

# **Allegro<sup>®</sup>**

## **Free Physical Viewer**

**Product Version 15.7**  
**July 2006**

© 1999-2006 Cadence Design Systems, Inc. All rights reserved.  
Printed in the United States of America.

Cadence Design Systems, Inc., 555 River Oaks Parkway, San Jose, CA 95134, USA

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

All other trademarks are the property of their respective holders.

**Restricted Print Permission:** This publication is protected by copyright and any unauthorized use of this publication may violate copyright, trademark, and other laws. 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. This statement grants you permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used solely for personal, informational, and noncommercial purposes;
2. The publication may not be modified in any way;
3. Any copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement; and
4. Cadence reserves the right to revoke this authorization at any time, and any such use shall be discontinued immediately upon written notice from Cadence.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

# Contents

<u>blank waived drcs</u>	8
<u>Menu Path</u>	8
<u>Procedure</u>	8
<u>color</u>	9
<u>Syntax</u>	10
<u>Dialog Boxes</u>	12
<u>Procedures</u>	15
<u>Examples</u>	22
<u>color priority</u>	23
<u>Priority Dialog Box</u>	23
<u>Defining Color Priority</u>	23
<u>dehighlight</u>	25
<u>Options Tab for the dehighlight Command</u>	25
<u>Procedure</u>	25
<u>exit</u>	26
<u>grid toggle</u>	27
<u>highlight</u>	28
<u>open</u>	32
<u>Menu Path</u>	32
<u>Syntax</u>	32
<u>Examples</u>	32
<u>Dialog Box</u>	33
<u>Procedure</u>	33
<u>plot</u>	34
<u>Print Dialog Box</u>	34
<u>Procedures</u>	35
<u>plot setup</u>	38
<u>Plot Setup Dialog Box</u>	38
<u>Procedure</u>	40
<u>rats all</u>	42
<u>Procedure</u>	42
<u>rats component</u>	43

## Allegro Free Physical Viewer

---

<u>Procedure</u> .....	43
<u>rats net</u> .....	44
<u>Procedure</u> .....	44
<u>redisplay</u> .....	45
<u>script</u> .....	46
<u>Scripting Dialog Box</u> .....	46
<u>Procedures</u> .....	47
<u>show element</u> .....	50
<u>Dialog Boxes</u> .....	50
<u>Procedures</u> .....	52
<u>show measure</u> .....	54
<u>Measure Dialog Box</u> .....	54
<u>Procedure</u> .....	55
<u>show property</u> .....	57
<u>Show Property Dialog Box</u> .....	57
<u>Procedures</u> .....	58
<u>show waived drcs</u> .....	61
<u>Menu Path</u> .....	61
<u>Procedure</u> .....	61
<u>status</u> .....	62
<u>Drawing Options Dialog Box</u> .....	62
<u>unrats all</u> .....	70
<u>Procedure</u> .....	70
<u>unrats component</u> .....	71
<u>Procedure</u> .....	71
<u>unrats net</u> .....	72
<u>Procedure</u> .....	72
<u>viewlog</u> .....	73
<u>Select File to View Dialog Box</u> .....	73
<u>Procedures</u> .....	74
<u>zoom center</u> .....	75
<u>Procedure</u> .....	75
<u>zoom fit</u> .....	76
<u>Procedure</u> .....	76
<u>zoom in</u> .....	77
<u>Procedure</u> .....	77

**Allegro Free Physical Viewer**

---

<u>zoom out</u>	78
<u>Procedure</u>	78
<u>zoom points</u>	79
<u>Procedure</u>	79
<u>zoom previous</u>	80
<u>zoom world</u>	81
<u>Procedure</u>	81

## **Allegro Free Physical Viewer**

---

---

# Allegro Free Physical Viewer

---

## 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.

## color

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

Controls the following aspects of design display:

- Colors for:
  - ☐ Subclasses
  - ☐ Grids\*
  - ☐ Ratsnest lines\*
  - ☐ Design window background\*
  - ☐ Highlighted objects during interactive commands\*
- Visibility of classes and subclasses\*

\* You can set and change these items using command arguments or using the Color and Visibility dialog box.

Because you can set the colors and visibility for each subclass of your design with this command, you can also use the `color` command to view all the classes and subclasses, and the groups under which they are located.

Lastly, the tool provides a global color palette, which you can modify and save as a local or database (design-specific) palette with the `color` command.

For more details on customizing colors used in your design, see *Getting Started* in the user guide of your documentation set.

### Menu Path

*Display – Color/Visibility*

## Syntax

```
color [-gridcolor | -ratscolor | -highlightcolor | -permcolor "<color_number>"]  
      [-background "<RGB_value>"] [-group "<group_name>"] [-globvis "{off | on}"]  
      [-vis "{<class_name> | <class/subclass_name>}"] [-invis "{<class_name> |  
      <class/subclass_name>}"] [-toggle "<class_name> | <class/subclass_name>}"]  
      [-help]
```

**-gridcolor "<color\_number>"**

Sets the color of the grids in the design window. Choose the color from the current palette.

This argument is the same as the *Grids* field in the *Display* section of the Color and Visibility dialog box. (*Group* is set to *Display*.)

**-ratscolor "<color\_number>"**

Sets the color of the ratsnest lines. Choose the color from the current palette.

This argument is the same as the *Ratsnest* field in the *Display* section of the Color and Visibility dialog box. (*Group* is set to *Display*.)

**-highlightcolor  
"<color\_number>"**

Sets the color of objects that are temporarily highlighted during an interactive command, such as copy or move. Choose the color from the current palette.

This argument is the same as the *Temporary highlight* field in the *Display* section of the Color and Visibility dialog box. (*Group* is set to *Display*.)

**-permcolor "<color\_number>"**

Sets the permanent highlight color, the color of objects when you run the hilight command. Choose the color from the current palette.

This argument is the same as the *Permanent highlight* field in the *Display* section of the Color and Visibility dialog box. (*Group* is set to *Display*.)

## Allegro Free Physical Viewer

### color

---

**-background "<RGB\_value>"**

Sets the color of the design window's background. The RGB value is the 8-bit representation of a color value, ranging from 0 to 255. For example:

■ 0xff0000 is red.

■ 0xffffffff is white.

■ 0x000000 is black.

This argument is the same as the *Background* field in the *Display* section of the Color and Visibility dialog box. (*Group* is set to *Display*.)

**-group "<group\_name>"**

Opens the Color and Visibility dialog box, with the *Group* field set to the specified group name.

**-globvis "{off | on}"**

Controls visibility for all subclasses.

This argument is the same as the *Global visibility* field in the Color and Visibility dialog box.

**-vis "{<class\_name> | <class/subclass\_name>}"**

Enables visibility for an entire class or subclass.

This argument is the same as enabling the check box for a class or subclass in the Color and Visibility dialog box.

**-invis "{<class\_name> | <class/subclass\_name>}"**

Makes the specified class or subclass invisible.

This argument is the same as disabling the check box for a class or subclass in the Color and Visibility dialog box.

