

Q 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
Q Commands	4
quickplace	5
Quickplace Dialog Box	5
Schematic Page Number Dialog Box	6
Placing Components in a User-Defined Location	9
Placing Unplaced, Logic-Bearing Components in the Database to Areas Outside the Board Outline	10
Placing Components in a Design Partition	11
Placing Components by Schematic	11
quit	13
qvupdate	14
Running Qvupdate on Windows	14
Running Qvupdate on UNIX	17

Q Commands

quickplace	quit	qvupdate
------------	------	----------

quickplace

The `quickplace` command places components on the board using the the Quickplace dialog box.

 The `place manual` command addresses several aspects of component placement, such as orientation and re-layering. This command is apt for placing primary components in the design where placement is critical. The `quickplace` command just places the components on the board. This command is useful when getting a large number of discrete components into a design.

Using the *Quickplace* dialog box, you can create the floorplan of a design faster in the layout editor. You place logic-bearing symbols outside the board outline, either in a default or user-defined configuration. The default configuration consists of unplaced components in rows, inward to outward of the board outline, at the top edge of the board, in descending pin count order, with the components on the top layer of the board. You can also place components in rooms you specify.

The `quickplace` command also provides options to place the components as they are placed in the schematic and preserves their relative positions during placement.

Related Topics


- [Overview of Placing Elements](#)
- [Quickplace Dialog Box](#)

Quickplace Dialog Box

Access using:

- *Menu path: Place – Quickplace*
- *Command line: quickplace*

<i>Placement Filter</i>	Places components according to criteria you choose.
<i>Place by property/value</i>	Places components by a property and its value. The pull-down menu list all properties, including user-defined, assigned to unplaced component instances and definitions in the current design. The pull-down list displays all values for the chosen property name, including a blank value, that places all unplaced component or definitions with the associated property whatever its value.
<i>Place by room</i>	Places components in one or all rooms simultaneously. If the room is too small for all components, the layout allows overlapping. Choosing <i>Place by Room</i> disables options in the <i>Placement Position</i> section. To place by room within the room boundary, use <i>Place by Property/Value</i> and select the <i>Room</i> property from the pull-down list.
<i>Place by part number</i>	Places all unplaced components with the part number you specify. Allegro X PCB Editor and Allegro X Advanced Package Designersupport wildcard characters (* and ?).
<i>Place by net name</i>	Places all unplaced parts associated by the chosen net name. Click the <i>browse</i> button to display the <i>Select Net Name</i> data browser from which you can choose a net name. You can use wildcard characters to specify multiple nets.
Place by net group name	Places all unplaced parts associated by the chosen net group name. Click the <i>browse</i> button to display the <i>Select net group name</i> browser from which you can choose net group name which are associated with the unplaced parts. You can use wildcard characters to specify multiple net groups.

<i>Place by schematic page number</i>	<p>Places all unplaced parts simultaneously from multiple schematic pages while maintaining an update of those with unplaced components. Click the browse button to display the Schematic Page Number Dialog Box from which you can choose schematic hierarchical blocks and pages.</p> <div style="border: 1px solid #fde725; padding: 10px; margin-top: 10px;"> <p> By default, this option ignores components which are part of modules. Enable <i>Place components from modules</i> to quick place.</p> </div>
<i>Place all components</i>	Places all unplaced components (excluding components from modules) in the database. Enabled by default, which disables the <i>Place by refdes</i> section.
<i>Place by associated components</i>	Places unplaced components with their associated discrete components in the database. Select <i>All</i> from the drop-down to place all associated components close to their parents components in the database.
<i>Place by refdes</i>	This section lets you filter the unplaced parts.
<i>Type</i>	Choose the type of components to place.
<i>Refdes</i>	Places or blocks out specific groups of components by refdes name. Click <i>Include</i> or <i>Exclude</i> and enter an alphanumeric character in the corresponding field. This option accepts wildcard characters.
<i>Number of pins</i>	Enter the minimum or maximum number of pins allowable for placement. Defaults are <i>0</i> for <i>Min</i> and <i>100000</i> for <i>Max</i> .
<i>Placement position</i>	Specifies placement location.
<i>Place by partition</i>	Places components in a design partition that you specify.
<i>Place associated components on parent pins</i>	Places associated discrete components on their respective parent pins. Enabled by default. If disabled, places the associated components outside the board outline.

