

M 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 | 6 |
| M Commands | 6 |
| mail | 7 |
| Allegro Mail Dialog Box | 8 |
| manage_settings | 9 |
| Manage UI Settings Dialog Box | 9 |
| Managing User Interface Settings | 10 |
| mark fanout | 11 |
| Associating Clines and Vias with Component Symbol Instances | 13 |
| mbs2brd | 15 |
| Translating Mentor Board Station Designs into Allegro PCB Editor | 16 |
| mbs2lib | 19 |
| Mentor PCB Library Translator Dialog Box | 20 |
| Translating Mentor Libraries | 22 |
| metal density scan | 24 |
| metal usage report | 26 |
| Metal Usage Report Dialog Box | 27 |
| Generating Metal Usage Report | 28 |
| mirror | 30 |
| mirror Command: Options Panel | 31 |
| Relocating Symbols to the Opposite Side of a Board or Substrate | 32 |
| Mirroring Design Objects on the Same Subclass | 33 |
| miter_by_pick | 34 |
| Mitering Corners | 35 |
| model editor | 36 |
| | 37 |

| | |
|---|----|
| Features of Model Editor | 41 |
| model integrity | 42 |
| mod padstack | 43 |
| | 43 |
| Editing Design Padstacks | 44 |
| modpaste | 45 |
| mosaic_cmd | 46 |
| mouse_pos | 47 |
| move | 48 |
| | 48 |
| Move Command: Pop-Up Menu Options | 52 |
| Move Command: Options Panel | 54 |
| Moving One or More Elements | 57 |
| Moving Elements by Incremental Distance | 59 |
| Moving RF Clearance Assembly Group | 61 |
| Moving Objects Relative to Other Objects | 62 |
| move vertex | 63 |
| | 63 |
| Move Vertex Command: Options Panel | 64 |
| Moving a Vertex | 66 |
| movewindow | 67 |
| Positioning the Window | 68 |
| muconnect | 69 |
| | 69 |
| Symphony Dialog Box | 70 |
| Connecting to Symphony Server Application | 74 |
| multpadedit | 75 |
| | 75 |
| Multpadedit Command: Options Panel | 76 |
| Modifying Pad Shapes | 77 |
| muserver | 78 |
| | 78 |
| Symphony Server Window | 81 |
| Managing Symphony Server | 86 |
| muservermgr | 87 |
| | 87 |
| Symphony Server Manager Dialog Box | 89 |

| | |
|--|----|
| Managing Designs Using Symphony Server Manager | 91 |
|--|----|

M Commands

| | | |
|--------------------|--------------------|----------------------------------|
| mail | manage_settings | Managing User Interface Settings |
| mark fanout | mbs2brd | mbs2lib |
| metal density scan | metal usage report | mirror |
| miter_by_pick | model editor | model integrity |
| mod padstack | modpaste | mosaic_cmd |
| mouse_pos | move | move vertex |
| movewindow | muconnect | multpadedit |
| muserver | muservermgr | |

mail

The `mail` command displays the Allegro X Mail dialog box that you use to facilitate communication among team members working with partitioned designs. An e-mail program must exist on your system, and you must be able to log on to it to use the tool's mail. Attachments are not supported.

On Windows, the command uses Windows MAPI to send e-mail messages; on UNIX/Linux, the results of the call `which mail`.

Allegro Mail Dialog Box

| | |
|------------------|---|
| <i>To</i> | Enter recipient names to which to send a message, separated by a semicolon. |
| Stored Addresses | Lists all designers' names associated with the partitioned design. |
| <i>cc</i> | Enter recipient names to which to carbon copy (cc) the message. |
| <i>Subject</i> | Enter the message topic. |
| Message Body | Enter the message content. |
| <i>Send</i> | Click to deliver the message to the specified recipients. |
| <i>Cancel</i> | Click to preclude sending the message. |

Related Topics

- [design partitioning](#)

manage_settings

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

Related Topics

- [Manage UI Settings Dialog Box](#)

Manage UI Settings Dialog Box

Use this dialog box to manage the custom settings.

Access Using:

- Menu path: *View – UI Settings – Manage Settings*

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

Managing User Interface Settings

The `manage_settings` command provides various ways to customize the user interface (UI) settings. You can do the following to modify the UI:

Applying Custom UI Settings

Perform the following steps to apply custom UI settings to your layout editor:

1. Choose *View – UI Settings – Manage Settings*.
Alternately, run `manage_settings` from the command window prompt.
2. Select the name of an already saved custom settings in the *Custom Settings* section.
3. Click *Apply*.
The selected settings are applied to the active database.

Importing Custom UI Settings

Perform the following steps to import custom UI settings to your layout editor:

1. Choose *View – UI Settings – Manage Settings*.
Alternately, run `manage_settings` from the command window prompt.
2. Click *+* in the *Custom Settings* section.
A file browser opens.
3. Browse the location of the configuration file and click *Open*.
The selected settings are now available under the *Custom Settings* section.

Exporting Custom UI Settings

Perform the following steps to export custom UI settings from your layout editor:

1. Choose *View – UI Settings – Manage Settings*.
Alternately, run `manage_settings` from the command window prompt.
2. Select the name of an already saved settings in the *Custom Settings* section.
3. Click *Export*.

A file browser opens.

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

Restoring Legacy UI Settings

Perform the following steps to restore legacy UI settings to your layout editor:

1. Choose *View – UI Settings – Manage Settings*.
Alternately, run `manage_settings` from the command window prompt.
2. Click *+* in the *Custom Settings* section.
A file browser opens.
3. Browse to default configuration file `AllToolbars*.ini`, which is located at
`<installation_directory>/share/pcb/text`.
The *AllToolbars* settings is now available in the *Custom Settings* section.
4. Select *AllToolbars* and click *Apply*.
The legacy settings are applied to the active database.

 You can also use *View – UI Settings – Reset UI to All Toolbars* menu option.

Related Topics

- [manage_settingsManage UI Settings Dialog Box#69430](#)

mark fanout

The `mark fanout` command associates clines and vias with their respective component symbol instances, to ensure that fanouts created with Spectra or third-party tools are identified as such once the design is read into the layout or board editor. Fanouts already identified as such appear dimmed and cannot be chosen. Only clines and vias not yet marked as fanouts display with full intensity.

In APD, you can use the *Route* menu options, such as *Offset Via Generator* (`offset via gen`), *Flip Chip Die Escape Generator* (`die escape gen`), and *Wire Bond Die Escape Generator* (`wire bond escape`), to generate fanouts which can then be attached to the symbol using `mark fanout`.

⚠ In APD, the `mark fanout` command reassigns the via-structure clines and vias to the die; therefore, the associativity of the clines and vias to the via-structure will be lost. Replacing the via-structure will not change any clines or vias that had since been associated (Marked) with the die.

⚠ You need not run `mark fanout` when fanouts in your design have been created with the `create fanout` command.

Fanouts typically end on a via, but may also terminate on a cline. Clines and vias can only be fanouts if all existing clines and vias connected between them and their pin are fanouts too.

A marked fanout comprises clines and vias connected to a pin and associated with the component symbol instance, as does an unmarked fanout, however, the latter is not associated with the component symbol instance. You may unmark fanouts using the `unmark fanout` command.

⚠ The `mark fanout` command associates the clines and vias with an instance and not with the definition of the symbol. Therefore, refreshing a symbol from its definition removes any fanout information from the symbol instance; for example, if you unplace or replace a symbol instance or refresh from the library definition.


When you choose pins, this command searches each connection for the pin. If the connection ends without reaching another pin, all clines and vias are associated with the symbol instance, becoming a marked fanout. However, if the connection ends at another pin, then only the first cline and via connected to the pin are associated with the symbol instance. Pins already having a hole are ignored, as are pins whose clines and vias are already associated with a symbol instance. Once


marked, fanouts move along with symbol instances.

This command functions in both the noun-verb (pre-selection) mode and verb-noun mode. In the pre-selection use model in the Etch Edit application mode, you choose an element first, then from the pop-up menu (right-click) choose and execute the command.

Valid objects are:

- Pins
- Clines
- Vias
- Wire Bond Fingers (APD)

 Wire bond fingers attached to a design are treated as marked fanouts and are associated with components.

 In the menu-driven editing mode, existing fanouts appear dimmed and cannot be chosen. Only the clines and vias not yet marked as fanouts display with full intensity. This does not occur when you are working in the pre-selection use model.

Access Using:

- Menu path: *Route – Convert Fanout – Mark*

Related Topics

- [create fanout](#)
- [unmark fanout](#)

Associating Clines and Vias with Component Symbol Instances

1. Choose *Route – Convert Fanout – Mark*

Alternately, you could run the `mark fanout` command from the command line.

2. Another way to access this command is as follows:

- a. Choose *Setup – Application Mode – Etch Edit* to access the etchedit application mode.
- b. Hover your cursor over the element or window select to choose several elements. The layout editor highlights the element, and a datatip identifies its name.
- c. Right click to choose *Mark Fanout* from the pop-up menu to automatically launch the command and mark the fanouts.

3. Choosing pins associates the clines or vias connected to those pins with the symbol instance, along with any vias and clines in between.
4. Choosing vias and clines associates them with their pin's symbol instance, along with any vias and clines in between. When more than one pin causes ambiguity in determining the symbol owner, a failure occurs.

