cadence®

L Commands

Product Version 23.1
September 2023

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

Contents

1 7
L Commands 7
label device 8
8
Options Panel for the Label Device Command 9
Adding Device Label Text to Package or Part Symbols 10
label part 11
11
Label Part Command: Options Panel 12
Adding Part Number Text to Package/Part Symbols 13
label refdes 14
14
Label Refdes Command: Options Panel 15
Adding Reference Designators to Package/Part Symbols 16
label tolerance 17
17
Label Tolerance Command: Options Panel 18
Adding Tolerance Text to Package/Part Symbols 19
label value 20
20
Label Value Command: Options Panel 21
Adding Value Label Text to Package/Part Symbols 22
layer compare 23
23
Layer Compare Dialog Box 24
Comparing Layers Interactively 26
layer compare batch 27
27
Batch Layer Compare Dialog Box 29
Batch Compare Layers Dialog Box 35
Batch Compare Layer Dialog Box for Change Layer 36
Objects Specifications Dialog Box 37
Nets Specifications Dialog Box 38

Components Specifications Dialog Box 39
Specify Region Dialog Box 40
Batch Layer Compare Control File 41
Logical Operations in Interactive and Batch Modes of Layer Compare 45
Adding a New Comparison in Batch Compare 48
Removing All Comparisons in Batch Layer Compare 49
Changing Master, Compare, or Destination Layer in Batch Layer Compare 50
50
Specifying Expansion or Contraction to Shapes in Batch Layer Compare 51
Specifying Minimum Shape Area and Size in Batch Layer Compare 52
Rotating Shapes using Batch Layer Compare 53
Mirroring Shapes using Batch Layer Compare 54
Filtering Objects by Net, Type, Components, and Region using Batch Layer Compare 55
Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare 55
layer compare diff 57
57
Compare Differences Dialog Box 58
Viewing and Resolving Differences in Batch Layer Compare 61
layer estimation 62
62
Creating Layer Estimation Report 63
layer priority 65
65
Display Priority Dialog Box 66
Assigning a Display Priority To Layers 67
layout wizard 68
68
Board Wizard Window 69
Creating a New Layout Using Board Wizard 76
lead editor 79
79
Supported Lead Types 80
Assign Pin Leads Dialog Box 83
Assigning Leads to Pins 85
lef lib 86
86

LEF Library Manager Dialog Box 87
Filter options Dialog Box 89
Creating New Library Definition File Using LEF Library Manager 93
lef pin param 96
96
LEF Pin Parameters Dialog Box 97
Associating Die Pins with LEF Macro Cells 99
Reassigning Die Pins with LEF Macro Cells 100
Removing Die Pins and LEF Macro Cell Association 101
license_use 102
102
Licenses in Use Window 103
Displaying a Report of Product, Options, and Licenses Used 104
104
line fattening 104
104
Via-Via Line Fattening Dialog Box 107
Eliminating Acid Traps 108
linefont 110
110
Options Panel for the Linefont Command 111
list 112
Load Cadence Artwork Dialog Box 113
load gerber 115
115
Loading Vector-Based Data 116
load photoplot 118
load plot 119
119
Load Plot Dialog Box 120
Previewing IPF Files 122
Creating Penplot Files for Negative Plane Layers 123
Viewing a Plot File 124
load stream 125
125
Stream In Dialog Box 126
Stream In View Data Dialog Box 128

Stream In Edit Layer Mapping Dialog Box 131
Creating a Design from a GDSII Stream File 135
Viewing and Importing Data on GDSII Stream Layers 137
Viewing Data on GDSII Stream Structures 139
Mapping or Unmapping GDSII Stream Layers 141
Adding a User-Defined Subclass for Mapping Purposes 144
Creating a Stream Layer Conversion File Using a Text Editor 145
lock symbol 146

1

L Commands

label device	label part	label refdes
label tolerance	label value	layer compare
layer compare batch	layer compare diff	layer estimation
layer priority	layout wizard	lead editor
lef lib	lef pin param	license_use
line fattening	linefont	list
Load Cadence Artwork Dialog Box	load gerber	Loading Vector-Based Data
load photoplot	load plot	load stream
lock symbol		

label device

The `label device` command adds a label displaying the device name of a symbol.

When you place a symbol in a design, a label is placed according to the information specified in the Options panel. You can move, rotate, or change the text size, or change the content of the label.

> This command is available only in Symbol Editor mode of $BRANDX $PRODUCTPCB.

Related Topics

- Adding Device Label Text to Package/Part Symbols

Options Panel for the Label Device Command

Access Using

- Menu Path: *Layout – Labels – Device*

Mirror	Displays a mirror image of the label text.
Marker size	Specifies the size of the label.
Rotate	Rotates the label by the specified degree.
Text block	Defines the size and spacing of the text. You can define up to 16 text blocks.
Text name	Specifies the label text.
Text just	Specifies the typographical alignment or justification of the text in the label.

Related Topics

- prmed
- define text
- change

Adding Device Label Text to Package or Part Symbols

To add a label with device text to a symbol in Symbol Editor, do the following:

1. Choose *Layout – Labels– Device* or type `label device` in the Command window.
2. Specify the appropriate subclass in the Options panel.
3. Click the location on the canvas to place the label text.
4. Type the label text at the command line and press `Enter`.
5. Right-click and choose *Done*.

Related Topics

- label device
- Defining and Developing Libraries

label part

The `label part` command adds a label displaying the part number of a package or part symbol drawing.

When you place a symbol in a design, a label is placed according to the information specified in the Options panel. You can move, rotate, or change the text size, or change the content of the label.

ⓘ This command is available only in Symbol Editor mode of Allegro X PCB Editor.

Related Topics

- Adding Part Number Text to Package/Part Symbols

Label Part Command: Options Panel

Access Using

- Menu path: *Layout – Labels– Part Number*

Mirror	Displays a mirror image of the text.
Marker Size	Specifies the size of the marker.
Rotate	Rotates the label by the specified degree.
Text Block	Defines the size and spacing of the text.
Text name	Specifies the label text.
Text Just	Specifies the typographical alignment or justification of the text in the label.

Related Topics

- prmed
- define text
- change

Adding Part Number Text to Package/Part Symbols

To add a label with part number to a symbol in Symbol Editor, do the following:

1. Choose *Layout – Labels– Device* or type `label part` in the Command window.
2. Specify the appropriate subclass in the Options panel.
3. Click the location on the design canvas to place the label text.
4. Type the label text at the command line and press `Enter`.
5. Right-click and choose *Done*.

Related Topics

- label part
- Defining and Developing Libraries

label refdes

The `label refdes` command adds a label displaying the reference designator (RefDes) of a symbol.

When you place a symbol in a design, a label is placed according to the information specified in the Options panel. You can move, rotate, or change the text size, or change the content of the label.

> This command is available only in Symbol Editor mode of Allegro X PCB Editor.

Related Topics

- Adding Reference Designators to Package/Part Symbols

Label Refdes Command: Options Panel

Access Using

- Menu path: *Layout – Labels – Refdes*

Mirror	Displays a mirror image of the text.
Marker Size	Specifies the size of the marker.
Rotate	Rotates the label by the specified degree.
Text Block	Defines the size and spacing of the text.
Text name	Specifies the label text.
Text Just	Specifies the typographical alignment or justification of the text in the label.

