

Allegro[®] Free Physical Viewer

Product Version 17.2-2016

April 2016

Document Last Updated: March 15, 2013

© 1991–2016 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.

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>	7
Options tab for the <u>assign color</u> Command	7
Assigning a Custom Color or Highlighting an Element	8
<u>blank waived drcs</u>	10
Menu Path	10
Procedure	10
<u>cns_show</u>	11
Generating a Constraint Report	12
<u>color192</u>	13
Procedures	22
<u>colorview create</u>	33
Color Views Dialog Box	33
Procedures	34
<u>colorview restore</u>	36
Applying the Previous Color Visibility View	36
<u>custom datatips</u>	37
DataTips Customization Dialog Box	37
Procedure	40
<u>define embedded layer</u>	41
Embedded Layer Setup	41
<u>dehighlight</u>	45
Options Tab for the <u>dehighlight</u> Command	45
Procedure	46
<u>diestack editor</u>	48
Dialog Boxes	49
<u>exit</u>	56
<u>flipdesign</u>	57
Menu Path	57
Toolbar Icon	57
<u>grid toggle</u>	58
<u>help</u>	59
<u>highlight</u>	60

Allegro Free Physical Viewer

<u>Options Tab for the hilight Command</u>	61
<u>layer priority</u>	62
<u>Menu Path</u>	62
<u>Display Priority Dialog Box</u>	62
<u>Assigning a Display Priority To Layers</u>	63
<u>open</u>	64
<u>Menu Path</u>	64
<u>Syntax</u>	64
<u>Examples</u>	64
<u>Dialog Box</u>	64
<u>Procedure</u>	65
<u>plot</u>	66
<u>Print Dialog Box</u>	66
<u>Procedures</u>	67
<u>plot setup</u>	70
<u>Plot Setup Dialog Box</u>	70
<u>Procedure</u>	72
<u>prmed</u>	74
<u>Design Parameter Editor Dialog Box</u>	74
<u>Procedures</u>	74
<u>rats all</u>	77
<u>Procedure</u>	77
<u>rats component</u>	78
<u>Procedure</u>	78
<u>rats net</u>	79
<u>Procedure</u>	79
<u>redraw</u>	80
<u>reset dockwindows</u>	81
<u>Menu Path</u>	81
<u>Syntax</u>	81
<u>Displaying All Foldable Window Panes to Default Positions</u>	81
<u>script</u>	82
<u>Scripting Dialog Box</u>	82
<u>Procedures</u>	83
<u>show allpanes</u>	86
<u>Syntax</u>	86

Allegro Free Physical Viewer

<u>Displaying All Foldable Window Panes</u>	86
<u>showhide find</u>	87
<u>Syntax</u>	87
<u>Controlling the Visibility of the Find Window Pane</u>	87
<u>showhide options</u>	88
<u>Syntax</u>	88
<u>showhide text</u>	89
<u>Syntax</u>	89
<u>showhide view</u>	90
<u>Syntax</u>	90
<u>showhide vis</u>	91
<u>Syntax</u>	91
<u>show element</u>	92
<u>Dialog Boxes</u>	92
<u>Procedures</u>	93
<u>show measure</u>	96
<u>Measure Dialog Box</u>	96
<u>Procedure</u>	97
<u>show property</u>	99
<u>Show Property Dialog Box</u>	99
<u>Procedures</u>	100
<u>show waived drcs</u>	103
<u>Menu Path</u>	103
<u>Procedure</u>	103
<u>status</u>	104
<u>unrats all</u>	109
<u>Procedure</u>	109
<u>unrats component</u>	110
<u>Procedure</u>	110
<u>unrats net</u>	111
<u>Procedure</u>	111
<u>viewlog</u>	112
<u>Select File to View Dialog Box</u>	112
<u>Procedures</u>	113
<u>xsection</u>	114
<u>zoom center</u>	115

Allegro Free Physical Viewer

<u>Procedure</u>	115
<u>zoom fit</u>	116
<u>Procedure</u>	116
<u>zoom in</u>	117
<u>Procedure</u>	117
<u>zoom out</u>	118
<u>Procedure</u>	118
<u>zoom points</u>	119
<u>Procedure</u>	119
<u>zoom previous</u>	120
<u>zoom world</u>	121
<u>Procedure</u>	121