The command then exits, and you may choose other fanouts to mark.

mbs2brd

The `mbs2brd` command converts Mentor designs from Mentor Board Station (versions C2 and B4) to an Allegro X PCB Editor board file.

```
mbs2brd  -a <geom_ascii_file_name>
```

```
-t <tech_file_name>
```

```
-n <nets_file_name>
```

```
-c <comps_file_name>
```

```
-p <pins_file_name>
```

```
-r <traces_file_name>
```

```
-s <testpoints_file_name>
```

```
-e <template_board_name>
```

```
-d { Suppress dump libraries }
```

```
-f { Suppress db fix }
```

```
-y { Use symbol names as device type }
```

```
-pn { Use part numbers as device type }
```

```
-u <output units> { microns, mm, cm, or mils }
```


-z <device_class_filters_filename>

-pg <pwr_gnd_nets_filename>

-lm <usr_lyr_map_filename>

-ts { Build stackup from tech file }

-mva { Minimum Void Area }

-sct {Suppress Constraint Translation}

-log <log_file_name> { If not specified 'importMentor' will be created in the current directory }

<output_board_name> (Required)

| | |
|--|---|
| -a <geom_ascii_file_name> | The name of a single geometry ASCII file to use as a Mentor Board Station source (for example, export.geoms). |
| -t <tech_file_name> | The name of a Board Station technology file (for example, export.tech). |
| -n <nets_file_name> | The name of a Board Station nets file (for example, export.nets). |
| -c <comps_file_name> | The name of a Board Station components file (for example, export.comps). |
| -p <pins_file_name> | The name of a Board Station pins file (for example, export.pins). |
| -r <traces_file_name> | The name of a Board Station traces file (for example, export.traces). |
| -s <testpoints_file_name> | The name of a Board Station testpoints file (for example, export.testpoints). |

| | |
|---|--|
| -e <template_board_name> | The name of the board template to use while translating. |
| -d | Suppresses the creation of dump libraries. |
| -f | Suppresses db fix, and does not run dbdoctor against the generated board to check for and fix issues. |
| -y | Uses symbol names as device types. |
| -pn | Uses part numbers as device types. |
| -u | Specifies the output units. The valid options are: microns, mm, cm, or mils. |
| -z <device_class_filters_filename> | The name of the device class filter file, providing device class filters to the translator. |
| -pg <power_ground_nets_file_name> | The name of the power and ground nets file, providing which nets the translator should save as power and ground. |
| -lm <usr_layer_map_file_name> | The name of a user layer map, providing how the translator should map objects on non-standard layers. |
| -ts | Builds a stackup from the information in the tech file. |
| -mva | Optimizes the board to minimize the void area. |
| -sct | Suppresses constraint translation, and the constraints are saved in a DCF file and are not loaded to the output board file. |
| -log <log_file_name> | Specifies the name of the log file for the translation process. If this is not specified, the logs are generated in the <code>importMentor.log</code> file in the working directory. |
| <output_board_name> | The name of the Allegro brd file. |

Related Topics

- Transferring Logic Design Data
- [mbs2lib](#)

Translating Mentor Board Station Designs into Allegro PCB Editor

Perform these steps to translate mentor board station designs into your layout editor:

1. At your operating system prompt, enter `mbs2brd` to display the translator options.
2. When you have entered the information on the command line, press the Return key to run the translator.

The screen displays the status of the translation, similar to the following:

```
***** Starting Translation using version: vl6-3-81A_9/24/2009

Creating independent data.

Performing a partial database check before saving.

Writing database to disk.

'via20_40s_fl.dra' saved to disk.

Performing a partial database check before saving.

Writing database to disk.

'via20_40s_fl.bsm' saved to disk.

Creating independent data.

Performing a partial database check before saving.

Writing database to disk.

'via15_35s_fl.dra' saved to disk.

Performing a partial database check before saving.

Writing database to disk.

'via15_35s_fl.bsm' saved to disk.
```

The log files and `importMentor.log` that are created in the current working directory provide comprehensive details of the translation process.

Related Topics

- [mbs2lib](#)

mbs2lib

The `mbs2lib` command converts Mentor libraries from Mentor Board Station (versions C2 and B4) to a format that can be used in Cadence designs. Graphical user interface and batch versions of the translator let you create Cadence versions of all or part of a library via regularly scheduled incremental updates or in a "one shot" complete update of all Mentor-formatted libraries.

`mbs2lib -file <input filename>`

```
-dir <input directory name>
-list <input file-list filename>
-output <output directory>
-units <output units> { inch, mm, cm, or mil }
-psm { create package symbol files }
-map <user layer-map filename>
-nogui
-check {works only with -nogui
  Outputs missing depednency list to file "missingDep.lst" and exit}
-geompath <input directory-list filename>
-v { verbose mode for more error messages }
-help
```

| | |
|---|---|
| -help | Prints the usage description to the screen. |
| -file <input name> | The name of a single geometry ASCII file to use as a Mentor source (for example; geoms.ascii). |
| -dir <input directory name> | The name of a directory in which to look for one or more ASCII files to use as a Mentor source. |
| -list <input file list name> | The name of a text file containing a list of multiple ASCII files to use as Mentor sources. |

| | |
|---|---|
| -output <directory name> | The output directory within which a Symbols subdirectory is created to hold library data. The default location is the current working directory. |
| -units <output units> | The output units for all library objects created by the translator (inch, mm, cm, or mil). The translator defaults to the unit most closely resembling that of each original object. -psm Directs the translator to create .psm (symbol) files for all packages. The default is off. |
| -map <user layer.map> | The name of a user layer map, providing how the translator should map objects on non-standard layers. |
| -nogui | Directs the command to run in batch mode, not through the graphical user interface. |
| -check | Outputs missing dependency list to file <code>missingDep.lst</code> and exits. This option only works in the <code>-nogui</code> mode. |
| -geompath <input file list name> | The name of a text file containing a list of directories that contain ASCII files to use as Mentor sources. |
| -v | Verbose mode for error logging. |

Related Topics

- [Translating Mentor Board Station Designs into Allegro X PCB Editor](#)
- Transferring Logic Design Data
- [mbs2brd](#)

Mentor PCB Library Translator Dialog Box

The dialog box appears when Mentor source files have been identified and an output directory and user-defined layer map file selected. The controls in the dialog box correspond to the options available in batch mode.

ADD FILES buttons let you browse for and choose Mentor ASCII files from three sources:

| | |
|-----------------------|--|
| <i>One</i> | Identify a single ASCII file to use as a Mentor source (for example; 0.040x0.060x0.015) |
| <i>Directory</i> | Identify a directory containing one or more ASCII files to use as a Mentor source |
| <i>List</i> | Identify a text file containing a list of multiple ASCII files to use as Mentor sources. The file should contain one ASCII file name per line of text. |
| <i>Directory List</i> | Identify a text file containing a list of directory paths that contain Mentor sources. The file should contain one directory path per line of text. |

EXCLUDE buttons let you remove source files from the list:

| | |
|---------------------|--|
| <i>Selection</i> | Removes the highlighted file |
| <i>All</i> | Removes the entire list of files |
| <i>Deselect All</i> | De-highlights files selected for removal |

Output of the translated files is determined in the Output section of the dialog box:

| | |
|---------------------|--|
| <i>Units</i> | Control the units of measurement by selecting a units option for the translated library objects. The default selection is the unit most closely resembling that of each original object. |
| <i>Directory</i> | Browse for or enter the location of a Symbols directory that the translator creates to hold library data. The default location is the current working directory. |
| <i>Symbol Files</i> | Check to create .psm (symbol) files for all packages. The default is off. If you need to exercise greater control of layer mapping—for example, where package geometry cannot be created on BOARD GEOMETRY subclasses—a layer map file may be necessary in instances where you must override automatic map translations, providing information as to how the translator should map objects on non-standard layers. |

| | |
|---------------------|---|
| <i>Layer Map</i> | Browse for or enter the name of a user layer map file that provides how the translator should map objects on non-standard layers. |
| <i>Dependencies</i> | Click to display a list of dependencies that the objects might have on missing objects. |
| <i>Import</i> | Click to start the translation process. |

Translating Mentor Libraries

You can convert Mentor libraries either from GUI or from the command line using the following steps:

Using the Mentor PCB Library Translator Dialog Box

1. At your operating system prompt, enter `mbs2lib` to display the *Mentor PCB Library Translator* dialog box.
2. Click *One*, *Directory*, *List*, or *Directory List* buttons in the *Input Mentor ASCII Files* area to specify the locations of the Mentor sources.
3. Specify the Allegro Output options for *Units*, *Directory*, and *Symbol Files* settings.
4. Specify a *Layer Map* file to specify any non-standard layer mapping.
5. Click *Import* to start the process.
6. The translation log file is displayed. The log file and `MentorLibs.log` that are created in the current working directory provide comprehensive details of the translation process.

Using the Command Line

1. At your operating system prompt, enter `mbs2lib -help` to display the translator options.
2. When you have entered the information on the command line, press the Return key to run the translator.

The screen displays the status of the translation, similar to the following:

```
$ mbs2lib -file /hm/taylor/MentorLibs/plexus/geoms_to_cadence/tssop_56 -output /hm/taylor/MentorLibs/devices -nogui
```

```
Creating independent data.
```

```
... 50 entities converted.
```

```
... 100 entities converted.
```

```
... 150 entities converted.
```

```
... 200 entities converted.
```

```
Performing a partial dbcheck before saving.
```

```
Writing database to disk.
```

```
'tssop_56.dra' saved to disk.
```

The log file and `MentorLibs.log` that are created in the current working directory provide comprehensive details of the translation process.

Related Topics

- [mbs2lib](#)
- [Translating Mentor Board Station Designs into Allegro X PCB Editor](#)

metal density scan

The `metal density scan` command assesses and reports the localized metal density across each layer of the design based on a specified window size and a scan step size.

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

Access using:

- Menu path: *Manufacture – Metal Density Scan*

metal usage report

The `metal usage report` command provides you with an accurate assessment of the percentage of metal in a user-specified region of the design.

Related Topics


- [Generating Metal Usage Report](#)
- [Preparing the Layout](#)

Metal Usage Report Dialog Box

Access using:

- Menu path: *Tools – Metal Usage Report*

| | |
|-----------------------------|--|
| <i>File Name</i> | Specifies the name of the report file to be written to disk. If you specify the <i>View Report</i> option, this field is ignored. You can type in the name of the report, or use the Browse button (...) to browse to the file name and location where you want the report to be saved. If you set the ADS_SDREPORT environment variable in your user preferences (<i>envded</i> command), the report file will be written to that subdirectory of the current working directory or the directory to which you browsed. |
| <i>Use board outline</i> | Click this button to use board outline or design extents as the boundary for the computation to generate a report for metal usage. |
| <i>Select window region</i> | Click this button to make a two-pick window selection in the design window. This bounding box is used as the extents for the computation to generate a report for metal usage. Any object only partially enclosed within this region is trimmed at the boundary to ensure accurate calculations. |
| <i>Select symbol</i> | <p>Click this button to select a placed symbol to be used as the boundary for computation, for example, a die. This symbol's place bound box is used as the extents for the computation to indicate how much metal is on the selected layers underneath the die.</p> <p>Note: If you select multiple shapes in this mode, the layout editor will generate a separate metal usage report for each of the shapes selected.</p> |

| | |
|--|---|
| <i>Select shape</i> | <p>Click this button to select a shape to be used as the boundary for the computation that indicates how much metal is on the selected layers underneath the shape. This shape does not have to be on a layer being reported on.</p> <div style="border: 1px solid #fde725; padding: 10px; margin: 10px 0;"> <p> In creating this report, the layout editor looks at any voids in the shape boundary itself and removes those prior to beginning its computations. If you select a crosshatched shape, only objects directly under the metal of the cross hatching are accounted for, just as with regular voids in the shape.</p> </div> <p>Note: If you select multiple shapes in this mode, the layout editor will generate a separate metal usage report for each of the shapes selected.</p> |
| <i>Include ratio table for selected layers</i> | <p>By default, this report lists (in tabular form), each layer with its metal area and the percentage of that metal area versus the area you selected. Check this box to generate an additional, tabular report at the bottom of the report which lists the ratio of metal on each layer versus the metal on each other layer in the report. The default setting for this option is <i>off</i>. A text description including the formula used for generating the values in this report is included directly in the report itself when you select this option.</p> |
| <i>Layer table</i> | <p>This table lists all layers in the design (planes, conductors, and named dielectric layers), and the soldermask top and bottom layers. Only those layers with an "X" in the first column are processed. By default, all layers are enabled. However, if you change the settings, the layout editor remembers the new settings for the next run of the report.</p> |
| <i>Write Report</i> | <p>If you check this box, the report is written to disk using the file name that you specified. The default setting is <i>on</i>. You can choose to write and view the report.</p> |
| <i>View Report</i> | <p>If you check this box, when the report is complete, it appears in a window for viewing. The default setting is <i>on</i>. You can choose to write and view the report.</p> |
| <i>Report</i> | <p>If you click this button, the layout editor generates a Metal Usage Report based on the current layer and area selections. The report is either be written to disk or opened for viewing or both, based on the current settings.</p> |
| <i>Close</i> | <p>Click this button to close the Metal Usage Report dialog box. The layout editor does not generate a report.</p> |

Generating Metal Usage Report

Perform these steps to generate a metal usage report for your designs:

1. Choose *Tools – Metal Usage Report*.

Alternately, run the `metal usage report` command at the console window prompt.

