

Allegro® X APD Free Physical Viewer

Product Version 23.1
September 2023

© 2023 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. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR52.227-7013 et seq. or its successor.

Contents

<u>3d</u>	7
<u>Menu Path</u>	7
<u>Toolbar Icon</u>	7
<u>3D Canvas User Interface</u>	7
<u>Navigating in 3D canvas</u>	12
<u>Menus</u>	12
<u>assign color</u>	16
<u>Options tab for the assign_color Command</u>	16
<u>Assigning a Custom Color or Highlighting an Element</u>	17
<u>blank waived drcs</u>	18
<u>Menu Path</u>	18
<u>Procedure</u>	18
<u>capture image</u>	19
<u>Procedure</u>	19
<u>cns_show</u>	20
<u>Generating a Constraint Report</u>	21
<u>color192</u>	22
<u>Procedures</u>	30
<u>colorview create</u>	39
<u>Color Views Dialog Box</u>	39
<u>Procedures</u>	40
<u>colorview restore</u>	43
<u>Applying the Previous Color Visibility View</u>	43
<u>custom datatips</u>	44
<u>DataTips Customization Dialog Box</u>	45
<u>Procedure</u>	47
<u>define grid</u>	48
<u>Define Grids Dialog Box</u>	48
<u>Procedures</u>	49
<u>dehighlight</u>	51
<u>Options Tab for the dehighlight Command</u>	51
<u>Procedure</u>	52

Allegro X Package Design Plus Free Physical Viewer

<u>display toolbar param</u>	54
<u>Customize Dialog Box</u>	55
<u>Procedures</u>	55
<u>exit</u>	58
<u>flipdesign</u>	59
<u>Menu Path</u>	59
<u>Toolbar Icon</u>	59
<u>help</u>	60
<u>highlight</u>	61
<u>Options Tab for the highlight Command</u>	61
<u>layer priority</u>	63
<u>Menu Path</u>	63
<u>Display Priority Dialog Box</u>	63
<u>Assigning a Display Priority To Layers</u>	64
<u>manage settings</u>	65
<u>Manage UI Settings Dialog Box</u>	65
<u>Procedure</u>	65
<u>open</u>	67
<u>Menu Path</u>	67
<u>Syntax</u>	67
<u>Examples</u>	67
<u>Dialog Box</u>	67
<u>Procedure</u>	68
<u>opengl report</u>	69
<u>plot</u>	70
<u>Print Dialog Box</u>	70
<u>Procedures</u>	70
<u>plot setup</u>	71
<u>Plot Setup Dialog Box</u>	71
<u>Procedure</u>	73
<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

Allegro X Package Design Plus Free Physical Viewer

<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>save settings</u>	82
<u>Dialog Box</u>	82
<u>Procedure</u>	82
<u>script</u>	83
<u>Scripting Dialog Box</u>	83
<u>Procedures</u>	84
<u>show allpanes</u>	86
<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>Controlling the Visibility of the Options Window Pane</u>	88
<u>showhide text</u>	89
<u>Syntax</u>	89
<u>Controlling the Visibility of the Command Window Pane</u>	89
<u>showhide view</u>	90
<u>Syntax</u>	90
<u>showhide views1</u>	91
<u>showhide viewsh</u>	92
<u>showhide viewsv</u>	93
<u>showhide vis</u>	94
<u>Syntax</u>	94
<u>show element</u>	95
<u>Dialog Boxes</u>	95
<u>Procedures</u>	98
<u>show measure</u>	100

Allegro X Package Design Plus Free Physical Viewer

<u>Measure Dialog Box</u>	101
<u>Procedure</u>	102
<u>show property</u>	103
<u>Show Property Dialog Box</u>	103
<u>Procedures</u>	104
<u>show waived drcs</u>	107
<u>Menu Path</u>	107
<u>Procedure</u>	107
<u>status</u>	108
<u>Status Tab</u>	108
<u>unrats all</u>	111
<u>Procedure</u>	111
<u>unrats component</u>	112
<u>Procedure</u>	112
<u>unrats net</u>	113
<u>Procedure</u>	113
<u>viewlog</u>	114
<u>Select File to View Dialog Box</u>	114
<u>Procedures</u>	114
<u>xsection</u>	116
<u>zoom center</u>	117
<u>Procedure</u>	117
<u>zoom fit</u>	118
<u>Procedure</u>	118
<u>zoom in</u>	119
<u>Procedure</u>	119
<u>zoom out</u>	120
<u>Procedure</u>	120
<u>zoom points</u>	121
<u>Procedure</u>	121
<u>zoom previous</u>	122
<u>zoom world</u>	123
<u>Procedure</u>	123

Allegro Package Design Plus Free Physical Viewer

3d

Launches Allegro 3D Canvas, which lets you visualize and analyze a three-dimensional model of a design as a manufactured output. You can also view mechanical objects such as shields, fans, heat sinks and housings.

For additional information, see the [Allegro PCB Editor 3D Canvas](#) guide.

Menu Path

View – 3D View

Toolbar Icon



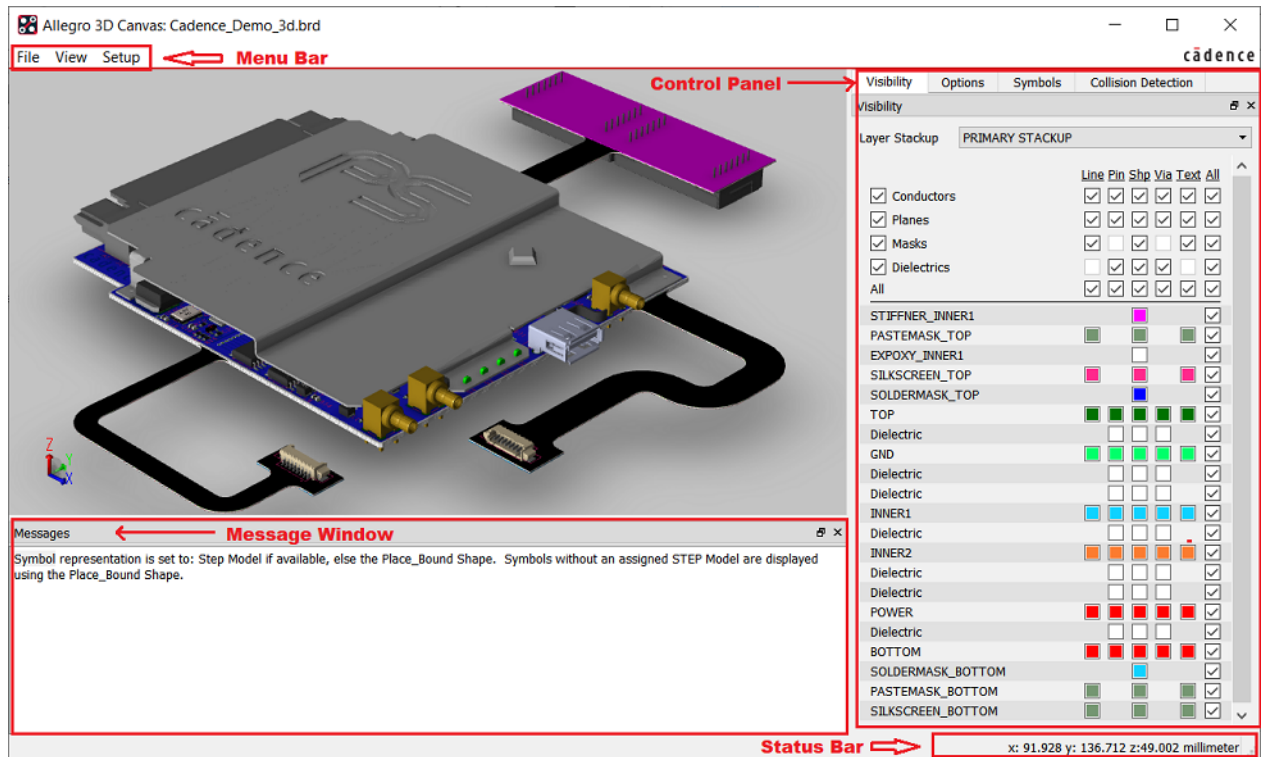
3D Canvas User Interface

When you launch a design in 3D Canvas, it displays the zoomed-fit isometric view of the design in the window frame. The name of the design is displayed in the title bar.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

The pull-down menus provide commands to view a design in 3D canvas and also to setup the viewing preferences.



