edit the new padstacks as necessary.

## show allpanes

Restores the *Options*, *Worldview*, *Find*, *Visibility*, and *Command* foldable window panes to display in the positions in which you last viewed them.

To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default* (`reset dockwindows` command).

### Menu Path

*View – Windows – Show All*

### Syntax

`show allpanes`

## Displaying All Foldable Window Panes

1. Choose *View – Windows – Show All*.

The *Options*, *View*, *Find*, *Visibility*, and *Command* foldable window panes display in the positions in which you last viewed them.



## showhide find

Toggles the visibility of the *Find* window pane.

A check mark next to *View – Windows – Find* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Find* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Find* window pane to expand it, or clicking the X to hide it.

**Note:** To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default* (`reset dockwindows` command).

### Menu Path

*View – Windows – Find*

### Syntax

```
showhide find [show] [hide]
```

<b>showhide_find</b>	Displays or hides the pane, depending on its current state.
<b>show</b>	Displays the pane if it is hidden. If it is already visible, no action occurs.
<b>hide</b>	Hides the pane if it is visible. If it is already hidden, no action occurs.

## Controlling the Visibility of the Find Window Pane

1. Choose *View – Windows*.
  - a. To hide the pane, click *Find* if a check mark appears next to it.
  - b. To display the pane, click *Find* if no check mark appears next to it.

## showhide options

Toggles the visibility of the *Options* window pane.

A check mark next to *View – Windows – Options* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Options* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Options* window pane to expand it, or clicking the X to hide it.

**Note:** To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default to Default* (`reset dockwindows` command).

### Menu Path

*View – Windows – Options*

### Syntax

```
showhide options [show] [hide]
```

<b>showhide_options</b>	Displays or hides the pane, depending on its current state.
<b>show</b>	Displays the pane if it is hidden. If it is already visible, no action occurs.
<b>hide</b>	Hides the pane if it is visible. If it is already hidden, no action occurs.

## Controlling the Visibility of the Options Window Pane

1. Choose *View – Windows*.
  - a. To hide the pane, click *Options* if a check mark appears next to it.
  - b. To display the pane, click *Options* if no check mark appears next to it.

## showhide text

Toggles the visibility of the *Command* window pane.

A check mark next to *View – Windows – Command* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Command* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Command* window pane to expand it, or clicking the X to hide it.

**Note:** To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default* (`reset dockwindows` command).

### Menu Path

*View – Windows – Command*

### Syntax

```
showhide text [show] [hide]
```

<b>showhide_text</b>	Displays or hides the pane, depending on its current state.
<b>show</b>	Displays the pane if it is hidden. If it is already visible, no action occurs.
<b>hide</b>	Hides the pane if it is visible. If it is already hidden, no action occurs.

## Controlling the Visibility of the Command Window Pane

1. Choose *View – Windows*.
  - a. To hide the pane, click *Command* if a check mark appears next to it.
  - b. To display the pane, click *Command* if no check mark appears next to it.

## showhide view

Toggles the visibility of the *Worldview* window pane.

A check mark next to *View – Windows – Worldview* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Worldview* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Worldview* window pane to expand it, or clicking the X to hide it.

**Note:** To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default UI* (`reset dockwindows` command).

For more information on the *Worldview* window pane, see the *Getting Started with Physical Design* user guide in your documentation set.

### Menu Path

*View – Windows – Worldview*

### Syntax

```
showhide view [show] [hide]
```

<b>showhide_view</b>	Displays or hides the pane, depending on its current state.
<b>show</b>	Displays the pane if it is hidden. If it is already visible, no action occurs.
<b>hide</b>	Hides the pane if it is visible. If it is already hidden, no action occurs.

## Controlling the Visibility of the Worldwide Window Pane

1. Choose *View – Windows*.
  - a. To hide the pane, click *Worldview* if a check mark appears next to it.
  - b. To display the pane, click *Worldview* if no check mark appears next to it.

## showhide vis

Toggles the visibility of the *Visibility* window pane.

A check mark next to *View – Windows – Visibility* indicates that the window pane is visible. Choosing the menu option with a check mark next to it hides the pane. When you hide and then re-display a window pane, it appears in the same position and size as before. Dock or undock the *Visibility* window pane by left clicking to choose it and moving it anywhere within or outside the design window.