**-toggle "{<class\_name> | <class/subclass\_name>}"**

Reverses the current visibility for the specified subclass. If the subclass is visible, this command makes it invisible. If the subclass is invisible, this command makes it visible.

This argument is the same as clicking the check box for a class or subclass in the Color and Visibility dialog box.

**-help**

Displays command syntax in the console.

## Dialog Boxes

### Color and Visibility Dialog Box

Use this dialog box to control the colors and the visibility of the various classes and subclasses in the tool. You can also manage the colors of the design window's display: grids, ratsnest lines, highlighted elements, the background, and the brightness of active elements during interactive commands.

**Note:** If you set the `color_lastgroup` environment variable, the Color and Visibility dialog box appears with the *Group* field selection that you last used. If you do not set this environment variable, the Color/Visibility dialog box appears with the current active layer in the Options panel displayed in the *Group* field.

<i>Group</i>	<p>Specifies the group that you want to view.</p> <p>See the <i>Getting Started with Physical Design</i> user guide in your documentation set for the list of all classes and subclasses under each group.</p>
<i>Global visibility</i>	<p>Controls the visibility of all subclasses.</p> <p>Click the box to the left of a subclass' color to change that individual element's visibility.</p> <p>Running the <code>-globvis</code> argument with the <code>color</code> command performs the same function.</p>
<i>Classes</i>	<p>For all but the <i>Display</i> group, classes appear horizontally as column headings. The check box indicates whether the class is visible (selected) or invisible (deselected). By default, classes are visible.</p> <p>Running the <code>-vis</code>, <code>-invis</code>, or <code>-toggle</code> argument with the <code>color</code> command performs the same function.</p>
<i>Subclasses</i>	<p>For all but the <i>Display</i> group, subclasses appear vertically below each class name.</p> <p>The check box indicates whether the subclass is visible (selected) or invisible (deselected). By default, subclasses are visible. Running the <code>-vis</code>, <code>-invis</code>, or <code>-toggle</code> argument with the <code>color</code> command performs the same function.</p> <p>The color box indicates the color of the object in the subclass. The default is pink.</p>

## Allegro Free Physical Viewer

### color

---

<i>Display</i>	The following six fields appear when you set <i>Group</i> to <i>Display</i> . They control the appearance of features of the design window.				
<i>Grids</i>	<p>Specifies the color of the grids. The default is white.</p> <p>Running the <code>-gridcolor</code> argument with the <code>color</code> command performs the same function.</p>				
<i>Ratsnest</i>	<p>Specifies the color of ratsnest lines. The default is pink.</p> <p>Running the <code>-ratscolor</code> argument with the <code>color</code> command performs the same function.</p>				
<i>Temporary highlight</i>	<p>Specifies the color of objects that are temporarily highlighted during an interactive command, such as copy or move. The default is white.</p> <p>Running the <code>-highlightcolor</code> argument with the <code>color</code> command performs the same function.</p>				
<i>Waived DRC</i>	Specifies the color of waived DRC error markers. The default is yellow. The text within the marker retains the color setting for DRC error markers specified in <i>Group: Stack-Up</i> .				
<i>Permanent highlight</i>	<p>Specifies the color of objects when you run the <code>highlight</code> command. The default is white.</p> <p>Running the <code>-permcolor</code> argument with the <code>color</code> command performs the same function.</p>				
<i>Background</i>	<p>Specifies the color of the design window's background. The default is black.</p> <p>Running the <code>-background</code> argument with the <code>color</code> command performs the same function.</p>				
<i>Shadow mode</i>	<p>These settings appear when you set <i>Group</i> to <i>Display</i>. They allow you to highlight individual objects without affecting the visibility of that object's entire subclass.</p> <p>The <code>shadow</code> command performs the same function.</p> <table><tr><td><i>On/Off</i></td><td>Activates and deactivates <i>Shadow mode</i>, which darkens the colors of objects and elements of your design. Use this with the <code>highlight</code> command.</td></tr><tr><td><i>Brightness</i></td><td>Specifies the percentage of brightness applied to colors when <i>Shadow mode</i> is set to <i>On</i>. The default is 50%.</td></tr></table>	<i>On/Off</i>	Activates and deactivates <i>Shadow mode</i> , which darkens the colors of objects and elements of your design. Use this with the <code>highlight</code> command.	<i>Brightness</i>	Specifies the percentage of brightness applied to colors when <i>Shadow mode</i> is set to <i>On</i> . The default is 50%.
<i>On/Off</i>	Activates and deactivates <i>Shadow mode</i> , which darkens the colors of objects and elements of your design. Use this with the <code>highlight</code> command.				
<i>Brightness</i>	Specifies the percentage of brightness applied to colors when <i>Shadow mode</i> is set to <i>On</i> . The default is 50%.				

## Allegro Free Physical Viewer

### color

---

	<i>Dim active layer</i>	Applies the <i>Brightness</i> percentage to the colors of objects in the active layer, darkening the colors so that the highlighted objects stand out more.
<i>Palette</i>	Displays the color palette you want to use. By default, the palette saved with the design appears. For a new design, the global palette appears.  These are the options in the <i>Palette</i> field's menu:  <i>Read Database</i> Displays the database palette—that is, the color settings last saved with this design.  <i>Read Global</i> Displays the global palette provided by the tool.  <i>Read Local</i> Displays the color settings in the local palette.  <i>Write Local</i> Saves the current colors to the local palette, automatically stored in the <code>l1allegro.col</code> file.	
<i>Modify</i>	Opens the Color Editor, where you can change a color element in a palette. For details, see <a href="#">Color Editor Dialog Box</a> below.	
<i>Apply</i>	Applies color and visibility changes you have made to the current design. The dialog box remains open so you can continue modifying different classes and subclasses.	
<i>Reset</i>	Resets color changes you have made to the previous color.	

### Color Editor Dialog Box

Use this dialog box to set specific shades and hues of color for your color palette or for the design window's background.

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>Color   Solid</i>	Displays the resulting color.
<i>Hue</i>	Represents the selected color's hue.

## Allegro Free Physical Viewer

### color

---

<i>Sat</i>	Represents the selected color's saturation.
<i>Lum</i>	Represents the selected color's luminosity.
<i>Red</i>	Represents the amount of red in the selected color.
<i>Green</i>	Represents the amount of green in the selected color.
<i>Blue</i>	Represents the amount of blue in the selected color.