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

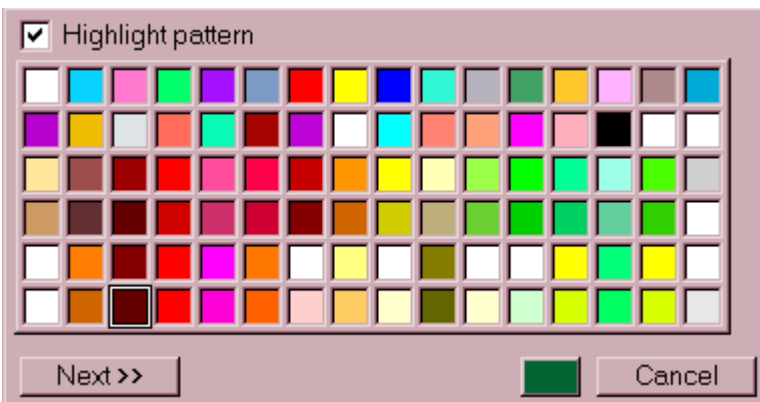
The following display only when you choose the *Display – Assign Color* menu item.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

<i>Selected color</i>	Indicates the currently chosen color.
<i>More Colors</i>	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. <i>Next/Prev</i> toggle the primary and secondary color palettes. The primary palette is the Cadence default, organized by a rainbow spectrum of 96 colors. The secondary palette comprises 96 colors used for customization. The first 24 positions are reserved for colors used in pre-16.0 databases.
<i>Highlight Pattern</i>	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 <i>Display</i> category of the <i>Color</i> dialog box. Once the element becomes highlighted in the design canvas, its name also displays in the bold font in the <i>Nets</i> section of the <i>Color</i> dialog box. Striping is only visible when the <code>display_nohilitefont</code> 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.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

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 accentuate certain elements with a pattern in the selected color if required.

The element becomes highlighted in the design canvas and its name displays in a bold 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.

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.

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

The Nets grid is used to customize color settings on nets or across their elements which include pins, vias, clines, shapes, or rats. Colors can be applied at the bus, diff pair, xnet and net level. Colors applied to hierarchical objects descend to their membership. Filtering and sorting controls are available to customize the display of nets. Custom color settings can be temporarily disabled, which reverts the color display back to layer- based settings while preserving the net coloring scheme for future use.

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

Allegro Free Physical Viewer

<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>Layers</i>	Choose to display the <i>Layers</i> grid, which lets you control the color and visibility of classes and subclasses, along with levels of transparency for the design and shapes.
<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> <p>Note: For SiP Free Physical Viewer, the Color Dialog has a <i>Bond Wires Profiles</i> category also. Choose on this category to set the color and visibility of bond wires based on the profiles. Bond wires do not reside on any subclass. From this location, you can control the color for the bond wires or set the display, based on the profiles.</p>

Allegro Free Physical Viewer

Allegro Free Physical Viewer

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

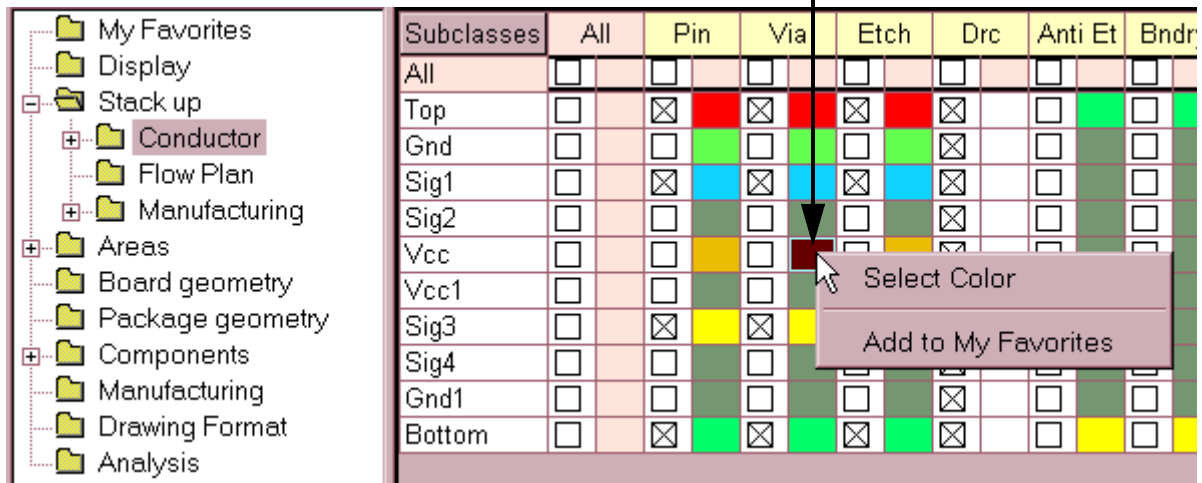
Subclasses	All	Pin	Via	Etch	Drc	Anti Et	Bndry
All	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Top	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<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

Allegro Free Physical Viewer

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



This color box becomes active



Allegro Free Physical Viewer

Allegro Free Physical Viewer

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 Favorites* 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.

Stacked via label: Specifies the color of stacked span of via labels. The default is white.

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

Allegro Free Physical Viewer

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.