You can also control the visibility by clicking the arrow on the *Visibility* window pane to expand it, or clicking the X to hide it.

**Note:** To show all window panes in their original positions, use *View – Windows – Reset UI to Cadence Default UI* (reset dockwindows command).

### Menu Path

*View – Windows – Visibility*

### Syntax

```
showhide vis [show] [hide]
```

<b>showhide_vis</b>	Displays or hides the pane, depending on its current state.
<b>show</b>	Displays the pane if it is hidden. If it is already visible, no action occurs.
<b>hide</b>	Hides the pane if it is visible. If it is already hidden, no action occurs.

## Controlling the Visibility of the Visibility Window Pane

1. Choose *View – Windows*.
  - a. To hide the pane, click *Visibility* if a check mark appears next to it.
  - b. To display the pane, click *Visibility* if no check mark appears next to it.

## show element

[Dialog Box](#) | [Procedure](#)

The `show element` command lets you list the attributes of a graphic element. It displays all values relevant to the element, such as its graphic coordinates, segment coordinates (for lines, connect lines, rectangles, and shapes), segment length, center and radius (for arcs), symbol type and reference designator (for package symbols), attached properties.

The `show element` command shows the schedule for user schedule nets.

### Menu Path

*Display – Element*

### Toolbar Icon



## Dialog Boxes

### Show Element Dialog Box

The Show Element dialog box is a text box. It contains the following controls:

<i>File – Save As</i>	Saves the information in a text file. When you issue this command, Allegro PCB Editor and Allegro Package Designer prompts you for a file name and appends the <code>.txt</code> extension.
<i>File – Print</i>	Prints the contents of the window on either UNIX or Windows systems.
<i>File – Stick</i>	Makes the window remain on screen until you close the window, or the program terminates. Use this option to compare information between two windows. For example, you may use <code>show element</code> to obtain information about two design elements and use <i>File – Stick</i> to compare the contents of each window.

## Allegro Free Physical Viewer

### show element

---

You can click on the x y coordinates in the Show Element dialog box and zoom center on the location in the Design window.

### Find By Name/Property

Use this dialog box to set up search criteria so you can find element types quickly.

<i>element Type</i>	Defines the element type you want to select.
<i>Available elements</i>	Lists all the available elements in the design.
<i>Name Filter</i>	Lets you narrow the element list of names by typing in names, parts of names, and using wildcards.
<i>Value Filter</i>	Lets you narrow the element list of values by typing in values, parts of values, and using wildcards.
<i>All -&gt;</i>	Lets you move all the <i>Available elements</i> into the <i>Selected element</i> list.
<i>&lt;-All</i>	Lets you move all the <i>Selected elements</i> into the <i>Available element</i> list.
<i>Selected elements</i>	Lists all the elements you chose.

Double clicking an element in either the *Available element* list or the *Selected element* list results in the element moving to the other column.

When you click *Apply*, the Show Element dialog box appears and the Find by Name/Property dialog box remains open. When you click *OK*, the elements are found but the Find by Name/Property dialog box closes.

## Procedures

### Displaying Design Attributes for an element

This procedure lets you display element attributes. You can also find instances of inherited properties on parent and child elements using this method. This depends on where you start to search for inherited properties. If you add the FIXED property to a net and, by inheritance, to its associated pin, only the first instance of the inherited property (attached to the pin) is

printed. Since the attachment does not exist on the pin, it is reported as being inherited from the net.

1. Run the `show element` command.
2. In the Find filter, choose the design elements you want to display.
3. Position the cursor over an element and click to select.

The element is highlighted and the Show Element dialog box appears. It contains all values relevant to the element you picked.

4. Choose additional elements for display or click right and choose *Done* from the pop-up menu.

**Note:** You can print a listing of the highlighted design element or you can save the listing to a file.

### Finding an element by its Property

1. Run the `show element` command.
2. Click *More* in the Find Filter.

The Find by Name/Property dialog box appears.

3. Choose the property from the *Available Properties* list box.

The property appears in the *Name* field.

4. To display all elements that have the chosen property, click *Apply*.

A Show Element dialog box appears, listing all elements to which the chosen property currently is attached.

Any elements on the design that have the chosen property are highlighted. If there are no such elements, a message is displayed in the command console:

```
No instances of <property_name> found.
```

5. To display attributes for the chosen element, click *Show*.

The Find by Property Show dialog box appears.

