

V Commands

Product Version 23.1
September 2023

© 2024 Cadence Design Systems, Inc.
Printed in the United States of America.

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

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.

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 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 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 DFAR252.227-7013 et seq. or its successor.

Contents

1	5
V Commands	5
3d	6
3d_export	7
3d_model_export	9
3d_xprobe	10
3dx	11
variant assembly	12
Variant Assembly Dialog Box	13
Creating Variant Assembly Drawing	15
variant bom	16
Variant BOM Dialog Box	17
Generating a Variant Bill of Materials Report	18
version	19
vertex	20
Options Panel for the Vertex Command	21
Creating or Editing a Vertex on Line and Arc Segments	24
Deleting Vertex on Line and Arc Segments	26
vi	27
via align	28
Options Panel for the Via Align Command	29
Aligning Vias in Horizontally or Vertically	30
via array	31
Options Panel for the via array Command	32
Adding a Via Array to Cline	36
Adding a Via Array to Shape	37
Updating a Via Array	38
Deleting a Via Array	39
via assign net	40
Assigning Net to a Via	41
Options Panel for the via assign net Command	42
view 3d	43
viewlog	44

Select File to View Dialog Box	45
Viewing Log Files	46
vision manager	47
Visions Dialog Box	49
Viewing Vision Analysis Results	51
void adjacent layer shapes	53
Options Tab for the void adjacent layer shapes Command	54
Creating Shape Voids	55

V Commands

3d	3d_export	3d_model_export
3d_xprobe	3dx	variant assembly
variant bom	version	vertex
vi	via align	via array
via assign net	view 3d	viewlog
vision manager	void adjacent layer shapes	

3d

The `3d` command launches Allegro 3D Canvas, which lets you visualize and analyze a three-dimensional model of a design as a manufactured output. You can visually check whether the symbol placement, position, and proximity to other symbols is proper and decide if a violation of design constraints occur. You can also view mechanical objects such as shields, fans, heat sinks and housings and run checks for verifying any collisions or other placement issues.

Access Using

- Menu path: *View – 3D Canvas*
- Toolbar icon:



Related Topics

- [Allegro X 3D Canvas](#)

3d_export

The `3d_export` command exports the 3D design to a disk file containing data based on the specified options. The first two arguments of this batch command specify the output file name and the unit, respectively. The supported output file formats are STEP (*.stp, *.step), IGES (*.iges), ACIS (*.asat *.asab *.sat *.sab), and PDF (*.pdf).

```
3d_export outputFile millimeter|micron|inch [OPTIONS]
```

-e	electrical_component	Export electrical component
-m	mechanical_component	Export mechanical component
-X	3d_model	Export 3d models
-p	place_bound	Export place bound
-d	dfa_bound	Export DFA bound
-E	external_conductor	Export external conductor
-i	internal_conductor	Export internal conductor
-D	dielectric	Export dielectric
-s	soldermask	Export soldermask
-S	silkscreen	Export silkscreen
-M	mechanical_hole	Export mechanical hole
-P	through_pin_hole	Export through pin hole
-V	via_hole	Export through via hole
-f	fully_bent	Export design in a fully bent state
-l	separate_file_per_layer	Separate file per layer
-C	compress	Compress output file
-A	all	Export everything

Examples

- `3d_export abc.stp millimeter electrical_component external_conductor`
- `3d_export abc.stp millimeter -eE`
- `3d_export abc.stp millimeter -e -E`

3d_model_export

The `3d_model_export` command exports the 3D mapped models or a map file based on the specified options. The `-f` argument specifies the output file format for the models or map files. If no options are specified, the command runs with the default options.

Syntax

`3d_model_export map|model [OPTIONS]`

<code>-m</code> <code>method</code>	Export method for model files = [<code>file_first</code> <code><attachment_first></code> <code>attachment_only</code>]
<code>-d</code> <code>dir</code>	Export output dir for model files = <code><./step></code>
<code>-p</code> <code>path</code>	Export output path for map file = <code><./export.map></code>
<code>-V</code> <code>via_hole</code>	Export through via hole
<code>-f</code> <code>format</code>	Export format = [<code><stp></code> <code>acis_bin</code> <code>acis_txt</code>]
<code>-i</code> <code>item</code>	Export items for map file = [<code>def</code> <code>inst</code> <code><def_inst></code>]

Examples

- The following example shows the syntax of the command using the default options:

```
3d_model_export map format=stp path="./export.map" item=def_inst" or  
"3d_model_export model format=stp dir="./step" method=attachment_first
```

- `3d_model_export map -f stp -i inst`
- `3d_model_export model -m file_first format=acis_txt`

3d_xprobe

The `3d_xprobe` command enables cross-probing between Allegro 3Dx Canvas and 2D layout design. To cross-probe objects between 3D and 2D designs, you must invoke Allegro 3DX Canvas. This command works in a post-select model. Select an object in the 2D design canvas, right-click and choose *Cross probe in 3D* from the pop-up menu. The selected object is highlighted and zoomed into the Allegro 3DX canvas. Similarly, select an object in Allegro 3DX Canvas and choose *Cross Probe* from the right-click menu. The selected object is highlighted and zoomed into the 2D design canvas.

3dx

The `3dx` command launches Allegro 3Dx Canvas, which lets you visualize and analyze a three-dimensional model of a design as a manufactured output. You can visually check whether the symbol placement, position, and proximity to other symbols is proper and decide if a violation of design constraints occur. You can also view mechanical objects such as shields, fans, heat sinks and housings and run checks for verifying any collisions or other placement issues.