An orientation marker appears in the lower right corner of the canvas that indicates the current spatial orientation of the model in space, showing the X, Y, and Z axis lines. The orientation marker adjusts automatically as you manipulate the viewpoint.

The 3D canvas user-interface has following elements:

- **Menu bar:** The pull-down menus provide commands for viewing a design and for setting up viewing preferences.

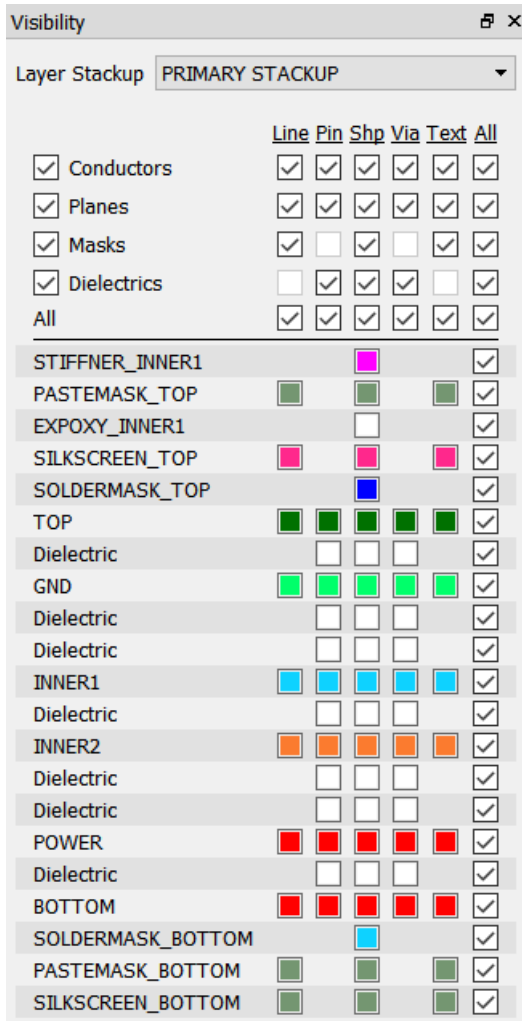
File View Setup

- **Control Panel:** The control panel has following window panes:

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

- ❑ **Visibility:** This window lets you selectively display or hide design layers (conductor, plane, mask, and dielectric) in 3D Canvas. You can turn off or on the visibility of any layer by clicking the associated check box.

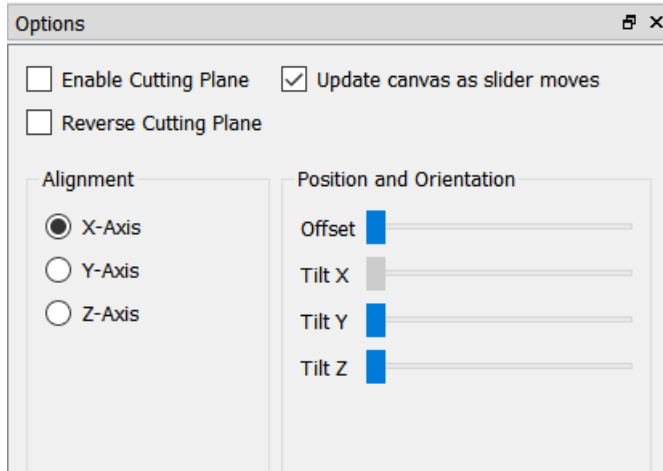


For a rigid-flex design, multiple stackups are available. Select either primary stackup or any flex stackup from the *Layer stackup* pull-down menu to manage the visibility of layers specified in that stackup.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

- ❑ **Options:** This window displays options for pop-up menu commands for *Cutting Plane*.

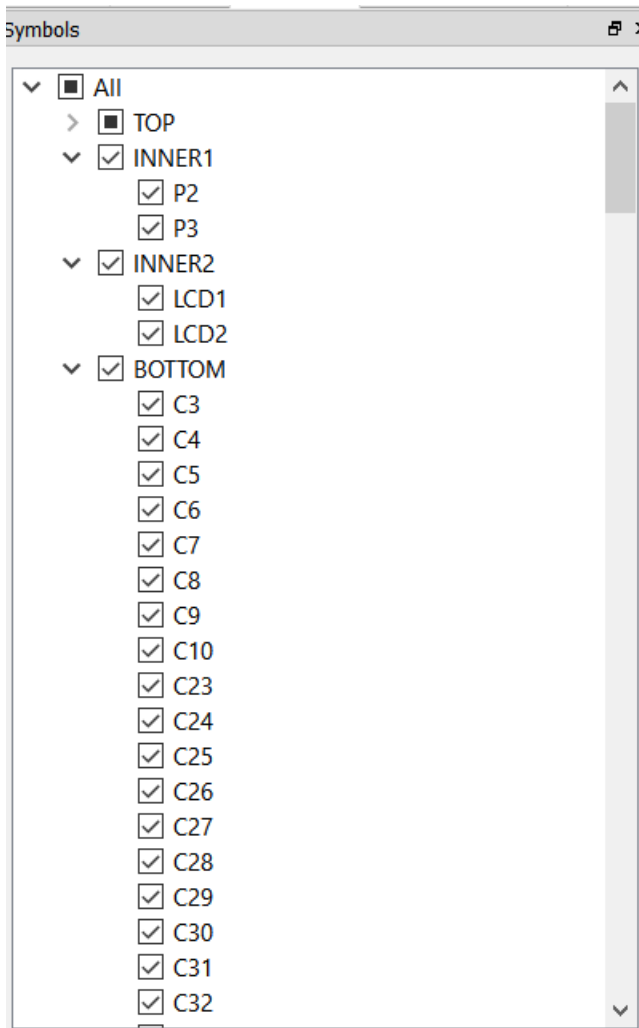


Cutting plane options

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

- ❑ **Symbols:** This window controls the visibility of symbols placed on conductor and plane layers. By default, all symbols are visible in 3D canvas. To hide any symbol, uncheck the checkbox associated with its reference designator.



- **Messages window:** This window displays all types of messages from 3D canvas. You can stop display of new messages by enabling *Silent mode* checkbox in the *Messages* category of the *Allegro 3D Canvas Preferences* dialog.

If you wish to review the messages later, use *File – Output – Messages* menu option to save them to a text file.

Messages in 3D canvas follow a color scheme to easily detect their type:

- ❑ Red color: Messages displayed in red color are errors.
- ❑ Pale green color: Messages showing warnings are appeared in pale yellow color.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

- ❑ **Black Color:** Messages stating information are displayed in black color.