Nets

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

Allegro Free Physical Viewer

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

Allegro Free Physical Viewer

<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>Stipple patterns</i>	<p>Allows you to assign stipple patterns to objects in addition to assigning color.</p> <p>To assign a stipple pattern, select the pattern and click on the object. The stipple pattern displays in the corresponding color box.</p>
<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 Color Dialog Box .
<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 >></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*.

Allegro Free Physical Viewer

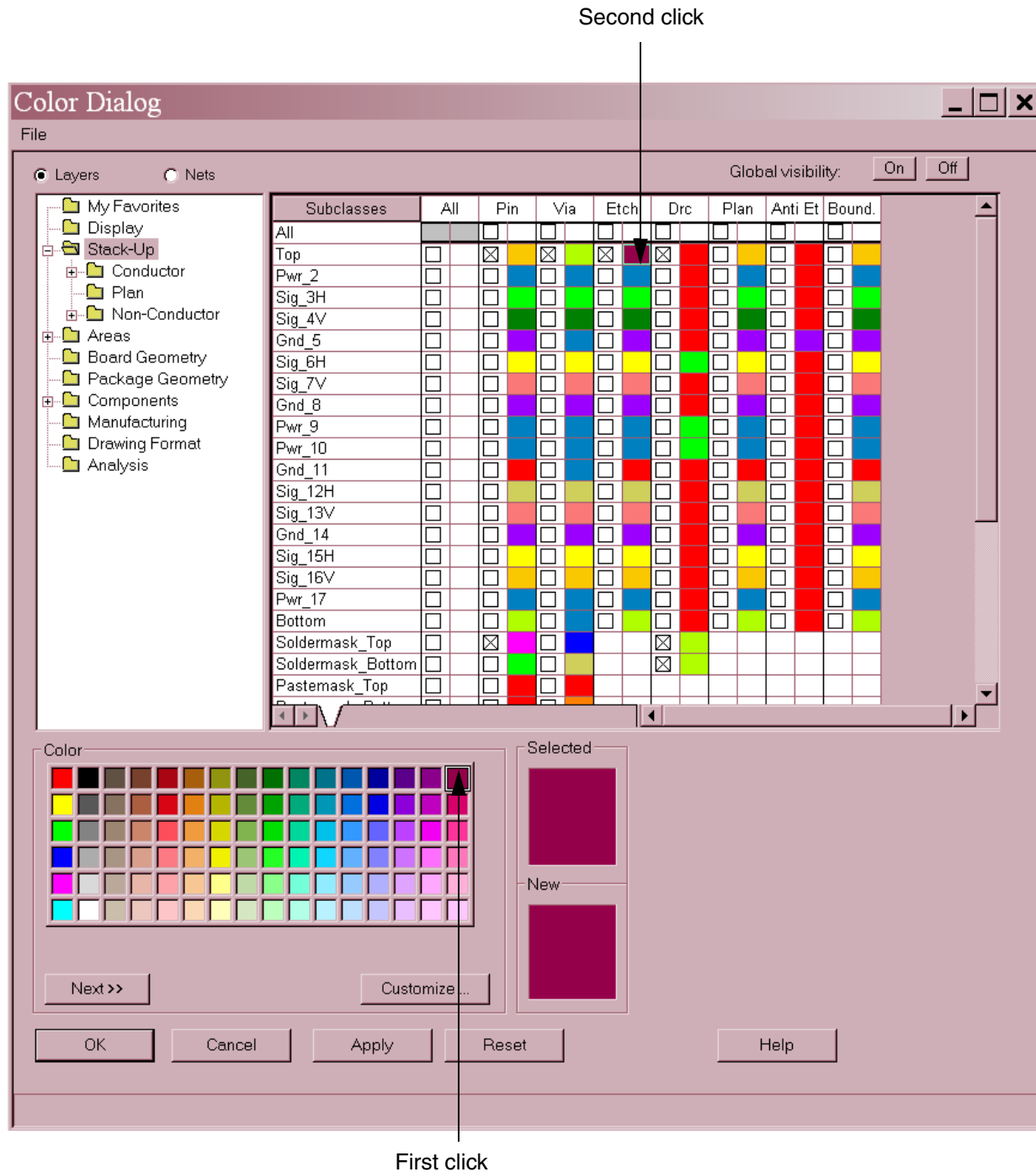
Allegro Free Physical Viewer

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

Allegro Free Physical Viewer

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 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

Allegro Free Physical Viewer

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"/>

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"/>
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"/>

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.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

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"/>
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"/>

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"/>
Net	Ad2	<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.
-----------------	---

Allegro Free Physical Viewer

Allegro Free Physical Viewer

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`.

custom datatips

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

Lets you customize a context-sensitive datatip that identifies an element when the cursor hovers over it. The datatip configuration file `custdatatips.cdt` contains default datatip information, which loads in the local `pcbenv` directory when the tool launches. Otherwise, only element names display in datatips.