### Finding an element by its Name

1. Click the arrow next to the drop-down list box at the bottom of the Find Filter.
2. Choose the type of element from the list.



## Allegro Free Physical Viewer

### show element

---

3. Enter the name of the element in the *Name* field to the right of the drop-down list box.
4. Click *Enter*.

The Show Element dialog box appears and the element on the design is highlighted.

## show measure

[Dialog Box](#) | [Procedure](#)

The `show measure` command lets you calculate the distance between two user-defined points on your design and displays the following information:

- Distance
- Total distance
- Manhattan distance
- Change along the x-axis
- Change along the y-axis

### Menu Path

*Display – Measure*

### Toolbar Icon



## Measure Dialog Box

<i>Dist</i>	Displays the distance between two markers shown on the elements you picked.
<i>Total Dist</i>	Displays the accumulated total of all values displayed in the Dist field since you chose the second element or since you last chosen Next from the pop-up.
<i>Manhattan Dist.</i>	Displays the absolute sum of the x-distance and the y-distance between two markers. This is always a positive value
<i>Dx</i>	Displays the absolute x-distance (horizontal) between two markers.
<i>Dy</i>	Displays the absolute y-distance (vertical) between two markers.

## Allegro Free Physical Viewer

### show measure

---

**Note:** Manhattan Dist = Dx + Dy

*Width* Displays the width of line segments along a connect line.

If you have a connection path joining two elements, the following options appear on the Measure dialog box:

*Etch/Conductor Dist* Displays the distance along the center lines of the connect lines connecting the two elements.

*Total Etch/Conductor* Displays the accumulated connection path length from the first selection you made.

*Via Count* Displays the number of vias on the path joining the last two points you picked.

*Air Gap* Displays the minimum distance between the two elements you picked. If either element is a DRC marker, NCDrill figure, or a point not on any element, then a message displays indicating that no Air Gap was measured. A similar message displays if both picks are on the same etch/conductor type element.

*On Subclass* Displays the subclass that is common to both elements, if they have one. This field does not display if there is no common subclass.

## Procedure

1. Run `show measure`.
2. Adjust the Find Filter to choose specific design elements,.
3. Position the cursor and click to highlight the first element.

The Measure dialog box displays and identifies the element and its location.

4. Position the cursor and click to highlight the second element.

The Measure dialog box is updated with the second element and its location, and displays the distance between the two points you chose.

The following temporary markers on each element appear:

- ☐ A cross indicates the center of a pad or the vertex of a connect line or filled rectangle.

- A square at the nearest grid point identifies all other picks.

If you pick two different elements and an air gap has been defined between them, a line showing the air gap between the nearest points on the two elements is displayed.

The command finds the connecting path, if it exists, between the two elements you pick, highlights it, and displays the distance in the Dist field of the Measure dialog box. If more than one connecting path joins the two elements, one of them is found and highlighted.

- a. To measure any other path, indicate it by picking intermediate points along it and read the Total Dist field of the Measure dialog box.

5. When you are finished, click right to display the pop-up menu, and choose *Done*.

## show property

[Dialog Box](#) | [Procedure](#)

The `show property` command identifies the properties in your current design in the Show Property dialog box. You can list all design elements assigned to a property/value or view a property definition.

### Menu Path

*Display – Property*

## Show Property Dialog Box

Use this dialog box to find elements with a specific property/value or view the definition of a property.

### Information Tab

<i>Available Properties</i>	Displays a list of all Allegro PCB Editor and Allegro Package Designer properties. Click a property to choose it. The property name appears in the <i>Name</i> field.				
<i>Filter</i>	Limits the properties you want displayed in the <i>Available Properties</i> list.				
<i>Name</i>	Searches for the property name entered in this field.				
<i>Value</i>	Searches for the property value entered in this field. A property must be defined in the <i>Name</i> field before this field is active.				
<i>Type</i>	Indicates the property type after you have chosen a property.				
<i>Sort By</i>	Sorts elements in one of the following ways: <table><tr><td><i>Element</i></td><td>(Default) Lists properties by design element.</td></tr><tr><td><i>Property</i></td><td>Lists design elements by property.</td></tr></table>	<i>Element</i>	(Default) Lists properties by design element.	<i>Property</i>	Lists design elements by property.
<i>Element</i>	(Default) Lists properties by design element.				
<i>Property</i>	Lists design elements by property.				
<i>Show Val</i>	Displays a list of all the elements that have the chosen property/value. The list appears sorted in a separate window that remains open until you close it.				