Messages

The outline of the design cannot be rendered because the Board Geometry Design_Outline is not present.
Symbol representation is set to: Step Model if available, else the Place_Bound Shape. Symbols without an assigned STEP Model are displayed using the Place_Bound Shape.
Creating 3D SILKSCREEN_BOTTOM with default thickness 0.0127 mm because SILKSCREEN_BOTTOM is not defined in the cross section.
Creating 3D PASTEMASK_BOTTOM with default thickness 0.0127 mm because PASTEMASK_BOTTOM is not defined in the cross section.
Creating 3D SOLDERMASK_BOTTOM with default thickness 0.0127 mm because SOLDERMASK_BOTTOM is not defined in the cross section.
Creating 3D SOLDERMASK_TOP with default thickness 0.0127 mm because SOLDERMASK_TOP is not defined in the cross section.
Creating 3D PASTEMASK_TOP with default thickness 0.0127 mm because PASTEMASK_TOP is not defined in the cross section.
Creating 3D SILKSCREEN_TOP with default thickness 0.0127 mm because SILKSCREEN_TOP is not defined in the cross section.

- **Status bar:** The Status bar shows the X, Y, and Z coordinates as you move the mouse over the canvas. The points of reference are either of the mouse when it is in a free state or of a selected symbol during the **Move** command. The status bar also displays design units set in the 2D design.

x: 44.585 y: 27.317 z: -27.075 millimeter

Navigating in 3D canvas

Use mouse buttons to rotate, move, zoom in, zoom out and to measure distance between two objects. 3D Canvas supports following mouse actions terminologies:

Mouse Button	Usage
Left mouse button	Single and group selection
Right mouse button	3D interactive pop-up menu
Middle mouse button	Panning operations
Mouse wheel	Zoom operations
Shift + middle mouse button	Rotate operations

Menus

The following sections describe the command menus in Allegro 3D Canvas window.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

Table 1-1 File Menu Commands

This Command...	Does this...
<i>File</i>	
<i>Export</i>	Saves the 3D information into industry-related formats. For example, 2D PDF, 3D PDF, STEP, ACIS, and so on. Note: This option is disabled in Allegro Free Physical Viewer.
<i>Output Messages</i>	Saves messages from 3D Canvas to a text file.
<i>Close</i>	Closes 3D Canvas

Table 1-2 View Menu Commands

This Command...	Does this...
<i>View</i>	
<i>Camera</i>	
<i>Zoom Fit</i>	Sizes the entire design to fit within the Allegro 3D Canvas window
<i>Top</i>	Sets to display the top of the design
<i>Bottom</i>	Sets to display the bottom of a design
<i>Front</i>	Sets to display the front of a design
<i>Back</i>	Sets to display the back of a design
<i>Left</i>	Sets to display the left side of a design
<i>Right</i>	Sets to display the design from the right
<i>Isometric</i>	Sets to display the isometric view of the design
<i>Windows</i>	
<i>Messages</i>	Controls display of <i>Messages</i> window
<i>Options</i>	Controls display of <i>Options</i> pane
<i>Visibility</i>	Controls display of <i>Visibility</i> pane
<i>Symbols</i>	Controls display of <i>Symbols</i> pane

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

Table 1-3 Setup Menu Commands

This Command...	Does this...
<i>Preferences</i>	Provide options to Customizes 3D Canvas capabilities for the following categories:
<i>Appearance</i>	Provides controls to edit and save color themes and shadow preferences for 3D canvas.
	<div>Color Themes</div> <div>Provides options to set default and custom color themes.</div>
	<div>Shadows</div> <div>Controls following shadowing parameters:</div> <div> <div>■ Ambient shadows</div> <div>□ Intensity</div> <div>■ Simple shadow</div> <div>□ Blur</div> <div>□ Opaque</div> </div>
	<div><i>Projection</i></div> <div>Provides options to set projection:</div> <div> <div>■ Perspective</div> <div>■ Orthographic</div> </div>
<i>Messages</i>	Controls the display of new messages.
	<div><i>Silent mode</i></div> <div>If enabled prevents new messages to appear in the <i>Messages</i> window.</div>
<i>Interactive</i>	Enables interaction between 2D and 3D canvas.
	Note: This option is disabled in Allegro Free Physical Viewer.
	<div><i>Enable 2D/3D Interactive</i></div> <div>If enabled, highlights 2D design window and 3D Canvas interactively</div>
	<div><i>Enable Mask Layers Interactive Update</i></div> <div>If enabled, updates mask layers interactively when following operations are performed in the 2D design window.</div>

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

Symbol Representation Specifies options to display based on STEP model or place-bound shape or both

- Only the Boundary Shape
- Only the STEP model
- Both the Boundary Shape and the STEP model
- The STEP model, if available, else the Boundary Shape
- The Boundary Shape, if available, else the STEP model

Boundary Shape Source

- DFA Bound
- Place Bound

Transparency_ Sets mechanical symbol transparency.

By default, the symbol is opaque.

assign color

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.

- | | |
|--------------------------|--|
| <i>Selected color</i> | Indicates the currently chosen color. |
| <i>More Colors</i> | Displays a palette to choose colors. |
| <i>Highlight Pattern</i> | Choose the pattern for highlighting objects from a pull-down list. two options are available: <ul style="list-style-type: none">■ Solid■ Selected Pattern |

Selected Pattern

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

Assigning a Custom Color or Highlighting an Element

1. Choose *Display – Assign Color* from the top menu.

The color palette displays in the Options pane.

2. Click the color box of the new color for the element. The selected color displays in the top right of the palette.
3. Choose a pattern from the *Selected Pattern* to accentuate the element in the selected color if required.
4. Click to select an element.

The color of element changes in the design canvas.

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

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.

capture image

The `capture image` command captures the screen shots of the selected part of the design canvas and save in Jpeg (.jpg) format. The other formats for saving an image are TIFFT (*.tif), Bitmapap file (*.bmp), Portable Network Graphics (*.png), and Windows Device Independent Bitmap(*.dib).

When you capture an image a file browser is opened to save the image at a desired location. By default, the command is saved in the working directory.

Menu Path

File – Capture Canvas Image

Procedure

1. Zoom to that part of the design which you want to capture.
2. Choose *File – Capture Canvas Image*.

The *Capture Image* file browsers appears. By default, it shows the working directory.

3. Alternatively, browse a directory location.
4. Enter name of the image file.
5. Choose an image type.
6. Click *OK* to save.

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
- shapes
- ratsnests
- fingers
- bond wires
- cline segments

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. Use the *Save*, *Print*, or *Search* functions in the Show Constraints window to work with the report information.

color192

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, differential 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 Dialog

OK

Closes the dialog box after saving changes.

Cancel

Closes the dialog box without saving any changes. All palette changes are retained in the database, if saved.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<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>Controls Common to All Tabs Except Visibility Pane</i>	
<i>Load default color palette</i>	Loads the <code>\$HOME/pcbenv/allegro_192.col</code> and if not found, loads the Cadence defaults. The first column comprises popular colors typically used in designs.
<i>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>Save default color palette</i>	Exports the current design's customized color palette to a <code>.col</code> file stored in the <code>pcbenv</code> directory.
<i>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>Available colors</i>	Select a color from a table of all the possible color swatches.
<i>Highlight unused colors</i>	Select to highlight colors that have not been used. Not selected by default.
<i>Selected</i>	<p>Displays the selected color and pattern and allows customization of color.</p> <p>Three boxes are displayed.</p> <p>The first box from the left displays the selected color and the third box from the left displays the selected pattern. The middle box displays the look on combining the selected color and pattern.</p> <p>To display only the pattern, remove selection from the first check box from the left. Similarly, to display only the color, remove selection from the second check box.</p> <p>Click the first box showing the color to open the Set Color dialog box and add a custom color.</p>
<i>Available patterns</i>	Select a pattern to apply to an element.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

Layers Tab

Global Visibility

Controls whether or not all classes and subclasses are visible.

Filter layers

Searches every class and subclass for specific or groups of subclass layers.