2. In the Metal Usage Report dialog box, specify the report name.
3. Choose the selection type, and then select the region in the Design Window for the metal usage density.
4. Check the *Include ratio table for selected layers box* to generate a report section comparing the metal usage on one layer to the other layers.
5. Check the box in Column 1 for each layer on which you want the layout editor to generate results.
For improved performance, you may want to turn off some layers.
6. Check the *Write Report* or *View Report* box or both.
7. Click the *Report* button to start the process.

This may take some time. If you selected multiple layers, a progress meter is shown and updated as each layer is processed.

The report provides the total metal coverage of the area, which is the sum total of all via, pin pads, clines, shapes, and so on present on the layer and the percentage coverage. It presented in a tab-delimited file for easy import and processing by Excel or another spreadsheet program.

Note: If you select multiple shapes in the *Select shape* or *Select symbol* mode, the layout editor will generate a separate metal usage report for each of the shapes selected.

1. When finished generating reports, click the *Close* button.

Related Topics

- [metal usage report](#)

mirror

The `mirror` command lets you choose between two methods of duplicating elements. *Standard Mirror* and *Mirror Geometry*.

- *Standard Mirror* relocates geometry to the opposite side of the board or substrate, occurring about the stackup.
- *Mirror Geometry* creates an element (or a group of elements) on the current subclass layer that is a mirror image of the original, occurring around the Y-axis.

This command functions in a pre-selection use model, in which you choose an element first, then right click and execute the command from the pop-up menu. Valid objects are symbols and text.

In the pre-selection use model, the command is only available if you choose a homogeneous selection set, that is, symbols and text. If you choose components and clines, for example, a warning displays for each invalid element, and the layout editor ignores it.

When a via, pin, or symbol is mirrored, the padstack is mirrored at the instance level. For example, the top pad becomes the bottom pad. The next-to-top pad becomes the next-to-bottom pad, etcetera. The `pad_designer` always shows the padstack definition, which is never shown mirrored.

Related Topics

- [Relocating Symbols to the Opposite Side of a Board or Substrate](#)
- [Mirroring Design Objects on the Same Subclass](#)

mirror Command: Options Panel

When you access the command in the pre-selection use model from the right mouse button pop-up menu, the *Options* foldable window pane is not available for you to change settings.

Access Using:

- Menu path: *Edit – Mirror*

| | |
|------------------------|--|
| <i>Standard Mirror</i> | Relocates symbols to the opposite side of the board or substrate. Using the symbol origin as the location point, the layout editor anchors the symbol at the same grid point. Any test, etch/conductor, or vias built with the symbol are mirrored, and any attached connections and ratsnest lines become dynamic rubber band lines. |
| <i>Mirror Geometry</i> | <p>Creates an element (or a group of elements) on the current subclass layer that is a mirror image of the original, around the Y-coordinate of the copy origin. Valid elements are:</p> <ul style="list-style-type: none">■ Vias (their padstacks are not mirrored)■ Connect line and line segments■ Rectangles■ Text■ Shapes |

Related Topics

- [Mirroring Design Objects on the Same Subclass](#)

Relocating Symbols to the Opposite Side of a Board or Substrate

Perform these steps to relocate symbols in your design:

1. Choose *Edit – Mirror*.
Alternately, run the `mirror` command at the console window prompt. Then the layout editor will prompt you to choose the element that you want to mirror.
2. You can also access the command from the design workspace:
 - a. Hover your cursor over the symbol to be mirrored or window select to choose several symbols. The layout editor highlights the element, and a datatip identifies its name.
 - b. Right click to choose *Mirror* from the pop-up menu.

If you are mirroring a single symbol to which connect lines are attached, ratsnest lines replace them and all ratsnest lines then become dynamic.

If you are mirroring several symbols, the layout editor prompts you to pick the origin. Click at a location as the origin of the entire group.


3. Pick the destination point for the mirrored symbols.
The symbols relocate to the opposite side of the board or substrate.

Related Topics

- [mirror](#)

Mirroring Design Objects on the Same Subclass

When you choose a die symbol or choose elements that are part of symbols, but do not choose the entire symbol, elements mirror around the Y-coordinate of the copy origin on the same subclass.

 You can also mirror elements on the same subclass by running the `copy` command, setting the *Options* foldable window pane for *Rectangular* mode, and right mouse clicking to use the *Mirror Geometry* command on the pop-up menu that displays.

Following are the steps to mirror design objects in the same subclass:

1. Hover your cursor over the symbol to be mirrored or window select to choose several symbols. The layout editor highlights the element, and a datatip identifies its name.
2. Right click and choose *Mirror* from the pop up menu.
3. Pick the destination point for the mirrored symbols.
The elements appear mirrored in their new location.
4. Click right and choose *Done* from the pop-up menu.

Related Topics

- [mirror](#)
- [mirror Command: Options Panel](#)

miter_by_pick

The `miter_by_pick` command lets you change 90-degree wire corners to 45 degrees for wires exiting pins and vias.

Access Using:

- Menu path:
 - *Route – PCB Router – Miter by Pick* (Allegro X PCB Editor, Allegro X PCB EditorSI)
 - *Route – Router – Miter by Pick* (APD)

Mitering Corners

Perform these steps to miter corners:

1. Choose *Route – PCB Router – Miter by Pick* if you're working on Allegro X PCB Editor.
If you are working with Allegro X Advanced Package Designer (APD), then the menu path will be *Route – Router – Miter by Pick*.
Alternately, run the `miter_by_pick` command.
2. Right-click to display the pop-up menu and choose *Setup*.
The Automatic Router Parameters dialog box appears with the Miter Corners tab selected.
3. Make your selections. For additional information, see the [Miter Corners](#) tab in the description of the Automatic Router Parameters dialog box.
4. Click *OK* to save the changes and dismiss the dialog box.
5. Click on a net or a group of nets.
The 90-degree wire corners change to 45 degrees.
6. Choose one of the options from the pop-up menu, as described below:

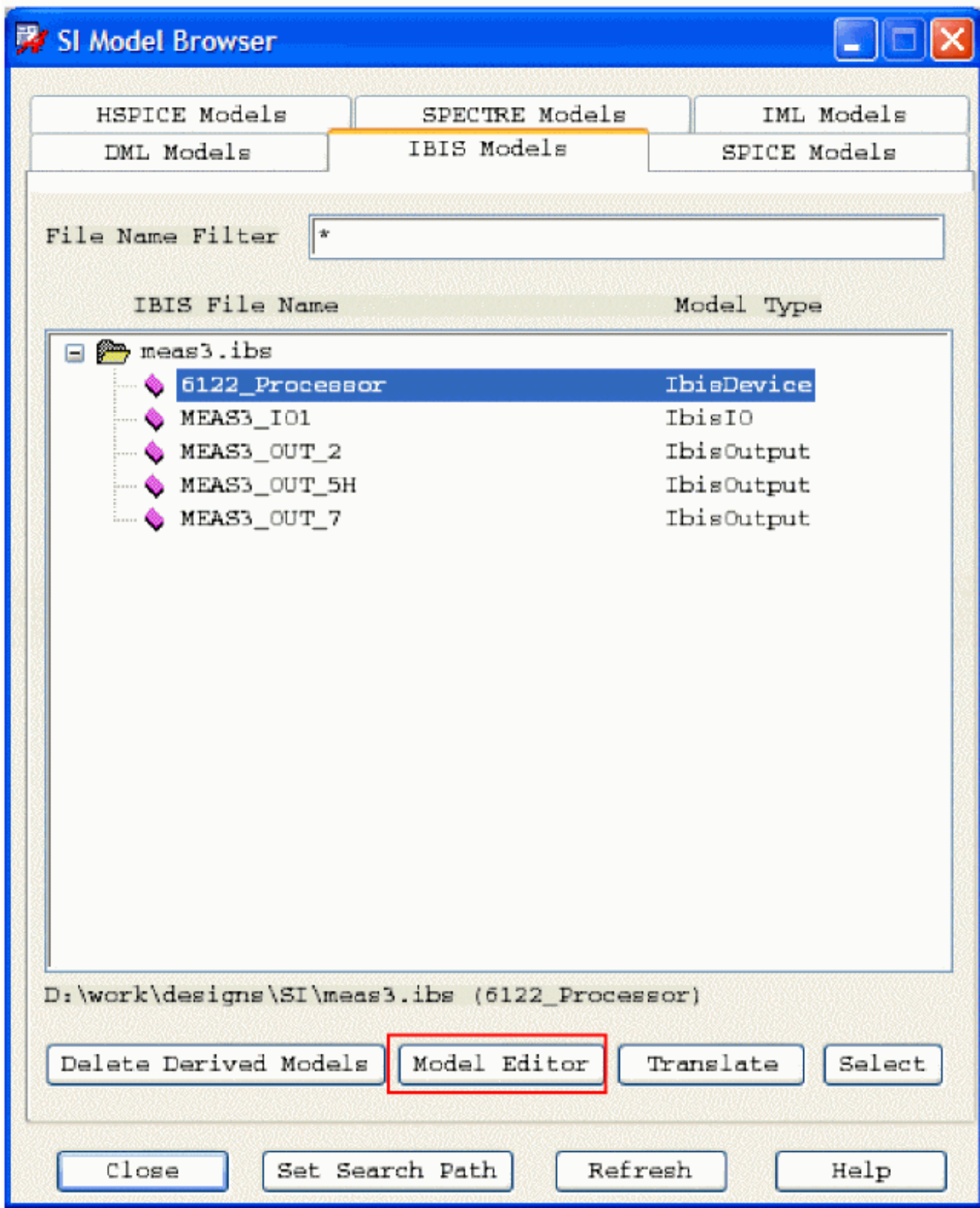
| | |
|-------------------|---|
| <i>Done</i> | Terminates the command, saving any routing performed while the command was active. |
| <i>Oops</i> | Removes the results of the last route. |
| <i>Cancel</i> | Terminates the command without saving any routing. |
| <i>Temp Group</i> | Enables you to route groups of connections. |
| <i>Complete</i> | Completes the selection of the items to group. |
| <i>Setup</i> | Opens the Automatic Router Parameter dialog box. |
| <i>Results</i> | Opens the routing results form to display the results of the current routing session. |