## Procedures

### Displaying Groups of Classes and Subclasses

1. Run the `color` command.

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

2. From the *Group* list, choose a group. The associated classes and subclasses appear.

**Note:** The *Display* group does not have classes and subclasses associated with it.

### Assigning Colors to Subclasses

1. Run the `color` command.

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

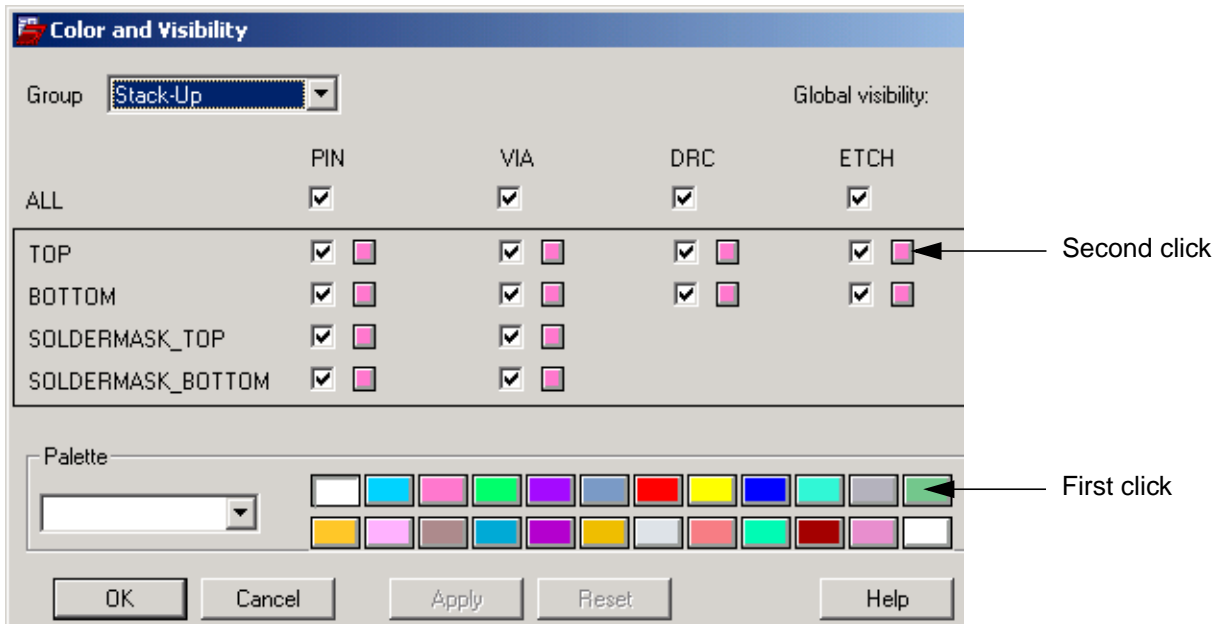
2. From the *Group* list, choose the group containing the subclass whose color you want to change.
3. In the *Palette* section of the dialog box, click one of the color boxes. This is the first click identified in the [Assigning Colors](#) figure in the next step. It is the new color you are assigning to a subclass.
4. Click the color box next to the subclass whose color you want to change. This is the second click identified in Figure 1-1. The color box for the subclass changes to the color you selected from the *Palette*.

For example, if *Group* is set to *Stack-Up*, you might want to change the color of the etch on the top layer. Click the color box for *TOP ETCH*.

## Allegro Free Physical Viewer

### color

Figure 1-1 Assigning Colors



5. Click *Apply* to update the drawing and continue changing colors.
6. When you are done making changes in this dialog box, click *OK* to save changes and close the dialog box.

The Options tab displays the color assigned to a subclass in a box next to the name of the subclass.

### Assigning Colors to Non-Class Design Elements

You can control the color of design elements in two ways:

- Using the Color and Visibility dialog box, described here.
- Running the `color` command with one of these arguments: `-gridcolor`, `-ratscolor`, `-highlightcolor`, or `-permcolor`. See [“Syntax”](#) on page 10 for details.

To change the color of grids, ratsnest lines, and highlighting using the user interface:

1. Run the `color` command.

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

2. From the *Group* list, choose *Display*.

3. In the *Palette* section of the dialog box, click one of the color boxes. This is the first click identified in the [Assigning Colors](#) figure in the previous section. It is the new color you want to assign.
4. In the *Display* section, click the color box next to the item (*Grids*, *Ratsnest*, *Temporary highlight*, *Permanent highlight*, or *Waived DRC*) whose color you want to change. This is the second click, similar to the one shown in the [Assigning Colors](#) figure.

The color box for this item changes to the color you selected from the *Palette*.

5. Click *Apply* to update the drawing and continue changing colors.
6. When you are done making changes in this dialog box, click *OK* to save changes and close the dialog box.

### Changing the Background Color

You can change the background color of the design window in two ways:

- Using the Color and Visibility dialog box and Color Editor, described here.
- Running the `color` command with the `-background` argument. See [Syntax](#) for details.

To change the background color using the user interface:

1. Run the `color` command.

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

2. From the *Group* list, choose *Display*.
3. In the *Display* section, click the *Background* color box.
4. In the Color Editor, set the color you want for the background and then click *OK*. For details, see [Color Editor Dialog Box](#).

The color next to the *Background* box in the Color and Visibility dialog box changes as does the color of the design window's background.

5. In the Color and Visibility dialog box, click *OK*.

## Controlling Class and Subclass Visibility

You can control the visibility of classes and subclasses in two ways:

- Using the Color and Visibility dialog box, described here.
- Running the `color` command with one of the visibility arguments: `-globvis`, `-vis`, `-invis`, or `-toggle`. See [Syntax](#) for details.

To control class and subclass visibility using the user interface:

1. Run the `color` command.

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

2. From the *Group* list, choose a group.

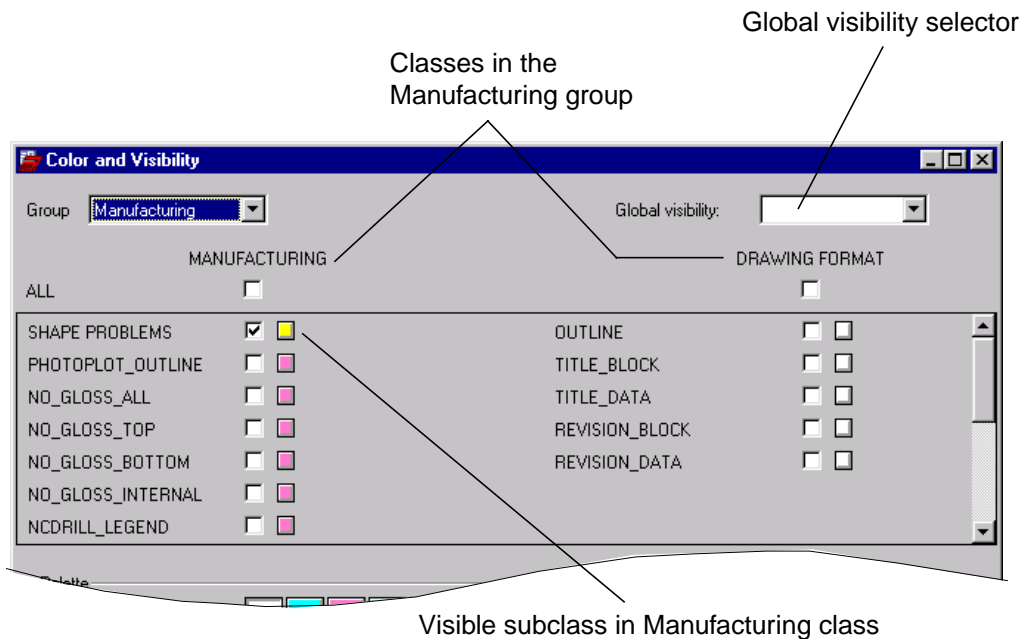
The classes and subclasses associated with that group appear in the dialog box.