## Allegro Free Physical Viewer

### show property

---

<i>Show Def</i>	Displays the definition of the chosen property, which appears in a separate window that remains open until you close it.
<i>Reset</i>	Clears the fields and resets to the defaults.

### Graphics Tab

<i>Available Properties</i>	Displays a list of all Allegro PCB Editor and Allegro Package Designer properties. Click on a property to choose it.
<i>Filter</i>	Limits the properties on display in the <i>Available Properties</i> list.
<i>Selected Properties</i>	Displays the name of the property for which to create text.
<i>Subclass</i>	Identifies the manufacturing subclass on which to create text for the chosen properties.
<i>Text Block</i>	Specifies the size of the text.
<i>Property Name</i>	If chosen, property text includes both property name and value.
<i>Reset</i>	Clears the fields and resets to the defaults.
<i>Create</i>	Click to create text for properties listed in <i>Available Properties</i> .
<i>Delete</i>	Deletes all text on the subclass.

### Procedures

#### Finding elements with a specific property/value

1. Choose *Display – Property* (show property command).

The Show Property dialog box appears.

2. Click the Information tab.

3. Choose a property from the *Available Properties* list.

–or–

Enter a property name in the *Name* field.

You can enter the property name in uppercase or lowercase.

**Note:** You can click *Filter* to limit the listed properties. By default, all properties

appear.

4. If needed, enter a property value in the *Value* field.
5. If needed, change the *Sort by* method.
6. Click *Show Val* for a list of elements that have the property—and its value, if specified.  
—or—  
Click *Show Def* for a definition of the property.

The Show window appears.

7. Click *OK* to close the Show Property dialog box.

To allow you to view property information while using other commands, the Show window does not disappear when you close the main Show Property dialog box. Close the Show window when you are done.

### Graphically displaying properties

1. Choose *Display – Property* (`show property` command).

The Show Property dialog box appears.

2. Click the Graphics tab.
3. Choose a property from the *Available Properties* list, moving it to the *Selected Properties* section, which displays the name of the property for which to create text.
4. Choose a manufacturing subclass on which to create text for the chosen properties in the *Subclass* field. If you specify a user-defined subclass to which to add properties, you must define them up prior to instantiating any properties using *Setup – Subclasses*.
5. Choose a value in the *Text Block* field, to specify the size of the text.
6. Enable the *Property Name* field to allow property text to include both the property name and value.
7. Click *Create* to create text. The status bar in the dialog box shows the number of text instances added.
8. Click *OK* to close the dialog box.
9. Choose *Display – Color Visibility* or click the color icon in the tool bar to display the Color dialog box.
10. In the *Package Geometry* section, click the ASSEMBLY TOP and BOTTOM subclasses to display them.

## Allegro Free Physical Viewer

### show property

---

11. Set the *Global Visibility* to *All Invisible*.
12. Click *Yes* in the confirmer that appears.
13. Set *Group* to *Manufacturing* and click any user-defined subclasses to display them; otherwise, Allegro PCB Editor adds the text instances to the *PROPERTIES* subclass by default.
14. Click *Apply* on the Color dialog box.
15. Click the Show Element icon. Set the *Find Filter* to *All Off* and enable *Text*.
16. Window select to zoom in. The elements with the property name and value text appear.



## show waived drcs

The `show waived drcs` command lets you display all waived DRC error markers on the board. This command is the opposite of the `blank waived drcs` command.

For more information on waiving DRC errors, see `blank waived drcs`.

### Menu Path

*Display – Waive DRCs – Show*

### Procedure

#### Showing Waived DRC Error Markers in the Design

**Note:** This command displays waived DRC errors that already exist in the design but are invisible, but will not waive DRC errors.

- Run the `show waived drcs` command. The waived DRC error markers appear on the board.

## status

In the layout mode, you can use the Status tab to verify the current state of shapes and DRCs and update them if they are out of date. An out of date dynamic shape is one for which the *Dynamic Fill* mode has been set to *Rough* or *Disabled* on the *Global Dynamic Shape Parameters* dialog box (non-*Smooth Dynamic Fill* mode). You can also assess the number of unplaced symbols or unrouted nets. In the symbol mode, you can view the number of connect and mechanical pins in the design.