Related Topics

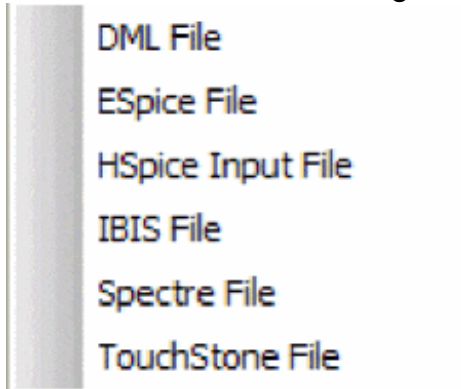
- [Automatic Router Parameters](#)

model editor

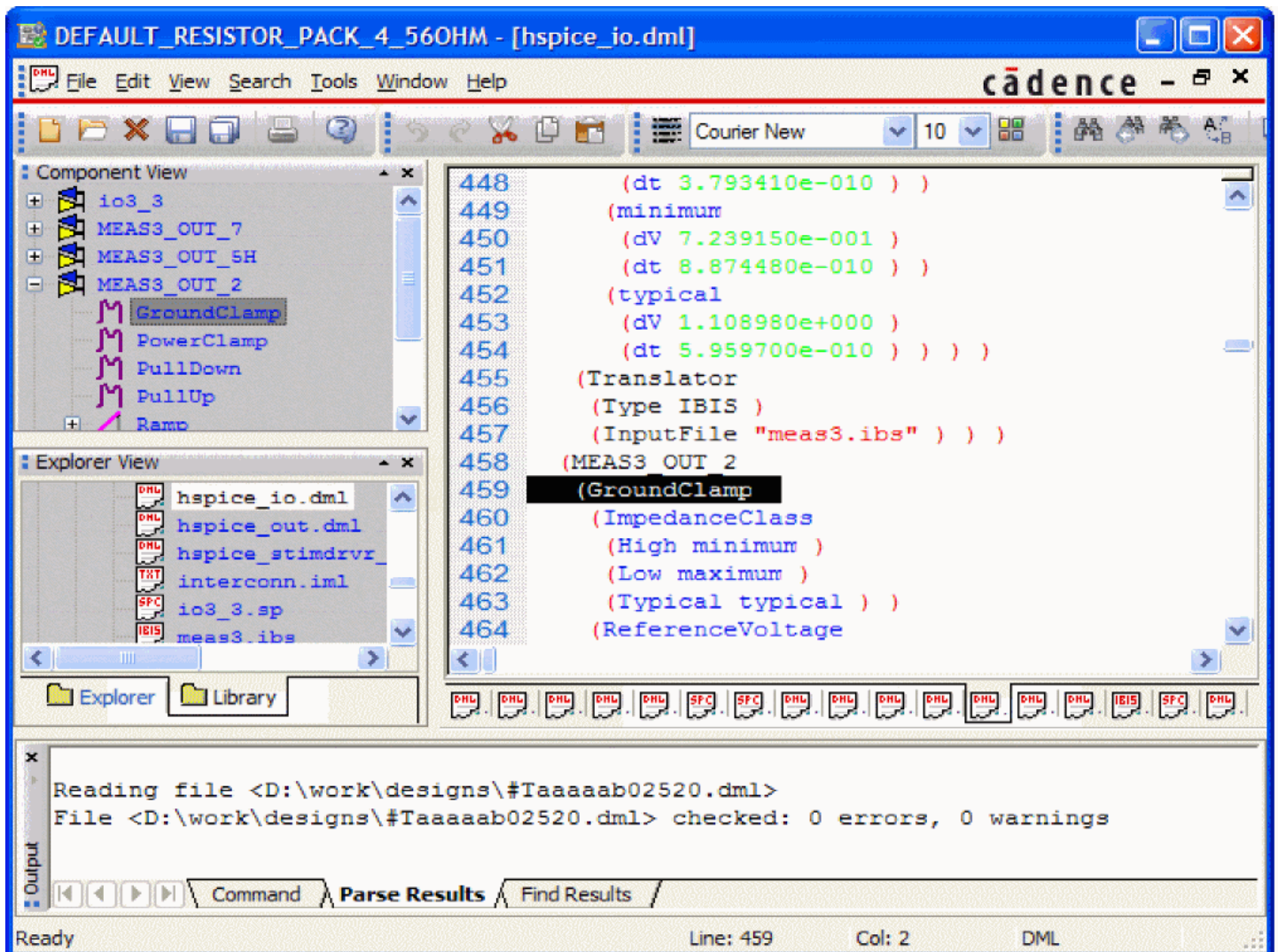
The `model editor` command assists in reviewing and validating models that you create or edit. When you make a selection in SI Model Browser for any of the models generated by the translation process and click the *Model Editor* button at the bottom of the SI Model Browser, the model is opened in its native format for editing in the Model Editor window.



Model Editor is a high-speed design editing tool that helps you ensure the integrity of the model data required for high-speed circuit simulations. It allows you to create, manipulate, and validate models quickly in an easy-to-use editing environment. Model Editor provides a model browser and syntax checker (parser) for models written in IBIS as well as for advanced models written in Cadence's device modeling language, DML. The following device model formats are supported:



Model Editor has color-coded keywords and has the complete model and all of its sub-models included in the Component View.

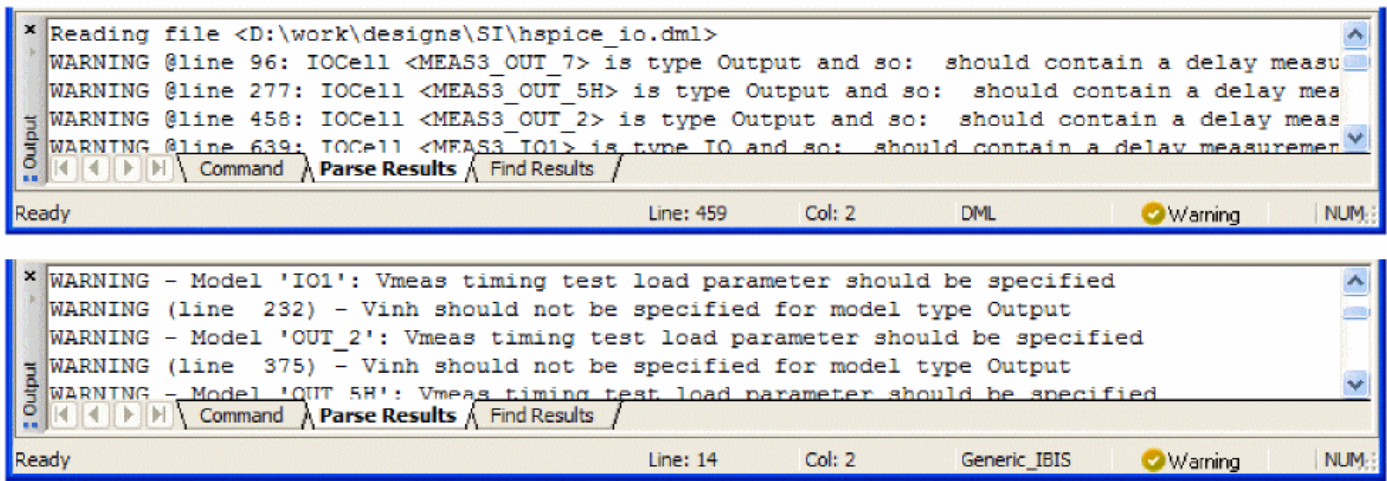


You can use the Component View to navigate to a specific model or subsection of a model.

Features of Model Editor

In addition to opening and editing model files in Model Editor, you can also parse model files to determine syntax errors. When you open a valid model file in Model Editor, it is automatically parsed using the parser appropriate for the file type.

You can also request to parse an open file at any time while you work with it in Model Editor. To do so, choose *Tools – Parse* in the Model Editor window. When it completes parsing the file, Model Editor displays errors or warnings in the Output window to mark any syntactical problems encountered within each model object contained in the file.



model integrity

For more information, see the *Model Integrity Command Reference*.

mod padstack

The `mod padstack` command lets you choose one or all padstack instances from your design for modification.

This command functions in a pre-selection use model, in which you choose an element first, then right-click and execute the command. Elements ineligible for use with the command generate a warning and are ignored. Valid objects are pins and vias.

Related Topics

- [padeditdb](#)

Editing Design Padstacks

Following are the steps to edit design padstacks:

1. Hover your cursor over the pin or via whose padstack you want to modify. The layout editor highlights the element and a datatip identifies its name.
2. Right click and choose *Modify Design Padstack* from the pop-up menu.
3. Choose one of the options from the pop-up menu, as described below:

| | |
|------------------------|---|
| <i>Single Instance</i> | Choose to modify one padstack instance. |
| <i>All Instances</i> | Choose to modify the padstack definition. |

4. The Padstack Designer opens and loads the padstack that is assigned to the pin or via you chose.
5. Specify the padstack parameters and layers as described in the section [Padstack Designer](#).

Related Topics

- [padeditdb](#)
- [Padstack Designer](#)

modpaste

Obsolete and no longer supported.

mosaic_cmd

Internal command.

mouse_pos

The `mouse_pos` command has been added to scripts to forcibly update the rubber band and cursor buffer dynamics used in some etch edit commands, thereby ensuring that script replay results are identical to those obtained during the recording phase. It is also useful to record a script as part of a test case to reproduce problems with the dynamics display update; for this purpose, use the command without input coordinates.

The x and y coordinates are optional and if not provided, a `mouse_pos` command is scripted using the current mouse position (that is, cause the script replay to have an dynamics update at this database coordinate).

```
mouse_pos [x coordinate] [y coordinate]
```

| | |
|----------------------------|--|
| x/y coordinates | In database units. Simulates the user moving the mouse to the specified location, updating the rubber band and cursor buffer dynamics accordingly. Prompts for a y coordinate only if you enter an x coordinate. |
|----------------------------|--|

move

The `move` command relocates the position of elements in a design. This command functions in a pre-selection use model, in which you choose an element first, then right click and execute the command from the pop-up menu. Valid objects are:

- Groups
- Symbols
- Pins
- Vias
- Clines
- Lines
- Shapes
- Figures
- Text
- Rat Ts
- Cline Segs (only if the environment variable `mv_cline_segs` is set). After moving the cline segments, check the design to ensure that the connectivity is maintained. You might be required to do some manual routing.