Access Using

- Menu path: *View – 3DX Canvas*
- Toolbar icon:



Related Topics

- [Allegro X 3DX Canvas](#)

variant assembly


The `variant assembly` command opens the *Variant Assembly* dialog box from which you set the options to generate an assembly drawing layer for components belonging to a specific variant of the current design.

Variant Assembly Dialog Box

Use this dialog box to generate assembly drawing using components that are used in a design variant.

Access Using

- Menu path: *Manufacture – Variants – Create Assembly Drawing*

<i>Variant</i>	Specify one variant name from the list available for the current design. Variants for a design are defined in the <code>variants.lst</code> file that can be created using the Allegro Design Entry HDL variant editor.
<i>Top Side</i>	Choose this to create an assembly drawing for components found on the top side of the board for the variant. The drawing is created under the <code>MANUFACTURING</code> class, with a subclass of <code><VARIANT NAME>_TOP</code> , where <code><VARIANT NAME></code> is the variant name converted to uppercase characters.
<i>Bottom Side</i>	Choose this to create an assembly drawing for components found on the bottom side of the board for the variant. The subclass will be <code><VARIANT NAME>_BOTTOM</code> .
<i>Use Assembly Data</i>	Choose this to create component outlines for the assembly drawing from the appropriate <code>ASSEMBLY_TOP/BOTTOM</code> subclass of the <code>PACKAGE GEOMETRY</code> class.
<i>Use Place Bound Data</i>	Choose this to create component outlines for the assembly drawing from the appropriate <code>PLACE_BOUND_TOP/BOTTOM</code> subclass of the <code>PACKAGE GEOMETRY</code> class. If the <code>PLACE_BOUND</code> outline should be a filled shape, it will appear on the variant assembly drawing as an unfilled shape.
<i>OK</i>	<p>Creates the variant assembly drawing. If the subclass does not exist, it is automatically created. If the subclass does exist, a menu appears asking if you want to overwrite the current subclass or not. The appropriate subclass for the Package Geometry Option need not be visible at the time of creation.</p> <div style="border: 1px solid #fde725; padding: 10px; margin-top: 10px;"> <p> Shapes, lines, and text on any other visible subclasses are included on the variant assembly drawing subclass. The <code>BOARD GEOMETRY/OUTLINE</code> and appropriate <code>REF DES/ASSEMBLY</code> subclasses are typical other subclasses that you may want to have visible.</p> </div>

Cancel

Exit from this dialog box without creating a variant assembly drawing.

Creating Variant Assembly Drawing

To create an assembly drawing using variants for components, as defined in the schematic, perform the following steps:

1. Choose *Manufacture – Variants – Create Assembly Drawing*.
Alternatively, type `variant assembly` in the Command window.
The *Variant Assembly* dialog box is displayed.
2. Fill out the controls in the dialog box as required.
3. Click *OK* to run the program.

variant bom

The `variant bom` command displays the *Variant BOM* dialog box from which you set the options for generation of a bill of materials report for components belonging to a specific variant of the current design. The format of the report is similar to that of the standard Bill of Materials report for a design.

Variant BOM Dialog Box

Use this dialog box to specify details for generating a bill of materials report for a variant of the current design.

Access Using

- Menu path: *Manufacture – Variants – Create Bill of Materials*

<i>Variant</i>	Specify one variant name from the list available for the current design. Variants for a design are defined in the <code>variants.lst</code> file created using the System Connectivity Manager variant editor.
<i>Include Components Not Installed</i>	Click this check box to append a special section to the end of the report that lists the components of the design that do not appear in the specified variant of the design.
<i>OK</i>	Creates the variant bill of materials report. The report file is named <code>var-<variant name>.rpt</code> , where <code><variant name></code> is the variant name converted to lowercase characters.
<i>Cancel</i>	Exit the form without creating a variant bill of materials report.

Generating a Variant Bill of Materials Report

To generate bill of materials report for the design variant of an active design:

1. Choose *Manufacture – Variants – Create Bill of Materials*.
Alternatively, type `variant bom` in the Command window.
The *Variant BOM* dialog box is displayed.
2. Fill out the controls in the dialog box as described above.
3. Click *OK* to run the program.

version

The `version` command identifies the release number of the layout editor operating in your environment when you type in the command in the Command window of layout editor. You can display additional information about the version by running the `about` command.

Syntax

`version`

vertex


The `vertex` command inserts vertex (corner) into existing lines. Line elements include connect lines and shape and void boundaries. You can move and alter vertices on shapes, rectangles, filled rectangles, and line and arc segments using this command.

While you can delete vertices using this command, you can also remove them with the [delete vertex](#) command.

Options Panel for the Vertex Command

Access Using

- Menu path: *Edit – Vertex*

 When you edit a vertex on a non-etch/conductor layer, the following settings are not available.

<i>Net name</i>	Indicates the net name of the selected cline.
<i>Bubble</i>	<p>Controls any automatic bubbling (moving of existing connections) to resolve DRC errors. Enabling either of the hug modes or shove-preferred bubble mode sets the <i>Line lock</i> field to <i>Line</i> to prevent you from adding arcs while in shove- or hug-preferred mode. Bubble mode does not support arcs. These are the choices:</p> <ul style="list-style-type: none">• <i>Off</i>: The tool flags all clearance violations with error markers.• <i>Hug Only</i>: Where possible, the routed cline contours other etch/conductor objects to avoid spacing DRC errors. Other etch/conductor remains unchanged.• <i>Hug preferred</i>: Where possible, the routed cline segments contours other etch/conductor objects to avoid spacing DRC errors. If not possible, the tool tries shoving other etch/conductor objects to open routing paths. <p>Note: This method is more aggressive than <i>Hug Only</i>.</p> <ul style="list-style-type: none">• <i>Shove preferred</i>: Where possible, the routed cline pushes and shoves other etch/conductor objects to avoid spacing DRC errors. If not possible, the tool tries hugging other etch/conductor objects. This is true for vias only if you enable the <i>Shove via</i> capability.

