

# **H 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	4
H Commands	4
help	5
helpcmd	5
Command Browser Dialog Box	6
Displaying List of All Commands	7
hide symbol drc	8
highlight sov	10
highlighting Segments Over Voids	11
highlight sov clear	11
Dehighlighting Segments Over Voids	12
hilight	13
Highlight Command: Options Panel	13
Highlighting Objects in Design Canvas	14
Cross-Probing in APD	15
hilight sym net	16
hilight sym net	16
Highlighting Signal Nets Associated with Symbols	17
history	18
http	19

---

# H Commands

---

help	helpcmd	hide symbol drc
highlight sov	highlight sov clear	hilight
hilight sym net	history	http

# help

The `help` command displays information available in the help system.

## Syntax

`help <command_name>`

<b>&lt;command_name&gt;</b>	Name of command for which help is required. For example, <code>help add shape</code> .
-----------------------------	--

## Access Using

- Toolbar icon:



## Related Topics

- [Cadence Help User Guide](#)

# helpcmd

The `helpcmd` command displays a list of all commands that can be entered in the command line.

## Command Browser Dialog Box

### ***Access Using***

- Menu path: *Tools – Utilities – Keyboard Commands*

<i>Execute</i>	Runs the command you selected from the displayed list.
<i>Help</i>	Displays documentation for the command you chose from the displayed list.
<i>Filter</i>	Limits the display of command selections in conjunction with wildcards (? for any single character and * for multiple characters).