When dynamic shapes are out of date, changing the dynamic fill mode on the *Status* tab produces the following behaviors:

### Menu Path

*Display – Status*

### Status Tab

Changing fill mode from	and using this button	produces this result
<i>Disabled to Rough</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Rough</i>
<i>Disabled to Smooth</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>
<i>Rough to Smooth</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>
any selection/no selection	<i>Update to Smooth</i>	updates dynamic shapes to <i>Smooth</i> changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>

## Menu Path

*Display – Status*

## Status Tab

<i>Connect pins</i>	Displays the number of connect pins in the design. (symbol mode only).
<i>Mechanical pins</i>	Displays the number of mechanical pins in the design. (symbol mode only).
<i>Symbols and Nets</i>	

*Unplaced symbols:* Displays the number and percentage of *<unplaced symbols>/<total symbols>* in the design. A green color box means all symbols are placed; yellow, some placed; and red, none placed. (layout mode only). Clicking the color box produces the Unplaced Symbol Availability Check report that lists the availability of unplaced symbols and their location on disk.

*Unrouted nets:* Displays the number and percentage of *<unrouted or partially nets>/<total nets>* in the design. A green color box means all symbols are placed; yellow, some placed; and red, none placed. (layout mode only).

*Unrouted connections:* Displays the number and percentage of *<unrouted connections>/<total pin-to-pin connections>* in the design, including nets with the NO\_RAT property. A green color box means all connections are routed; yellow, some routed; and red, none routed (layout mode only). The value derives from the netlist's From-To connections and is based on placed components, as is the percentage. Clicking the color box produces the Unconnected Pins report, which lists all unconnected pins in the design with hyperlinks to X/Y coordinates, net names, and total unconnected pins.

## Shapes

*Isolated shapes:* Displays the number of shapes on nets without connections, known as isolated shapes. Isolated shapes may occur during voiding, or when you add shapes to nets

without pins or vias to which to connect. A green color box means no shapes are isolated; yellow, some shapes remain isolated. Clicking the color box produces a report summarizing the data.

*Unassigned shapes:* Displays the number of copper shapes unassigned to a net. A green color box means no shapes are unassigned; yellow, some shapes remain unassigned. Clicking the color box produces a report summarizing the data. Clicking on the hyperlinked x/y coordinates in the report brings you to that shape location in the design.

*Out of date shapes:* Displays the number of *<non-Smooth dynamic shapes>/<total dynamic shapes>* in layout mode only.

A red color box indicates the *Dynamic Fill* mode for all dynamic shapes has been set to *Rough* or *Disabled* on the Global Dynamic Shape Parameters dialog box, making all dynamic shapes out of date (non-*Smooth Dynamic Copper Fill* mode) as a result. Out of date dynamic shapes prevent artwork output when you run `film param`, `odb_out`, and `stream out`.

A yellow color box indicates a portion of all dynamic shapes are out of date in the design.

A green color box indicates the *Dynamic Fill* for all dynamic shapes has been set to *Smooth*, making all dynamic shapes up-to-date (*Dynamic Fill* set to *Smooth*).

Clicking the color box produces a report, sorted by layer, showing the status of each dynamic shape on the board as follows:

*Smooth:* ready for artwork

*Out of date:* update required

*No Etch:* shape has no etch, possibly due to a route keepout. Delete the dynamic shape or add etch to produce artwork.

*Update to Smooth:* Click to automatically void and run DRC on all dynamically filled shapes, making all dynamic shapes up-

to-date (*Dynamic Copper Fill* mode set to *Smooth*) and produce artwork quality output (regardless of whether you chose *Rough* or *Disabled* in the *Fill Mode* field above). Changes the current *Dynamic Copper Fill* mode on the *Global Dynamic Shape Parameters* dialog box to *Smooth*.

To cancel dynamic filling of complex shapes for a large design, you can use the `ESC` key to stop the process, which leaves the shapes out of date. If several shapes are in the midst of dynamically filling when you invoke the `ESC` key:

Shapes already dynamically filled remain completed.

Shapes in the process of dynamically filling remain unfilled and marked out of date.

Shapes whose dynamic fill is yet to be updated remain filled but marked out of date.