<i>Shove vias</i>	<p>Allows the bubble functionality in shove mode to move vias when you are editing etch/conductor. It is only active when the bubble functionality is enabled. These are the choices:</p> <ul style="list-style-type: none">• <i>Full</i>: Vias are shoved in a <i>Shove preferred</i> manner. Vias are not moved unless there is no way to draw a connect line around them.• <i>Minimal</i>: Vias are shoved in a hug preferred manner. Any new or edited etch/conductor always shoves vias out of the way.• <i>Off</i>: Vias are not shoved.
<i>Clip dangling clines</i>	<p>This option clips dangling clines that are too close (violate spacing constraints) to any line segments you are editing. It is active only when bubble functionality is enabled in shove mode.</p>
<i>Smooth</i>	<p>Lets you control post-bubble smoothing. If you have not used bubbling in your etch editing, no smoothing occurs. The option is disabled while <i>Bubble</i> is off. The extent of smoothing that is done is determined by your choice of the these values:</p> <ul style="list-style-type: none">• <i>Off</i>: No smoothing occurs.• <i>Minimal</i>: Executes dynamic smoothing to minimize unnecessary segments.• <i>Full</i>: Executes more extensive smoothing to remove any unnecessary jogs. Full smoothing could, in some cases, hamper your ability to successfully edit a vertex. Use of smoothing in bubble mode could result in some elements getting bubbled unnecessarily.
<i>Allow DRCs</i>	<p>Specifies that the tool can violate design rules to make a connection. When this option is turned on, vertex locations that would generate a DRC changes your cursor display to a DRC <i>bow-tie</i> shape. The violating segment then appears in the temporary highlight color. You must resolve the violations for a successful design. When <i>Allow DRCs</i> is turned off, the <code>vertex</code> command rejects any bubbling that generates a DRC error by reverting to the last good result. If bubble is turned off, the editor sets the vertex at a point (between the last good point and the current point) that does not cause a DRC.</p>
<i>Allow gridless</i>	<p>Specifies whether the connect line or via you slide has to adhere to the routing grid. When you enable gridless routing, the tool can slide connections at maximum density while accommodating varying design rules and line widths. In addition, bubbled vertices are snapped to grid. This affects your connections only if <i>Allow DRCs</i> is inactive.</p>

Snap to
45(hold
Ctrl to
toggle)

When set slow cursor movement will provide resistance at 45 or 90 degree segment angles. Click to add a 45 degree segment as the cursor pauses on the 45 or 90 angles. If cursor movement continues, the segment snaps back to the cursor location.

- ✓ The *Design Parameter Editor* is also available for editing the parameters listed on the *Options* tab. Choose *Setup – Design Parameters* (prmed command), click the *Etch Edit* tab, and select the *Edit Vertex* folder.

Related Topics

- [Deleting Vertex on Line and Arc Segments](#)

Creating or Editing a Vertex on Line and Arc Segments

To add vertices (corners) to connect lines, shape and void boundaries, do the following:

1. Choose *Edit – Vertex*

Alternatively, type `vertex` in the Command window.

2. Configure the Options panel as necessary, based on the descriptions.

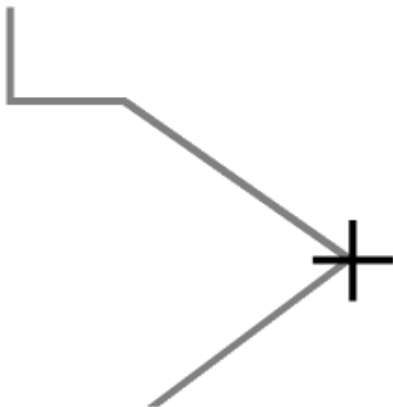
3. Click the line on which to add a vertex, or click the vertex to be changed.

If you click a point in the middle of a line segment where no vertex exists, one is inserted.
Vertex insertion operates on all lines and connect lines.



The line becomes dynamic. You can slide the cursor up and down the connect line to create the vertex at any point or angle.

4. Stretch the connect line or vertex to the new location.



5. Click to secure the connect line in the new location.

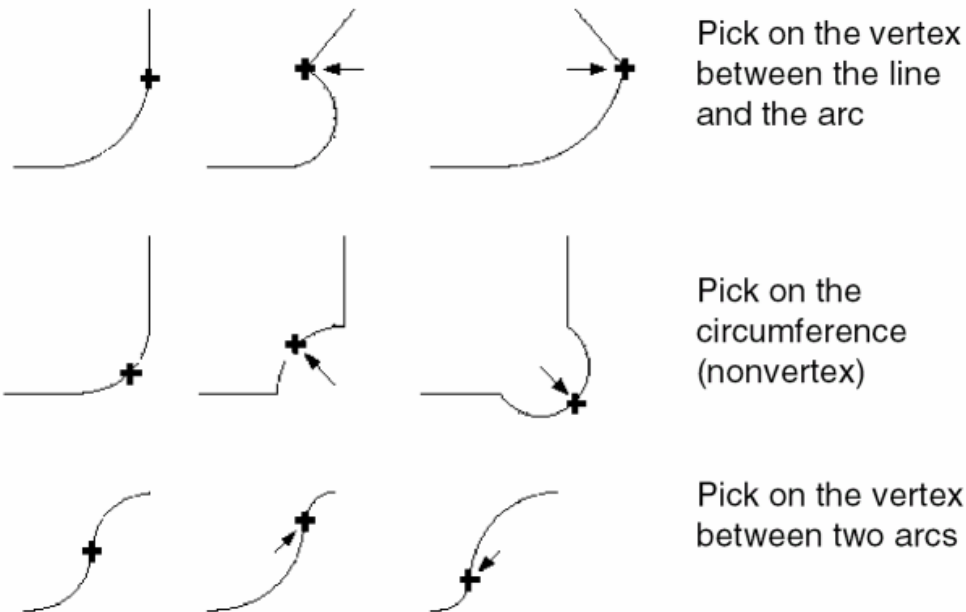
- ✓ Consider using the right mouse button pop-up menu option *Snap pick to*, which snaps the connect line to database elements such as segment vertex or grid point or intersection and so on.