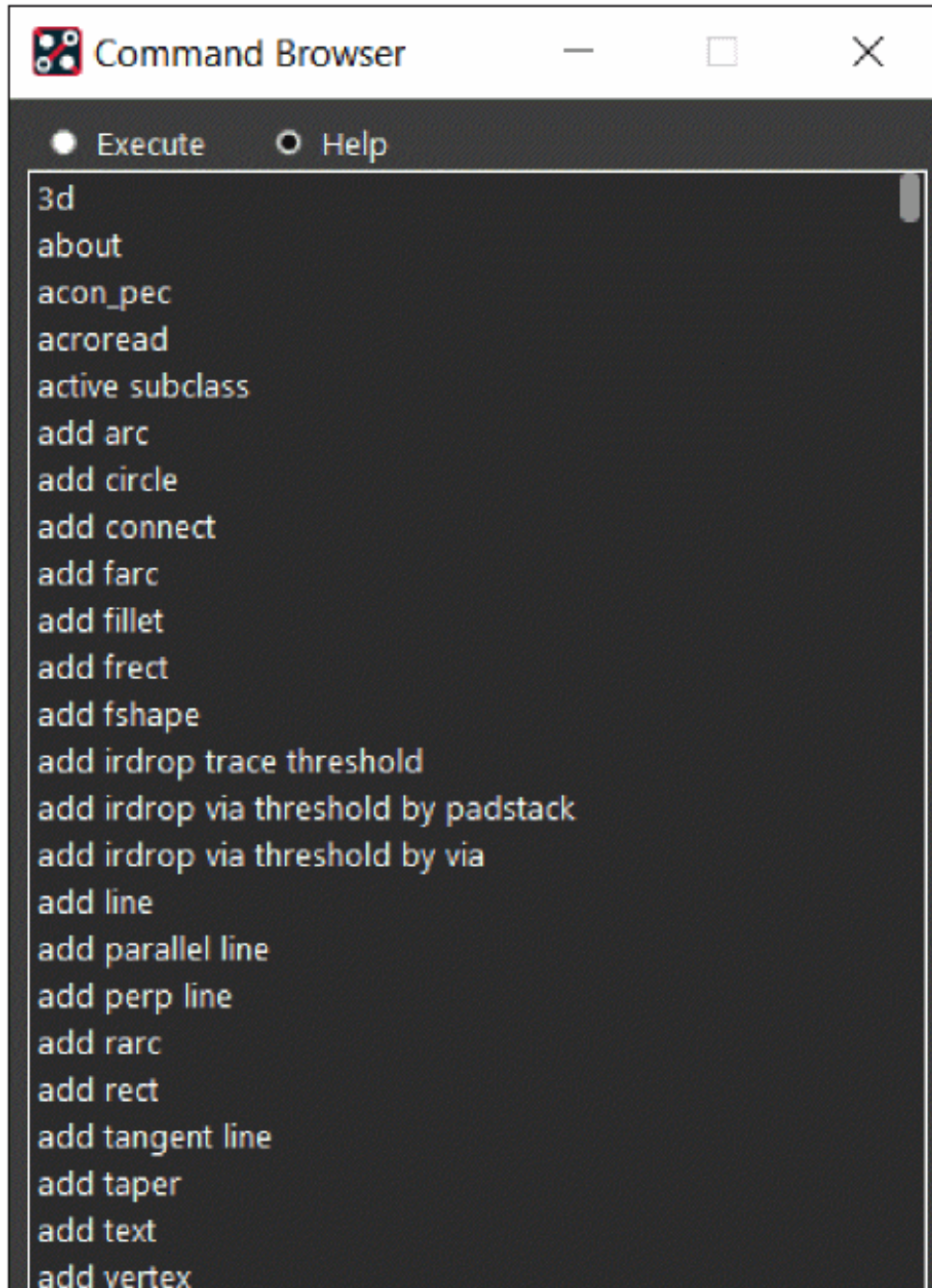
## Displaying List of All Commands

You can see a list of all the commands in your layout editor and then execute a specific command or access online help for it if help is available.

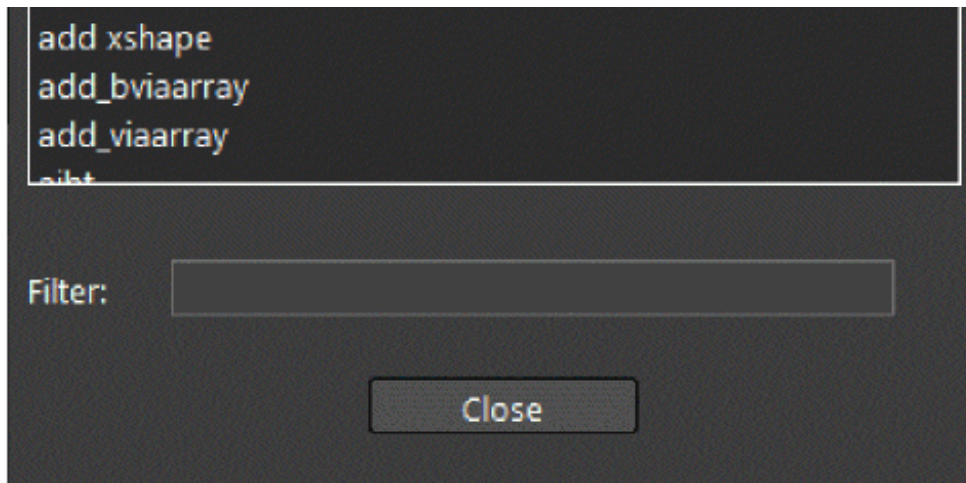
To view a list of available commands, do the following steps:

1. Type `helpcmd` at the Command window.


The *Command Browser* appears listing the commands in your layout editor.







2. Click either *Execute* (to run the command) or *Help* (to display documentation).

 Not every command has associated documentation.

3. Click the command name.
4. To limit the display of command selections, use the *Filter* field and wildcards (? for any single character and \* for multiple characters).
  - a. Type a command name or enter a partial string with wildcards to display one or more commands.
  - b. Press the `Tab` key. For example: Typing `?o1*` results in the display of the following commands: `color192 colorview create colorview load colorview restore polar`

## hide symbol drc

The `hide symbol drc` command hides all pin-to-pin minimum spacing DRC markers for large symbols. This applies to all the symbol definitions and instances in the design.

When a symbol with a high pin count (pin count over 10,000) is imported to the design, the `NODRC_SYM_SAME_PIN` property is applied to it by default. This property hides the pin-to-pin minimum spacing DRC markers for the pins on the symbol. The `hide symbol drc` command applies the property to all the symbol definitions and instances in the design to hide the symbol DRCs.

This leads to a smaller design file size and allows DRCs to run faster compared to a design with visible DRC markers for large symbols. The symbol import also speeds up significantly because the property is added before the symbol instance is placed in the design.

### ***Access Using***

- Menu Path: *Display – Large Symbol DRCs – Blank*

### **Related Topics**

- [NODRC\\_SYM\\_SAME\\_PIN](#)

# highlight sov

The `highlight sov` command locates and highlights the following violations:

- Segments of nets where signals overlap voids.
- Segments that run off an adjacent plane layer shape (partial shape coverage).
- Clines that are on a layer adjacent to a plane layer and are entirely within a void.

⚠ The `highlight sov` command does not check clines of nets that are assigned the `VOLTAGE` property.