3. Do any of the following, keeping in mind that each setting overrides the previously listed setting. Refer to [Figure 1-2](#) for the location of fields and check boxes.
  - ☐ To make *all* classes and subclasses visible or invisible, make a selection from the *Global visibility* field in the top right corner of the dialog box.
  - ☐ To control class visibility, click the check box under the appropriate class name. All the subclasses in that class become visible.
  - ☐ To control the visibility of individual subclasses, click the associated check box.

## Allegro Free Physical Viewer

### color

**Figure 1-2 Controlling Visibility**



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

**Note:** If objects that should be visible are not, check the Priority dialog box by running the `color priority` command. The check box for that color may be disabled.

5. When you are done making changes in this dialog box, click *OK* to save changes and close the dialog box.

### Modifying a Color Element

1. Run the `color` command.

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

2. In the *Palette* section of the dialog box, click the color box for the color you want to change.
3. Click *Modify*.
4. In the Color Editor, choose a new color. For details, see [Color Editor Dialog Box](#).
5. Click *OK*. The color box in the Color and Visibility dialog box changes to the new color.

6. Click *Apply* in the Color and Visibility dialog box to update the design with your color change.
7. If you want to save this color change for future use, continue with Saving the Modified Color Palette below.

### **Saving the Modified Color Palette**

After modifying the color palette (see the previous instructions, Modifying a Color Element), you can save the palette for use with other designs and for future use with the current design.

**Note:** A local palette is not specific to a design; it can be read into any other design. A design-specific color scheme is a database palette. A database palette is any set of colors saved with the design.

#### ***Saving a local palette***

1. Open the *Palette* field's menu after making your color changes.
2. Choose *Write Local*.

The current color selections are saved as your local palette. By default, local color selections are saved in a file, `lallegro.col`, in the `pcbenv` subdirectory of your home directory.

**Note:** Do not change the name or location of `lallegro.col`.

#### ***Saving a database palette***

1. Make sure you clicked *Apply* in the Color and Visibility dialog box after making your color changes.
2. Run the save or save as command and save the current design. The design is saved with the current (database) palette.

**Note:** The database palette remains current until you apply a different color palette to the design. For example, if you apply the global palette to your design and save the design, the database palette then matches the global palette.

### **Using Shadow Mode**

Refer to the shadow command.

### **Removing an Existing Placement Evaluator Display from View**

1. Choose *Display – Color/Visibility* (color command) to display the Color and Visibility dialog box.
2. Choose *Analysis* from the *Group* field.
3. In the *Global Visibility* field, choose *All Invisible* to turn off all subclasses.  
–or–  
Clear the *All* check box.
4. Click *Apply* to change the visibility settings in the design.

### **Displaying just the Ratsnest Histogram**

1. Choose *Display – Color/Visibility* (color command) to display the Color and Visibility dialog box.
2. Choose *Analysis* from the *Group* field.
3. Turn on only the *ANL\_RAT\_HISTOGRAM* subclass.
4. Click *Apply* to change the visibility settings in the design.

### **Displaying just the Router Color Map Relating to Vertical Layers**

1. Choose *Display – Color/Visibility* (color command) to display the Color and Visibility dialog box.
2. Choose *Analysis* from the *Group* field.
3. Turn on only those subclasses beginning with *ANL\_V\_*.
4. Click *Apply* to change the visibility settings in the design.

### **Displaying just the Router Color Map Relating to Horizontal Layers**

1. Run `color` to display the Color and Visibility dialog box.
2. Choose *Analysis* from the *Group* field.
3. Turn on only those subclasses beginning with *ANL\_H\_*.
4. Click *Apply* to change the visibility settings in the design.

## Displaying just the Router Color Map Relating to Overflow Areas

1. Run `color` to display the Color and Visibility dialog box.
2. Choose *Analysis* from the *Group* field.
3. Turn on only the following subclasses:
  - ☐ `ANL_H_100_COLORMAP`
  - ☐ `ANL_V_100_COLORMAP`
  - ☐ `ANL_VIA_100_COLORMAP`

## Examples

This example changes the color of ratsnest lines:

```
color -ratscolor "20"
```

The next example sets global visibility to *off*. Nothing appears in the design window.

```
color -globvis "off"
```

The next example sets the background color of the design to turquoise:

```
color -background 0x00ffff
```

This example sets the ASSEMBLY\_NOTES subclass of the BOARD GEOMETRY class visible:

```
color -vis "BOARD GEOMETRY/ASSEMBLY_NOTES"
```

The final example changes the visibility of the OUTLINE subclass of the BOARD GEOMETRY class. If visibility for this subclass is enabled, this command disables it—and vice versa.

```
color -toggle "OUTLINE"
```

## color priority

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

Assigns the order in which colors are displayed on top of each other in the active design for easier recognition. Also controls the visibility of a color. Use this command after assigning colors to subclasses with the `color` command.

For more details on assigning a display priority to colors, see *Getting Started* in the user guide.

### Menu Path

*Display – Color Priority*

### Priority Dialog Box

Use this dialog box to control the order in which colors appear in your design. For example, when you look at the design, the color at the top of the list appears on top of the color that appears second in the list. You can also turn off the display of a color.

*On* Click to display all colors that appear in your design.

*Off* Click to hide all colors that appear in your design.

### Defining Color Priority

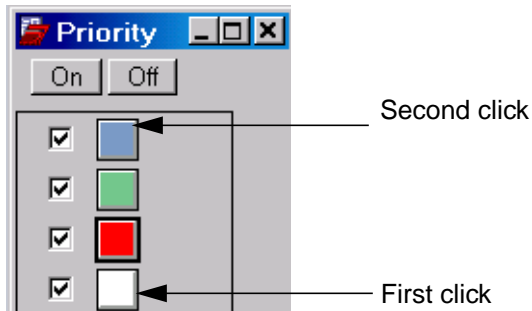
1. Run the `color priority` command.

The Priority dialog box appears.

The color with the highest priority appears at the top of the color panel. All elements of that color appear on top of other elements in the drawing.

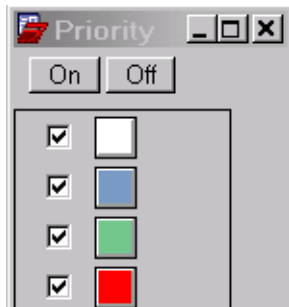
2. Click the color box of the color (not the check box) you want to move in the priority list. This is the first click identified in the [Assigning Color Priority](#) figure in the next step, in this case, white.
3. Click the color box (not the check box) of the color that occupies the position to which you want to move the selected color. This is the second click identified in Figure [1-3](#), in this case, blue.

**Figure 1-3 Assigning Color Priority**



The first color you selected moves to the new location. The previous color for that position moves down one level. [Figure 1-4](#) shows the new order in the example: white is listed first and blue has moved to the second position. All other colors shift down.

**Figure 1-4 Result of Assigning Color Priority**



4. If you want to prevent a subclass of objects from appearing, click the check box next to its color.

**Note:** This visibility check box takes precedence over visibility you set with the color command.

5. Click *OK*.

## dehighlight

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

Lets you deselect elements to be displayed in the permanent highlight color with the highlight pattern.