6. Choose *Done* from the pop-up menu.

If you click the endpoint of an arc, the editor fixed the center, radius, and the other endpoint of the curve. You must enter the second endpoint.

If you click the boundary of an arc, the editor fixes the two endpoints and waits for a third point to be selected. It provides three ways to edit the vertex of an arc:

- Click the vertex between the line and arc.
- Click the circumference (not a vertex).
- Click the vertex between two arcs.



Deleting Vertex on Line and Arc Segments

To delete vertices (corners) to connect lines, shape and void boundaries, do the following:

1. Choose *Edit – Vertex*

Alternatively, type `vertex` in the Command window.

2. Click on the vertex you want to delete, and choose *Delete Vertex* from the pop-up menu. –or–
Double-click on the vertex.

The vertex is deleted.

Related Topics

- [vertex](#)
- [delete vertex](#)

vi

The `vi` command lets you open a file from the Command window of your layout editor user interface.

If you enter the command without specifying a file name, a dialog box opens to specify the file name. If you type `vi` along with the file name in the Command window, the file opens in a text editor displays.

via align

The `via align` command aligns selected vias in both vertical and horizontal directions according to specified placement options. You can choose the type of alignment either from the Options panel or from pop-up menus.

Options Panel for the Via Align Command

Access Using

Menu path: *Route – Resize/Respace – Align Vias*

<i>Vertical placement</i>	Specifies the placement option for vertical alignment. You can choose from any one of the following: <i>None</i> : There is no change in the Y coordinates of the selected vias. This is selected by default. <i>Top</i> : The Y coordinate of the topmost selected via is assigned to all the selected vias. <i>Center</i> : The Y coordinate of the midpoint between the topmost and the bottommost selected vias is assigned to all the selected vias. <i>Bottom</i> : The Y coordinate of the bottommost selected via is assigned to all the selected vias. <i>Evenly Spaced</i> : The vias are spaced evenly between the topmost and the bottommost vias. <i>User Pick</i> : The vias are spaced according to a user-defined location in the design canvas. The pick point is determined by the <i>Snap pick to pop-up</i> menu setting.
<i>Horizontal placement</i>	Specifies the placement option for horizontal alignment. You can choose from any one of the following: <i>None</i> : There is no change in the X coordinates of the selected vias. This is selected by default. <i>Left</i> : The X coordinate of the left-most selected via is assigned to all the selected vias. <i>Center</i> : The X coordinate of the position between the left-most and the right-most selected vias is assigned to all the selected vias. <i>Right</i> : The X coordinate of the right-most selected via is assigned to all the selected vias. <i>Evenly Spaced</i> : The selected vias are spaced evenly between the left-most and the right-most vias. <i>User Pick</i> : The vias are spaced according to a user-defined location in the design canvas. The pick point is determined by the <i>Snap pick to pop-up</i> menu setting.
Stretch routing	Stretches all connected clines, including bond wires, to maintain connection to a via when it is moved. Note that vias that are directly connected to the via being aligned will be stretched as well.

Aligning Vias in Horizontally or Vertically

To align selected vias in vertical or horizontal directions, perform the following steps:

1. Choose *Route – Resize/Respace – Align Vias*.

Alternatively, type `via align` in the Command window.

2. Configure the Options panel.
3. Select the vias in the design canvas you want to align.
4. Choose *Done* from the pop-up menu when completed.

The pop-up menu shows the alignment status and allows you to choose any of the options.

The currently chosen option is grayed out. If you change any of the options for a selected set from the pop-up menu, select the vias again to change their alignment.

via array

The `via array` command lets you place a group of vias or structures in various patterns into a specified region of your design. The region may be the entire board, a bounding box that you draw with your mouse, or a shape. You can add via arrays to cline, shape, pin and via objects. When active, the command also places the properties attached to vias or structures. The command provides three modes to place, update and delete different types of array.