The general settings for datatips' can be specified using the *User Preferences Editor*. Choose *Setup User — Preferences* to display the User Preferences Editor. Select *Display — Datatips* to specify the general options.

Menu Path

Setup – Datatip Preferences

DataTips Customization Dialog Box

File – Load default CDT file Loads settings from the default `custdatatips.cdt` file.

File – Save default CDT file Saves modifications to the default settings in the `custdatatips.cdt` file.

File – Load Custom CDT File Imports your customized datatip settings from an external `.cdt` file and applies them to the current design. A file browser appears with the filter set to `*.cdt` and a list of all `.cdt` files available in the current local working directory. You can manually browse to other directories to open a `.cdt` file. For instance, you may create a file with settings that suit a particular design's requirements; each time you open that design, import settings from that `.cdt` file.

File – Save Custom CDT File Exports the current design's customized datatip settings to an external `.cdt` file stored in your local working directory. A file browser appears with the filter set to `*.cdt` and a list of all `.cdt` files in the current local working directory. You can manually browse to other directories to specify an alternate save location.

File – Close Closes the Datatips Customization dialog box.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

<i>Object type</i>	Choose to customize datatips for clines, nets, symbol instances, pins, vias, or DRCs.
<i>General tab</i>	<p>Lists information to display in a datatip for the element chosen in <i>Object Type</i>.</p> <p>Click to Check the <i>Name</i> box to the right of the information to include it in the datatip; the <i>Value</i> box gets checked automatically, indicating its inclusion in the datatip as well.</p> <p>Select the <i>Value</i> box to only include the alphanumeric character string associated with the information in the datatip, which displays as \$<value>, such as \$COMMENT for instance, in <i>Specify DataTips Format</i>.</p> <p>Choose <i>All</i> to display all information available for the chosen element in the datatip.</p>
<i>Advanced tab</i>	<p>Displays all properties applicable to the chosen <i>Object Type</i> and available for inclusion in the datatip.</p> <p>Select the <i>Name</i> box to the right of the information to include it in the datatip; the <i>Value</i> box gets checked automatically, indicating its inclusion in the datatip as well.</p> <p>Select the <i>Value</i> box to only include the alphanumeric character string associated with the information in the data tip, which displays as \$<value>, such as \$COMMENT for instance, in <i>Specify DataTips Format</i>.</p> <p>Select the <i>Save</i> box, which only appears next to the user-defined attributes, to include these properties in the CDT file on saving it.</p> <p>Select <i>All</i> to check the check boxes in the column to display all information available for the chosen element in the data tip.</p> <p>Note: For Net objects, <i>Path length</i> and <i>Manhattan length</i> are included in the Advanced tab.</p>
<i>Property filter</i>	Enter whole words or character strings to locate a subset of the properties available for the chosen element. To specify a character string, use the asterisk (*) as a wildcard character. Displays only when the you choose the <i>Advanced tab</i> .

Allegro Free Physical Viewer

Allegro Free Physical Viewer

Apply filter Choose to display a subset of the available properties using the string entered in the *Property filter*.



Tip

To use the Enter key to apply the filter, choose *Setup — User Preferences — Ui — Input*, and enable `form_oldreturn`.

Specify DataTips Format

Customize the order in which to display datatip information using the following keys:

Up arrow: Appends the selected datatip entry to the line above it.

Down arrow: Appends the selected datatip entry to the line below it.

Left arrow: Transposes the selected datatip entry with that immediately to the left of it (if the first entry in a line is selected, nothing happens).

Right arrow: Transposes the selected datatip entry with that immediately to the right of it (if the last entry in a line is selected, nothing happens).

ENTER: Inserts a line break, and moves all the data tips immediately after the selected space onto the next line when you choose a space between data tip entries on the same line.

BACKSPACE and DELETE: Removes a line break and places all data tips immediately after the selected space on the same line when you choose a space between data tip entries on adjacent lines (line break).

OK

Saves settings to the `.cdt` file currently loaded and closes the dialog box.

Cancel

Closes the dialog box without saving any changes.

Reset to defaults

Removes all datatips customization and restores original settings.

Procedure

Customizing Datatips

1. Choose *Setup – Datatip Preferences*.
2. Open a `.cdt` file containing the datatips customization required, or use the default `.cdt` that loads automatically.
3. Choose the *General* or *Advanced* tab.
4. Choose an element in *Object type*; all information related to the element displays.
5. Choose the information and values to display in the datatips as required.
6. Specify the datatips format.
7. Choose OK.

define embedded layer

Dialog Box

Displays the Embedded Layer Setup dialog box, used to specify the layers in the PCB stackup that can be used for component placement.