### Menu Path

*Display – Dehighlight*

### Options Tab for the dehighlight Command

A Find Filter tab displays with selections for symbols, functions, nets, pins, and DRC markers. Choose elements for dehighlighting by clicking or by name, using the name search fields in the find filter dialog box.

An Options tab displays that shows you the Class and Subclass of the element that you have chosen for dehighlighting, as well as the color that is assigned to it. It also provides these buttons so that you can dehighlight a whole group of elements at the same time:

- Nets
- Symbols
- Functions
- Pins

### Procedure

#### Dehighlighting Design Objects

1. Run the `dehighlight` command.

The Find Filter appears, listing the valid objects for the active command.

2. Click on a highlighted object  
or

Hold down the left mouse button to define an area that contains the design objects, then release the button.

The highlight color disappears from the chosen object(s).

## exit

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

### Menu Path

*File –Exit*

## **grid toggle**

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

This command is not available in Allegro PCB SI or AP SI.

## highlight

### Procedure

Lets you choose the following elements for highlighting.

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

The elements are displayed in the permanent highlight color with the highlight pattern. The highlight pattern is an alternating checkerboard of the element's color and the selected permanent highlight color. Elements highlighted with the *Display – Highlight* command stay highlighted until you choose *Display – Dehighlight* to turn off the highlighting.

### **Menu Path**

*Display – Highlight*

### **Procedure**

### **Highlighting Elements**

1. Run `highlight`.

The Find Filter or the Visibility dialog box display, depending on which one you choose before you run `highlight`.

## Allegro Free Physical Viewer

### hilight

---

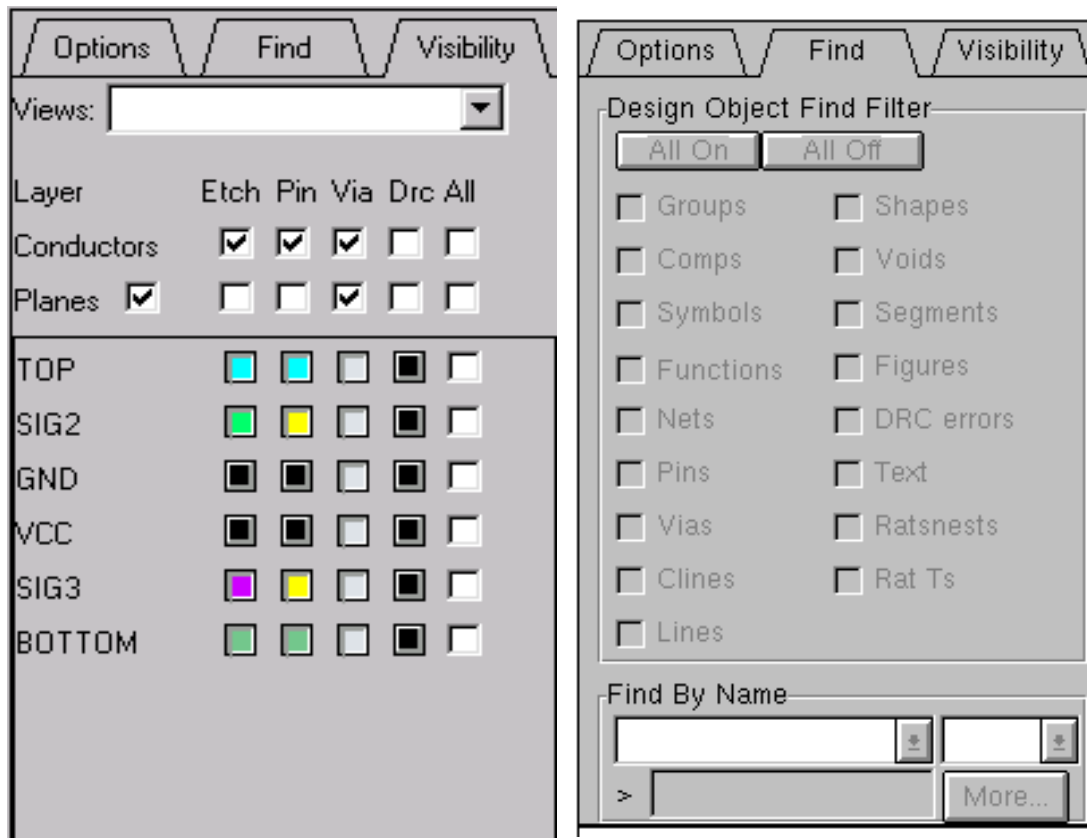
Options Find Visibility

Views:

Layer	Etch	Pin	Via	Drc	All
Conductors	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Planes	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

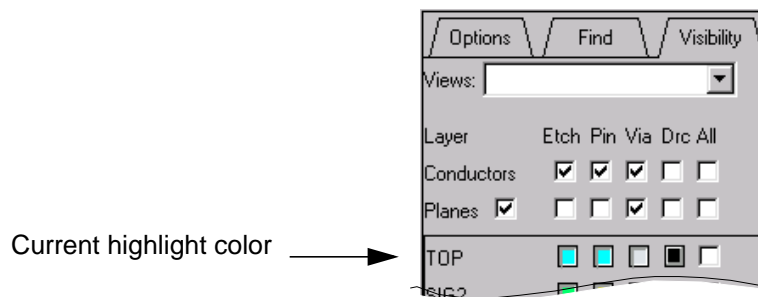
TOP	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
SIG2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
GND	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
VCC	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
SIG3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
BOTTOM	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

## Allegro Free Physical Viewer highlight



The Find Filter lists the objects that you can highlight for this command. The Options tab shows the colors that you can use to highlight a design object. The current highlight color appears in the PERM HIGHLIGHT field.

By default, the Options tab contains 24 colors. To change any of these colors, run the [color](#) command, described in the *Allegro PCB and Package Physical Layout Command Reference*. You want to modify a color element.



## Allegro Free Physical Viewer

### highlight

---

2. Make sure that the available selections in the Find Filter include the design object that you want to highlight.

To deselect any checked objects, click the check box.

3. Click the highlight color in the *Options* tab that you want to assign.

The color in the *PERM HIGHLIGHT* fields changes to the color you selected.

4. Click on the design objects in the design window.

—or—

Hold down the left mouse button to define an area that contains the design objects, then release the button.

The object(s) is highlighted with the color that you selected in the *Options* tab.

This procedure changes the color only of the design object that you selected. It does not change the color of all instances of that type of object.

## open

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

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

No warning occurs if you are switching your design from the tier where it was last saved to another tier. To display the warning, set the `db_tier_nomsg` environment variable using the *Setup – User Preferences – Drawing (envded)* menu command.

### Co-Design Environment

When you open a new .sip or .mcm design, and an existing drawing that has co-design dies with unsaved changes is open, you are asked whether you want to save the design. If you choose to save the changes, then the File - Save command is invoked to save the data. If you choose to discard the changes, then any temporary Open Access (OA) library/cell/views containing unsaved data are deleted before the new design is opened.