Options Panel for the via array Command

Access Using

- Menu path: *Place – Via Array*

<i>General Options</i>	
<i>Place</i>	When enabled, select one or more objects to add new arrays
<i>Delete</i>	When enabled, select one or more existing arrays or objects to delete the arrays from the design
<i>Update</i>	When enabled, select one or more existing arrays or objects to update either the array type or array values
<i>Enable DRC check</i>	Enables DRC checking for the via array placed during the command. If placing a via array results in a design rule violation, and this option is enabled, the via array is placed by removal of vias with DRC. If this option is disabled, the via array is placed with a DRC error.
<i>Enable preview</i>	Enables preview of the generated via array.
<i>Enable existing cline</i>	Enables selection of cline branches. If enable, via arrays are placed on the selected cline branches.
Enable origin point	Defines the origin point for connected cline from which end the via array start being placed
Via definition	
Via net	Enter a net name or browse to the net you want, or choose <i>Assign Net</i> from the pop-up menu and then click on a net on the layout.
Via padstack/structure	Lists vias and structures.
Angle	Specify to place a via/via structure at a predefined angle

Thermal relief type	<p>Specifies the thermal relief type for the vias and defines how the vias with the same net name as the shape should be connected to the shape. The settings in this option attach the <code>DYN_THERMAL_CON_TYPE</code> property to the vias.</p> <ul style="list-style-type: none">• <i>Full contact</i>: Creates no voids. For solid shapes, the shape completely fills around the via. For crosshatched shapes, the hatch lines provide the connections or Allegro X PCB Editor adds short connect lines.• <i>Orthogonal</i>: Connects straight up-down or left-right. The via connects directly to the void outline or hatch lines.• <i>Diagonal</i>: Connects diagonally upper left to lower right and lower left to upper right.• <i>8 way connect</i>: Connects lines from the thermal relief to the via both diagonally and orthogonally.• <i>None</i>: Contact is not made between the via and the shape.
Array parameters	Specify to set array parameters depending on the selected array type.

Type	<p>Select to specify an array pattern. Each array type comes with its own unique graphic to help explain the functionality. Swapping between the array types will toggle all appropriate settings, name, and graphics to match.</p> <ul style="list-style-type: none">• <i>Single side</i>: Add an array along one side of one or more selected object• <i>Both sides</i>: Add an array on both sides of one or more selected objects• <i>Centered</i>: Add an array centered on one or more selected objects• <i>Surrounding</i>: Add an object surrounding the selected objects• <i>Between</i>: Add an array between all selected objects that are parallel to each other• <i>Radial</i>: Add a circular or radial pattern of vias or structures around one or more selected objects• <i>Across board</i>: A matrix of vias or structures is added filling the board outline• <i>Across shape</i>: Matrices are added filling one or more selected shapes• <i>Across windowed area</i>: A matrix of vias or structures is added to a windowed area
Via to object gap (A)	Sets the distance of the first via from the boundary. This offset is the shortest distance from the center of the first via to the edge of the object. The first via references the bounding box of the shape instead of the shape boundary.
Via to via offset (B)	Sets the horizontal distance between two vias in a row. This offset is the shortest distance from the center of the first via to the center of the adjacent via.
Max Via displacement	Controls how much the array can move a via before requiring it to be removed per DRC conditions.
Row count	Specify the number of rows in a via array.
Row to row offset (C)	Sets the vertical distance between two vias in a column. This offset is the shortest distance from the center of the first via to the center of the next via.
Horizontal offset (B)	Sets the horizontal center to center spacing between via columns.

Vertical offset (C)	Sets the vertical center to center spacing between via rows.
Angle (B)	Sets the minimum angle between the center of two vias for a circular array.
Radius (A)	Sets the distance between the center of the object and the center of the via
Staggered vias	When enabled, arranges the vias in the group into a staggered pattern. Otherwise, the vias are arranged in horizontal rows and vertical columns.

Pop-Up Menu Options

When you are in `via array`, right-click in your design canvas to display the pop-up menu.

<i>Place</i>	Places the via array on the board.
<i>Assign Net</i>	Assigns a net associated with an object to the via array. For example, when you click on a pin, the pin net name is assigned to the net of the via array.


Related Topics:

- [Allegro User Guide: Preparing the Layout](#)
- [Adding a structure to the structure list](#)
- [Adding a Via Array to Shape](#)
- [Updating a Via Array](#)
- [Deleting a Via Array](#)

Adding a Via Array to Cline

To add a via array on one side of a cline or cline segments, perform the following steps:

1. Choose *Place – Via Array*.
Alternatively, type `via array` in the Command window.
2. Open the *Options* panel.
3. In the *General Options*, choose *Place*.
4. Ensure that *Enable DRC check* and *Enable Preview* options are enabled.
5. In the *Via net* field, enter an existing net name or browse to select a net.
6. In the *Via Padstack/Structure* field, select a via or structure from the pull-down list.

 You may need to add vias or via structures using Constraint Manager if they do not appear in the drop-down list.

7. In the *Array Parameters*, set the following:
 - a. Select *Type* as *Single Side*
 - b. Specify the value for *Via to object gap (A)*.
 - c. Specify the value for *Via to via offset (B)*.
8. Click to select a cline in the design canvas.
A via array is displayed along the cline. Moving the cursor to the other side of the cline changes the direction of the array.
9. Click on the board in the blank space to place the array on one side of the cline.
10. Right-click and choose an *Done* from the pop-up menu.


Related Topics:

- [Updating a Via Array](#)
- [Deleting a Via Array](#)

Adding a Via Array to Shape

For adding a via array to a dynamic shape, do the following:

1. Choose *Place – Via Array*.
Alternatively, type `via array` in the Command window.
2. Open *Options* panel.
3. In the *General Options*, choose *Place*.
4. Ensure that *Enable DRC check* and *Enable Preview* options are enabled.
5. In the *Via net* field, enter an existing net name or browse to select a net.
6. In the *Via Padstack/Structure* field, select a via or structure from the pull-down list.

 You may need to add vias or via structures using Constraint Manager if they do not appear in the drop-down list.

7. In the *Array Parameters*, set the following:
 - a. Select *Type* as *Across shape*
 - b. Specify the value for *Horizontal offset (B)*.
 - c. Specify the value for *Vertical offset (C)*.
 - d. Specify the value for *Max Via displacement*.
8. Click to select a shape in the design canvas.
A via array is displayed around the shape.
9. Click on the board in the blank space to place the array.
10. Right-click and choose an *Done* from the pop-up menu.