Layers

Choose to display the *Layers* grid, which lets you control the color and visibility of classes and subclasses, along with levels of transparency for the design and shapes.

Left Pane: Displays each class associated with a group. The color and visibility of the subclasses associated with that class display horizontally.

Right Pane: For all but the *Display* group, which has no associated classes or subclasses, each row lists a subclass. An check mark indicates the subclass is visible. The color box indicates the color assigned to the subclass element.

Clicking the *All* column or *All* row enables visibility for the entire row or column.

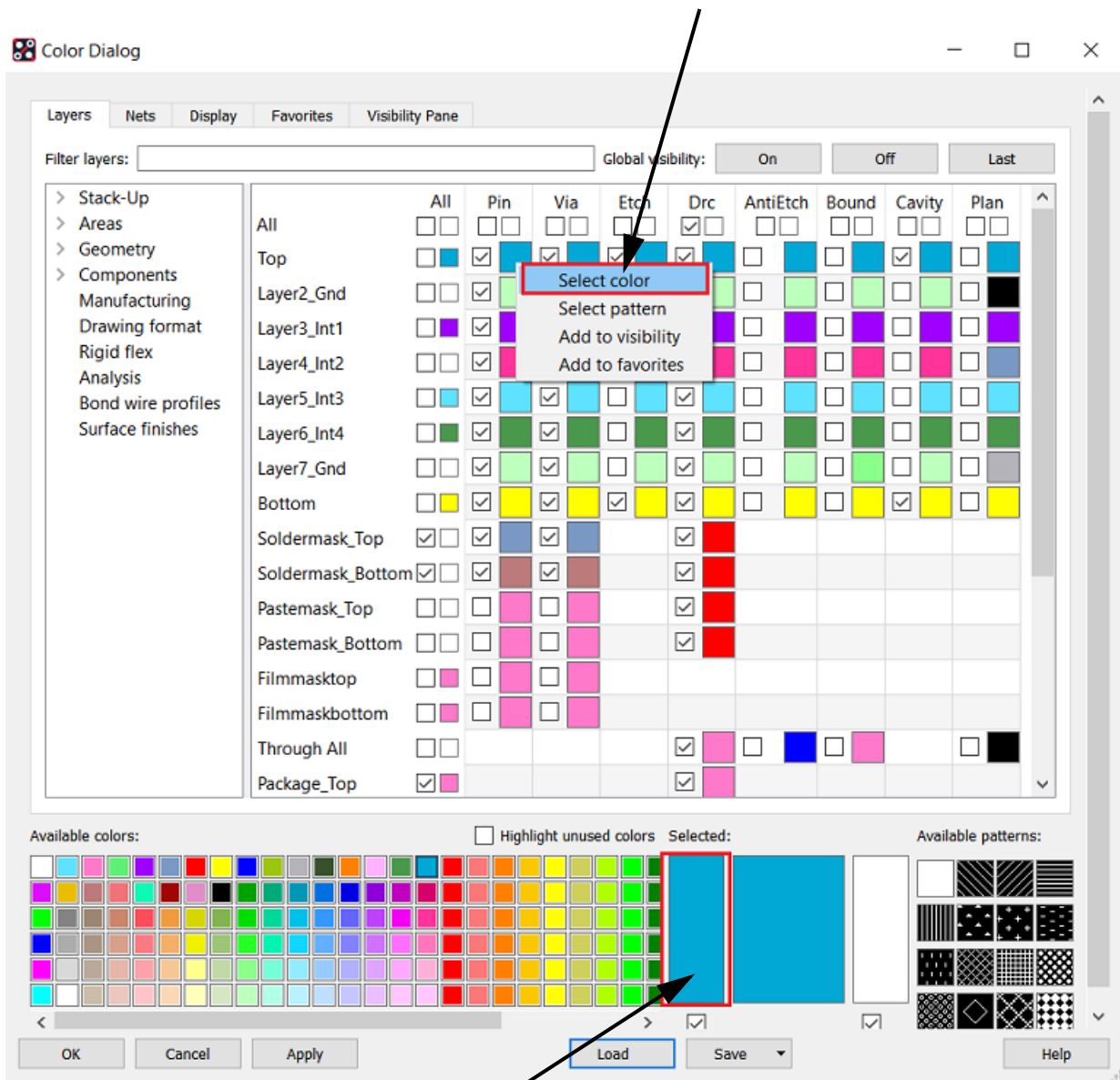
Clicking the intersection of the *All* row and *All* column cell (*All/All* cell) enables visibility globally. By default, subclasses are visible.

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 X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

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



This color box becomes active

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 Favorites*. The subclasses are copied, rather than moved, to the *Favorites* folder. Any changes to the *Favorites* folder updates the `CVSettings.xml` file in the `pcbenv` directory.

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

Display Tab

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.

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

Backdrill holes: Specifies the color of backdrill holes. The default is green.

Drill holes: Specifies the color of drill holes. The default is grey.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

Drill labels: 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 drill labels: 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.

Alignment Guides: Specifies the color of alignment guides. The default is pink.

Shadow mode

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

Enabled: Activates and deactivates *Shadow mode*, which darkens the colors of objects and elements of your design. Use this with the hilight command.

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.

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.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<i>Shapes Transparency</i>	<p>Assigns varying degrees of transparency to shapes only. Sliding the bar completely to the right (100%) represents a pre-16.0 graphics display.</p> <p>Sliding the bar completely to the left (0%) causes shapes to display with less intensity.</p>
<i>Object filter</i>	<p>Filters objects (vias, pins, or DRCs) from the design canvas. Objects smaller than the specified size (in pixel) are not displayed in the canvas when zooming or panning the design.</p> <p>Sliding the bar sets the objects size in pixel (px).</p>
<i>Nets Tab</i>	<p>Displays the Nets grid, which alphabetically lists nets hierarchically by bus, differential pair, Xnet, match group, or net group.</p> <p>Left pane: Alphabetically lists nets hierarchically by bus, differential pair, Xnet, or net.</p> <p>Right pane: Displays each class associated with the selected net object. The color and visibility of the subclasses associated with that class display horizontally.</p> <p>An check mark 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.</p>
<i>Hide Custom Colors</i>	<p>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 <i>Nets</i> grid.</p>
<i>Sort ascending</i>	<p>Choose to display list of nets in ascending order.</p>
<i>Sort descending</i>	<p>Choose to display list of nets in descending order.</p>
<i>Exclude default nets</i>	<p>Choose to exclude the default nets from the list.</p>
<i>Filter nets</i>	<p>Use to search a net from the list</p>
<i>Clear All Nets</i>	<p>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 <i>Nets</i> grid.</p>
<i>Visibility Pane</i>	

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<i>Visible classes</i>	Lists the classes that are visible in the Visibility window pane in your workspace. Click the Up and Down buttons () to add or remove classes. You can also drag and drop classes.
<i>Available classes</i>	Lists all the classes that are available and can be added to the list of visible classes.
<i>Show global visibility</i>	Displays the Global Visibility field in the Visibility window pane. Selected by default.
<i>Show view selection</i>	Adds the View field to enable view listing. Selected by default.
<i>Show stackup selection</i>	Enables display of layer stackup information. Not selected by default. Note: Available only if design has more than one zone in stackup.
<i>Show conductors</i>	Enables the display of Conductors under Layer. Selected by default.
<i>Show planes</i>	Enables the display of Planes under Layer. Selected by default.
<i>Show masks</i>	Enables the display of Masks under Layer. Selected by default.
<i>Show wire profiles</i>	Enables the display of wire profiles.
<i>Show layer numbers</i>	Enables the display of layer numbers with their names.
<i>Button size</i>	Specifies button size.
<i>Spacing</i>	Specifies the spacing between color boxes.
<i>Reset Default</i>	Restore the default visibility settings

Select 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>Pick Screen Colors</i>	Click to choose user-defined colors.
<i>Custom Colors</i>	Displays user-defined colors.