During the move process, the command highlights and displays the new location of the rats dynamically. This behavior avoids unnecessary picks and ensures placement optimization. Displaying rats dynamically is a default behavior, however, does not applied for the following objects in a design:

- User-defined and system-defined net schedules
- Power and ground nets
- Nets with pin-count greater than 20
- Components with pin-count greater than 100
- Net that has fixed connection between pins of two components
- Net shared by multiple pins of a single component

To disable this behavior, set the environment variable *no_dynamic_ratsnest* in the *User Preferences Editor* dialog box. This restores the behavior where rats are displayed as elastic rubber bands in dynamic mode. The `move` command also allows you to interactively change the grid definition and move on-grid/off-grid objects in-line with other objects. The *Relative Grid* option modifies the X and/or Y grid values that are either based on the selected object(s) origin or an alternate relative grid origin. This option is active during the `move` command and reverts to default settings when the command ends. To aid component alignment and placement with already placed components while moving, the command provides option to enable dynamic component alignment behavior. When enabled, align guidelines appear during move process that are configured for either component origin or place bound edges or for both. For performance reasons, the `move` command does not force a shape update until the placement pick.

Using the move command with Allegro X Advanced Package Designer (APD)

When APD detects that it is moving a die or BGA symbol in possession of an IC group, it moves everything associated with the die as a single object instead of moving just the die bump symbol. This means that any tiles and via structures that also belong to the die move along with the symbol. You can move die and BGA elements only in the APD.

Related Topics

- [Move Command: Options Panel](#)
- [Moving One or More Elements](#)
- [Moving Elements by Incremental Distance](#)
- [Moving RF Clearance Assembly Group](#)
- [Moving Objects Relative to Other Objects](#)
- [Preparing the Layout](#)

When working with this command, you can right-click in your design canvas to display the pop-up menu and choose the following options

Access Using:

- Menu path: *Edit – Move*
- Toolbar icon:



| | |
|------------------------|---|
| <i>Done</i> | Commits the current action and returns the editor to the idle state. |
| <i>Oops</i> | Reverses the action of the last pick. |
| <i>Cancel</i> | Reverses results of the current route and returns the editor to the idle state. |
| Persistent select | Specify the selection mode (Select by Polygon, Select by Lasso, and Select on Path) for selecting multiple objects |
| Select by Polygon | Lets you choose multiple elements by drawing a polygon around the elements during an editing session. |
| Select by Lasso | Lets you choose multiple elements by drawing a open loop path around the elements during an editing session. |
| Select by Path | Lets you choose multiple elements by drawing a straight line path during an editing session. |
| Temp Group | Lets you to choose multiple elements for simultaneous editing. |
| Reject | Lets you deselect and dehighlight an element(s) selected during the current interactive command, continues the find process at the same location selected, and highlights the next element found. |
| <i>Alt Symbol</i> | Lets you cycle through a list of alternate package symbol names that can be substituted for the primary package symbol. |
| <i>Mirror Geometry</i> | Creates an element (or a group of elements) on the current subclass layer that is a mirror image of the original, occurring around the Y-axis. |

| | |
|-------------------------|--|
| <i>Mirror</i> | Available only when you have chosen a symbol instance or module. Relocates symbols to the opposite side of the board or substrate. Using the symbol origin as the location point, the layout editor anchors the symbol at the same grid point. Any test, etch/conductor, or vias built with the symbol are mirrored, and any attached connections and ratsnest lines become dynamic rubber band lines. |
| Set Rotate Angle | Specify angle for rotation. |
| <i>Rotate</i> | Turns an element or group of elements around an axis. |
| <i>Change User Pick</i> | Available only when <i>Rotation Point</i> is set to <i>User Pick</i> . Allows you to quickly choose a new move origin as an aid in placement. |
| <i>Options</i> | Displays all parameters relevant to the command from the right mouse button instead of the <i>Options</i> foldable window pane. Changing a parameter here automatically updates its value on the <i>Options</i> foldable window pane. |
| <i>Snap pick to</i> | Snaps to destinations other than the grid. For additional information, see the <i>Getting Started with Physical Design</i> user guide in your documentation set. |

Related Topics

- [Moving One or More Elements](#)
- [Moving Elements by Incremental Distance](#)
- [Moving RF Clearance Assembly Group](#)
- [Moving Objects Relative to Other Objects](#)

Move Command: Pop-Up Menu Options

When working with this command, you can right-click in your design canvas to display the pop-up menu and choose the following options

Access Using:

- Menu path: *Edit – Move*
- Toolbar icon:

| | |
|------------------------|---|
| <i>Done</i> | Commits the current action and returns the editor to the idle state. |
| <i>Oops</i> | Reverses the action of the last pick. |
| <i>Cancel</i> | Reverses results of the current route and returns the editor to the idle state. |
| Persistent select | Specify the selection mode (Select by Polygon, Select by Lasso, and Select on Path) for selecting multiple objects |
| Select by Polygon | Lets you choose multiple elements by drawing a polygon around the elements during an editing session. |
| Select by Lasso | Lets you choose multiple elements by drawing a open loop path around the elements during an editing session. |
| Select by Path | Lets you choose multiple elements by drawing a straight line path during an editing session. |
| Temp Group | Lets you to choose multiple elements for simultaneous editing. |
| Reject | Lets you deselect and dehighlight an element(s) selected during the current interactive command, continues the find process at the same location selected, and highlights the next element found. |
| <i>Alt Symbol</i> | Lets you cycle through a list of alternate package symbol names that can be substituted for the primary package symbol. |
| <i>Mirror Geometry</i> | Creates an element (or a group of elements) on the current subclass layer that is a mirror image of the original, occurring around the Y-axis. |

| | |
|-------------------------|--|
| <i>Mirror</i> | Available only when you have chosen a symbol instance or module. Relocates symbols to the opposite side of the board or substrate. Using the symbol origin as the location point, the layout editor anchors the symbol at the same grid point. Any test, etch/conductor, or vias built with the symbol are mirrored, and any attached connections and ratsnest lines become dynamic rubber band lines. |
| Set Rotate Angle | Specify angle for rotation. |
| <i>Rotate</i> | Turns an element or group of elements around an axis. |
| <i>Change User Pick</i> | Available only when <i>Rotation Point</i> is set to <i>User Pick</i> . Allows you to quickly choose a new move origin as an aid in placement. |
| <i>Options</i> | Displays all parameters relevant to the command from the right mouse button instead of the <i>Options</i> foldable window pane. Changing a parameter here automatically updates its value on the <i>Options</i> foldable window pane. |
| <i>Snap pick to</i> | Snaps to destinations other than the grid. For additional information, see the <i>Getting Started with Physical Design</i> user guide in your documentation set. |


Related Topics


- [Moving One or More Elements](#)
- [Moving Elements by Incremental Distance](#)
- [Moving RF Clearance Assembly Group](#)
- [Moving Objects Relative to Other Objects](#)

Move Command: Options Panel

In addition to setting parameters relevant for this command on the *Options* foldable window pane, you may also set them by right clicking to display the pop-up menu from which you may choose *Options*. Changing a parameter using either location automatically updates the other.

| | |
|----------------------|--|
| <i>Ripup etch</i> | Rips up any connection elements to the closest pin, T connection, or via, and unsets <i>Stretch etch</i> and <i>Slide etch</i> . If you do not choose this field, all connections associated with the element on the board/substrate remain as dangling lines. If the element is a package/part symbol with connect lines to any pins and you enable <i>Ripup etch</i> , the connect lines erase and become dynamic rubber bands. |
| <i>Slide etch</i> | Slides any connection elements to the closest pin, T connection, or via, and unsets <i>Stretch etch</i> and <i>Slide etch</i> . |
| <i>Stretch etch</i> | Rips up the first line segment of any connection attached to the element and adds an odd angle segment between the rotated element and the rest of the connection. If you enable <i>Stretch etch</i> , the segments that connect to the symbol/via appear as rubber bands and the rubber band lines go to the other end of the segment that was erased. If you enable <i>Stretch etch</i> , the rubber band lines appear from the segment entering the pin. Otherwise, the rubber band lines appear from the connecting pin. If you disable <i>Stretch etch</i> , the ratsnest lines rubber band (for example, from pin to pin on another symbol). |
| <i>Rotation Type</i> | Determines the rotation type: <ul style="list-style-type: none">• <i>Absolute</i> rotates the element once to place it at the angle specified in the <i>Angle</i> field.• <i>Incremental</i> provides a dynamic handle for controlling the element. It uses the number in the <i>Angle</i> field as the amount by which to increment the element as you rotate it. |

| | |
|------------------------------|---|
| <p><i>Rotation Angle</i></p> | <p>Determines the angle of rotation, but has a different meaning depending on the rotation mode:</p> <ul style="list-style-type: none"> • For <i>Incremental</i> mode, <i>Angle</i> specifies how many degrees comprise each increment as you rotate the element. • For <i>Absolute</i> mode, <i>Angle</i> specifies the final degree of rotation from the 0, 0 orientation. <p>When you execute the command, the element immediately rotates to that angle. You can enter a number between 0 and 360, or you can choose one of the following numbers from the pop-up menu: 0, 45, 90, 135, 180, 215, 270, and 315. Accuracy is up to three decimal places.</p> <div style="border: 1px solid #fde725; padding: 10px; margin-top: 10px;"> <p> Rotating objects on non-orthogonal angles may round off the values you defined for spacing constraints, which may result in disconnects and DRCs.</p> </div> |
| <p><i>Rotation Point</i></p> | <p>Indicates the anchor point around which the element turns. <i>Sym Origin</i> is the 0, 0 point of the element (symbol).</p> |
| | <p><i>Body Center</i>: Specifies the point at the center of an invisible boundary that the program draws around the edge of the element.</p> |
| | <p><i>User Pick</i>: Specifies a mouse click or typed coordinates that indicate the point and causes the <i>Change User Pick</i> option to display on the right-mouse-button popup menu.</p> |
| | <p><i>Symbol Pin Number</i>: Invokes a field where you enter the number. The <i>Symbol Pin #</i> field appears only when you choose <i>Sym Pin Number</i> as the rotation point and where you enter a pin number.</p> |
| <p><i>Relative Grid</i></p> | <p>Re-centers the grid to the origin of the selected object(s). This option is enabled only when <i>Rotation Point</i> is set to <i>Sym Origin</i>.</p> |
| | <p><i>Spacing X</i>: Specify the length of the grid in the X (horizontal) direction.</p> |
| | <p><i>Spacing Y</i>: Specify the length of the grid in the Y (vertical) direction.</p> |
| | <p><i>Alternate Origin</i>: Click to choose an alternate grid origin. This option can also be set from pop-up menu <i>Set Relative Grid Origin</i>. To restore the default origin click <i>Restore Relative Grid Origin to Default</i> option.</p> |

| | |
|--------------------------|--|
| <i>Dynamic Alignment</i> | <p>Displays alignment guidelines in both the horizontal and the vertical directions that match with placed components during component movement. By default, the guidelines are configured for component origin. To change the configuration, use <i>Preferences</i> button. If this option is checked, can be disabled during the move operation by unchecking from pop-up menu, <i>Options – Dynamic Alignment – Enable</i>.</p> <div> The alignment guides are not visible for non-orthogonally placed components.</div> |
| <i>Preferences</i> | <p>Invokes the <i>Display – Align Guides</i> section in the <i>User Preferences Editor</i> dialog box for enabling the behavior and setting the configuring guidelines. Options available as environment variables are:</p> <ul style="list-style-type: none">• <i>align_guides</i>: If enabled, displays align guides during the move operation. By default, it is off.• <i>align_guides_component_origin</i>: If enabled, aligns the origins during the move operation. By default, it is on.• <i>align_guides_place_bound</i>: If enabled, aligns the place bound extent of the components during the move operation. By default, it is off.• <i>align_guides_ratsnest</i>: If enabled, aligns ratsnest between two off-grid pins during the move operation. By default, it is on. |

Related Topics

- [move](#)
- [Moving Elements by Incremental Distance](#)
- [Moving RF Clearance Assembly Group](#)
- [Moving Objects Relative to Other Objects](#)

Moving One or More Elements

Perform these steps to move a single element or multiple elements in a design:

1. Choose *Edit – Move*.

Alternately, run the `move` command from the command line prompt. The layout editor then asks you to select the element or elements to be moved.