Related Topics

- prmed
- define text
- change

Adding Reference Designators to Package/Part Symbols

To add a label with reference designator (RefDes) to a symbol in Symbol Editor, do the following:

1. Choose *Layout – Labels – Refdes*.
2. Complete the Options panel.
3. Click the location on the canvas to place the label text.
4. Type the label text at the command line and press *Enter*.
5. Right-click and choose *Done*.

Related Topics

- label refdes
- Defining and Developing Libraries

label tolerance

The `label tolerance` command adds a label displaying tolerance of symbol.

When you place a symbol in a design, a label is placed according to the information specified in the Options panel. You can move, rotate, or change the text size, or change the content of the label.

ⓘ This command is available only in Symbol Editor mode of Allegro X PCB Editor.

Related Topics

- Adding Tolerance Text to Package/Part Symbols

Label Tolerance Command: Options Panel

Access Using

- Menu path: *Layout – Labels – Tolerance*

Mirror	Displays a mirror image of the text.
Marker Size	Specifies the size of the marker.
Rotate	Rotates the label by the specified degree.
Text Block	Defines the size and spacing of the text.
Text name	Specifies the label text.
Text Just	Specifies the typographical alignment or justification of the text in the label.

Related Topics

- prmed
- define text
- change

Adding Tolerance Text to Package/Part Symbols

To add a label with tolerance to a symbol in Symbol Editor, do the following:

1. Choose *Layout – Labels – Tolerance*.
2. Complete the Options panel.
3. Click the location on the canvas to place the label text.
4. Type the label text at the command line and press *Enter*.
5. Right-click and choose *Done*.

Related Topics

- label tolerance
- Defining and Developing Libraries

label value

The `label value` command adds a label displaying value of symbols.

When you place a symbol in a design, a label is placed according to the information specified in the Options panel. You can move, rotate, or change the text size, or change the content of the label.

ⓘ This command is available only in Symbol Editor mode of Allegro X PCB Editor.

Related Topics

- Adding Tolerance Text to Package/Part Symbols

Label Value Command: Options Panel

Access Using

- Menu path: *Layout – Labels– Value*

Mirror	Displays a mirror image of the text.
Marker Size	Specifies the size of the marker.
Rotate	Rotates the label by the specified degree.
Text Block	Defines the size and spacing of the text.
Text name	Specifies the label text.
Text Just	Specifies the typographical alignment or justification of the text in the label.

Related Topics

- prmed
- define text
- change

Adding Value Label Text to Package/Part Symbols

To add a label displaying value to a symbol in Symbol Editor, do the following:

1. Choose *Layout – Labels– Value*
2. Complete the Options panel.
3. Click the location on the canvas to place the label text.
4. Type the label text at the command line and press `Enter`.
5. Right-click and choose *Done*.

Related Topics

- label value
- Defining and Developing Libraries

layer compare

The `layer compare` command compares a specified region for two layers in the same design database and creates a new destination layer with the differences. Performs `AND`, `OR`, `XOR`, and `ANDNOT` operations on the layers being compared.

You can create new destination layers only on non-etch layers. For the `CONDUCTOR` destination class, existing layer names can be selected.

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

Related Topics

- Comparing Layers Interactively
- Logical Operations in Interactive and Batch Modes of Layer Compare

Layer Compare Dialog Box

Access Using

- Menu path: *Tools – Layer Compare – Interactive Layer Compare*

Use substrate outline	Select to define the comparison region using substrate extents. The substrate extents are defined by the shape on the substrate geometry or the outline layer. If the substrate geometry or outline layer does not exist, the substrate extents are the same as the database extents. ⚠ If plating bar tails extend beyond the boundary of the substrate outline, and you want the entire tail to be part of the comparison, use the Window mode or adjust the substrate outline shape.
Select window region	Click to define the comparison region by specifying a rectangular window on the canvas. Click to specify the two ends of a diagonal for the rectangle. Rectangle is the default region.
Select symbol	Click to define the comparison region using the place bound extents of a selected symbol.
Select shape	Click to define the comparison region using a selected shape, such as a filled or unfilled rectangle, a filled or unfilled shape, or a crosshatched shape. The shape need not be a conductor shape and can be present in layers other than the reference layer or comparison layer. For example, you can define a layer and draw a shape that is the extents you want to use with layer compare. This is the only option which allows for a region that includes voids.
Reference Layer	Specify the class and subclass for the reference layer for the comparison operation. By default, the top conductor layer is selected. Specify if you want to use the *Positive* (default) or *Negative* regions of the layer.
Comparison Layer	Configure the class and subclass for the compared layer for the comparison operation. By default, the top conductor layer is selected. Specify if you want to use the *Positive* (default) or *Negative* regions of the layer.

Destination Layer	Configure the layer where shapes are to be created for the differences found in the comparison operation. If you specify a new name, a layer is created by the layout editor. Else, you can specify an existing layer. For an existing layer, select *Remove existing shapes on layer* to remove any existing shapes. ⓘ You can create new subclasses only on non-etch layers. If the destination class is `CONDUCTOR`, you can only select an existing layer name.
Operation	Select the logical operation to be performed between the layers being compared. The default is `AND`.
Remove existing shapes on layer	Select to remove the existing shapes on the specified destination layer. ⚠ The shapes are removed after the comparison is made and before the results are generated.
OK	Click to save the comparison results to the database and close the dialog box.
Cancel	Click to exit without saving the comparison results.
Generate	Click to compare and update the database.

Related Topics

- Logical Operations in Interactive and Batch Modes of Layer Compare

Comparing Layers Interactively

To compare layers in the active design database and create a destination layer, do the following steps:

1. Choose *Tools – Layer Compare – Interactive Layer Compare*
   The *Layer Compare* dialog box appears.
2. Select the comparison region.
   By default, the comparison region is defined by a window.
3. Select the reference layer.
   This is the master layer.
4. Select either *Positive* or a *Negative* to specify a region for the reference layer.
5. Select the comparison layer.
6. Select to *Positive* or a *Negative* to specify a region for the comparison layer.
7. Specify the destination layer.
8. Select the operation.
9. Click *Generate* to compare and update the database.
10. Vary the option selections to generate different results, if needed.
    For example, you can perform an `AND` operation followed by and `XOR` operation. You can also choose to compare different layers.
11. Click *OK* to update all the comparison results and close the dialog box.

Related Topics

- Logical Operations in Interactive and Batch Modes of Layer Compare

layer compare batch

The `layer compare batch` command performs multiple sequential comparisons between layers in a design or across designs using a control file. Performs `AND`, `OR`, `XOR`, Master `ANDNOT` Compare, and Compare `ANDNOT` Master operations on the layers being compared.

You can define highlight colors for the results of the different operations performed. This makes it possible to distinguish the results of different operations placed on a single destination result.

The control file used with the batch command is a text file that defines the comparison operations and their sequence. This file can be created using any text editor or defined using the tree view of the layer compare batch command user interface.

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

Syntax

```
layer_compare [-e] <ref_design> <control_file> <comp_design>
```

-e	Specifies that the tool proceed even with errors and send errors to the log file. If this argument is not set, the layout editor stops when the first error is detected. This is an optional argument.
ref_design	Specifies an existing readable and writable design database to be used as the reference for layer comparison. The output results will be created on this design based on the control file specifications. This can be a `.sip`, `.mcm`, or `.brd` file.
control_file	Specifies an existing text file that contains the instructions for comparisons.
comp_design	Specifies an existing readable design database to be used as a comparison design for layer comparison. If you do not use this argument, the reference design is used as the comparison design. This can be a `.mcm` or `.brd` file.