Menu Path

Setup – Embedded Layer Setup

Embedded Layer Setup

The information displayed in the ID, Layer Name, Type and Thickness columns of the dialog box is same as displayed in the Cross Section Report (*Setup – Cross-Section*) and cannot be modified here.

Embedded Status	<p>Specify whether the layer can be used for component placement and if used, the orientation of the component on the layer. The supported values are:</p> <ul style="list-style-type: none">■ Not Embedded Components cannot be placed in this layer■ Body Up The layer can be used for placement of packaged component; and the body of the component placed on this layer is oriented toward the Top surface of the PCB.■ Body Down The layer can be used for component placement, however, the body of the component is oriented toward the Bottom surface of the PCB.■ Protruding Allowed This enables the embedded component placed on the adjacent signal layers to cut across the current layer. This option allows placement of embedded components for which package height is greater than the dielectric thickness between two layers.
-----------------	--

Allegro Free Physical Viewer

Allegro Free Physical Viewer

Attach Method	<p>Specify the method to be used for connecting the components to the embedded layer. The options supported are:</p> <ul style="list-style-type: none">■ Direct Attach <p>The component is placed (soldered) directly to the etch layer.</p> ■ Indirect Attach <p>Component is not placed on the etch layer directly. It is suspended in the dielectric material and vias are used to connect the component and the etch layer.</p> <p>This field is not valid/required for layers with <u>Embedded Status</u> set to Protruding Allowed.</p>
Embedded Global Parameters	<p>These parameters are defined to ensure that appropriate error margins are provided to accommodate manufacturing errors.</p>

Allegro Free Physical Viewer

Allegro Free Physical Viewer

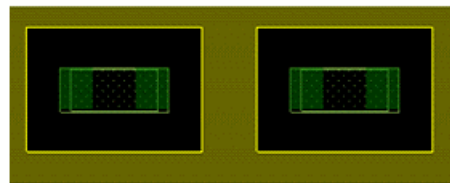
- Package height buffer

Specifies the minimum gap to be maintained between the embedded component and the etch layer.

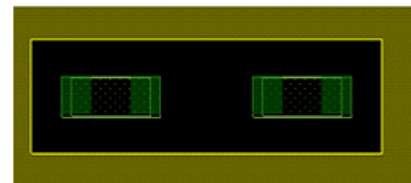
For example, if the gap between two layers is 20 mils, and you specify the Package height buffer as 3 mils, the maximum height of the component on this layer can only be 17 mils. If component height is larger than this value, DRC errors are generated.

- Minimum cavity gap for merging

Specifies the minimum gap in the x-y direction that is to be maintained between two cavities before the cavities are merged.



Cavity separation greater than parameter value



Cavity merger – separation at parameter value or less

- Placebound to via keepout expansion

Specifies the additional area beyond placebound used for via keepout.

- Package to cavity spacing

Specifies the minimum gap to be maintained between the embedded component and the cavity surrounding it. Thus effectively it is the minimum distance between the placebound and cavity outline specified on component symbol.

- Via connect height

This parameter is defined only when Indirect Attach method of component placement is used. It specifies the height of the vias used for connecting an embedded component to the etch layer. This value gets added to the PACKAGEHEIGHT to calculate effective package height.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

- **Default via connect padstack**

This parameter is defined only when Indirect Attach method of component placement is used. Specifies the default padstack to be used if the EMB_VIA_CONNECT_PADSTACK property is not specified on the drawing.

- **Cavity to route keepout expansion**

This parameter is defined only when Protruding Allowed option is enabled for the etch layer. This indicates the minimum distance between a protruding cavity and the routes on the etch layer.

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, clear 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.

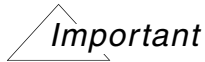
8. To remove only the highlight state from an element, click *Retain Objects Custom Color*, which also preserves the display of the element's custom color in the design canvas, while retaining its custom color assignment in the *Nets* grid of the Color dialog box. To remove both the highlight state and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

diestack editor



This command is available with SIP Free Physical Viewer.

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

The `diestack editor` command lets you visualize the side views of die stacks that may include various combinations of dies (both standard and co-design), spacers, interposers, and adhesives or epoxy layers necessary for the manufacture of die stacks. It also lets you edit vertical dimensions, spacer and interposer material data, and flip-chip bump dimensional data.

How the Diestack Editor Works

Die stacks always appear with the die stack up, but the die-stack editor indicates whether the die stack is located on the TOP or BOTTOM substrate surface. You can have only one die stack active in the die-stack editor at a time. When the die-stack editor is active, you cannot interact with the substrate until you complete the die-stack editing session. You can edit the X- and Y-axes in the Design Window with the tool's standard editing commands, such as, `add`, `move`, `spin`, and `delete`; you can edit the Z-axis dimensions using the die-stack editor (side view). However, you cannot edit in the SiP Layout Design Window (top view) when the die-stack editor is active.