2. Another way to access this command is as follows:


- a. Hover your cursor over an element or window select a group of elements. The layout editor highlights the element.
- b. Double click the element(s) to activate the command.

When you choose multiple elements, they move (and rotate) as if they were a single unit.

When you specify an origin, that point is the origin for the entire unit.

The element attaches to your cursor, any attached etch/conductor or ratsnest lines become dynamic rubber band lines. As you move, the rats are dynamically updated to their new location. You have dynamic control over the element's movement until you click a location in the design or type its coordinates.

3. Right-click and choose *Options* from the pop-up menu, or in the *Options* panel, complete parameters as required.

 Choosing `rotate` from the pop-up menu suspends the command until you rotate the element; then you can move the element at the new rotation, until you pick another location.

4. Optionally, enable *Dynamic Alignment* option.

Align guidelines becomes visible and dynamically changes as you move the elements.

5. Click to choose the location to which to move the element.

The element appears at its new location.

If any element had connected lines to other elements, the connected lines become permanently deleted, stretched and connected, or left alone depending on how the *Ripup Etch/Conductor* and *Stretch Etch/Conductor* fields are set.

Related Topics

- [move](#)
- [Move Command: Pop-Up Menu Options](#)
- [Moving RF Clearance Assembly Group](#)
- [Moving Objects Relative to Other Objects](#)
- [Move With Dynamic Alignment ON](#)

Moving Elements by Incremental Distance

You can move elements incrementally in two ways:

- Using ix or iy Coordinates
- Using Relative Grid Option

Using ix or iy Coordinates

To move any element relatively or incrementally you can use ix and iy to specify the relative distance. To move an element in the X direction do following steps:

1. Hover your cursor over an element or window select a group of elements. The layout editor highlights the element. When you choose multiple elements, they move (and rotate) as if they were a single unit. When you specify an origin, that point is the origin for the entire unit. The element attaches to your cursor, any attached etch/conductor or ratsnest lines become dynamic rubber band lines. As you move, the rats are tonically updated to their new location. You have dynamic control over the element's movement until you click a location in the design or type its coordinates.
2. At the command prompt, type the command `move` or use the menu selection *Edit – Move*.
3. Right-click and choose *Options* from the pop-up menu, or in the *Options* panel, complete parameters as required.
4. Select an origin for the move. The origin will not matter as you are moving the element(s) relative to their location.
5. When you get the command prompt to pick new location for the element(s): type ix 100. This will move the element by 100 database units in the positive X-direction.
If you give the new location for the element(s) as ix -100. It will move the element by 100 database units in the negative X-direction.

Using Relative Grid Option

To move any element incrementally using Relative Grid option, do following steps:

1. At the command prompt, type the command `move` or use the menu selection *Edit – Move*.
2. In the *Options* panel, ensure *Rotation Point* is set as *Sym Origin*.
3. Enable *Relative Grid* option and specify the X and Y spacing values.

Grids are dynamically updated.

4. In *Find* filter, select the design objects to move.
5. Hover your cursor over an element or window select a group of elements.
6. Click to specify an origin for the move.
7. Move the object(s) incrementally inline from the original position.
8. Click to choose the location to which you want to place the element.
The selected objects appears at its new location.

Related Topics

- [move](#)
- [Move Command: Pop-Up Menu Options](#)
- [Move Command: Options Panel](#)
- [Moving One or More Elements](#)
- [Moving Objects Relative to Other Objects](#)

Moving RF Clearance Assembly Group

Following are the steps to move an RF Clearance Assembly group:

1. In the *Find* from enable *Groups*.
2. Hover your cursor over an RF clearance assembly group or window select a group of RF elements.
3. Right-click and choose *Move* from the pop-up menu.
Alternately, type the command `move` in the command line prompt or use the menu selection *Edit – Move*.
4. Enter the desired parameters in the *Options* panel or right-click and choose *Options* from the pop-up menu.
5. Click to choose the location to which to move the RF clearance assembly group.
The RF clearance assembly group appears at its new location.

Related Topics

- [move](#)
- [Move Command: Pop-Up Menu Options](#)
- [Move Command: Options Panel](#)
- [Moving One or More Elements](#)

Moving Objects Relative to Other Objects

To move any element relative to another object at a certain distance do the following steps:

1. At the command prompt, type the command `move` or use the menu selection *Edit – Move*.
2. In *Find* filter, select the design objects to move.
3. Hover your cursor over an element or window select a group of elements. The layout editor highlights the element.
The element attaches to your cursor, any attached etch/conductor or ratsnest lines become dynamic rubber band lines. As you move, the rats are tonically updated to their new location. You have dynamic control over the element's movement until you click a location in the design or type its coordinates.
4. Right-click and choose *Options* from the pop-up menu, or in the *Options* panel, enable *Relative Grid* and specify the X and Y spacing values.
Grids are dynamically updated.
5. Click *Alternate Origin* in the *Options* panel. Click any point in the design canvas.
Grid is temporarily changed to the new alternate origin.
6. Click to choose the location to which you want to place the element.
The selected objects appears at its new location.

Related Topics

- [move](#)
- [Move Command: Pop-Up Menu Options](#)
- [Move Command: Options Panel](#)
- [Moving One or More Elements](#)
- [Moving Elements by Incremental Distance](#)

move vertex

The `move vertex` command repositions an existing vertex.

This command functions in a pre-selection use model, in which you choose an element first, then right click and execute the command from the pop-up menu.

Prior to using the command, set relevant parameters in the *Edit Vertex* section of the *Route* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* ([prmed](#) command). You may also set them by right clicking to display the pop-up menu from which you may choose:

- Design Parameters to access the Design Parameter Editor
- Options

Changing a parameter using either of these pop-up menu choices automatically updates the Options foldable window pane as well.

Valid elements are cline segments and other segments.

Related Topics

- [Moving a Vertex](#)
- [prmed](#)

Move Vertex Command: Options Panel

When you access the command in the pre-selection use model from the right mouse button pop-up menu, the *Options* foldable window pane is not available for you to change settings.

| | |
|----------------------------------|--|
| <i>Active Class and Subclass</i> | The upper drop-down list box displays the current class; the lower drop-down list box, the current subclass with choices for modifying the value. |
| <i>Net</i> | Identifies the net assigned to the element you select. If no net is assigned, the value is NULL NET. To the left of the field name is an indicator for the nets. When you are routing a single net, the indicator shows one net. When you are performing differential pair routing, the indicator shows two nets. If you are performing group routing, the indicator shows multiple nets. If you are in single trace mode in differential pair routing or group routing, the indicator shows only the control trace highlighted. The field always shows the net name of the control trace (for both differential pairs or single traces) and never the differential pair name. |
| <i>Bubble</i> | Controls any automatic bubbling (moving of existing connections) to resolve DRC errors with the following options: <i>Off</i> : The clines you route start at the location you indicate, and no bubbling occurs. DRC flags all clearance violations with error markers. <i>Hug Only</i> : Where possible, the routed cline contours around other etch/conductor objects to avoid spacing DRCs. Other etch/conductor remains unchanged. <i>Hug Preferred</i> : Where possible, the routed cline contours around other etch/conductor objects to avoid spacing DRCs. If not possible, the layout editor tries shoving other etch/conductor objects to open routing paths. Note : This method is more aggressive than <i>Hug Only</i> . <i>Shove Preferred</i> : Where possible, the routed cline pushes and shoves other etch/conductor objects to avoid spacing DRCs. If not possible, the layout editor attempts to hug other etch/conductor objects. |
| <i>Shove vias</i> | Allows the bubble functionality in shove mode to move vias when you are editing etch/conductor. It is only active when <i>Bubble</i> is enabled. The following are the options <i>Full</i> : Vias are shoved in a shove-preferred manner. Any new or edited etch/conductor always shoves vias out of the way. <i>Minimal</i> : Vias are shoved in a hug-preferred manner. Vias are not moved unless there is no way to draw a connect line around them. <i>Off</i> : Vias are not shoved |

| | |
|-------------------------------------|--|
| <i>Clip dangling clines</i> | Active for shove-preferred mode and controls whether the layout editor clips back dangling clines to fix DRC errors. When disabled, the dangling cline endpoints remain unchanged, and the layout editor corrects the DRC errors, if possible, by bubbling the new cline around the dangling endpoints (similar to hug-preferred mode). |
| <i>Smooth</i> | Active when you set the <i>Bubble</i> field to hug- or shove-preferred mode and controls whether smoothing occurs on the cline to minimize segments between the start and finish points. Smoothing occurs dynamically as you move the mouse on cline segments close to the segment you selected. Performance with the <i>Smooth</i> option active may be somewhat slower than when it is inactive. These are the choices: <i>Minimal</i> : Executes dynamic smoothing to minimize unnecessary segments. <i>Full</i> : Executes more extensive smoothing to remove any unnecessary jogs. <i>Off</i> : Disables smoothing. Note: Full smoothing does not smooth the cline you are adding back to its source. Rather, it smooths the newly created etch/conductor back to your last pick. Additionally, parts of other clines that are shoved during this procedure may also be smoothed. |
| <i>Allow DRCs</i> | Specifies that design rules can be violated to make a connection. If <i>Bubble</i> is disabled, the vertex is set at a point between the last good point and the current point that does not cause a DRC error. |
| <i>Allow Gridless</i> | Specifies that the etch/conductor can go off the routing grid. Gridless routing lets the layout editor add connections at maximum density while accommodating varying design rules and line widths. The DRC minimum space separates objects. When <i>Bubble</i> is disabled, the <i>Allow Gridless</i> field controls the removal of a small segment at the end of the new route when in <code>add connect</code> mode. Normally, if the last segment is small, the layout editor does not add it (to avoid adding a little jog). If <i>Allow Gridless</i> is off, the layout editor adds the segment. |

Moving a Vertex

To move a vertex, follow these steps:

1. Hover your cursor over the vertex to move. The layout editor highlights the element and a datatip identifies its name.
2. Right click and choose *Move Vertex* from the pop-up menu.