### Menu Path

*File – Open*

### Syntax

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

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

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

### Examples

```
open master.brd
```

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

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

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

## Dialog Box

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

## Procedure

### Opening an Existing File

1. Run the `open` command.

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

2. Choose a file from the list.

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

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

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

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

5. Click *Open* to open the file.

## plot

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

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



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

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

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

### Menu Path

*File – Plot*

### Print Dialog Box

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

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

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

## Allegro Free Physical Viewer

### plot

---

OK

Creates the plot and closes the dialog box.

## Procedures

### Plotting Prerequisites on a Unix Workstation

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

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

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

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

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

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

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

### Plotting Your Design on Unix

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

—or—

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

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

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

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

### Plotting Your Design on Windows

You can preview your plot on Windows before producing it.

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

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

## Allegro Free Physical Viewer

### plot

---

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

## plot setup

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

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

### Menu Path

*File – Plot Setup*

### Plot Setup Dialog Box

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

#### General Tab

##### *Plot scaling*

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

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

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

##### *Plot orientation*

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

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

##### *Plot method*

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

## Allegro Free Physical Viewer

### plot setup

---

	<i>Black and white</i> : Directs the output to print in black and white.
<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. The environment variable <code>PLOT_VECTORIZE_TEXT</code> determines whether the <i>Vectorize text</i> option is enabled or disabled by default.  <i>Width</i> : specifies the width for simulating the text characters. The width used is established with the environment variable <code>PLOT_VECTEXT_WIDTH</code> . 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.  Environment variables are the initial default settings for the Plot Setup dialog box. If you modify the settings in the dialog box, the new settings override any environment variable settings that you may have specified. When you exit, the current dialog box settings are saved in the <code>.ini</code> file. These <code>.ini</code> file settings are then used in the next session, and again override any specified environment variable settings.  If you modify the environment variable settings after changing settings in the <i>Plot Setup</i> dialog box, these new environment variable settings are not used. You must delete the <code>.ini</code> file, and then the new environment variable settings take effect.  You can set environment variables using procedures, based on the platform you are running. You can also set these environment variables using <i>Setup – User Preferences – Plot</i> .
<i>OK</i>	Saves the settings and closes the dialog box.

## Allegro Free Physical Viewer

### plot setup

---

*Cancel* 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

*Non-Vectorized Text* Choose to generate plot files with true font text , which lets you generate PDF-format plot output with searchable text.

*Font* Specifies a font to use; defaults to Courier.

*Font Height* Enter a percentage scaling factor for the character height to closely match font text with that of the normal vectorized text display/plot.

*Font Width* Enter a percentage scaling factor for the character width to closely match font text with that of the normal vectorized text display/plot.

*View Available Fonts* Click to review the available text fonts for the plot device.

### Margin Control

*Margin Width* Specify the desired margin width in user units. The default equates to 0.25 inches, or 0.0 if the `noplotmargins` environment variable is set.

*OK* Saves the settings and closes the dialog box.

*Cancel* Closes the dialog box without saving the settings.

## Procedure

### Setting Parameters for Plotting a Design

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

## Allegro Free Physical Viewer

### plot setup

---

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

### rats all

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

To control the way in which the ratsnest lines are displayed, use them with the following commands:

color Controls the color of ratsnest lines.

color priority Prioritizes color and sets visibility according to color.

To display ratsnest lines as straight or jogged lines, set the ratsnest geometry in the *Display* tab of the Drawing Options dialog box (status).

### Menu Path

*Display – Show Rats– All*

### Procedure

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

- Run `rats all`.

All ratsnest lines in the design are displayed.

## **rats component**

Displays existing ratsnest lines attached to component pins.

To control the way in which the ratsnest lines are displayed, use them with the following commands:

color Controls the color of ratsnest lines.

color priority Prioritizes color and sets visibility according to color.

To display ratsnest lines as straight or jogged lines, set the ratsnest geometry in the *Display* tab of the Drawing Options dialog box (sstatus).

### **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 way in which the ratsnest lines are displayed, use them with the following commands:

color Controls the color of ratsnest lines.

color priority Prioritizes color and sets visibility according to color.

To display ratsnest lines as straight or jogged lines, set the ratsnest geometry in the *Display* tab of the Drawing Options dialog box (status).

### 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.

## **redisplay**

Updates and redraws the current design window. Similar to redraw.

### **Menu Path**

*View – Refresh*

## script

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

The `script` command records a series of actions. It creates a text file containing the commands that you execute and adds a `.scr` extension to the file name. You can use scripts to perform global tasks such as setting up dialog box options, adding objects 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.

### Environment Variables

To keep the Scripting dialog box open, set the `script_keepformopen` environment variable using *Setup – User Preferences* and selecting *UI* from the *Categories* frame. When you set this variable, the dialog box does not close when you click the *Replay* button. To specify a script to run on startup, set the `script_startup` environment variable using *Setup – User Preferences* and selecting *UI* from the *Categories* frame.

### Scripting in Allegro Package Designer XL and SiP Digital Architect/SiP Layout

For information on scripting in Allegro Package Designer XL and SiP Digital Architect/SiP Layout, see *Generating a Co-Design Die* in the *Placing the Elements* User Guide.

### Menu Path

*File – Script*

### Scripting Dialog Box

*Script File*

## Allegro Free Physical Viewer

### script

---

<i>Name</i>	Specifies the name of the file in which you record your actions. Allegro PCB Editor adds the <code>.scr</code> extension to the file name.
<i>Browse</i>	Displays the script file data browser that lets you choose a script file to replay.
<i>Library</i>	Displays the script file data browser that lets you choose a script file to replay. Opens to your script path location.
<i>Generate</i>	Displays a file browser from which you can choose a <code>.jrl</code> file to convert into a script without having to leave the current environment. To process the journal file and reconstruct the appropriate script outside of Allegro PCB Editor, run:  <pre>j2script &lt;source_jrl_file&gt; &lt;target_allegro_script&gt;</pre>
<i>Record/Replay</i>	
<i>Macro record mode</i>	Specifies whether or not you record as a macro. When replaying, a macro requires a starting point.
<i>Record</i>	Starts recording your actions.
<i>Stop</i>	Stops recording your actions or replaying a script.
<i>Replay</i>	Starts replaying a macro or script.
<i>Cancel</i>	Closes the dialog box.
<i>Help</i>	Displays the Help window.

## Procedures

### Creating a Script

1. Run the `script` command.

The Scripting dialog box appears.

2. In the *Name* text box, enter a name for the script.

3. Click *Record*.

The Scripting dialog box disappears.

4. Perform the tasks that you want the script to run.

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

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

### **Creating a Macro**

1. Run the `script` command.

The Scripting dialog box appears.

2. In the *Name text* box, enter a name for the macro.
3. Click *Macro Record Mode*.
4. Click *Record*.

The Scripting dialog box disappears.

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

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

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

### **Replaying a Script**

1. Run the `script` command.

The Scripting dialog box appears.

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

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

3. Click *Replay*.

The script replays.

### **Replaying a Macro**

1. Run the `script` command.

The Scripting dialog box appears.