Related Topics:

- [via array](#)
- [Adding a Via Array to Cline](#)
- [Deleting a Via Array](#)

Updating a Via Array

The via array can be updated by changing the spacing values without deleting the previous array.

1. Choose *Place – Via Array*.

Alternatively, type `via array` in the Command window.

2. Open *Options* panel.
3. In the *General Options*, choose *Update*.
4. Ensure that *Enable DRC check* and *Enable Preview* options are enabled.
5. In the *Via net* field, enter an existing net name or browse to select a net.
6. In the *Via Padstack/Structure* field, select a via or structure from the pull-down list.
7. In the *Array Parameters*, set the following:
 - a. Select a different *Type*.
 - b. Specify the value for *Via to object gap (A)*.
 - c. Specify the value for *Via to via offset (B)*.
8. Click to select a object or the array in the design canvas.
You can preview both the existing and the updated arrays.
9. Click on the board in the blank space to update the array.
10. Right-click and choose an *Done* from the pop-up menu.

Related Topics:

- [via array](#)
- [Adding a Via Array to Cline](#)
- [Adding a Via Array to Shape](#)

Deleting a Via Array

To remove a via array from an object, perform the following steps:

1. Choose *Place – Via Array*.
Alternatively, type `via array` in the Command window.
2. Open *Options* panel.
3. In the *General Options*, choose *Delete*.
4. Ensure that *Enable DRC check* and *Enable Preview* options are checked.
5. Click to select a via in the design canvas.
6. Click on the board in the blank space to delete the array.
7. Right-click and choose an *Done* from the pop-up menu.

Related Topics:

- [via array](#)
- [Adding a Via Array to Cline](#)
- [Adding a Via Array to Shape](#)

via assign net

The `via assign net` command lets you assign a net to a via. The command is available in *General Edit* and *Etch Edit* application modes.

Assigning Net to a Via

To assign a net to a via, do the following:

1. Hover over a single via or multiple vias.
2. Right-click and choose the `Assign net to via` pop-up menu command.
3. In the *Options* panel, choose net name from the pull-down menu or browse from the *Select a net* dialog box.
4. Right-click and choose *Done* to complete the command.

Options Panel for the via assign net Command

<i>Assign net</i>	
<i>Assign net name</i>	Choose net from the pull-down menu or browse the net from <i>Select a net</i> dialog box.

view 3d

The `view 3d` command launches the Cadence 3D Design Viewer which lets you visualize and analyze designs in three dimensions. Cadence 3D Design Viewer allows you to control the orientation of the 3D model, specify color assignments, and control which objects to display. In addition to visualization, Cadence 3D Design Viewer allows you to modify or create new wire profiles, parameters, and groups.

Cadence 3D Design Viewer provides additional features that include Markup (an easy way to make comments directly on the screen display) and 3D DRC (design rule checking in three dimensions).

- The Markup feature allows you to annotate the displayed image and export industry standard (.jpg) images with notes and diagrams.
- The 3D DRC engine checks spacing rules in full three dimensions for wires, metal, and component bodies. Errors in 3D are highlighted by small 3D markers. You can click on a marker to display the nature of the error and highlight the geometries that triggered the error. 3D DRC is an extra-cost option.

 This command is available only with Allegro Advanced Package Designer (APD).

Access Using

- Menu path: *View – 3D Model*
The *3D Viewer Design Configuration* dialog box displays. Select *View* button.
- Toolbar icon:




Related Topics

- Cadence 3D Viewer User Guide

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.

 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*.

The `viewlog -browse` command opens the *Select File to View* dialog box. You can select a log file from the file browser to open it in a file viewer.

Select File to View Dialog Box

Access Using

- Menu path: *File – Viewlog* or *File – File Viewer*

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, the editor 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.
<i>Close</i>	Dismisses the window.

Related Topics

- Managing Environment Variables

Viewing Log Files

After completing an action or process, you can open log file either by specifying its name or just by selecting it from the list of log files.