<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>Val</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>HTML</i>	Represents the color code in HTML
<i>Add to Custom Colors</i>	Moves the color you created with the vertical sliding bar and crosshair to the <i>Custom Color</i> section of available user-defined colors.
<i>OK</i>	Closes the dialog box and saves your modifications.
<i>Cancel</i>	Closes the dialog box and does not save your modifications.

Procedures

Assigning Colors to Subclasses

1. Choose *Display – Color/Visibility*.

The Color Dialog box appears.

2. Click *Layers*.

3. Choose a folder from the left pane that contains the subclass whose color you want to change.

4. Click the new color in the Color section.

The Selected color box shows the color you have chosen.

5. Click the color box next to the subclass whose color you want to change.

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*.

6. 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).
7. 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. Open the *Display* tab.
4. In the *Color* section, click a color box. It is the new color you want to assign.
5. In the *Display* tab, click the color box next to the item (*Grids*, *Ratsnest*, *Temporary highlight*, or *Waived DRC*) whose color you want to change.

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 a check mark appears in each box associated with that subclass.
- ☐ To control the visibility of an individual subclass, click its associated box, and a check mark 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 *Selected*. For details, see the [Select 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*, *Val*, *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 *Save – 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 color palette, choose *Save – Load default color palette*.

Importing a Customized Color Palette

1. Choose *Display – Color/Visibility*.

The Color Dialog box appears.

2. Choose *Load – 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 color palette, choose *Load – Load default color palette*.

Setting Transparency Globally

1. Choose *Display – Color/Visibility* to display the Color Dialog box.
2. Open the *Display* tab.

3. In the *Global transparency* section, use the 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. Open the *Display* tab.
3. In the *Shapes transparency* section, use the 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 *Favorites*.
5. Right-click and choose *Add to Favorites* from the pop-up menu.
6. Add as many subclasses as necessary.

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

7. A subclass stored in the *Favorites* folder can be removed by hovering your cursor over the color box associated with the subclass, right-clicking and choosing *Remove from 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. Open *Nets* tab.
3. Choose the type of element to display. *List...* displays nets from an external list.
4. Click the new color in the *Color* palette.

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

5. 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 check mark 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, (for example, all Pins,) click the color box in the *All* row.

6. 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

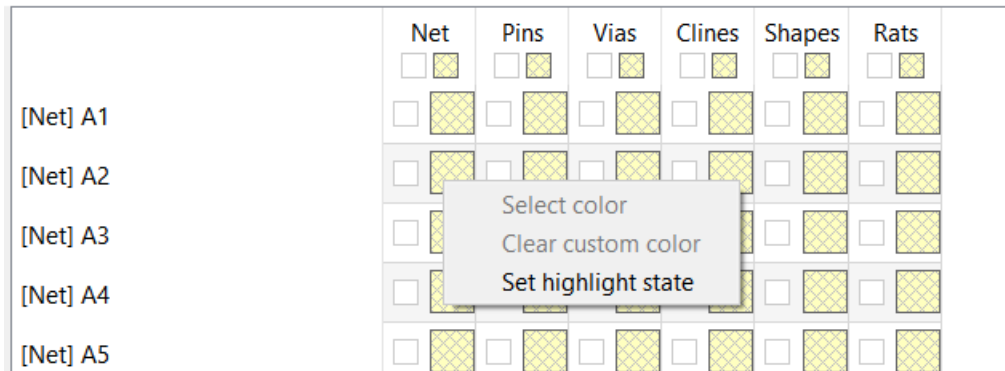
1. Run the `color192` command.

The Color Dialog box appears.

2. Open *Nets* tab.
3. Hover your cursor over a box with no check mark in it to the left of the element's color box.
4. Right-click and choose *Set Highlight State*.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer



The element becomes highlighted in the design canvas.

Removing the highlight state from nets and net elements

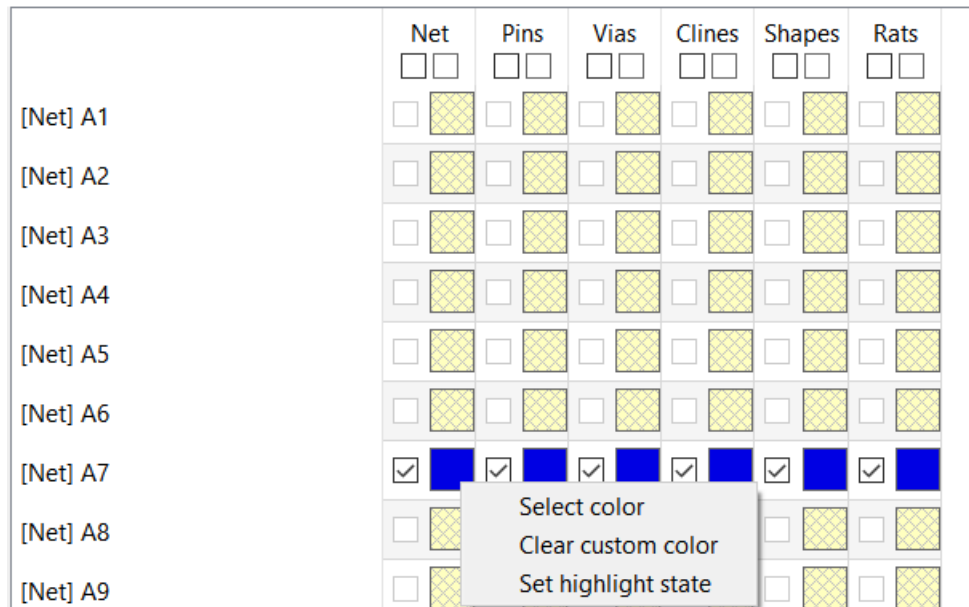
1. Hover your cursor over an element's color box.
2. Right click and choose *Clear Highlight State*.

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

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

1. Run the `color192` command.
The Color Dialog box appears.
2. Click *Nets*.

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



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.

Overriding Custom Colors

1. Run the `color192` command.

The Color Dialog box appears.

2. Open *Nets* tab.
3. Click the new color in the *Color* palette.

The *Selected* 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 check mark 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. Open *Nets* tab and choose *Filter* to narrow the number of displayed nets. The first found net appears in the grid.
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

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.

Save view

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 `.color` file extension and stores the file in the current working directory.

Click ... to browse for an existing filename and overwrite its contents.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<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.
	<i>Partial</i>	<p>Allows a color view file to be created that stores only <i>changes</i> 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.</p> <p>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.</p>
	<i>Partial with toggle</i>	<p>Functions the same as the <i>Partial</i> view replacement method because the color view file only stores changes. However, settings that you change toggle when you load the color view file.</p> <p>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.</p>
<i>Preserve zoom level</i>	Enable to save the zoom level	
<i>Preserve flip state</i>	Enable to save the flip state	
<i>Save</i>	Saves a new or changed file	
<i>Close</i>	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 <i>Save</i> before clicking <i>Close</i> .	

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.
- 3.

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

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 Customization

DataTips Customization Dialog Box

<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 X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

Apply filter

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



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).

Colors

Click to open Select Color dialog for name and value

Load default CDT file

Loads settings from the default `custdatatips.cdt` file.