About Spacers

A spacer is a manufactured or molded block of depositing material, such as adhesives or epoxies. It is rectangular and provides clearance or adhesion, or both, between dies or other die-stack elements that may be necessary to manufacture a die stack. Use the `add spacer` command to add spacers and capture the values for materials and other properties for these items that are used in both electrical and thermal analyses.

About Interposers

An interposer is a substrate with a single conductor layer used in the manufacture of a die stack to support die connectivity. It is used with wire bond dies where the die-pad positions create wire bond lateral spans that are beyond the physical limits of a wire bonding machine. Use the `add interposer` command to add interposers and capture the values for materials and other properties for these items that are used in both electrical and thermal analyses.

The `diestack editor` command does not generate a log file.

Menu Path

Display – Die Stack

Toolbar Icon



Dialog Boxes

Die-stack Viewer Dialog Box

When you run the `diestack editor` command, the Die-stack Viewer dialog box appears. The dialog box contains four tabs and a two-dimensional drawing containing data related to the currently selected die stack. The tabs are:

- Die Stacks – contains general information about the selected die stack.
- Dies – used to access and edit information about dies.
- Spacers – used to access and edit information about spacers.
- Interposers – used to access and edit information about interposers.

Standard Buttons/Options

<i>Close</i>	Exits the command.
<i>View Orientation</i>	Displays the current view orientation. You can choose a different view orientation: <i>North</i> , <i>South</i> , <i>East</i> , or <i>West</i> . The North side of the tool's plan view is the top of that drawing area, and a north view orientation is viewing a die stack's north-facing side from the north. There is a graphical indicator in the side-view drawing that shows the current viewing orientation.
<i>Help</i>	Invokes the online help system for the <code>diestack editor</code> command.

Die Stacks Tab

Die-stack Name Displays the name of the selected die stack. Use the drop-down list to choose any other die stacks in the current package design. If you change the current die stack, be sure to apply your changes before you edit another die stack.

When displaying this dialog box, you can also change the active diestack by selecting another diestack in the Design Window.

Rename Disabled in Free Viewer

Substrate Location Shows whether the selected die stack is located on the TOP or BOTTOM substrate surface.

Outputs

Report Lets you produce a report that contains detailed information regarding the selected die stack. You can save the report to a file.

Execute Processes the items selected in the *Outputs* frame.

The *Disclaimer* statement clarifies questions of view orientation and accuracy.

Dies Tab

Refdes Specifies the reference designator of the selected die instance.

Type A read-only field that specifies the type of the selected die instance as either FLIP-CHIP or WIRE BOND and STANDARD or CODESIGN.

Orientation A read-only field that specifies the orientation of the selected die as either CHIP-UP or CHIP-DOWN.

Thickness Specifies the thickness of the selected die.

Bump Dimensions The fields in this frame apply only to flip-chip dies. For wire-bond dies, the fields are unused and are grayed out.

Dmax Specifies the maximum diameter of the bump.

Note: A value of zero for either *Dmax* or *HT* means that the overall height of the die does not include the bump measurements; therefore the bumps do not appear in the diestack-editor.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

D1 (at substrate) Specifies the diameter of the solder bump at the package side. The value of *D1* must be less than the value of *Dmax*.

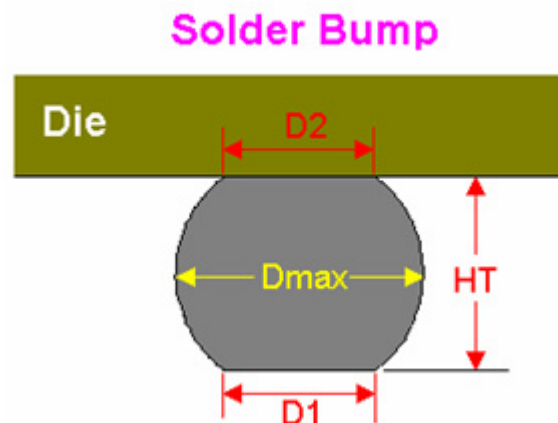
D2 (at die) Specifies the diameter of the solder bump at the die side. The value of *D2* must be less than the value of *Dmax*.

HT Specifies the height of the solder bump. This value does not include either the thickness of the die PASSIVATION layer or the thickness of the substrate solder mask.

Note: A value of zero for either *Dmax* or *HT* means that the overall height of the die does not include the bump measurements; therefore the bumps do not appear in the diestack-editor.

View/Unview Click to display or remove a graphic of the solder ball dimensions.

These dimensions are referenced from a die stack mounted on the TOP substrate surface as viewed in the die-stack editor 2D elevation view. Note that the dimensions are for bump dimensions after the die's assembly to the substrate surface and should represent, as accurately as possible, the bump dimensions after attachment to the substrate. The model for these dimensions is based on the PakSI 3D modeling interface used in the SI Analysis feature.