Example

The following command will compare the reference database ref.sip in the `/home/usr1/test1/` folder with the comparison design `test.mcm` in the `/home/usr1/des1/` folder using the control file `/home/usr1/test1/ctl_file.txt`. The output of the comparison is written to the reference database `ref.mcm`. This command does not proceed if it encounters an error.

```
layer_compare /home/usr1/test1/ref.sip /home/usr1/test1/ctl_file.txt
/home/usr1/des1/test.mcm
```

You can use the `-help` argument with this command (`layer_compare -help`) to get help on this command.

Related Topics

- Logical Operations in Interactive and Batch Modes of Layer Compare

Batch Layer Compare Dialog Box

Access Using

- Menu path: *Tools – Layer Compare – Batch Layer Compare*

Comparison database	Select this check box to specify a comparison database different from the current database. This is selected by default. If selected, you can either browse to a comparison database or type the database name in the text box. By default, the current database is specified. Note that the comparison database specified in this field and the reference database need not be in the same format. For example, you can compare an `.mcm` database with a `.sip` database or a `.sip` database with a `.brd`.
Control file	Select this check box to specify the control file to be used. This is selected by default. If you deselect this checkbox, a temporary control file is created by the tool. If selected, you can either browse to a control file or type the name of a control file in the text box. If you specify a control file, the file is updated when comparison is run. This overwrites the content of the control file. Similarly, if you change the control file, the tree is updated to match the file.
Show detailed differences instead of location and size of differences	Select to display detailed differences. This check box is selected by default.
Operation Tree	Lists the operations to perform. This tree is generated from the control file, if it is specified. You can edit the tree to add, remove, or modify comparisons, or to change the sequence of listed comparison operations. The root or top-level node of the tree is the Comparisons folder that contains one or more Single Comparison folders. The Single Comparison folder contains a Master Layer folder, a Compare folder, a Destination folder, and an Operations folder.

	• The top-level *Comparisons* folder contains all the comparisons to be performed in the sequence in which they are run . You can right-click this folder, and choose to add a comparison (*Add*) or to remove all comparisons (*Remove All*). • The *Single Comparison* folder defines a single comparison to be performed. The name of this folder is generated using the layers to be compared and the operations to be performed. You can right-click this folder, and choose to perform the following operations on the comparison entry: ◦ move up (*Move Up*) ◦ move down (*Move Down*) ◦ copy (*Copy*) ◦ remove (*Remove*) This is defined by the *COMPARE* keyword of the control file that starts a comparison in the file. Note that the name displayed does not correspond to any particular name in the control file but provides an easy reference in the tree view.

	• he *Master* folder describes the reference (master) layer in the current database for the comparison. You can right-click this folder, and choose to change the reference layer (*Change Layer*) or add the following options and filters from the listed items: ◦ Options ▪ *Expand Value* ▪ *Contract Value* ▪ *Minimum Shape Size* ▪ *Rotation, Mirror* ▪ *Negative Artwork* ▪ *Solid Crosshatch Shapes* ◦ Filters ▪ *Object Filter* ▪ *Net Filter* ▪ *Component Filter* ▪ *Region Filter* The selected options and filters are removed from the menu and are listed as entries under this folder. The master layer is defined by the first and second arguments of the *COMPARE* keyword of the control file. The options are defined by the *OPTION* keyword and the filters are defined by the *FILTER* keyword. The first argument of *OPTION* and *FILTER* specifies the database name and the remaining arguments define the name of the option and filter followed by any required values.

- TThe *Compare* folder describes the compare layer in the comparison database to be used in this operation. You can right-click this folder, and choose to change the compare layer (*Change Layer*) or add the following options and filters from the listed items:
  - Options
    - *Expand Value*
    - *Contract Value*
    - *Minimum Shape Size*
    - *Rotation, Mirror*
    - *Negative Artwork*
    - *Solid Crosshatch Shapes*
  - Filters
    - *Object Filter*
    - *Net Filter*
    - *Component Filter*
    - *Region Filter*

  The selected options and filters are removed from the menu and are listed as entries under this folder.
  The compare layer is defined by the *COMPARE* keyword of the control file. The options are defined by the *OPTION* keyword and the filters are defined by the *FILTER* keyword. The first argument of *OPTION* and *FILTER* specifies the database name and the remaining arguments define the name of the option and filter followed by any required values.

	• The *Destination* folder defines the destination (output) layer to place the results of the comparison. This layer exists in the active drawing, and is created automatically by the tool if it does not exist. You can right-click this folder, and choose to change the destination layer (*Change Layer*) or add options (*Expand Value*, *Contract Value*, *Minimum Shape Size*, *Rotation*, *Mirror*). The selected options are removed from the menu and are listed as entries under this folder. ⓘ You can create new subclasses only on non-etch layers. If the destination class is `CONDUCTOR`, you can only select an existing layer name. The destination layer and the options are defined by the `OUTPUT` keyword of the control file. The first two arguments specify the class and subclass, and the third argument specifies the options. • The Operations folder lists the operations to be performed. The default operations are `AND, MANDNOTC` (Master `ANDNOT` Compare), and `CANDNOTM` (Compare `ANDNOT` Master). The default operations are defined by the *icp_layer_compare_default_operations* variable under lc_packaging category in User Preferences Editor (*Setup – User Preferences*). You can set a different default by specifying a list of operations separated by spaces in the variable text field. The valid values are `AND, OR, XOR, MANDNOTC, CANDNOTM`. At least one operation must be specified for any comparison operation.
Operation Colors	Select the color to be used for the results of the available logical operations. Numbers shown are the color palette index of the color along with the RGB value. A color swatch is included for reference. The default color for all operations is `index 2` in the drawing until changed or read from a control file. The colors are defined by the `COLOR` keyword of the control file. ⚠ The color settings apply globally and for all the comparison output operations. They cannot be configured separately for individual comparisons.

Compare	Click to start the comparison batch operation. A progress meter is shown with updates on the progression of the tool. If a control file is specified, and it needs updating, it is written before the comparator is launched.
Cancel	Click to close the dialog box without updating the control file or database.

Related Topics

- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare
- Batch Layer Compare Control File
- Logical Operations in Interactive and Batch Modes of Layer Compare

Batch Compare Layers Dialog Box

Master and Compare classes	Select the classes and subclasses for the reference and compare layers. You can select any existing layers in the design for the master and compare layer fields. You can also select from the following two special layers as master or compare: • *ALL_CONDUCTOR*: Combines the pin, via, and conductor classes for the specified etch layer name. • *ALL_SOLDERMASK*: Combines the via, pin, substrate geometry, and component for the geometry solder mask layer specified.
Destination classes	Select the class and subclass for the destination layer. You can select any existing layer in the active drawing as a destination. You can also specify a new subclass name to have the tool create this layer on the fly for generating the output. ⓘ You can create new subclasses only on non-etch layers. If the destination class is `CONDUCTOR`, you can only select an existing layer name.
OK	Click to save the changes and close the dialog box.