<i>By user pick</i>	Places all the unplaced components at a user-defined location inside the drawing extent in conjunction with the <i>Edge</i> and <i>Board Side</i> options. When enabled, selecting <i>Place</i> attaches the outlines of all the unplaced components to the cursor. Click to choose a location in the design canvas to place all the components at the cursor location. If <i>Schematic type placement</i> option is also enabled in the <i>Schematic Page Number</i> dialog box the component outlines attached to the cursor retain their positions as they are defined on the schematic page. The relative positions of the component outlines are determined by the SCHEMATIC_XY property, which is transferred to the layout on importing schematic.
<i>Around package keepin</i>	Places components around the edges of the package keepin. You specify the edge direction as <i>Top</i> , <i>Bottom</i> , <i>Left</i> , or <i>Right</i> in the <i>Edge</i> field.
<i>Edge</i>	Places components on one or more board edges. The default is <i>Top</i> .
<i>Board layer</i>	Places components on the layer. The default is <i>Top</i> .
<i>Overlap components by</i>	Controls the amount of overlap. The default is <i>50%</i> . This option is disabled if <i>Schematic type placement</i> option is enabled in <i>Schematic page number</i> dialog.
<i>Symbols placed</i>	Displays the number of components placed and the number of available components for placement, as determined by the <i>Place by refdes</i> settings.
<i>Undo last place</i>	Removes the last place operation specified by the <i>Place by refdes</i> settings.
<i>Place components from modules</i>	Filters components that are part of the module. Enabling this field breaks modules apart.
<i>Unplaced symbol count</i>	Displays the number of remaining unplaced components.
<i>Viewlog</i>	Displays the <code>quickplace.log</code> report for the last place operation.
<i>Close</i>	Saves the changes and closes the dialog box.
<i>Unplace</i>	Removes all the components placed during the current session.
<i>Place</i>	Places the specified components in the specified configuration.

Schematic Page Number Dialog Box

<i>Select page number(s) for placement</i>	Displays all schematic pages with unplaced components with branches in the tree control representing hierarchical blocks. Choose multiple pages or levels of hierarchy for placement.
<i>Schematic type placement</i>	If enabled, components are placed according to the schematic. This option does not work if multiple pages are selected for placement.
<i>Scaling factor</i>	Controls the spacing between components before placement. The default value is 1. A value greater than 1 increases the spacing between components. To reduce the spacing, specify a positive value which is less than 1.
<i>OK</i>	Saves the changes and closes the dialog box.
<i>Cancel</i>	Closes the dialog box.
<i>Reset</i>	Returns settings to their original state.

Related Topics

- [quickplace](#)
- [Placing Components in a User-Defined Location](#)
- [Placing Unplaced, Logic-Bearing Components in the Database to Areas Outside the Board Outline](#)
- [Placing Components in a Design Partition](#)

Placing Components in a User-Defined Location

Following are the steps to place components in a user defined location:

1. Choose *Place – Quickplace* (`quickplace` command).
The Quickplace dialog box appears.
2. Choose *By user pick* in the *Placement position* section.
3. Choose a board edge in the *Edge* field. You can select only one at a time, when placing by user pick.
4. Choose a *Board layer*.
5. Click *Place* to add the components.
The outlines of all the unplaced components are attached to the cursor. You can cancel the cursor attachment using pop-up menu command *Cancel*.
6. Pick a location in the design.
The components are placed at the selected location.
7. Click *Close* to save your changes and close the dialog box.

Related Topics

- [quickplace](#)
- [Quickplace Dialog Box](#)
- [Placing Unplaced, Logic-Bearing Components in the Database to Areas Outside the Board Outline](#)
- [Placing Components in a Design Partition](#)

Placing Unplaced, Logic-Bearing Components in the Database to Areas Outside the Board Outline

Following are the steps for placing unplaced, logic bearing components in the database to areas outside the board outline:

1. Choose *Place – Quickplace* (`quickplace` command).
The Quickplace dialog box appears.
2. Configure the dialog box to add unplaced components to the outside edges of your board outline.
3. Click *Place* to add the components.
4. Continue to place components.
5. Click *Close* to save your changes and close the dialog box.