Save default CDT file

Saves modifications to the default settings in the `custdatatips.cdt` file.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<i>Load Custom CDT File</i>	Imports your customized datatip settings from an external <code>.cdt</code> file and applies them to the current design. A file browser appears with the filter set to <code>*.cdt</code> and a list of all <code>.cdt</code> files available in the current local working directory. You can manually browse to other directories to open a <code>.cdt</code> 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 <code>.cdt</code> file.
<i>Save Custom CDT File</i>	Exports the current design's customized datatip settings to an external <code>.cdt</code> file stored in your local working directory. A file browser appears with the filter set to <code>*.cdt</code> and a list of all <code>.cdt</code> files in the current local working directory. You can manually browse to other directories to specify an alternate save location.
<i>OK</i>	Saves settings to the <code>.cdt</code> file currently loaded and closes the dialog box.
<i>Cancel</i>	Closes the dialog box without saving any changes.
<i>Reset to defaults</i>	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 grid

Displays the *Define Grids* dialog box, used for controlling the X and Y grid values for both etch and non-etch grids and for customizing the grid for each etch layer.

The non-etch/non-conductor grid is for interactive commands, such as manual placement, drafting, and the like. The same single-increment grid, with grid points spaced uniformly apart across the grid, is used for all non-etch layers.

Etch/conductor grids are used for interactive routing and etch editing. A separate X,Y grid exists for each etch layer in the design (TOP, INTERNAL, BOTTOM, and so on.). For each etch grid, you can set a single increment value, or up to a maximum of 20 grid increments for a grid of repeating pattern with different spacing between grid points.

Note: Many automatic tools, such as test prep, Allegro PCB Router, and automatic placement, provide their own grid setup.

The default point of origin for all layers is 0, 0. The default increment setting for non-etch layers is 100, 100. For etch layers, the default setting is 25, 25.

For additional information about defining a variable grid for etch/conductor subclasses, see the *Routing the Design* user guide in your documentation set.

You may also access the *Define Grids* dialog box by:

- Choosing *Setup – Design Parameters* (prmed command) to access the Design Parameter Editor's *Display* tab and clicking *Setup Grids*.

Menu Path

Setup – Grids

Define Grids Dialog Box

Use this dialog box to reset the point of origin for X and Y, as well as the spacing between the grid points for X and Y.

Grids On

Toggles the grid on and off for all layers. The default is off (invisible). A check mark in the box indicates that the grid is visible.

Layer

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<i>Non-Etch/Non-Conductor</i>	<p>Sets a single-increment X,Y grid for all non-etch/non-conductor layers.</p> <p>The spacing fields in the Non-Etch/Conductor section do not control the route grid. Non-etch/conductor grids are used for all classes other than ETCH/CONDUCTOR and do not support multiple spacing values.</p>
<i>All Etch/All Conductor</i>	<p>Sets global etch/conductor layer values. Each field sets the corresponding field in each etch/conductor layer to the grid value that you enter here.</p>
<i>TOP/SURFACE</i>	<p>Indicates the grid for the TOP etch/ Surface conductor layer. The list of etch/conductor layers must include at least TOP/SURFACE and BOTTOM/BASE, with any internal etch/conductor layers listed below TOP/SURFACE. Each etch/conductor layer has default settings that are identical to those in the layer above it.</p>
<i>BOTTOM/BASE</i>	<p>Indicates the grid for the BOTTOM/BASE conductor layer.</p>
<i>Offset:</i>	<p>The X, Y coordinate is the point at which the grid begins in relation to the 0,0 point of the design. The grid begins at 0,0 plus the values you specify for the X and Y offset. You can use this box to offset the grid without having to relocate the drawing origin to obtain a shifted grid pattern.</p> <p>It can be difficult to edit a design where grids have been shifted, because items are located off-grid. Cadence recommends that you shift grids back to their original location before editing interactively.</p>
<i>Spacing:</i>	<p>The X, Y coordinate controls the placement of all components and for all classes other than etch/conductor. X is the length of the grid in the X (horizontal) direction. Y is the length of the grid in the Y (vertical) direction.</p>

Procedures

Creating a Routing or Non-Etch Grid

1. Run the `define grid` command.

The *Define Grids* dialog box appears.

2. Set *Spacing* and *Offset* for all layers.

You can set the same route grid for all layers by entering values in the *All Etch/All Conductor* fields, or you can set different route grids for each layer by entering values in the individual layer fields.

3. If you want to display the grid, check *Grids On*.
4. Click *OK* to close the *Define Grids* dialog box.

Controlling the Visibility of the Non-Etch Grid

1. Run the `define grid` command.
The *Define Grids* dialog box appears.
2. Check the *Grids On* box to display the grid.
or
deselect the *Grids On* box to hide the grid.

dehighlight

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:

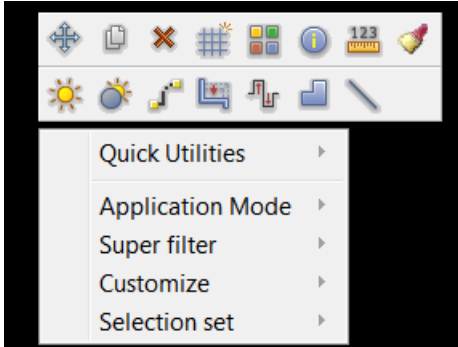
```
<element type><element name> dehighlighted
```

10. Right-click and choose *Done*.

display toolbar param

Use the Customize dialog box to control the look of the toolbar on your user interface.

Apart from standard task-specific toolbars, a *ContextMenu* toolbar is also available that contains frequently-used commands.



If enabled, the *ContextMenu* toolbar starts appearing on the right mouse button pop-up menu.

Menu Path

View – Customize Toolbar

Customize Dialog Box

Toolbars Tab

<i>Toolbars</i>	Lets you display or hide categories of toolbar buttons. Checking off all the toolbars extends the size of the working area in the user interface.
<i>Toolbar Name</i>	Displays the name of the selected toolbar.
<i>New</i>	Opens the <i>New Toolbar</i> dialog box that lets you add a user-defined toolbar category to the list. You can drag existing toolbar buttons into this new category and add them to the toolbar in the user interface.
<i>Delete</i>	Deletes the user-defined toolbar selected in the <i>Toolbars</i> section.

Commands Tab

<i>Select a toolbar to rearrange</i>	Select a toolbar category from the pull-down menu to modify.
<i>Add Command</i>	<p>Opens the <i>Add Command</i> dialog box that lets you add commands to the selected toolbar category.</p> <p>In the <i>Add Command</i> dialog box, commands are listed in order of menu and submenus. A list of <i>All Commands</i> is also available, which is sorted by command name.</p>
<i>Buttons</i>	Displays the buttons that appear in the selected toolbar category.
<i>Move Up</i>	Moves up the selected button in the <i>Buttons</i> window. You can also drag the button to change its position in the toolbar
<i>Move Down</i>	Moves down the selected button in the <i>Buttons</i> window. You can also drag the button to change its position in the toolbar.
<i>Delete</i>	Deletes the selected button from the <i>Buttons</i> window.
<i>Reset</i>	Returns the toolbars to their default configurations.

Procedures

Hiding Toolbar Categories

You can hide all toolbar categories, or only those that you do not use.

1. Choose *View – Customize Toolbar*.

The *Customize* dialog box appears.

2. Choose the *Toolbars* tab.

A listing of all the available Toolbars appears.

3. Deselect the boxes next to the toolbar categories in the Toolbars window that you want to hide.

The buttons disappear. If you deselect all the categories, the toolbar disappears, giving you more design workspace.

Note: You can see the different toolbar categories by clicking the *Commands* tab on the *Customize* dialog box and highlighting a category. The corresponding buttons display in the *Buttons* window.

Rearranging Buttons

You can rearrange buttons for any toolbar category. This is useful if you only use one button from a category. You can add the button to another category and remove the toolbar category that you do not use from the design window.

1. Choose *View – Customize Toolbar*.

The *Customize* dialog box appears.