Related Topics

- Adding a New Comparison in Batch Compare
- Batch Layer Compare Control File
- Logical Operations in Interactive and Batch Modes of Layer Compare

Batch Compare Layer Dialog Box for Change Layer

Class and Subclass	Select the Class and Subclass for the reference, compare, or destination (output) layer depending on the folder from which you choose the Change Layer. You can select any existing layers in the design for the master and compare layer fields. You can also select from the following two special layers as master or compare: • ALL_CONDUCTOR: Combines the pin, via, and conductor classes for the specified etch layer name • ALL_SOLDERMASK: Combines the via, pin, substrate geometry, and component for the geometry solder mask layer specified. You can select any existing layer in the active drawing as a destination. You can also specify a new subclass name for the layout editor create this layer on the fly for generating the output. ⓘ You can create new subclasses only on non-etch layers. If the destination class is CONDUCTOR, you can only select an existing layer name.
OK	Click to save the changes and close the dialog box.
Cancel	Click to close the dialog box without saving changes.

Related Topics

- Batch Layer Compare Control File
- Logical Operations in Interactive and Batch Modes of Layer Compare
- Changing Master, Compare, or Destination Layer in Batch Layer Compare

Objects Specifications Dialog Box

Available Items	Click to add the listed items to the Selected Items list. Lists all the available object types, such as shapes, fingers, pins, vias, clines, and lines
Selected Items	Lists the items selected for the filter. Click to remove an item from the list.
OK	Click to save the changes and close the dialog box.
Cancel	Click to close the dialog box without saving changes.

Related Topics

- Batch Layer Compare Control File
- Logical Operations in Interactive and Batch Modes of Layer Compare
- Nets Specifications Dialog Box

Nets Specifications Dialog Box

Available Items	Click to add the listed items to the Selected Items list. Lists all the available nets in the design.
Selected Items	Lists the items selected for the filter. Click to remove an item from the list.
OK	Click to save the changes and close the dialog box.
Cancel	Click to close the dialog box without saving changes.

Related Topics

- Batch Layer Compare Control File
- Batch Compare Layers Dialog Box

Components Specifications Dialog Box

Available Items	Click to add the listed items to the Selected Items list. Lists all the available components including dies and BGAs in the design.
Selected Items	Lists the items selected for the filter. Click to remove an item from the list.
OK	Click to save the changes and close the dialog box.
Cancel	Click to close the dialog box without saving changes.

Related Topics

- Batch Layer Compare Control File
- Batch Compare Layers Dialog Box

Specify Region Dialog Box

Lower Left X	Specify the X-coordinate of the lower-left corner of the filter region.
Lower Left Y	Specify the Y-coordinate of the lower-left corner of the filter region.
Upper Right X	Specify the X-coordinate of the upper-right corner of the filter region.
Upper Right Y	Specify the Y-coordinate of the upper-right corner of the filter region.
OK	Click to save the changes and close the dialog box.
Cancel	Click to close the dialog box without saving changes.

ⓘ The lower-left coroner and upper-right corner cannot not be the same points. These are absolute points in the drawing to which the change is applied. It is the current design for master, or comparison database when adding a region filter for the compare layer.

If a region is not specified, the drawing outline is used in both the master and the compare databases, and they are automatically adjusted to be centered on top of each other. This lets you compare two designs that use different units, or have a different origin, without having to set up the region all the time.

The coordinates can be specified in any design units. If the compare database uses different design units, specify the value in the units to have the value automatically converted to the design units of the current drawing.

Related Topics

- Batch Layer Compare Control File
- Batch Compare Layers Dialog Box

Batch Layer Compare Control File

The control file used by the *layer compare batch* command follows a keyword-value pair format.

> ⚠ If you use a name with space in the control file, enclose it using either straight single quotes (`'`) or straight double quotes (`"`). You can use the hash (`#`) character for comment lines.

Use the file to define the master, compare, and destination layers and various operations and filters using different keywords.

Supported Operations

The control file supports the following operations:

- *MASTER_AND_COMPARE*: Logical AND of the master layer and the compare layer. Returns *True* if the items are in both the databases.
- *MASTER_OR_COMPARE*: Logical OR of the master layer and the compare layer. Returns *True* if the items that are in either the master database or the compare database.
- MASTER_XOR_COMPARE: Logical XOR of the master layer and the compare layer. Returns *True* if the items are in either of the databases but not in both.
- MASTER_ANDNOT_COMPARE: Logical ANDNOT of the master layer with the compare layer. Returns *True* if the items are in the master database but not in the compare database.
- COMPARE_ANDNOT_MASTER: Logical ANDNOT of the compare layer with the master layer. Returns *True* if the items are in the compare database but not in the master database.

Supported Keywords

The following keywords are supported:

- DETAILED_DIFFERENCES: Set to `TRUE` to display the detailed differences instead of location and size of differences. This is an optional keyword.
  TRUE is the default.
  For example, the following line sets to display only location and size of differences:
  ```
  DETAILED_DIFFERENCES FALSE
  ```
- COLOR : Specifies the color to be used for the result of a logical operation. Syntax is:
  ```
  COLOR <operation> <color index in the color palette >
  ```

For example, the following line specifies the color with index 10 for the AND operation:

```
COLOR MASTER_AND_COMPARE 10
```

- COMPARE : Specifies the class and subclass of the compare layer and the master layer. Syntax is:

  ```
  COMPARE <master class> <master subclass> <compare class> <compare subclass>
  ```

  For example, the following line specifies the master classes as `ALL CONDUCTORS` and `TOP_COND` and the compare classes as `CONDUCTOR` and `TOP_COND`.

  ```
  COMPARE 'ALL CONDUCTOR' TOP_COND CONDUCTOR TOP_COND
  ```

- OUTPUT : Specifies the destination or output class and subclass and, optionally, the comparison operations. Syntax is:

  ```
  OUTPUT <Destination class> <Destination subclass> [<operations>….]
  ```

  For example, the following line specifies the destination class as `COMPONENT GEOMETRY` and the subclass as `RESULTS`. It also specifies the operations `MASTER_AND_COMPARE`, `MASTER_ANDNOT_COMPARE`, and `COMPARE_ANDNOT_MASTER`

  ```
  OUTPUT 'COMPONENT GEOMETRY' RESULTS MASTER_AND_COMPARE MASTER_ANDNOT_COMPARE
  COMPARE_ANDNOT_MASTER
  ```

  The destination layer will be created if it does not exist, and can be used in subsequent compare statements. If no arguments are specified, the defaults will be used.

- FILTER Specifies a filter to apply. Syntax is:

  ```
  FILTER <DATABASE> <FILTER_TYPE> <MODIFERS….>
  ```

  The `DATABASE` can be either `MASTER` or `COMPARE`.

  The valid `FILTER_TYPE` values and their respective modifiers are:

  - NETS: Modifier is a list of nets.
  - OBJECTS: Modifier is a list of any of the following types of object: PINS, SHAPES, VIAS, FINGERS, CLINES, or LINES
  - COMPONENTS: Modifier is a list of one or more component reference designators.
  - REGION: Modifier is a list of four values, specifying the lower-left corner and upper-right corner of a region, in the format:

    ```
    MINX:<value> MINY:<value> MAXX:<value> MAXY:<value>
    ```

    If units are not supplied, the units of the master database are assumed.