*Dynamic Fill*: Controls automatic voiding and edge smoothing for all dynamically filled shapes. Use this field to change the dynamic copper fill mode while you are evaluating the status of dynamic shapes without opening the *Global Dynamic Shape Parameters* dialog box. The setting you choose here then defaults to the *Global Dynamic Shape Parameters* dialog box.

*Smooth*: Choose to automatically void and run DRC on all dynamically filled shapes and produce artwork quality output.

*Rough*: Select to see connectivity without full edge smoothing and thermal hookups in a fast fill mode to obtain true clearances around elements and resolve intersections with other voids. Artwork quality results and artwork are not created.

*Disabled*: Select to globally defer dynamically filling all dynamic shapes you subsequently create or modify to speed performance. Use this option to edit etch for medium to large ECOs, manual ECOs or to run batch programs such as netin, glossing, testprep add/replace vias, for example. Shapes created under this global setting are not voided, and DRC does not run. They are marked out of date to be filled later. Artwork cannot be produced.

## Allegro Free Physical Viewer

### status

---

#### *DRCs*

*DRC errors*: Indicates whether DRC markers are up-to-date. The status can be Out Of Date or Up to Date.

A red color box indicates DRC is out of date or Batch DRC is required.

A yellow color box indicates DRC is up to date, but DRC errors exist.

A green color box indicates DRC is up to date and no DRC errors exist.

*Update DRC* : Click to display the total number of errors. It is only enabled when online DRC is enabled.

*Waived DRC errors* : Displays the count of waived DRC errors that exist in the design. Waived DRC errors are never considered out-of-date.

A green color box indicates there are no waived DRC errors present in the design.

A yellow color box indicates there are waived DRC errors.

*On-Line DRC*: Specifies whether you run DRC online (*On*) or in batch mode (*Off*). Default is *On*. You should leave DRC mode on so that as you change the design, you get immediate feedback about design rule violations. For better performance, turn it off, but you should run a batch DRC update before manufacturing the board.

#### *OK*

Closes the dialog box.

#### *Refresh*

Click to display the most recent status for symbols, nets, and shapes.

## unrats all

The unrats all command hides all ratsnest lines in your design.

### Menu Path

*Display – Blank Rats – All*

### Toolbar Icon



## Procedure

### Hiding Ratsnest Lines

1. Run `unrats all`.  
All ratsnest lines in the design disappear.
2. Run *View – Refresh* to clean up the appearance of your design.

## unrats component

Hides visible ratsnest lines to pins on an individual component or a group of components in a design. Click to select the components or select the appropriate symbol name or symbol list from the Find by Name section of the Find filter.

### Menu Path

*Display – Blank Rats – Component*

### Procedure

#### Hiding Ratsnest Lines to Pins on Components

1. Run `unrats component`.
2. All ratsnest lines to pins on the components that you select disappear.

Optionally, you can extend your selection by clicking right and choosing *Refdes List* or *Refdes Name* from the pop-up menu.



## unrats net

Hides visible ratsnest lines to pins on an individual net or a group of nets in a design. To select the nets to be invisible, select the pins on the appropriate net or select the appropriate net name or net list from the Find by Name section of the Find filter.

### Menu Path

*Display – Blank Rats – Net*

## Procedure

### Hiding Ratsnest Lines to Pins on Nets

Hides visible ratsnest lines to pins on an individual net or a group of nets in a design. To select the nets to be invisible, select the pins on the appropriate net or select the appropriate net name or net list from the Find by Name section of the Find filter.

1. Run unrats net.
2. All ratsnest lines to pins on the nets that you select are removed.

Optionally, you can extend your selection by Net by clicking right and choosing Net List or Net Name from the pop-up menu.

## viewlog

[Dialog Box](#) | [Procedures](#)

The `viewlog` (also `viewlog -last`) command lets you view log files created by an automatic process, such as AutoRoute, NC Drill, and Silkscreen. The windows in which log files appear contain menu controls that let you save and print the logs.

You can click on the x y coordinates in the Viewlog dialog box and zoom center on the location in the Design window.

To be able to search a text file when you use the *File – File Viewer*, *File – Viewlog*, or *Display – Element* menu commands, be sure to set the `allegro_html` environment variable by choosing *Setup – User Preferences – Ui*.