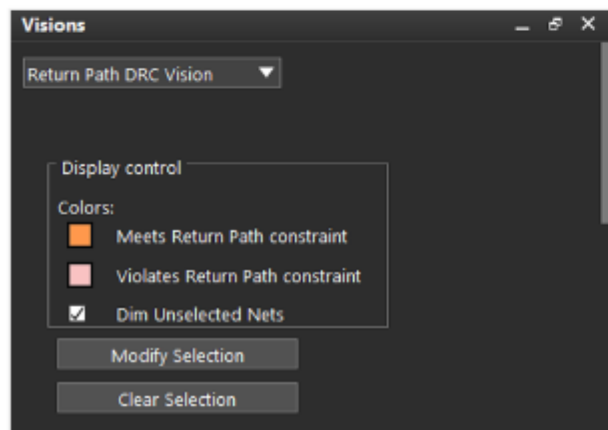
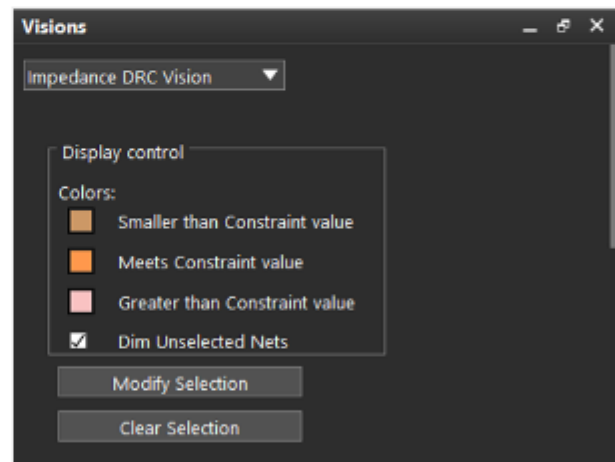
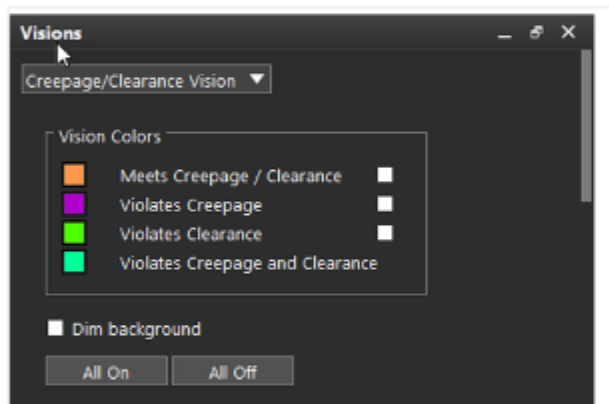
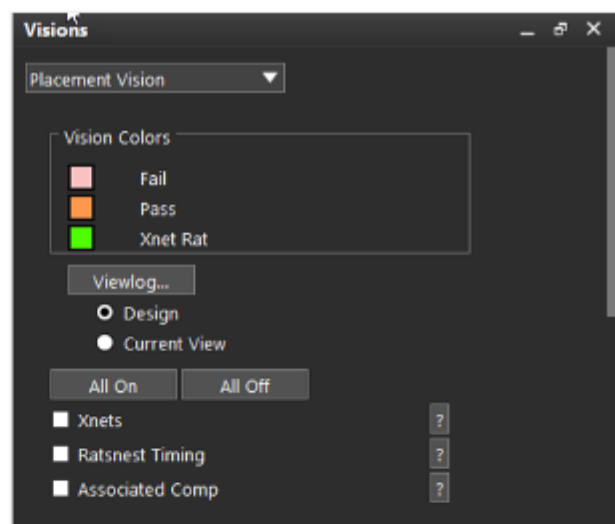
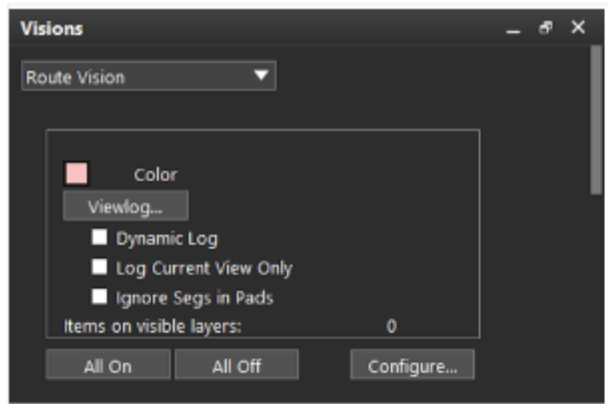
1. To open a log file, try either of the following methods:
 - Choose *File – Viewlog*.
Alternatively, type `viewlog` in the Command window.
 - Type `viewlog` in the Command window followed by the name of the file you want to view.
2. In the *Select File to View* dialog box, select a log file to view and click *Open*.
The log file viewer window displays the selected file.
3. To zoom in on the location in the design canvas, click the x y coordinates in the log file viewer window.

vision manager

The `vision manager` command displays selected nets or segments as a result of an analysis or DRC using color codes. The command provides four types of visions that run routing-based or placement-related graphical overlay checks and shows the results.

The options available are:

- Route Vision
- Placement Vision
- Creepage/Clearance Vision
- Impedance DRC Vision (available when *High Speed* option is selected)
- Return Path DRC Vision (available when *High Speed* option is selected)



Related Topics

- [Viewing Vision Analysis Results](#)

Visions Dialog Box



Access Using

- Menu path: *View – Vision Manager*

Placement Vision Options

Analyzes components and ratsnests in a design and view results by highlighting the objects that violates placement checks.

<i>Vision Colors</i>		
	<i>Fail</i>	Displays the color of objects that fail the analysis. Select color swatch to change the color. By default, red color is set.
	<i>Pass</i>	Displays the color of objects that pass the analysis. Select color swatch to change the color. By default, green color is set.
	<i>Xnet Rat</i>	Displays the color for viewing XNet rats of a design. Select color swatch to change the color. By default, magenta color is set.
<i>Viewlog</i>	Click to view <i>Placement Vision Report</i> for the last analysis. The report shows status of all the three visions in different tabs. Selecting an object row in the report zooms and highlights that object in the design window. Any design changes updates the log dynamically.	
<i>Design</i>	If enabled, all the objects in the design are reported in the log. This option is enabled by default.	
<i>Current View</i>	If enabled, only the objects that are visible in the current view of design window are reported in the log.	
<i>All On</i>	Enables all the three visions and displays results.	
<i>All Off</i>	Disables all the three visions.	
<i>Xnets</i>	Choose this option to view the XNet rats of a design. The vision reduces rats clutter and helps optimizing the placement of active components. The <i>XNets</i> vision ignores cline segments of a XNet connected to the discrete component and highlights all the XNets of a design in <i>XNet Rat</i> color.	