In the following example, the first line defines an `OBJECTS` filter for the `COMPARE` layer for the objects of types `VIAS`, `FINGERS`, and `SHAPES`. The second line defines a `NETS` filter for the `MASTER`

layer for the nets `VDD` and `VSS`. The third line defines a `COMPONENTS` filter for the `MASTER` layer for the component with the reference designator `DIE`.

```
FILTER COMPARE OBJECTS VIAS FINGERS SHAPES
FILTER MASTER NETS VDD VSS
FILTER MASTER COMPONENTS DIE
```

- OPTION : Specifies an option to apply. Syntax is:
  OPTION <DATABASE> <OPTION_TYPE> <MODIFIERS>
  The DATABASE can be MASTER or COMPARE or DESTINATION. The valid OPTION_TYPE values and their respective modifiers are:
  - POSITIVE / NEGATIVE: Layer is a positive or negative layer. Positive is default if neither option defined.
  - CROSSHATCH_AS_SOLID: Crosshatched shapes will be treated as solid-fill.
  - EXPAND / CONTRACT: Apply an expansion or contraction to shapes on the specified layer. Modifier is the amount to expand or contract. If no units are supplied with modifier, units of the master database will be used.
  - MIN_HEIGHT / MIN_WIDTH / MIN_SIZE: Apply a size filter. Any polygons not meeting the required minimum size restrictions will be removed and the count of filtered items reported in the log file. Modifier is the minimum linear dimension required.
    MIN_SIZE will set both MIN_HEIGHT and
    MIN_WIDTH to the value specified.
    If no units are supplied with modifier, units of the master database will be used.
  - ROTATE: Apply a rotation to the polygon set. All polygons will be rotated by the specified amount around the center of the region being considered. Modifier is the angle of rotation, given in degrees.
  - MIRROR: Mirror the polygons on this layer. Mirroring is done through the center of the region being considered.

  In the following example, the first line defines a `CROSSHATCH_AS_SOLID` option for the compare layer. The second line defines the `EXPAND` option with the value `25um`.

```
OPTION COMPARE CROSSHATCH_AS_SOLID OPTION MASTER EXPAND 25um
```

Related Topics

- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare
- Batch Layer Compare Dialog Box

Logical Operations in Interactive and Batch Modes of Layer Compare

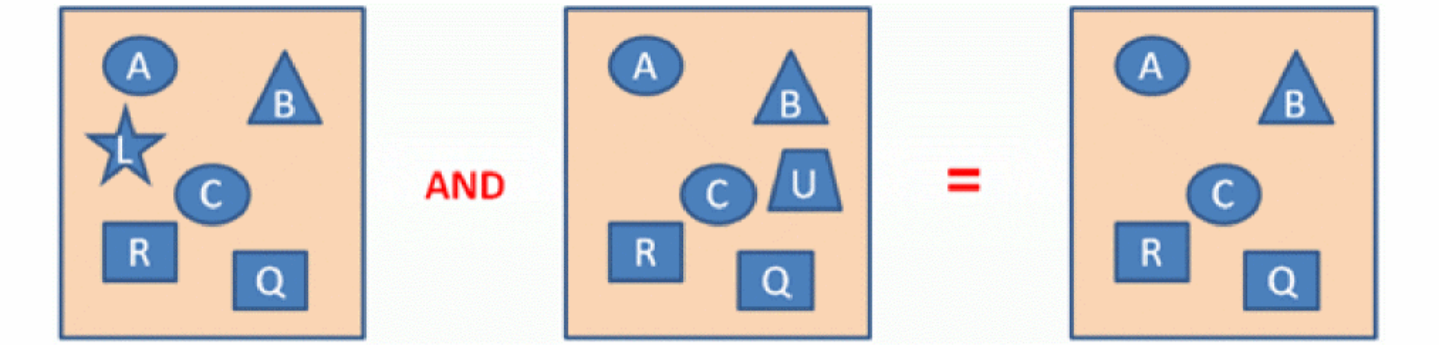
You can perform the following logical operations:

- AND
- OR
- XOR
- Master ANDNOT Compare
- Compare ANDNOT Master (Available only for the batch mode)

The AND Operation

Available for both the Interactive *Layer Compare* and *Batch Layer Compare* commands. The control file keyword is `MASTER_AND_COMPARE`.

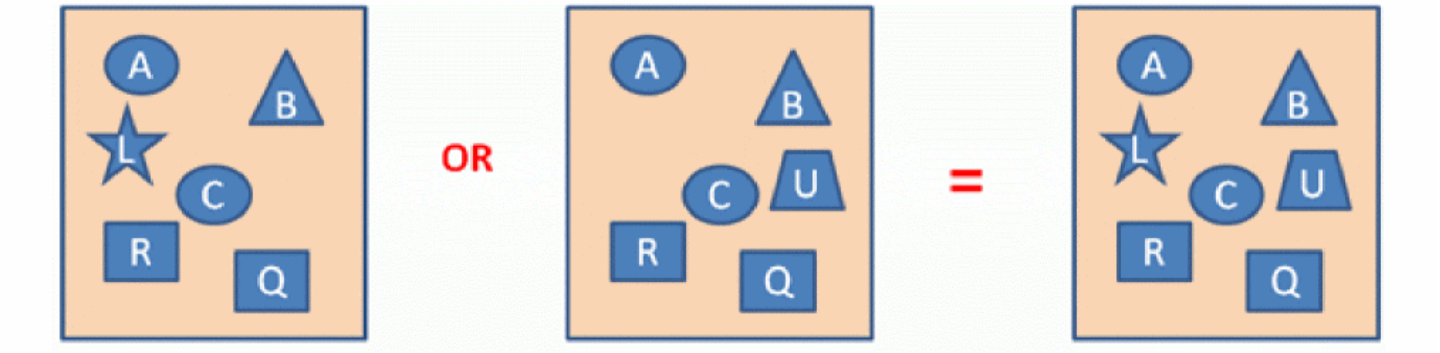
This comparison results in only those selected items present in both the reference and compare layers. Considering the first square is the reference, the second is the compare, and the third is the destination:



The OR Operation

Available for both the *Interactive Layer Compare* and *Batch Layer Compare* commands. The control file keyword is `MASTER_OR_COMPARE`.

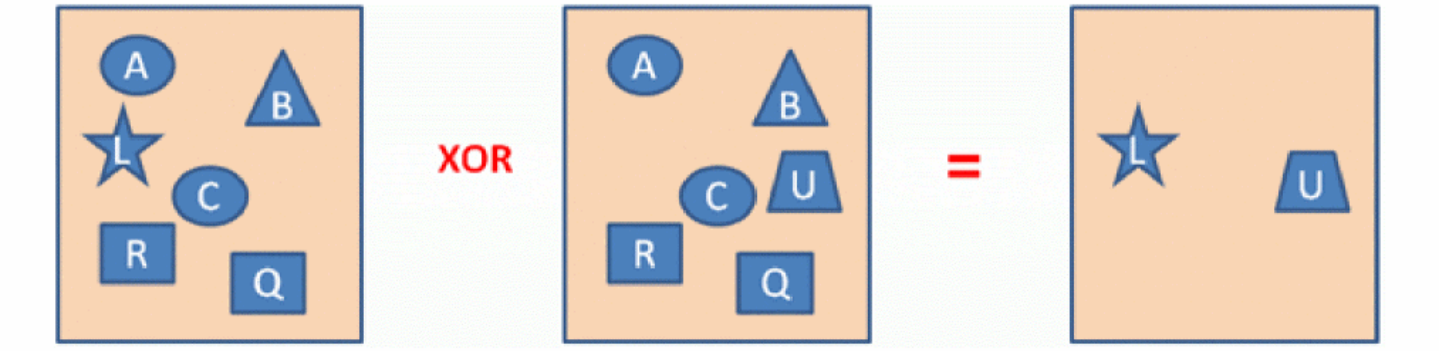
This comparison results in all selected items present in any one of the reference or the compare layers. Considering the first square is the reference, the second is the compare, and the third is the destination:



The XOR Operation

Available for both the *Interactive Layer Compare* and *Batch Layer Compare* commands. The control file keyword is `MASTER_XOR_COMPARE`.