Related Topics

- [quickplace](#)
- [Quickplace Dialog Box](#)
- [Placing Components in a User-Defined Location](#)
- [Placing Components in a Design Partition](#)

Placing Components in a Design Partition

Following are the steps for placing a component in a design partition:

1. Choose *Place – Quickplace* (`quickplace` command).
The Quickplace dialog box appears.
2. Choose the components you want to place in the *Placement filter* section.
3. Choose *Place by partition in the Placement position* section.
4. Select a partition name from the pull-down menu.
5. Click *Place* to add the components.
The components appear within the partition boundaries.
6. Click *Close* to save your changes and close the dialog box.

Related Topics

- [quickplace](#)
- [Quickplace Dialog Box](#)
- [Placing Components in a User-Defined Location](#)
- [Placing Unplaced, Logic-Bearing Components in the Database to Areas Outside the Board Outline](#)

Placing Components by Schematic

Following are the steps to place components by Schematic:

1. Choose *Place – Quickplace* (`quickplace` command).
The Quickplace dialog box appears.
2. Select *Place by schematic page number* in the *Placement filter* section.
3. Click browse button to open *Schematic Page Number* dialog box.
4. Select a single page in the *Schematic* tree structure.
5. Enable *Schematic type placement* option.
6. Choose *By user pick in the Placement position* section.
7. Choose *Edge* and *Board layer*.
8. Click *Place* to add the components.
The component outlines attaches to the cursor as specified in the schematic page.
9. Pick a location in the design.
The components are placed maintaining their relative positions as defined on the schematic page.
10. Click *Close* to save your changes and close the dialog box.

Related Topics

- [quickplace](#)
- [Quickplace Dialog Box](#)
- [Placing Components in a User-Defined Location](#)
- [Placing Unplaced, Logic-Bearing Components in the Database to Areas Outside the Board Outline](#)
- [Placing Components in a Design Partition](#)

quit

The `quit` command indicates the end of a design session and returns you to the host operating system. The command asks for confirmation by displaying a window in which you can choose to quit or cancel the command and stay in the user interface.

Access using:

- *Menu path: File – Quit*

qvupdate

Updates footprint information in design (.brd), drawing (.dra), padstack (.pad), or module (.mdd) databases that were created prior to release 14.0 so that text and graphics associated with them can be displayed in the Quickview window of file/library browsers.

⚠ Graphics are not available for padstacks; only text regarding the padstack name, type, units, accuracy, and geometry is available.

⚠ This command is no longer required in release 17.0 and may be removed in future releases.

qvupdate <database name>

qvupdate * <file extension>

database name	Updates the footprint information for the database.
* file extension	Updates all files of a particular type.

Related Topics

- [Running Qvupdate on Windows](#)
- [Running Qvupdate on UNIX](#)

Running Qvupdate on Windows

Follow the steps to run Qvupdate on Windows:

1. Open the Qvupdate dialog box by double-clicking the `qvupdate.exe` icon located in your install directory.
2. Enter, or browse for, the name of the single database you want to update. Include the full path if the database name you enter is in a different location than Qvupdate.
—or—
Enter `*` and the appropriate extension to update to update all files of a particular type; for example, `C:\Boards*.brd`.
3. Click *Execute*.

The program updates the footprint information for the database(s). When the operation is complete, a log file is displayed containing the status of the update.

Related Topics

- [qvupdate](#)
- [Running Qvupdate on UNIX](#)

Running Qvupdate on UNIX

Follow are the steps to run Qvupdate on UNIX:

1. Enter `qvupdate` and a single database name at the console window prompt.

—or—

Enter `qvupdate` and `*` with the appropriate extension to update to update all files of a particular type.

```
qvupdate /boards/pinesc.brd
```

```
qvupdate /boards/*.brd
```

The program updates the footprint information for the database(s) and returns information containing the status of the update.

Related Topics

- [qvupdate](#)
- [Running Qvupdate on Windows](#)