Conductivity Specifies the electrical conductivity of the solder bumps.

Placement

Layer Displays the stackup layer where the pads of the selected die are located.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

Rotation Displays in degrees the angular rotation of the currently active member of the diestack.

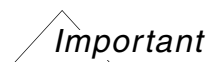
Stretch bond wires on move **Disabled for Free Viewer**
Specifies the behavior of the wirebonds when moving a die. Check this box to have the tool keep the bond fingers in their current locations and stretch the bond wires to the new pin locations during move operations (default setting). If you do not check this box, both the bond fingers and bond wires move with the die.

Delete **Disabled for Free Viewer**
Lets you delete the component and symbol.
This field applies to the Dies, Spacers, and Interposer tabs.

Move **Disabled for Free Viewer**
Lets you place the currently active member of the diestack in the drawing area on the cursor in the Design Window. You can choose a new location for the symbol by either picking a location or entering absolute or incremental coordinates at the console window prompt.
When satisfied with the location, right-click and choose *Done* from the pop-up menu, or let the tool automatically complete the operation when you pick another field or tab.
This field applies to the *Dies*, *Spacers*, and *Interposers* tabs.

Swap **Disabled for Free Viewer**
Lets you swap a die with another die or with an interposer. The target can be in the same diestack or a different one.

Spacers Tab



The options in this tab are disabled in the Free Viewer.

Ref ID Specifies the identifier used to reference a spacer instance. Because spacers do not have logical components, they do not have reference designators, as do dies.

Symbol Name A read-only field that specifies the name of the spacer symbol.

Part Number Specifies the part number of the spacer used in the Bill of Materials.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

Material

Name Specifies the name of the material that makes up the spacer. The thermal conductivity of a spacer is a property of its material. Click ... to launch a read-only version of the Materials Editor for selection of a spacer material. You must edit the material properties through the Materials Editor using the define materials command.

Thickness Specifies the thickness of the spacer.

Dimensions Specifies the length and width of the spacer.

Length Specifies the length of the spacer.

Width Specifies the width of the spacer.

Resize Spacer **Disabled for Free Viewer**

Clicking this button displays the *Resize Spacer Dialog Box*.

Placement

Layer Specifies the name of the non-substrate DIELECTRIC layer on which you place the spacer. You can use the pull-down menu to change the layer of the die to another named DIELECTRIC layer only.

If you change the layer of the spacer, you change the spacer's orientation from non-mirrored to geometry-mirrored (use the show element command).

If the placement of one spacer conflicts with the placement of another spacer, the SiP tool makes the placement, but also displays a DRC violation.

Rotation Specifies in degrees the angular rotation of the currently active member of the diestack. You can choose *0*, *90*, *180*, or *270* from the drop-down list or enter a specified angle (up to three decimal places).

Delete Lets you delete a spacer.

Allegro Free Physical Viewer

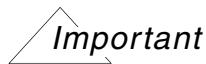
Allegro Free Physical Viewer

Move Lets you place the currently active member of the diestack in the drawing area on the cursor in the Design Window. You can choose a new location for the symbol by either picking a location or entering absolute or incremental coordinates at the console window prompt. This field applies to the *Dies*, *Spacers*, and *Interposers* tabs.

Complete the operation by right-clicking and choose *Done* from the pop-up menu or let the tool automatically complete the operation when you choose another field or tab.

Swap Lets you swap a spacer with another spacer. The target spacer can be in the same diestack or a different one.

Interposers Tab



The options in this tab are disabled in the Free Viewer.

Lets you display or edit data regarding any interposer members of the currently selected die stack.

Ref ID Specifies the reference identifier of the interposer instance.

Symbol Name A read-only field that specifies the name of the interposer symbol.

Part Number Specifies the part number of the interposer used in the Bill of Materials.

Conductor Material

Name Specifies the name of the conductor material used on the interposer. The electrical properties of an interposer are derived from its conductor material.

Click ... to launch a read-only version of the Materials Editor for selection of a conductor material. You can edit material properties through the Materials Editor using the define materials command.

Thickness Specifies the thickness of the interposer conductor material.