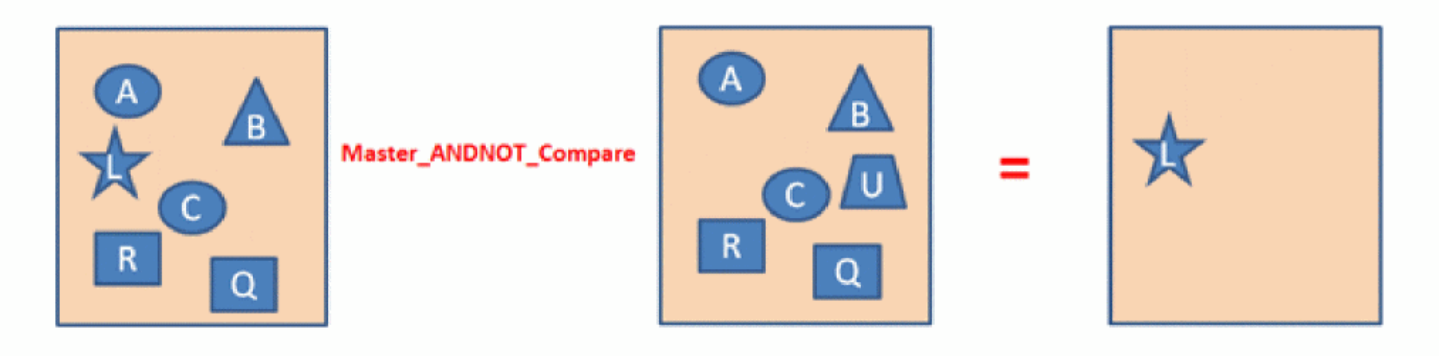
This comparison results in all selected items present in either the reference layer or the compare layer but not both. Considering the first square is the reference, the second is the compare, and the third is the destination:



The Master ANDNOT Compare Operation

Available for both the *Interactive Layer Compare* and *Batch Layer Compare* commands. The control file keyword is `MASTER_ANDNOT_COMPARE`.

This comparison results in those selected items present in the reference layer but not present in the compare layer. Considering the first square is reference, second is the compare, and the third is destination:

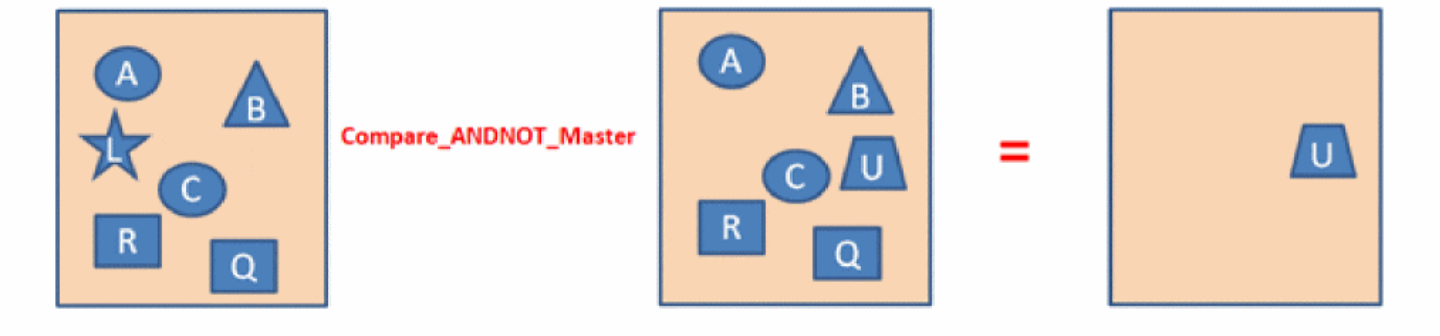


The Compare ANDNOT Master Operation

Available for only the *Batch Layer Compare* command. The control file keyword is

`COMPARE_ANDNOT_MASTER`.

This comparison results in those selected items present in the compare layer but not present in the reference layer. Considering the first square is the reference, the second is the compare, and the third is the destination:



Related Topics

- layer compare
- layer compare batch
- Batch Layer Compare Control File

Adding a New Comparison in Batch Compare

To add a new comparison, do the following steps:

1. In the Batch Layer Compare dialog box, choose *Add* from the pop-up menu of the *Comparisons* folder.
2. In the Batch Compare Layers dialog box that appears, select the classes and sub-classes for the reference and compare layers.
   You can select any existing layers in the design for the master and compare layer fields. You can also select from the following two special layers as master or compare:
   - ALL_CONDUCTOR: Combines the pin, via, and conductor classes for the specified etch layer name.
   - ALL_SOLDERMASK: Combines the via, pin, substrate geometry, and component for the geometry solder mask layer specified.
3. Select the class and sub-class for the destination layer.
   You can select any existing layer in the active drawing as a destination. You can also specify a new subclass name to have the tool create this layer on the fly for generating the output.

   > ⓘ You can create new subclasses only on non-etch layers. If the destination class is CONDUCTOR, you can only select an existing layer name.

4. Click *OK*.
   The new comparison is listed in the Batch Layer Compare dialog box under Comparisons folder with three new folders for Master, Compare, Destination, and Operations.

Related Topics

- Batch Compare Layers Dialog Box
- Batch Layer Compare Control File
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Removing All Comparisons in Batch Layer Compare

To remove all comparisons, do the following steps:

1. In the Batch Layer Compare dialog box, choose *Remove ALL* from the pop-up menu of the *Comparisons* folder.
   All listed comparisons are removed.

Related Topics

- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Changing Master, Compare, or Destination Layer in Batch Layer Compare

To change the reference (Master), compare, or destination (output) layer, do the following steps:

1. In the Batch Layer Compare dialog box, choose *Change Layer* from the pop-up menu of the listed master, compare, or destination folder.
2. In the dialog box that appears, select the Class and Subclass for the reference, compare, or destination (output) layer depending on the folder from which you chose Change Layer.
   You can select any existing layers in the design for the master and compare layer fields. You can also select from the following two special layers as master or compare:
   - ALL_CONDUCTOR: Combines the pin, via, and conductor classes for the specified etch layer name
   - ALL_SOLDERMASK: Combines the via, pin, substrate geometry, and component for the geometry solder mask layer specified.

   You can select any existing layer in the active drawing as a destination. You can also specify a new subclass name to have the tool create this layer on the fly for generating the output.

   > ⓘ You can create new subclasses only on non-etch layers. If the destination class is CONDUCTOR, you can only select an existing layer name.

3. Click *OK* to save the changes and close the dialog box.
   The listed layer class and subsclass names are changed to the new values in the Batch Layer Compare dialog box.