<i>Ratsnest Timing</i>	<p>Choose this option to view and modify the timing delay issues of ratsnests during components placement. The <i>Ratsnest Timing</i> vision compares Manhattan distance of the ratsnest with the DRC timing constraint and displays the results by highlighting ratsnests in <i>Pass/Fail</i> vision colors.</p> <p>Ensure that the <i>Propagation delay</i>, <i>Relative propagation delay</i>, and <i>Total etch length</i> checks are enabled in the Electrical section of the <i>Analysis Modes</i> dialog (<i>Setup – Constraints – Modes</i>).</p> <p>If <i>XNets</i> vision is also enabled, the results shows XNets only and ratsnests timing delay results are not shown.</p> <div> Only nets which have timing constraints are analyzed and highlighted by the vision. Nets without timing constraint are not analyzed at all.</div>
<i>Associated Comp</i>	<p>Choose this option to view the associated components that are placed outside the allowed distance. The <i>Associated Comp</i> vision checks the spacing between associated components and their parent components and displays the results by highlighting the associated components in vision <i>Pass/Fail</i> color.</p> <div> This option is not available with OrCAD PCB Editor.</div>

Viewing Vision Analysis Results

To view results for an analysis, do the following steps:

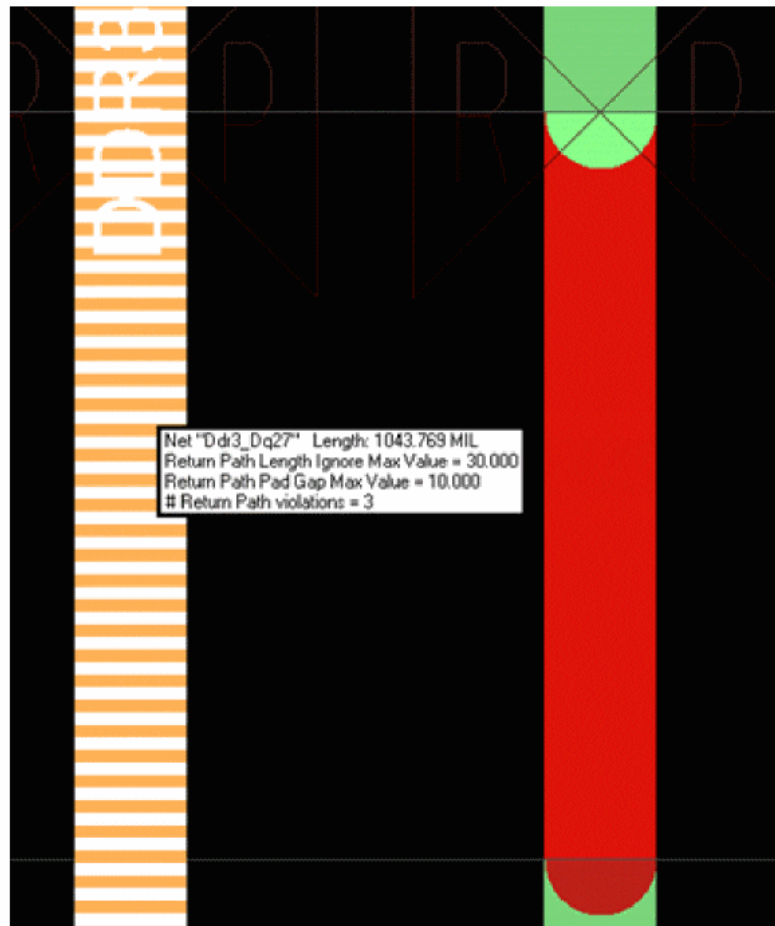
1. Select the analysis for which you want to see the color-coded results.
2. Depending on the vision specify the color code.
3. If needed, select the nets for which you want to view the vision.
 - a. To modify selected nets, click *Modify Selection*.
 - b. Right-click and choose *Done* when finished.

For route vision, cline segments will appear in different colors, depending on the rules you have enabled.

 Set *Ignore Segs in Pads* to skip segments that are entirely within a pad. These segments will not be analyzed by any of the algorithms.

4. Hover over a net to view details of the violations.

The violations are displayed in the selected color. For example, in the following image, segments with violation are shown in red and on hovering over a net shows details of the return path status.



Related Topics

- [vision manager](#)

void adjacent layer shapes

The `void adjacent layer shapes` command generates manual voids around objects of all kinds in shapes on specified layers; for example, to ensure that a high-speed signal has no interference from nearby power nets, or to reserve space in that shape for future routing.

Options Tab for the void adjacent layer shapes Command

Access Using

- Menu path: *Shape – Void Adjacent Layer Shapes*

<i>Create voids only if layer matches</i>	Specifies that voids should be created only if layers match
<i>Create voids in shapes on same net</i>	Specifies that voids should be created in the shape if it is on the same net as the selected item.
<i>Void Settings</i>	
<i>Clearances</i>	Select to specify clearance values for the void across layers. Selected by default.
<i>Merge options</i>	Select to specify merging rules for pad voids. Not selected by default.
	Depending on <i>Void Settings</i> option selected a table is displayed listing all the layers. The layer of the selected object is highlighted. If you selected <i>Clearances</i> , select layers for the void and specify the clearances for the selected layers. If you selected <i>Merge options</i> , select layers and specify the maximum and minimum separation values. By default, the maximum separation value for all layers is 10UM and the minimum is 0UM.
<i>Create Voids</i>	Click to create voids.

Creating Shape Voids

To create voids in shapes on selected layers, do the following:

1. Choose *Shape – Void adjacent Layer Shapes*.
Alternatively, type `void adjacent layer shapes` in the Command window.
2. Select design objects in the design canvas.
The layers on which the selected objects exist are highlighted in the Options tab.
3. Configure the Options tab.
4. Click *Create Void*.

V Commands

V Commands--void adjacent layer shapes