2. Choose the *Commands* tab.

3. Select the toolbar category to rearrange from the pull-down menu.

4. Click *Add Command*. Select the toolbar that contains the button you want to add. Choose the commands and add them by clicking *Add*. Close *Add Command* dialog box once all the commands are added.

5. Choose and drag the buttons to rearrange the positions of the buttons.

Note: To return the toolbars to their default configuration, choose *Reset*.

Creating Your Own Toolbar

1. Choose *View – Customize Toolbar*.

The *Customize* dialog box appears.

2. Choose the *Toolbars* tab.

3. Choose *New*.

The *New Toolbar* dialog box appears.

4. Enter the name for the new toolbar in the name field.

5. Choose *OK* to close the *New Toolbar* dialog box.

The new toolbar is added to the list in the *Toolbars* window and a new toolbar appears. Choose the *Commands* tab and then add buttons you want to add, to your new toolbar.

The new toolbar expands to hold as many buttons as you want.

6. Drag your toolbar to the location you want to use it from.

You can use it as a floating vertical or a horizontal toolbar anywhere on your desktop, however, as soon as you drop it into the toolbar area, it becomes fixed.

Deleting a Custom Toolbar

1. Choose *View – Customize Toolbar*.

The *Customize* dialog box appears.

2. Choose the *Toolbars* tab.

3. Highlight the toolbar you want to delete, and then choose *Delete*.

The *Delete* button is not available unless you have chosen a toolbar you previously defined.

exit

Exits the active layout, closes the application, and returns to the host operating system. Any changes in the design are discarded.

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.

Menu Path

View – Flip Design

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.

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

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.

Menu Path

Display – Highlight

Options Tab for the highlight Command

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

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

Procedure

Highlighting Elements

1. Do one of the following:

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

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

```
<element type><element name> highlighted
```

—or—

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

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

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

- d. Click the element to highlight.

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

```
<element type><element name> highlighted
```

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

layer priority

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

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

Menu Path

Display – Layer Priority

Display Priority Dialog Box

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

<i>Default display 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.
<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.

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

Assigning a Display Priority To Layers

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

manage_settings

The `manage_settings` command lets you save and manage pre-defined toolbars and dock panes settings of the layout editor. Once created, you can export and import the custom settings across different systems.

Menu Path

View – UI Settings – Manage Settings

Manage UI Settings Dialog Box

Use this dialog box to manage the custom settings.

<i>Custom Settings</i>	Displays names of already saved UI settings
<i>+</i>	Opens a standard file browser to import the UI configuration (*.ini) file.
<i>Export</i>	Opens a standard file browser to export the existing UI configuration (*.ini) file. By default, the settings are saved to <HOME>/pcbenv directory.
<i>Apply</i>	Applies the selected UI settings to the active database.
<i>Delete</i>	Deletes the existing UI settings.

Procedure

Applying a Custom Setting

1. Run `manage_settings` from the command window prompt.
2. Select the name of an already saved custom settings in the *Custom Settings* section.
3. Click *Apply*.

The selected settings are applied to the active database.

Importing a Custom Settings

1. Run `manage_settings` from the command window prompt.
2. Click *+* in the *Custom Settings* section.

A file browser opens.

3. Browse the location of the configuration file and click *Open*.

The selected settings are now available under the *Custom Settings* section.

Exporting a Custom Settings

1. Run `manage_settings` from the command window prompt.
2. Select the name of an already saved settings in the *Custom Settings* section.
3. Click *Export*.

A file browser opens.

4. Browse the location to save the configuration file and click *Save*.

Restoring Legacy Settings

1. Run `manage_settings` from the command window prompt.
2. Click *+* in the *Custom Settings* section.

A file browser opens.

3. Browse to default configuration file `AllToolbars*.ini`, which is located at `<installation_directory>/share/pcb/text`.

The *AllToolbars* settings is now available in the *Custom Settings* section.

4. Select *AllToolbars* and click *Apply*.

The legacy settings are applied to the active database.

open

Opens an existing design file in the current directory. A file browser lets you search for the specified design file if you do not provide a file name. You can also open a file from the most recently used files using the *File – Recent Designs* menu.

Menu Path

File – Open

File – Recent Designs

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. You can type the name of the file you want in the *File name* field, or choose the file from the list. Selecting a directory in the list by double clicking on it pushes into the directory.

The *Look in* drop-down allows you to navigate higher in the directory hierarchy. You can drag and drop folders in the *Look in* column to create links to directories. To delete those link a *Remove* option is available on the right-click menu.

The *File of type* field, by default, provides the typical extension of the file required. The drop-down provides access to additional extensions. You can override the extensions by typing a name with one or more wildcard characters in the *File name* field. The preview button displays a simple graphic of the database, the image of which depends on the type of database you are viewing.

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 P button in the upper right corner to display the graphic preview of the design.
4. Click *Open* to open the file.

opengl report

Checks system graphic information and creates a report listing vendor card type and version.

Menu Path

Setup – OpenGL Status

Note: OpenGL must be enabled for this command to function.

plot

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.

Menu Path

File – Plot

Print Dialog Box

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

Procedures

Plotting Your Design

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.

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

The `plot setup` command lets you set parameters for plotting a design. (See the `plot` command for additional details on plotting.)

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

<i>Plot scaling</i>	<i>Fit to page:</i> Indicates the plot file is to be scaled to fit the entire plotted page.
	<i>Scaling factor:</i> Indicates the scale of the finished plot.
	<i>Default line weight:</i> Converts any zero width line to a width proportional to the setting. Aids in displaying very thin lines on high-resolution output.
<i>Plot orientation</i>	<i>Auto center:</i> Centers the design on the plot page. This control automatically invokes when you choose the Fit to page setting.
	<i>Mirror:</i> Flips the design end-for-end about the Y axis. Useful for viewing top and/or bottom layers.
<i>Plot method</i>	<i>Color:</i> 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 (<code>allegro_plot_param.stipples</code>); 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.
	<i>Black and white:</i> Directs the output to print in black and white.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<i>Plot contents</i>	<i>Screen contents:</i> Prints/plots the contents of what is currently displayed in the design area of the user interface. <i>Sheet contents:</i> Prints/plots the extents of all currently-visible graphics within the design (not the drawing extents).
<i>IPF setup</i>	<i>Vectorize text:</i> Specifies that text output to the IPF file is broken into line vectors. <i>Width:</i> 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 <i>Vectorize text</i> is enabled, and a negative value is entered in the <i>Width</i> 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 <i>Vectorize text</i> and <i>Width</i> settings apply as specified when the <code>create plot</code> command executes.
<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 researchable 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.
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

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 X Package Design Plus Free Physical Viewer

Allegro Package Design Plus 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 – Reset UI to Default

Syntax

`reset_dockwindows`

Displaying All Foldable Window Panes to Default Positions

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

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

save_settings

The `save_settings` command saves the currently active UI settings of the layout editor with a new name.

You can configure user interface of the application for a specific task and save the settings. Multiple settings can be saved based on the different task requirements. For example, you can save settings for placements, settings for routing, settings for design checks, and so on.

UI settings includes the state and the location of toolbars and docking panes. The settings are saved in the `.config` folder in `%USERPROFILE%` of Windows and `$HOME` of Linux.

Note: Saving a new setting becomes the current default. Opening a new session of layout editor shows this setting by default.

Menu Path

View – UI Settings – Save Settings

Dialog Box

The command displays *Save UI Settings* dialog box to enter a name.

Procedure

1. Run `save_settings` from the command window prompt.

If the name is new (has never been saved), the settings are written to memory.

If the name already exists, following error message is displayed:

```
Setting with name:<name> already exists. To save this UI  
configuration, use a unique name.
```

2. Click *OK* to save the settings with a different name.
3. Enter the new name into the field, and click *Save*.

script

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.