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

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

3. Click *Replay*.

The script replays.

### **Converting a .jrl File to a Script**

1. Run the `script` command.

The Scripting dialog box appears.

2. Click *Generate*.

A file browser appears.

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

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

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

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

## show element

Dialog Box | Procedure

The `show element` command lets you list the attributes of a graphic object. It displays all values relevant to the object, 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*

## 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. Use the User Preferences Editor dialog box to set the <code>print_unix_command</code> environment variable governing UNIX printing or the <code>print_nt_extension</code> environment variable governing Windows printing.
<i>File – Stick</i>	Makes the window remain on screen until you close the window, or the program terminates. Use this option to compare information between two windows. For example, you may use <code>show element</code> to obtain information about two design objects and use <i>File – Stick</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.

## Allegro Free Physical Viewer

### show element

---

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

To be able to access a web link as the value of a property, be sure to set the `allegro_html` environment variable by choosing *Setup – User Preferences – Ui*. For additional information on storing web links as the value of a property, see *Creating Design Rules* in the user guide of your product documentation.

### Find By Name/Property

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

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

Double clicking an object in either the *Available Object* list or the *Selected Object* list results in the object 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 objects are found but the Find by Name/Property dialog box closes.

## Procedures

### Displaying Design Attributes for an Object

This procedure lets you display object attributes. You can also find instances of inherited properties on parent and child objects 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 objects you want to display.
3. Position the cursor over an object and click to select.

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

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

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

### Finding an Object by its Property

1. Run the `show element` command.
2. Click *More* in the Find Filter.
3. Choose the property from the *Available Properties* list box.

The property appears in the *Name* field.

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

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

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

No instances of `<property_name>` found.

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

The Find by Property Show dialog box appears.

### **Finding an Object 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 object from the list.
3. Enter the name of the object in the *Name* field to the right of the drop-down list box.
4. Click *Enter*.

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

## show measure

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

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

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

### Menu Path

*Display – Measure*

### 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.  <b>Note:</b> Manhattan Dist = Dx + Dy
<i>Width</i>	Displays the width of line segments along a connect line.

## Allegro Free Physical Viewer

### show measure

---

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.

## Allegro Free Physical Viewer

### show measure

---

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

- a. To measure any other path, indicate it by picking intermediate points along it and read the Total Dist field of the Measure dialog box.
5. When you are finished, click right to display the pop-up menu, and choose *Done*.

## show property

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

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

For more details about properties, see the *Creating Design Rules* user guide in your product documentation.

### 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.

## Allegro Free Physical Viewer

### show 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.
<i>Reset</i>	Clears the fields and resets to the defaults.

### Graphics Tab

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

### Procedures

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

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

The Show Property dialog box appears.

2. Click the Information tab.
3. Choose a property from the *Available Properties* list.  
–or–  
Enter a property name in the *Name* field.

## Allegro Free Physical Viewer

### show property

---

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 and Visibility dialog box.

## Allegro Free Physical Viewer

### show property

---

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.
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 and Visibility 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](#) and *Waiving Design Rule Check Errors* in your product documentation.

### 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

### Dialog Box

The `status` command lets you establish the operating characteristics for the active design in either layout or symbol mode.

### Menu Path

*Setup – Drawing Options*

## Drawing Options Dialog Box

### Status Tab

In the layout mode, you can use the Status tab to verify the current state of dynamic shapes and DRCs and update them if they are out of date. An out of date dynamic shape is one for which the *Dynamic Copper Fill* mode has been set to *Rough* or *Disabled* on the *Global Dynamic Shape Parameters* dialog box (non-*Smooth Dynamic Copper Fill* mode).

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

Changing fill mode from	and using this button	produces this result
-------------------------	-----------------------	----------------------

<i>Disabled</i> to <i>Rough</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Rough</i>
<i>Disabled</i> to <i>Smooth</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>
<i>Rough</i> to <i>Smooth</i>	OK	no update of dynamic shapes changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>

## Allegro Free Physical Viewer

status

---

### Changing fill mode from any selection/no selection and using this button produces this result

any selection/no selection	<i>Update to Smooth</i>	updates dynamic shapes to <i>Smooth</i> changes fill mode in <i>Global Dynamic Shape Parameters</i> dialog box to <i>Smooth</i>
----------------------------	-------------------------	--

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.

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

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

<i>Isolated shapes</i>	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.
------------------------	---

<i>Unassigned shapes</i>	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.
--------------------------	--

## Allegro Free Physical Viewer

### status

---

#### *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 Copper Fill* mode for all dynamic shapes has been set to *Rough* or *Disabled* on the Global Dynamic Shape Parameters dialog box, making all dynamic shapes out of date (non-*Smooth Dynamic Copper Fill* mode) as a result. Out of date dynamic shapes prevent artwork output.

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 Copper Fill* mode for all dynamic shapes has been set to *Smooth*, making all dynamic shapes up-to-date (*Dynamic Copper Fill* mode 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.

#### *DRC errors*

Indicates whether DRC markers are up-to-date. The status can be *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.

## Allegro Free Physical Viewer

### status

---

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.

#### *Refresh button*

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

#### *DRC Controls*

##### *On-Line DRC*

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

##### *Default Symbol Height*

Indicates the default height of a symbol on the design if you did not specify the height when you created the symbol. Use this value to check package-to-package/part-to-part and package-to-keepout/part-to-keepout DRC. Defaults to 150 Mils.

##### *Update DRC button*

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

#### *Shapes (Dynamic Copper Pour)*

##### *Fill Mode*

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

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

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

*Disabled:* Select to globally defer dynamically filling all dynamic shapes you subsequently create or modify to speed

performance. Use this option to edit etch for medium to large ECOs, manual ECOs or to run batch programs such as netin, glossing, testprep add/replace vias, for example. Shapes created under this global setting are not voided, and DRC does not run. They are marked out of date to be filled later. Artwork cannot be produced.

### *Update to Smooth*

Click this button to automatically void and run DRC on all dynamically filled shapes, making all dynamic shapes up-to-date (*Dynamic Copper Fill* mode set to *Smooth*) and produce artwork quality output (regardless of whether you chose *Rough* or *Disabled* in the *Fill Mode* field above). Changes the current *Dynamic Copper Fill* mode on the *Global Dynamic Shape Parameters* dialog box to *Smooth*.

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

Shapes already dynamically filled remain completed.

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

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

## Display Tab

### *Connect Point Size*

Specifies the size of a connect point in user units. The default is 10. Cpoint sizes should be set high enough to be easily seen in a drawing. The appropriate setting (for visibility) depends on the line width in your design.

Etch/conductor lines that extend outside groups may sometimes have to be connected to new or existing etch/conductor. Because groups may have properties, for example `FIXED`, that cannot be modified when connections are made to objects outside the group, connection points (cpoints) are created when a connection line (cline) outside a group connects to a cline in a group.

The cpoint is created so that the original cline remains unaltered in the group. (Normally, the new cline would merge with the existing one, thus changing aspects of the group.) Cpoint functionality can minimize lost information that is often the result of merged, overlapped, and/or deleted clines. You cannot add cpoints through an interactive command or manipulate them, nor can you select them using *Display – Element* (show element command).