A report file containing the violations is generated. You can open the report file in the *Viewlog* dialog box, which allows you to click on the X,Y coordinates of a violation and center the design window on that object.

The user preference variable `sov_spacing` allows you to specify the minimum space that must exist between a void and a cline segment. The user preference variable `sov_active` allows you to limit the check to only the active layer.

## Access Using

- Menu path: *Display – Segments Over Voids – Highlight*
- Toolbar icon:



## Highlighting Segments Over Voids


The Highlight Segments Over Voids function checks the design and highlights any segments that overlap a void on either adjacent plane layer. (If a value is assigned to the user preference variable `sov_spacing`, then that offset is also considered. If `sov_active` is enabled only the active layer is checked.)

To highlight segments that overlap voids, do the following:

1. From the *Display* menu, choose *Segments Over Voids – Highlight*.

Alternatively, type `highlight sov` in the Command window.


The report file appears that lists all the violations.

 If no violations are found, a message appears in the command line, and the report file does not appear.

2. In the report file, click on the location coordinates (in parentheses) for a particular net segment or void to center the design window on that object.

The cursor jumps to the location of the selected object and centers it in the design canvas.

The segment and void involved in the violation are highlighted.

 Layers do not need to be visible to be checked. Centering the design canvas from the report file does not change the visibility of layers.

3. Manually reroute the cline segment around the pin void to correct the routing path or use the `spread between voids` command in Allegro X PCB Editor (or the `spread clines` command in APD) to slide the segments off voids.
4. Repeat Steps 2 & 3 as needed to clean-up all the violations.

## Related Topics

- [spread between voids](#)
- [spread clines](#)

# highlight sov clear

The `highlight sov clear` command removes the highlighting of the segments that were previously highlighted by the `highlight sov` command.

## ***Access Using***

- Menu path: *Display – Segments Over Voids – Dehighlight*

### Dehighlighting Segments Over Voids

1. From the *Display* menu, choose *Segments Over Voids – Dehighlight* or type `highlight sov clear` at the command prompt.

The command dehighlights segments that are highlighted as a result of `highlight sov` command.

# highlight

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

If a net becomes highlighted in the design canvas, its name also displays in bold text in the *Nets* section of the *Color* dialog box.

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

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

- Symbols
- Functions
- Pins
- Vias
- DRC error markers
- Nets
- Clines
- Lines
- Shapes
- Via structures

A related command is *Display – Assign Color* (`assign color` command), which assigns a color and highlights an element without requiring the use of the *Color* dialog box or this command.

## Highlight Command: Options Panel

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

### Access Using

- Menu path: *Display – Highlight*
- Toolbar icon:



<i>Default Highlight Color</i>	Displays the current <i>Temporary Highlight</i> color that will be used to highlight chosen elements.
--------------------------------	---

### Related Topics

- [Cross-Probing in APD](#)
- [assign color](#)
- [Working with Highlighting and Coloring](#)
- [color192](#)
- [dehlight](#)



## Highlighting Objects in Design Canvas

To highlight objects in the design canvas

1. Do one of the following:

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

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

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

—or—

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

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

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

- d. Click the element to highlight.

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

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

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

## Cross-Probing in APD

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

## Related Topics

- [highlight](#)

# highlight sym net

## highlight sym net

The highlight sym net command highlights all the signal nets connected to a symbol. This command however, does not highlight dummy, power, and ground nets.

Elements highlighted with highlight sym net command stay highlighted until you right-click and choose *Dehighlight associated nets* (`dehighlight sym net` command) to disable the highlighting.

The command functions in a pre-selection mode in which you choose an element first, then right-click and execute the command. This command is valid only for symbols.

If the nets become highlighted in the layout, they are also highlighted in the Constraint Manager. You can use this feature to cross-probe between layout design and Constraint manager.

## Highlighting Signal Nets Associated with Symbols

To highlight signal nets connected to a symbol, do the following:

1. Hover your cursor over a symbol or draw a window around the symbols whose associated nets are to be highlighted.  
The layout editor highlights the symbol and a datatip identifies its name.
2. Right-click and choose *Highlight associated nets* from the pop-up menu.
3. Choose highlight pattern from the *Assign Pattern* dialog box.  
The signal nets associated with symbol are highlighted.

## Related Topics

- [dehighlight sym net](#)

# history

Internal command.

# http

The `http` command entered at the console window prompt lets you specify a site to be opened on your web browser.

## Syntax

```
http <file>
```

<b>file</b>	The Web location to open.
-------------	---------------------------