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

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 views1

The `showhide views1` command opens a second work area. You can perform most of the actions inside the Split View window. You can also zoom and pan in the Split View window independent of the main design window.

Menu Path

View – Split View – Float

Toggles the visibility of the *Split View* window pane.

A check mark next to *View – Split View – Float* 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 *Split View* 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 *Split View* window pane to expand it, or clicking the X to hide it.

Controlling the Visibility of the Split View Window Pane

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

showhide viewsh

The `showhide viewsh` command divides the design canvas from left to right direction into two sections. The size of each section can be adjusted using resize cursor that appears when you hover the cursor over the splitted edge of the section. You can zoom and pan in both the windows independent of each other. You can perform design tasks in one of the window which are also replicated and displayed in the other window.

Menu Path

View – Split View – Horizontal

Toggles the visibility of the *Split View* window pane.

A check mark next to *View – Split View – Horizontal* 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.

showhide viewsv

The `showhide viewsv` command divides the design canvas in a top to down direction into two sections. The size of each section can be adjusted using resize cursor that appears when you hover the cursor over the splitted edge of the section. You can zoom and pan in both the windows independent of each other. You can perform design tasks in one of the window which are also replicated and displayed in the other window.

Menu Path

View – Split View – Vertical

Toggles the visibility of the *Split View* window pane.

A check mark next to *View – Split View – Vertical* 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.

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

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 – Pin</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 – Pin</i> to compare the contents of each window.

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.

Find by Query

You can use this method to issue a query to find the specific objects (nets, clines, shapes, voids, and so on) based on their attribute values in the database. The query returns all the objects that match a specified filter or criteria. You can define the criteria in the *Find By Query* dialog box available in the *Find* filter window pane.

<i>Objects</i>	Displays a list of different types of objects. For example, components, clines, text shapes, symbols, and so on.
<i>Configure</i>	Controls the display of objects in the <i>Objects</i> tab. De-selecting a check box will stop showing that object in the <i>Objects</i> tab.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<i>Fields</i>	<p>Displays selected objects and their attributes.</p> <p>To add objects to the <i>Fields</i>, either double-click or drag and drop them from the <i>Objects</i> tab.</p> <p>To remove objects from the <i>Fields</i>, use drag and drop method.</p>
<i>Filters</i>	<p>Display filtered objects and set the values of their attributes.</p> <p>To add objects to the <i>Filters</i>, either double-click use the arrow button or drag and drop them from the <i>Fields</i> tab.</p> <p>You can remove objects from the <i>Filters</i> by using the arrow button or the drag and drop method.</p> <p>The attributes of the objects are added with default value “*” . You can double-click to specify the values by setting values in the <i>Filter Setting</i> dialog box.</p>
->	Add the selected object from the <i>Fields</i> to the <i>Filters</i> section.
<-	Removes the selected object from the <i>Filters</i> section.
<i>AND</i>	Controls the filter operations. If enabled, the filter objects are added under node of AND. By default, this option is enabled.
<i>OR</i>	Controls the filter operations. If enabled, the filter objects are added under node of OR.
<i>Matching Objects</i>	<p>Displays the total number of matching objects depending on the criterion defined in the <i>Filters</i> section and display the results in a table format.</p> <p>A default list of attributes for each type of object is always displayed in the table. For example, <i>Text</i> objects attributes <i>Class Name</i>, <i>Subclass Name</i>, and <i>Text Contents</i> are always displayed in the table.</p>
<i>Defer Selection</i>	Suspends the selection of objects when clicked in the <i>Matching Objects</i> table for canvas selection or as an input to a command until the <i>Select</i> button is clicked.
<i>Select</i>	<p>Available when <i>Defer Selection</i> is enabled.</p> <p>Redirects selection of objects from the <i>Matching Objects</i> table for canvas selection or to a command as an input.</p>
<i>Display Search Result in Table</i>	Click to display the <i>Matching Objects</i> results in a table format.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

<i>Configure Search Result Table</i>	Click to control the display of object attributes for the object types in the <i>Matching Objects</i> result table.
<i>Save Query</i>	Saves the current query settings in a file (.qfnd). You can set <code>miscpath</code> env variable in the <i>User Preferences Editor</i> dialog box to specify the location for saving queries.
<i>Load Query</i>	Loads previously saved query files.
<i>Clear Query</i>	Removes the settings for current query.
<i>Rerun Query</i>	Updates the <i>Matching Objects</i> results for the current query.
<i>Export Result</i>	Exports the query results in an XML or CSV format.
<i>Close</i>	Displays <i>Close Dialog</i> for saving query.
<i>Cancel</i>	Closes the dialog box without saving any changes.

The *Objects* tab displays list of different types of objects that can be modified in the *Configure* tab. You can add required objects and their attributes in the *Fields* for creating a query. The *Filters* section defines the query that will be used to extract the information.

The search results are displayed on the fly in the *Matching Objects* section. The *Configure Search Table* icon lets you identify the fields you want to see in the result table.

You have options to save query in a file (.qfnd), clear query, and load query. If inactive for some time, the *Rerun Query* button becomes active to refresh the results. You can export the query in standard XML or CSV formats.

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.

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

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.
<i>Dy</i>	Displays the absolute y-distance (vertical) between two markers. Note: Manhattan Dist = Dx + Dy
<i>Pick Angle</i>	Displays the angle between two markers. This field is useful when doing offset routing.
<i>Width</i>	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:

<i>Etch/Conductor Dist</i>	Displays the distance along the center lines of the connect lines connecting the two elements.
<i>Total Etch/Conductor</i>	Displays the accumulated connection path length from the first selection you made.
<i>Via Count</i>	Displays the number of vias on the path joining the last two points you picked.
<i>Air Gap</i>	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.
<i>On Subclass</i>	Displays the subclass that is common to both elements, if they have one. This field does not display if there is no common subclass.

Procedure

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

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

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

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

The following temporary markers on each element appear:

- ☐ A cross indicates the center of a pad or the vertex of a connect line or filled rectangle.
- ☐ A square at the nearest grid point identifies all other picks.

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

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

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

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

show property

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:	
	<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.	
<i>Show Def</i>	Displays the definition of the chosen property, which appears in a separate window that remains open until you close it.	

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

Reset 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.
11. Set the *Global Visibility* to *All Invisible*.
12. Click *Yes* in the confirmer that appears.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus Free Physical Viewer

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. 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.

Menu Path

Display – Status

Status Tab

<i>Connect pins</i>	Displays the number of connect pins in the design. (symbol mode only).
<i>Mechanical pins</i>	Displays the number of mechanical pins in the design. (symbol mode only).
<i>Symbols and Nets</i>	<p><i>Unplaced symbols:</i> Displays the number and percentage of <i><unplaced symbols>/<total symbols></i> in the design. A green color box means all symbols are placed; yellow, some placed; and red, none placed. (layout mode only).</p> <p><i>Unrouted nets:</i> Displays the number and percentage of <i><unrouted or partially nets>/<total nets></i> in the design. A green color box means all symbols are placed; yellow, some placed; and red, none placed. (layout mode only).</p> <p><i>Unrouted connections:</i> Displays the number and percentage of <i><unrouted connections>/<total pin-to-pin connections></i> 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.</p>

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.

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.

Allegro X Package Design Plus Free Physical Viewer

Allegro Package Design Plus 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.

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.

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 netlist 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 netlist 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 Netlist or Net Name from the pop-up menu.

viewlog

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.

Menu Path

File – File Viewer

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 Windows systems.
<i>File – Pin</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 – Pin</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 F2.

–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 `F11`.

–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 F12.

–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–

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

Use dynamic zooming by way of the middle mouse button.