### Select File to View Dialog Box

The log file viewer contains the following menu bar options:

<i>File – Save As</i>	Saves the information in a text file. When you issue this command, you are prompted for a file name and the <code>.txt</code> extension appends.
<i>File – Print</i>	Prints the contents of the window on either UNIX or Windows systems. Use the User Preferences Editor dialog box to set the <code>print_unix_command</code> environment variable governing Unix printing or the <code>print_nt_extension</code> environment variable governing Windows printing. See the <i>Getting Started with Physical Design</i> user guide in your documentation set for more information.
<i>File – Stick</i>	Makes the window remain on screen until you close the window, or the program terminates. Use this option to compare information between two windows. For example, you may use <code>show element</code> to obtain information about two design elements and use <i>File – Stick</i> to compare the contents of each window.
<i>Close</i>	Dismisses the window.

## Procedures

### Viewing Log Files Without Specifying File Name

1. Type `viewlog`.

A file browser appears

2. Choose the log file you want to view and click *Open*.

The log file viewer window displays the selected file.

3. Click on the x y coordinates in the Viewlog dialog box and zoom center on the location in the Design window.

### Viewing Log Files Specifying File Name

1. Type `viewlog` followed by the name of the file you want to view.

The log file viewer window displays the specified file.

2. Click on the x y coordinates in the Viewlog dialog box and zoom center on the location in the Design window.

## **zoom center**

The zoom center command moves the indicated point in the drawing into the center of the window display.

### **Menu Path**

*View – Zoom Center*

## **Procedure**

### **Centering a Design Window About a Specific Point**

Use one of these methods:

- ☐ Run `zoom center`, then click the location you want to be the center of the new display.
- ☐ Use dynamic zooming by way of the middle mouse button.

## zoom fit

The `zoom fit` command fits your entire layout in the design window. In APD, the command focuses around ASSEMBLY\_TOP/ASSEMBLY\_BOTTOM shapes if there is no package substrate outline or if there are no keepouts in the design.

### Menu Path

*View – Zoom Fit*

### Toolbar Icon



## Procedure

### Fitting Your Layout in the Design Window

Use one of these methods:

- Type `zoom fit` at the console window prompt.

–or–

Press F9.

–or–

Use dynamic zooming by way of the middle mouse button.

A full view of the design, excluding legends and borders, is displayed in the Design window.

## zoom in

The `zoom in` command magnifies your view by a factor of two. You can continue to zoom in on a design by repeating this command.

### Menu Path

*View – Zoom In*

### Toolbar Icon



## Procedure

### Magnifying Your View

Type `zoom in` at the console window prompt.

–or–

Press F10.

–or–

Draw the Zoom stroke (z) with the mouse. )

–or–

Use dynamic zooming by way of the middle mouse button. A full view of the design, excluding legends and borders, is displayed in the Design window.

## zoom out

The `zoom out` command halves the magnification of your layout.

You can continue to zoom out on a design by repeating this command.

### Menu Path

*View – Zoom Out*

### Toolbar Icon



## Procedure

### Reducing the Magnification of Your Layout

Type `zoom out` at the console window prompt.

–or–

Press F11.

–or–

Use dynamic zooming by way of the middle mouse button.

## zoom points

The `zoom points` command lets you define an area of your layout to zoom in on (magnify).

### Menu Path

*View – Zoom By Points*

### Toolbar Icon



## Procedure

### Zooming in on a Specific Area of Your Design

Use one of these methods:

1. Type `zoom points` at the console window prompt.

–or–

Press F8.

–or–

Draw the Zoom stroke (z) with the mouse.

–or–

Use dynamic zooming by way of the middle mouse button.

2. Click in the layout to anchor the start coordinate.
3. Move the mouse pointer over the layout to define the zoom boundary.  
A bounding box expands as you move the mouse.
4. Click again to define the end coordinate.  
The selected area expands into view.



## **zoom previous**

The `zoom previous` command lets you to zoom back from the current window extents to the prior view.

### **Menu Path**

*View – Zoom Previous*

### **Toolbar Icon**



## zoom world

The `zoom world` command reduces the magnification of your design so you can view your entire drawing.

### Menu Path

*View – Zoom World*

### Procedure

To zoom out to a full view of your design, use one of the following methods:

1. Run the `zoom world` command.

or

Draw the Zoom-In stroke (z) with the mouse.

or

Use dynamic zooming by way of the middle mouse button.