Related Topics

- Batch Layer Compare Control File
- Batch Compare Layer Dialog Box for Change Layer
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Specifying Expansion or Contraction to Shapes in Batch Layer Compare

To apply an expansion or contraction to shapes on a specified layer of Master, Compare, or Destination, do the following steps:

1. In the Batch Layer Compare dialog box, choose either *Expand Value* for expansion or *Contract Value* for contraction from the pop-up menu of the listed master, compare, or destination folder.
2. In the dialog box that appears, specify the value.
3. Click OK to save the change and close the dialog box.
   The option is listed under the folder with the specified value.

Related Topics

- Batch Layer Compare Control File
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Specifying Minimum Shape Area and Size in Batch Layer Compare

To specify the minimum area for shapes on a layer of Master, Compare, or Destination, do the following steps:

1. In the Batch Layer Compare dialog box, choose *Minimum Shape Area* for area or *Minimum Share Size* for size from the pop-up menu of the listed master, compare, or destination folder.
2. In the dialog box that appears, specify the value.
   Any polygons not meeting the required minimum size and area restrictions will be removed and the count of filtered items reported in the log file when the batch layer compare command runs.
3. Click *OK* to save the change and close the dialog box.
   The option is listed under the folder with the specified value.

Related Topics

- Batch Layer Compare Control File
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Rotating Shapes using Batch Layer Compare

You can rotate all polygons by a specified angle around the center of the region being considered. To apply a rotation, perform the following steps:

1. In the Batch Layer Compare dialog box, choose *Rotation* from the pop-up menu of the listed master, compare, or destination folder.
2. In the dialog box that appears, specify the angle of rotation.
3. Click *OK*.
   The Rotation option is listed under the folder with the specified angle.

Related Topics

- Batch Layer Compare Control File
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Mirroring Shapes using Batch Layer Compare

You can mirror all polygons around the center of the region being considered. To apply mirroring, perform the following steps:

1. In the Batch Layer Compare dialog box, choose *Mirror* from the pop-up menu of the listed master, compare, or destination folder.
   The Mirror option is listed under the folder.

Related Topics

- Batch Layer Compare Control File
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Filtering Objects by Net, Type, Components, and Region using Batch Layer Compare

You can filter objects by net, object type, owning component, and region. Filtering can be applied only on master and compare folders.

To apply a filter, do the following steps:

1. In the Batch Layer Compare dialog box, choose any one of the following from the pop-up menu of a listed master or compare folder:
    - *Object Filter*: To filter based on object type such as shapes, fingers, pins, vias, clines, or lines.
    - *Net Filter*: To filter based on nets.
    - *Component Filter*: To filter based on owning components..
    - *Region Filter*: To filter based on a specified region.
2. For filtering by object, net, or component, select from the available list in the dialog box to the selected list and click *OK*.
   To filter by region, specify the lower left and upper right coordinates of the region and click *OK*.
   The specified filter is listed under the folder.

Related Topics

- Batch Layer Compare Control File
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Use Batch Layer Compare to perform multiple, sequential comparisons between layers of the same design or between layers in two different designs.

1. Choose *Tools – Layer Compare – Batch Layer Compare*
   The Batch Layer Compare dialog box appears.
2. Select the comparison database.

3. Specify the control file.
   Use the operations tree if you do not have a control file.
4. Add comparisons or remove all comparisons if needed.
5. Right-click *Comparisons* and choose *Add* to open the dialog box to specify new comparison.
6. Right-click *Comparisons* and choose *Remove All* to remove all comparisons.
7. Specify or change the Master layer; Right-click and choose *Change Layer*.
8. Specify or change the Compare layer; Right-click and choose *Change Layer*.
9. Specify or change the Destination layer; Right-click and choose *Change Layer*.
10. Add options to the destination layer; Right-click and choose the options from the listed items.
11. Specify one or more operations.
12. Specify the destination layer.
13. Click *Compare*.

Related Topics

- Batch Layer Compare Control File
- Viewing and Resolving Differences in Batch Layer Compare

layer compare diff

Displays and steps through the differences, identified by the `layer compare batch` command for a compare database. You can use this command to resolve the identified differences.

⚠ This command is to review differences between two different designs and can be used only with the output of the `batch layer compare` command.

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

Related Topics

- Viewing and Resolving Differences in Batch Layer Compare
- Compare Differences Dialog Box
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Compare Differences Dialog Box

Access using:

- Menu path: *Tools – Layer Compare – Difference Walker*

Compare Database	Select the compare database. Lists all the databases for which there are differences recorded in the active drawing. If multiple layer compare has been run against multiple different databases, each is listed. When the last difference versus a given compare database is removed, that database is removed. The default database is the first alphabetically.
Pan/Zoom in Compare Design	Select to zoom and update visibility of the compared design's display window to match the master design, as you move between the differences. The comparison database must have been opened in another window using the *Open Compare Database* button.
Open Compare Database	Click to browse to and open the comparison database in another window to see the differences and compare the full area surrounding a flagged difference. This compare window closes when you change the comparison database or exit from the layer compare batch command. ⚠ Opening a new window using this button uses an additional tool license.
Local Class/ Subclass	Select from the layers in the active drawing for which there are differences in the selected compare database. The class and subclass fields automatically filter to only those layers that have differences. The default layer is the first found alphabetically. You can also select from the following two special layers as master or compare: • ALL_CONDUCTOR: Combines the pin, via, and conductor classes for the specified etch layer name. • ALL_SOLDERMASK: Combines the via, pin, substrate geometry, and component for the geometry solder mask layer specified.

Compared Class/ Subclass	Displays the layers in the active compare database for which there are differences with the local layer. Most often, there will only be one compare layer and these fields do not need to be modified. You can also select from the following two special layers as master or compare: • ALL_CONDUCTOR: Combines the pin, via, and conductor classes for the specified etch layer name❑ • ALL_SOLDERMASK: Combines the via, pin, substrate geometry, and component for the geometry solder mask layer specified.
Differences	Displays the total number of active differences, the selected difference, and a list of all the differences.^The list of all the differences in the design is sorted by the size of the difference. The text entry for a difference lists the center point of the difference (for reference) and the size (area) of the region. As a difference is acknowledged, it is removed from this list.
Previous Difference	Click to move to the previous difference in the list and make it active. If the first difference is active, this wraps to the bottom of the list.
Next Difference	Click to move to the next difference in the list and make it active. If the last difference is active, this wraps to the top of the list.
Selected difference information	Displays information about the selected difference. Information includes the operation which resulted in this difference, the size of the bounding box, and the area of the region. Select to either zoom and center on difference (default) or to center window on difference. You can also choose to show selected difference layers only (not selected by default) and to restore original visibility when you exit the layer compare tool (selected by default).
Zoom and center on difference	Select to zoom in and center of the difference. Selected by default. Mutually exclusive with Center window on difference.
Center window on differece	Select to center the window on difference. Not selected by default. Mutually exclusive with Zoom and center on difference.
Show selected difference layers only	Select to change the visibility of layers in the design so that only the layer in the active drawing and the layer which contains the difference region shapes are active.

Restore original visibility on exit	Select to restore visibility, on exit, of all layers in the design to the state they were in immediately before launching the tool. This is selected by default.
Acknowledge Difference and Remove	Click to sign off a difference if the active difference has been corrected in the active drawing, or if the difference does not require a change in the active design. This removes the shape from the design and removes the entry for this difference in the list on the form.
OK	Click to save database and close the dialog box.
Cancel	Click to close the dialog box without saving the changes.