The vertex cursor appears when you hover over a vertex.



3. The console window prompt displays the following message:
Pick destination of vertex
4. Position the vertex in its 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.

Related Topics

- [move vertex](#)

movewindow

The `movewindow` command relocates and resizes the position of your Cadence user interface on your monitor. You enter the command name followed by two numbers (representing pixels) at the command console prompt of your user interface.

```
movewindow <1st number> <2nd number>
```

Positioning the Window

1. Type `movewindow` followed by two numbers representing the left/right and up/down directions. Use of the minus (-) sign before a number is supported. The user interface relocates or resizes accordingly.


muconnect

The `muconnect` command connects clients to Symphony Server application. This command launches the *Symphony* dialog box to query Symphony Server applications and databases, and has options to specify team design settings at the client end.

When connected, clients can view other users and the objects locked by them. The summary of team design session and user-activities are available in the log tab.

 This command is available in the following layout editors:

- Allegro X PCB Editor
- Allegro X Advanced Package Designer (APD)
- Allegro Sigrity SI

 This command is available with the *Symphony Team Design* option.

Related Topics

- [Connecting to Symphony Server Application](#)
- [Working with Symphony Team Design](#)
- [muserver](#)
- [muservermgr](#)



Symphony Dialog Box


Use this dialog box to connect clients to Symphony Server application.


Access Using:

- Menu path: *File – Symphony Connect*

| | | |
|-----------------|---|--|
| <i>Connect</i> | | |
| | <i>Host Name</i> | Specifies host name or IP address to connect and query the Symphony Server application for the databases available for team design. By default, <code>localhost</code> is set. |
| | <i>Query Host</i> | Queries specified host name and provides list of databases available for team design. |
| | <i>User Name</i> | Displays name of the current users. |
| | <i>Database</i> | Displays list of available databases. |
| Server Commands | Available when the same user has started the server application. | |
| Save | Saves the current database connected to the Symphony Server application. | |
| Close | Closes the Symphony Server application and prompts to save changes at the server end. The clients are automatically disconnected from the server application. | |
| <i>Users</i> | Displays list of users connected to the Symphony Server application and working on the selected database. | |
| <i>Locks</i> | Displays canvas locks, generated on selected objects by all the clients. Hovering over a lock provides the locking information (user name, user ID, IP, and host name) as data tip. Right-click to choose options to unlock, display, hilight, and dehilight the lock. | |
| <i>Log</i> | Displays user activities as well as summary of database check results. Hovering over a log entry provides the timestamp as data tip. | |

| <i>Options</i> | | |
|----------------|---|---|
| | <i>Query Port Range</i> | <p>Specifies the port range available for server to client communication.</p> <p>The default range is 4000 - 4005.</p> |
| | <i>Timeout (ms)</i> | <p>Specifies the time limit for querying databases available for team design.</p> <div style="border: 1px solid #f9e79f; padding: 10px; margin-top: 10px;"> <p> Reducing the <i>Timeout</i> below the default value of 5000 ms may cause the shared database from not being seen even if it is available for connection.</p> </div> |
| | <i>Maximum Recent Hosts</i> | <p>Specifies the number of recent hosts available in the Host Name pull-down list of the Connect tab.</p> |
| | <i>Show Own Command Locks</i> | <p>Suppresses the canvas lock display generated while selecting objects in an active command.</p> <p>By default this option is disabled.</p> <div style="border: 1px solid #f9e79f; padding: 10px; margin-top: 10px;"> <p> Locks are generated on selecting objects and prevent other clients to edit the selected objects.</p> </div> |
| | <i>Disable On-Line DRC for Incoming Updates</i> | <p>Disables DRC updates on integrated server changes for a performance improvement.</p> |
| | <i>Display Temp Updates</i> | <p>Displays updates from other users before they are integrated and accepted by the server application.</p> |

| | | |
|--|---|---|
| | <i>Display On-Line DRC for Incoming Updates</i> | Disables DRC updates on integrated server changes for improved performance. |
| | <i>Process Incoming Updates Between Cmds</i> | Allows integrating updates in the database when the active command is completed by switching between commands without performing a <i>Done</i> . |
| | <i>Display Peer Cursors</i> | Displays cursors of other clients on the canvas. This option is disabled by default. |
| | <i>Peer Cursor Size (pixels)</i> | <p>Displays size of the cursor in pixels from other clients. The default value is 10 pixels.</p> <div style="border: 1px solid #d4edda; padding: 10px; margin-top: 10px;">  The cursor size depends on the resolution set for the display. The default value may need to be adjusted. </div> |
| | <i>Send Cursor Information</i> | Send the cursor location to other clients. By default, this option is enabled. |
| | <i>Run Directory</i> | Specifies the working directory of the client where default client files (journal, reports, logs, and so on) are saved. A sub-folder is automatically generated matching the database name in the <i>Run Directory</i> location. By default, the directory location is set to <code>pcbenv/Symphony/<design_name></code> . Once connected to Symphony database, the title bar displays the <i>Run Directory</i> location. |
| | <i>Browse</i> | Specifies the client's working directory. |
| | <i>Default</i> | Reverts to default settings. |
| | <i>Reset</i> | Reverts to previously saved settings. |
| | <i>Apply</i> | Saves the changes made to the Symphony window. |

| | |
|------------|---|
| Connect | Connects to the selected database. |
| Refresh DB | <p>Refreshes database using the active copy available on the server application.</p> <div> This option is normally not required as database updates are sent to each client automatically so that all the users always stay in sync.</div> |
| Disconnect | Disconnects the active database from the Symphony Server application and returns to the standalone layout editor application. |

Related Topics

- [Working with Symphony Team Design](#)

Connecting to Symphony Server Application

To connect your layout editor to Symphony Server applications, follow these steps:

1. Run the *muconnect* command.
The *Symphony* window opens.
2. In the *Connect* tab, enter host name of the machine on which Symphony Server application is running.
3. Query host name to display list of available databases visible under a particular TCP Port on the host machine.
4. In the *Options* panel of the *Symphony* window, change the settings, if required, and click *Apply* to update the settings.
5. Select the database and click *Connect*.
The database is loaded from the server in the layout editor. The menus and toolbars are updated and show only Symphony team design environment functionality. The name (Symphony) is added to the title bar.

Related Topics

- [muconnect](#)

mulpadedit

The `mulpadedit` command lets you modify individual pad shapes or multiple instances of one pad shape.

Related Topics

- [Modifying Pad Shapes](#)


Multpadedit Command: Options Panel

| | |
|-----------------|---|
| <i>Geometry</i> | Specifies the standard shape of the pad you want to modify. You have a choice of Null (no shape), Circle, Square, Oblong, Rectangle, or Shape (custom pad). |
| <i>Width</i> | Specifies the width of the pad you want to modify if it is a square, oblong, or rectangle. If the pad is a circle, enter the diameter of the circle. In the unit of measurement that you set for the padstack. |
| <i>Height</i> | Specifies the height of the pad you want to modify if it is a square, oblong, or rectangle. If the pad is a circle, enter the diameter of the circle. In the unit of measurement that you set for the padstack. |
| <i>Layer</i> | Specify the layer for the pad you want to modify. |

Modifying Pad Shapes

To modify pad shapes in your design, follow these steps:

1. Specify the padstack shape that you want to modify in the Options foldable window pane.

 Enter an asterisk (*) in the *Geometry*, *Width*, and *Height* fields on the *Options* foldable window pane to select pads.


2. Modify the selected shape by specifying a new shape, size, layer, pad type or any combination of these characteristics.
3. Choose the pads that you want to edit in your drawing.
4. Enter a name for the modified pad shape in the Padstack Map dialog box.
5. In the Options foldable window pane, change the fields to modify the selected pad(s).

Related Topics

- [multpadedit](#)

muserver

The `muserver` command starts the Symphony Server application. This command launches the *Allegro Symphony Server* window to share a database with multiple clients to perform design activities in a concurrent environment.

 The command is available in the following layout editors:

- Allegro X PCB Editor
- Allegro X Advanced Package Designer (APD)
- Allegro X Sigrity SI

The command is available in the layout editor only when the *Symphony Team Design* option is selected. However, you can still start the Symphony Server application using the `muserver` batch command.



If the command is started inside the layout editor, it automatically connects the user to the server application and enables the current database for team designing.

There are two modes of team design:

- Network server-side and informal first-client driven: The server application is started by the design owner in a standalone mode on a dedicated network server machine. Users communicate with the server machine to connect to the server application for performing team design tasks.
- Informal first-client driven mode: The server application is started by one of the users to enable the currently opened database for team designing. The other users communicate with the local machine to connect with the server application. This is the preferred mode for short-term team-designing.

In both the modes, the master database is controlled by the server and the owner is responsible to open and save the databases.

```
muserver -port <min>[-<max>] -safe -nographic|-nograph -autosave -autosave_int -
autosave_name<filename> -autosave_vers [<database>]
```

| | |
|--|--|
| -port <min>[-<max>] | Starts server listening on minimum port. If it is not available and a maximum port value is provided, the command scans for the first available port between minimum and maximum. |
| -safe | Runs the server without user or site configuration files and settings. For more information, refer to <code>safe.txt</code> located at <code><installation_directory>/share/pcb/batchhelp</code> directory. |
| -nographic -nograph | Runs the server in pseudo non-graphic mode. On Unix, it requires an X server but does not display any graphics. Use VNC server (http://www.realvnc.com). <div> Specify a design name with this command-line option.</div> |
| -autosave | Enables automatic saving of the active database. |
| -autosave_int <# of minutes> | Enables automatic saving of the active database. You need to specify the auto-save interval in minutes. |
| - autosave_name <filename> | Enables automatic saving of the active database. You need to specify a custom name for the output file. <div> Specify a design name with this command-line option.</div> |
| - autosave_vers <# of versions> | Enables automatic saving of the active database and specifies the number of auto-save files. You can save a minimum of 1 version, and a maximum of 99 versions. |
| <database> | (Optional) Specifies a database for editing. Supported extensions are <code>.brd</code> , <code>.dpf</code> , and <code>.mcm</code> . Ignores initialization (<code>.ini</code>) files. |

Related Topics

- [Managing Allegro Symphony Server](#)
- [Working with Symphony Team Design](#)
- [muconnect](#)
- [muservermgr](#)

Symphony Server Window

Use this window to share databases with multiple clients to perform design activities in a concurrent environment.