Cpoints display as diamonds on the cline's subclass in plotting but not in the output of artwork.

**Figure 1-5 Cpoint on Two Clines**



<i>DRC Marker Size</i>	Determines the size, in user-defined units, of the DRC markers that appear in a design. The default is 25.
------------------------	--

<i>Rat T (Virtual pin) size</i>	Allows you to control the graphical size of a Rat T.
---------------------------------	--

<i>Max Rband Count</i>	Specifies an upper limit on the number of lines drawn during editing sessions. You should not have to change this value with modern graphic cards. The default is 500.
------------------------	--

<i>Ratsnest Geometry</i>	Displays a pop-up menu that lets you choose the shape of the ratsnest lines. The default is <i>Jogged</i> .
--------------------------	---

<i>Ratsnest Points</i>	Displays a pop-up menu that lets you choose the closest distance on a line ( <i>Closest endpoint</i> ) or between two pins ( <i>Pin to pin</i> ). The default is <i>Closest endpoint</i> .
------------------------	--

#### *Enhanced Display Modes*

<i>Display Plated Holes</i>	Displays plated drill holes in your design. Unchecked by default, because plating hole visibility often is not required due to the volume of entities and supporting geometries such as etch layer pads.
-----------------------------	--

*Display Non-Plated Holes*

## Allegro Free Physical Viewer

status

---

Displays non-plated drill holes in your design. You may want to make non-plated holes visible during placement and routing because they have no padstacks associated with them on etch layers and are normally invisible. As a result, they may lack proper keepout areas to guide you. Unchecked by default.

### *Filled Pads*

Controls onscreen design display and fills pins. Unchecked by default. Leaving pads unfilled can improve display performance.

### *Cline Endcaps*

Controls onscreen design display and rounds line vertices to more closely approximate artwork.

### *Thermal Pads*

Displays thermal pads when you have a negative plane. Unchecked by default.

### *Bus Rats*

Displays the middle portion of ratsnest lines with the same BUS\_NAME property so that they appear to be merged into a thick line. Unchecked by default.

### *Grids*

Displays the grid. Checked by default.

### *Waived DRCs*

Displays waived DRCs. Disabled by default.

## **Text Tab**

### *Justification*

Indicates the anchor point within text that you add to the layout. The anchor location determines how the text appears in the text block. The default is *Left*.

### *Parameter Block*

Defines the size and spacing of the text you add to the design. Using the define text command, you can define up to 16 text blocks. The default is *1*.

### *Text Marker Size*

Determines the size, in user units, of the displayed text markers. These markers indicate locations for text that will be entered later. The default is *50*.

### *Refdes*

Specifies the text size for reference designators in the Allegro PCB SI layout

### *Room Name*

Specifies the text size of room names in the Allegro PCB SI layout.

## Allegro Free Physical Viewer

status

---

<i>Notes</i>	Determines the text size of notes in the Allegro PCB SI layout.
--------------	---

### Line Lock Tab

Use this tab to specify the default values when you add lines.

<i>Lock Direction</i>	Displays a pop-up menu that lets you specify the direction of lines that you add to a drawing. The options are Off, 45, and 90. The default is Off.
<i>Lock Mode</i>	Specifies the type of segments to use when adding lines or connect lines. The options are <i>Line</i> and <i>Arc</i> . The default is <i>Line</i> .
<i>Minimum Radius</i>	Determines the minimum radius allowed for an arc. The default is 0.
<i>Fixed 45 Length</i>	Specifies the length, in user units, of 45-degree segments. By default, this is unchecked. If checked, the default is 25.
<i>Fixed Radius</i>	Specifies the radius, in user-defined units, of arcs. By default, this is unchecked. If checked, the default is 25.
<i>Tangent</i>	Specifies whether tangent lines are locked. If checked, arcs construct tangent to lines. By default, this is checked.

### Symbol Tab

Use this tab to specify the default values used when you place symbols.

<i>Angle</i>	Specifies the default angle that is used to place symbols. You can enter an angle with up to three decimal places, or you can display a pop-up menu and choose one of the available angles. The default is 0.
<i>Mirror</i>	Lets you mirror symbols that you add to a drawing. By default, this is not checked.
<i>Drawing size</i>	Displays the Drawing Parameters dialog box.

## **unrats all**

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

### **Menu Path**

*Display – Blank Rats – All*

## **Procedure**

### **Hiding Ratsnest Lines**

1. Run `unrats all`.

All ratsnest lines in the design disappear.

2. Run *View – Refresh* to clean up the appearance of your design.

## unrats component

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

### Menu Path

*Display – Blank Rats – Component*

### Procedure

#### Hiding Ratsnest Lines to Pins on Components

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

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

## unrats net

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

### Menu Path

*Display – Blank Rats – Net*

## Procedure

### Hiding Ratsnest Lines to Pins on Nets

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

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

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

## viewlog

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

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

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

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

### Select File to View Dialog Box

The log file viewer contains the following menu bar options:

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

## Procedures

### Viewing Log Files Without Specifying File Name

1. Type `viewlog`.

A file browser appears

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

The log file viewer window displays the selected file.

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

### Viewing Log Files Specifying File Name

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

The log file viewer window displays the specified file.

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

## zoom center

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

### Menu Path

*View – Zoom Center*

## Procedure

### Centering a Design Window About a Specific Point

Use one of these methods:

- Run `zoom center`, then click the location you want to be the center of the new display.

Use dynamic zooming by way of the middle mouse button. For details on dynamic zooming, see the *Getting Started with Physical Design* user guide in your documentation set.

## 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*

## Procedure

### Fitting Your Layout in the Design Window

Use one of these methods:

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

–or–

Press `F9`.

–or–

Use dynamic zooming by way of the middle mouse button. For details on dynamic zooming, see the *Getting Started with Physical Design* user guide in your documentation set.

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*

## Procedure

### Magnifying Your View

Type `zoom in` at the console window prompt.

–or–

Press F10.

–or–

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

–or–

Use dynamic zooming by way of the middle mouse button. For details on dynamic zooming, see the *Getting Started with Physical Design* user guide in your documentation set.

## 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*

## Procedure

### Reducing the Magnification of Your Layout

Type `zoom out` at the console window prompt.

–or–

Press F11.

–or–

Use dynamic zooming by way of the middle mouse button. For details on dynamic zooming, see the *Getting Started with Physical Design* user guide in your documentation set.

## 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*

### Procedure

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

Use one of these methods:

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

–or–

Press `F8`.

–or–

Draw the Zoom stroke (z) with the mouse.

–or–

Use dynamic zooming by way of the middle mouse button. For details on dynamic zooming, see the *Getting Started with Physical Design* user guide in your documentation set.

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*

## zoom world

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

### Menu Path

*View – Zoom World*

### Procedure

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

1. Run the `zoom world` command.

or

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

or

Use dynamic zooming by way of the middle mouse button. (For details on dynamic zooming, see the *Getting Started with Physical Design* user guide in your documentation set..

## **Allegro Free Physical Viewer**

### zoom world

---