Dielectric Material

Name Specifies the name of the dielectric material that makes up the interposer. The thermal conductivity of a spacer is a property of its material.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

<i>Thickness</i>	<p>Specifies the thickness of the interposer dielectric material.</p> <p>Click ... to launch a read-only version of the Materials Editor for selection of a dielectric material. You can edit material properties through the Materials Editor using the <u>define materials</u> command.</p>
<i>Placement Layer</i>	<p>Specifies the name of the non-substrate CONDUCTOR layer on which the interposer interconnect (vias, clines, and shapes) is placed. You can use the pull-down menu to change the layer of the die to another named conductor layer only.</p> <p>If you change the layer of the interposer to the opposite side of the substrate, the interposer's orientation changes from non-mirrored to geometry-mirrored (use the <code>show element</code> command).</p> <p>If the placement of one interposer conflicts with the placement of another spacer (you cannot mix spacers and interposers on the same layer), the SiP tool makes the placement, but also shows a DRC violation.</p>
<i>Rotation</i>	<p>Specifies in degrees the angular rotation of the currently active member of the diestack. You can choose <i>0</i>, <i>90</i>, <i>180</i>, or <i>270</i> from the drop-down list or enter a specified angle (up to three decimal places).</p>
<i>Delete</i>	<p>Lets you delete the interposer.</p> <p>This field applies to the <i>Dies</i>, <i>Spacers</i>, and <i>Interposer</i> tabs.</p>
<i>Move</i>	<p>Lets you place the currently active member on the cursor in the Design Window. You can choose a new location for the symbol by either picking a location or entering absolute or incremental coordinates at the console window prompt. This field applies to the <i>Dies</i>, <i>Spacers</i>, and <i>Interposer</i> tabs.</p> <p>Complete the operation by right-clicking and choose <i>Done</i> from the pop-up menu or let the tool automatically complete the operation when you choose another field or tab.</p>
<i>Swap</i>	<p>Lets you swap an interposer with another interposer or with a die. The target can be in the same diestack or a different one.</p>

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

flipdesign

Use this command to flip the design along the Y-axis on the drawing canvas. It sets the active layer to bottom etch when enabled and to top etch when disabled. Grids do not display when this command is active. The active *Flipboard mode* is indicated in the in the status bar at the bottom of the Allegro PCB Editor window; and in the title bar, with the design file name suffixed with the flip mode.

Run this command again to return to normal view.

This command is available only when the OpenGL integrated Application Programming Interface is enabled. OpenGL displays elements semi-transparently, allowing you to view any elements that lie beneath other elements. You can use *Tools – Utilities – OpenGL Status* for additional information.

Menu Path

View – Flipdesign

Toolbar Icon



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.

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 bold 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 hilight 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>-></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.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

<i>Bottom</i>	Moves the chosen layer to the bottom of the <i>Prioritized Layers</i> list.
<i><-</i>	Removes the chosen layer from the <i>Prioritized Layers</i> list.
<i><<-</i>	Removes all layers from the <i>Prioritized Layers</i> list.
<i>OK</i>	Saves your changes and closes the dialog box.
<i>Cancel</i>	Exits the dialog box.
<i>Apply</i>	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 Cadence Online Support.

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

Allegro Free Physical Viewer

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

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

Allegro Free Physical Viewer

<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

Allegro Free Physical Viewer

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*.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

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

Name Specifies the name of the file in which you record your actions. Allegro PCB Editor adds the `.scr` extension to the file name.

Browse Displays the script file data browser that lets you choose a script file to replay.

Library Displays the script file data browser that lets you choose a script file to replay. Opens to your script path location.

Generate Displays a file browser from which you can choose a `.jrl` 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>
```

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]
```

showhide_find	Displays or hides the pane, depending on its current state.
show	Displays the pane if it is hidden. If it is already visible, no action occurs.
hide	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]
```

showhide_options Displays or hides the pane, depending on its current state.

show Displays the pane if it is hidden. If it is already visible, no action occurs.

hide 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]
```

showhide_text	Displays or hides the pane, depending on its current state.
show	Displays the pane if it is hidden. If it is already visible, no action occurs.
hide	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]
```

showhide_view	Displays or hides the pane, depending on its current state.
----------------------	---

show	Displays the pane if it is hidden. If it is already visible, no action occurs.
-------------	--

hide	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]
```

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

Allegro Free Physical Viewer

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 -></i>	Lets you move all the <i>Available elements</i> into the <i>Selected element</i> list.
<i><-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

Allegro Free Physical Viewer

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
- Pick Angle

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.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

Dy Displays the absolute y-distance (vertical) between two markers.

Note: Manhattan Dist = Dx + Dy

Pick Angle Displays the angle between two markers. This field is useful when doing offset routing.

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.

Allegro Free Physical Viewer

Allegro Free Physical Viewer

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

Allegro Free Physical Viewer

<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

Allegro Free Physical Viewer

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</i> to <i>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</i> to <i>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</i> to <i>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

Connect pins

Displays the number of connect pins in the design. (symbol mode only).

Mechanical pins

Displays the number of mechanical pins in the design. (symbol mode only).

Symbols and Nets

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

Allegro Free Physical Viewer

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.

xsection

Displays the Layout Cross Section report, which provides information about each layer defined in your layout.

Menu Path

Setup – Cross-section

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.

Allegro Free Physical Viewer

Allegro Free Physical Viewer