Access Using:

- Menu Path: *File – Start Symphony Server*
- Command line: `muserver`

Allegro Symphony Server Window

| | |
|--------------------|--|
| <i>Open</i> | Browses for a database to open it in the server. |
| <i>Save</i> | Saves the database. |
| <i>Close</i> | Closes the database. |
| <i>Connections</i> | Displays a list of clients connected to the server application. The information for each client connection includes user name, machine name, unique ID, and color for easy and clear identification. |
| <i>Locks</i> | <p>Displays canvas locks generated by the layout editor for selected objects that are being edited by the clients.</p> <p>Hovering over a lock provides the locking information (user name, userID, IP, and hostname) in a data tip.</p> |
| <i>Log</i> | <p>Displays server activities as well as a summary of database check results.</p> <p>Hovering over a log entry provides the timestamp in the data tip.</p> |
| <i>Filter</i> | Filters <i>Log</i> data. |

Allegro Symphony Server Menu Options

| | | |
|--------------|---------------------------|--|
| <i>File</i> | | |
| | <i>Open</i> | Opens a file browser to search for the database. |
| | <i>Save</i> | Saves the active database with the current name. |
| | <i>Write As</i> | Saves a copy of the database to a new name without changing the name of the active database currently opened in the layout editor for team designing. |
| | <i>Close</i> | Closes the database and disconnects all clients. Prompts you to save the database prior to exiting the Symphony Server application. |
| | <i>Viewlog</i> | Displays database checks log file <code>dbdoctor.log</code> . |
| | <i>File Viewer</i> | Opens a standard file browser to open a <code>.log</code> file for viewing. |
| | <i>Options</i> | Open Server Options Dialog Box to change general and security settings of the server. |
| | <i>Recent Files</i> | Displays list of databases opened during a session of the Symphony Server application. You can clear the list by choosing the last entry <i>Clear Recent Files</i> . |
| | <i>Exit</i> | Exits the server application. |
| Tools | | |
| | <i>Database Check</i> | Performs basic database checks on the active database and reports the results in the <i>Log</i> tab. |
| | Enable On-Line DRC | Enables online DRC checks on the server database. |
| Help | | |
| | <i>About</i> | Displays version and platform of the server application. |

Server Options Dialog Box

| | | |
|-----------------|---|---|
| General | | |
| Server Settings | | |
| | Maximum Clients | Specifies the maximum number of clients that connect to a database at any given time. The default value is 10. |
| | Port Range | Specifies TCP-Port range when exporting a database to clients. The default value is 4000 - 4005. |
| | UID Block Size | <p>Specifies a reserved block of numbers to track database objects during team designing. Each database object added is assigned a unique number from the block as an ID for tracking its movements. The default block size is 100000.</p> <p>When tracking numbers are exhausted for a given client, a new block of numbers is provided automatically. Increasing the block size reduces the frequency of requests for new blocks.</p> |
| | Reset UIDs on Design Open | Resets UIDs when a design is opened in the Symphony Server. |
| | Save & Apply User Parameters | <p>Saves client-side parameter settings when disconnected from the server and applies when the client rejoins the session. This option is enabled by default.</p> <p>Client-side parameter settings include zoom-level, layers visibility, and so on. When modified, the client settings are saved in a sub-directory symphony_user_data, which is created in the working directory of the server database.</p> |

| | | |
|--------------------------|---------------------------------------|---|
| | <i>Enable Temp Updates</i> | <p>Displays temporary updates from other clients before they commit the changes to the server.</p> <p>By default, this option is enabled.</p> |
| | <i>Forward Peer Cursors</i> | <p>Forwards the cursor information of a client to other clients connected to a server database.</p> <p>This option is enabled by default.</p> |
| Autosave Settings | | |
| | <i>Enable</i> | <p>Enables automatic saving of the active database based on the values specified for auto-save interval and number of auto-save files.</p> |
| | <i>Custom Name</i> | <p>Specifies a custom name for autosaved databases using standard file versioning by appending a comma after the file extension (.brd).</p> <p>The default name is set to <design name>_autosave.brd.</p> |
| | <i>Interval</i> | Specifies the auto-save interval. |
| | <i>Number of Files</i> | Specifies the number of auto-saved files. |
| Design Settings | | |
| | <i>One Connection per User</i> | <p>Prevents the same user from connecting to the database more than once. By default, this option is disabled.</p> |

| | | |
|-----------------|---------------------------------------|---|
| | <i>Disable Permanent Locks</i> | <p>Prevents the permanent locking of database objects on the client side.</p> <p>Use this option when a user wants to work on a group of components or nets. Enabling this option prevents other users from interfering with design activities on the selected design objects.</p> <p>This option is disabled by default.</p> |
| Security | | |
| Access Settings | | |
| | <i>Allow List</i> | <p>Displays the list of users allowed to connect to the server. The default entry in the list is <i>(everyone)</i>. You can double-click the row after <i>(everyone)</i> and add a user name to the list.</p> |
| | <i>Deny list</i> | <p>Displays the list of users who are not allowed to connect to the server. The default entry in the list is <i>(everyone)</i>. You can double-click the row after <i>(everyone)</i> and add a user name to the list.</p> |
| | <i>Password</i> | <p>Displays a password to the server, required when connecting to the shared database. You can generate a new password or create a custom password:</p> <p>Select <i>Generate</i> button to auto-generate a password or select the <i>Custom</i> button to define a custom password.</p> |
| OK | | Click to apply the changes. |
| Cancel | | Closes without saving any changes. |

Related Topics

- [Working with Symphony Team Design](#)

Managing Symphony Server

You can perform the following steps to manage Symphony Server:

Starting Standalone Symphony Server Application

1. At the command prompt, run the `muserver` command.
The *Allegro* Symphony Server application starts as a standalone utility.
2. Click Open to browse a database.
The database is available for clients to connect.

Starting Symphony Server Application Inside Layout Editor

1. Choose *File – Start Symphony Server* or type `muserver` in the Command window of the layout editor.
The *Allegro* Symphony Server starts using the active database and is available for other clients to connect. The client started the Symphony Server automatically connected to it.

Changing Security Settings of Symphony Server

1. In the *Allegro* Symphony Server, choose *File – Options*.
The *Server Options* dialog box opens.
2. To restrict the access to the database, select the *Security* tab. Add user names to either *Allow List* or *Deny List*.
3. For additional security, enable *Password* in the *Security* tab.
4. Create a system-generated password by clicking *Generate* or click *Custom* to define a password manually.

Related Topics

- [muserver](#)


muservermgr

The `muservermgr`, when run from the operating system command prompt, command launches *Allegro X Symphony Server Manager* that allows you to manage PCB board designs and Symphony Servers within concurrent environment.

 This command is available in the following layout editors:

- Allegro X PCB Editor
- Allegro X Advanced Package Designer (APD)
- Allegro X Sigrity SI

`muservermgr <args> [<database repository path>]`

| | |
|---|--|
| <code>-port <min> [-<max>]</code> | Starts server listening on the minimum port number. If it is not available and a maximum port value is provided, the server manager scans for the first available port between minimum and maximum. |
| <code>-j <journal location></code> | Opens journal file. The default journal file <code><prog>.jrl</code> . |
| <code>-safe</code> | Runs the server without user or site configuration files and settings. See <code><installation_directory>/share/pcb/batchhelp/safe.txt</code> . |
| <code>- nographic - nograph</code> | Runs the server manager in pseudo non-graphic mode. On Unix it requires an X server, but does not display any graphics. Use VNC server (http://www.realvnc.com). <div> Specify a design with this command line option.</div> |
| <code><database repository path></code> | Specifies the location of directory where designs are kept to manage. |

| | |
|------------|---|
| -install | Installs Symphony Server Manager as a Windows service. This option is available only for Windows. |
| -uninstall | Uninstalls Symphony Server Manager as a Windows service. This option is available only for Windows. |
| -start | Starts Symphony Server Manager as a service, if it is installed and running. This option is available only for Windows. |
| -stop | Stops Symphony Server Manager as a service, if it is installed and running. This option is available only for Windows. |

Related Topics

- [Managing Designs Using Symphony Server Manager](#)
- [Working with Symphony Team Design](#)
- [muconnect](#)
- [muserver](#)

Symphony Server Manager Dialog Box

| | | |
|----------------|--|---|
| Base Directory | Browse for database to open in the server. | |
| <i>Browse</i> | Browse for directory location for managing designs. | |
| Databases | | |
| | <i>File</i> | Displays the name of the database. |
| | <i>Size</i> | Displays the size of the database. |
| | <i>Status</i> | Displays the status of the database. Values are <i>Starting</i> , <i>Available</i> , and <i>Hosted (Port number)</i> . |
| | <i>Owner</i> | Displays the name of the database owner. |
| | <i>Manage</i> | Click to start Symphony Server application for the selected design. Options are available to close, save, and delete the server database. |
| Connections | Display list of clients connected to the server manager application from the layout editor server manager remote dialog box. | |
| Log | Displays server manager activities and the server application status. | |
| | <i>Filter</i> | Filters the <i>Log</i> data. |
| <i>Options</i> | | |
| | <i>Port Range</i> | <p>Specifies TCP-Port range when exporting database to clients.</p> <p>The default value is 4000 - 4005.</p> |
| | <i>Starts Servers in No Graphic Mode</i> | Enable this option to start the server manager in non-graphic mode. This option is disabled by default. |

| | | |
|----------------|--|--|
| | <i>Journal Location</i> | Specifies the path of server journal file (<code>muservermgr.jrl</code>). Use this option to change the file path to a different location. By default, the <code>muservermgr.jrl</code> is saved in the same folder from where the server manager was started. |
| <i>Refresh</i> | Refreshes server manager and display status for each database. | |

Menu Options

| | | |
|-------------|-------------------------|--|
| <i>File</i> | | |
| | <i>Select Directory</i> | Specifies a directory where database files are saved for managing. By default, the directory location is set to <code>pcbenv/SymphonyMgr</code> . |
| | <i>Add File</i> | Opens a file browser to add new database files for managing. |
| | <i>Options</i> | Provides options to change server manager settings. |
| | <i>Viewlog</i> | Displays server manager activities. |
| | <i>Exit</i> | Closes the server manager application. |
| Help | | |
| | <i>About</i> | Displays version and platform of the server manager application. |

Managing Designs Using Symphony Server Manager

To manage your designs using the Symphony Server Manager, follow these steps:

1. On command prompt, type `muservermgr`.
Allegro Symphony Server Manager application starts.
2. Click File – Add file to browse a database.
The details of the database are displayed.
3. Click *Manage* button.
File Information dialog box is displayed with command buttons.
4. Click *Start Server* button in the *File Information* dialog box.
Symphony Server application starts for the selected database.
5. In the Allegro Symphony Server Manager dialog box, right-click on the database name and choose *Save* to save the changes to the database.
6. Click *Manage* or right-click to choose *Close Server* to stop the server application.
7. Click File – Exit to exit the server manager.

Related Topics

- [muservermgr](#)