Related Topics

- Viewing and Resolving Differences in Batch Layer Compare
- layer compare diff
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

Viewing and Resolving Differences in Batch Layer Compare

To step through and resolved the differences identified by Batch layer Compare, do the following:

1. Choose *Tools – Layer Compare – Difference Walker*.
   The Compare Layer Differences dialog box appears.
2. Select the compare database.
   You can click *Open Compare Database* to open the compared database in another window.
3. Specify the Local and Compared classes.
4. Select a difference from the list.
   You can resolve the difference or acknowledge and remove it from the list.
5. Click *Previous difference* or *Next difference* to walk through the differences.
6. Click *OK* to update changes and close the dialog box.

Related Topics

- Compare Differences Dialog Box
- layer compare diff
- Performing Multiple Simultaneous Comparisons Between Designs or Layers using Batch Layer Compare

layer estimation

The `layer estimation` command produces a report that estimates the number of layers needed to place and route a design.

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

Access Using

- Menu path: *Route – Flip-Chip Routing Layer Estimation*

Displays a report that estimates the number of layers needed to place and route a design.

Creating Layer Estimation Report

To generate a layer estimation report, do the following:

1. Choose *Flip-Chip Route – Routing Layer Estimation*.
   The Layer Estimation dialog box appears with a report similar to the following:

```
Include Nets with Voltage Property: No

Include Unassigned Pins:            No

Escape Distance From Die Outline:   0



Component = U1, # Pins = 1443, # Attempted Escapes = 1443


 Subclass             # Escapes % Escapes Pad Size(s)

 -------------------- --------- --------- -----------

 TOP                        292     20.2% 0.0450

 LA02                       256     17.7% 0.0450

 LA03                       114      7.9% 0.0450

 LA04                       108      7.5% 0.0450

 LA05                       194     13.4% 0.0450

 BOTTOM                      36      2.5% 0.0450

 Unsuccessful               443     30.7%
```

```
End of Layer Estimation Report.
```

layer priority

The `layer priority` command assigns a display priority to each layer. Overrides the default display order. Elements are drawn based on their assigned layer priority.

Your assignments are saved with the board. Always-on-top elements include:

- Temporary or transitory objects such as DRCs or ratsnests
- Active subclass

⚠ If an object present on a lower priority layer is highlighted or assigned color, the object gets a higher priority than the layer priority order.

Related Topics

- Assigning a Display Priority To Layers

Display Priority Dialog Box

Access Using

- Menu Path: *Display – Layer Priority*

Default display priority	Shows in a collapsing tree view, the default display priority for all the layers in your design.Where a number of layers are listed, the display area shows a folder icon. You can choose all layers by clicking the check box next to the folder icon or individual layers by clicking the check box next to the layer name.
Prioritized Layers	Specifies the layers that are drawn before those contained in the *Default Priority* list.
->	Moves the chosen layer from the *Default Priority* list to the *Prioritized Layers* list.
Up	Swaps the chosen layer with the layer immediately above it in the *Prioritized Layers* list.
Down	Swaps the chosen layer with the layer immediately below it in the *Prioritized Layers* list.
Top	Moves the chosen layer to the top of the *Prioritized Layers* list.
Bottom	Moves the chosen layer to the bottom of the *Prioritized Layers* list.
<-	Removes the chosen layer from the *Prioritized Layers* list.
<<-	Removes all layers from the *Prioritized Layers* list.
OK	Saves your changes and closes the dialog box.
Cancel	Exits the dialog box.
Apply	Saves layer priority assignments with the board.

Assigning a Display Priority To Layers

Using *Display Priority* dialog box you can control the order in which layers are drawn in your design. For example, the default layer at the top of the list appears on top of the layer that appears second in the list.To assign display priority to layers, do the following:

1. Choose *Display – Layer Priority.*
   Alternatively, type `layer priority` in the Command window.
   The *Display Priority* dialog box displays where you can assign a display priority to a layer.
2. Choose a layer from the *Default Priority* list and click -> to move it to the *Prioritized Layers* list.
3. Continue to move as many layers as required.
   The layers in the *Prioritized Layers* list are drawn before any layers in the *Default Priority* list.
4. Reorder any layers in the *Prioritized Layers* list by choosing layers.
5. Click <- to remove several layers from the *Prioritized Layers* list.
6. Click <<- to remove all layers from the *Prioritized Layers* list.
7. Click *Apply* to save layer priority assignments with the board.
   You can reuse the priority settings by exporting the settings into a .rpm file and then importing the file to a new design.

Related Topics

- layer priority
- `param out`
- param in
- techfile

layout wizard

The `layout wizard` command creates a design layout using the Board wizard.

Board Wizard Window

Access Using

- Menu path: *File – New*
  Select *Board (wizard)* from the *New Drawing* dialog box.

Board Wizard - Introduction	This page summarizes the capabilities and operating behavior of the wizard.	
Board Wizard - Template		
	Optionally enter a board template to import in this board	• Select *Yes* to import a template file. • Select *No* is selected by default.
Board Wizard - Tech File/Parameter File		
	Optionally enter a tech file to import in this board.	• Select *Yes* to import a tech file. • Select *No* is selected by default.
	Optionally enter a parameter file in this board.	• Select *Yes* to import a parameter file. • Select *No* is selected by default.

Board Wizard - Board Symbol	⚠ If you imported template data that included a board outline, you should not import a symbol from this dialog box. Doing so results in your new layout containing two board outlines.	
	Optionally enter a board symbol (.bsm) to import in this board.	• Select *Yes* to import a board symbol. • Select *No* is selected by default. ⓘ The *.bsm* template file must include the following: ◦ Board outline ◦ Company-standard design sheet
Board Wizard - Import Default Data	Displays only if you chose a data file (template, technology, or board symbol) to import into your new layout.	
	Import default parameter data now	Loads the selected data files at this stage in the process. This makes the data available for viewing and modifying in the subsequent pages. This option is selected by default.
	Import the parameter data at the end of the wizard	With this action, some of the parameters shown on subsequent pages and contained in the loaded data files, are disabled by the wizard. They are not available for viewing or modifying. Instead, those parameters are loaded into your new layout at the end of the wizard process when the board is created. This is not selected by default.
Board Wizard - General Parameters	Data that you specify in the General Parameters page takes precedence over the parameters in the data files. Select the units for the board drawing.	
	Units	Specify the unit for the design. *Mils* is selected by default.
	Accuracy	Determined by chosen design units. The value is `1` for Mils.

	Select the drawing size for the board drawing.	
	Size	Set the size of the drawing. *A* is selected by default. The field is disabled if a data file is chosen.
	Specify the location of the origin for this drawing.	Select one of the available options. The default selection and available options depend on the loaded data files.
	Next	Click to proceed to the General Parameters (Continued) page. However, one of the following two pages might appear if no data exist in `BOARD GEOMETRY/DESIGN_OUTLINE` or `ROUTE KEEPIN` and `PACKAGE KEEPIN`. • Board Outline page Appears only if no data files are loaded or data does not exist on `BOARD GEOMETRY/DESIGN_OUTLINE`. • Keepins page Appears if a loaded data file contains geometry on `BOARD GEOMETRY/DESIGN_OUTLINE`, but no data is present on `ROUTE KEEPIN` and `PACKAGE KEEPIN`.
Board Wizard - General Parameters (Continued)	You can add a maximum of 128 layers using the wizard when the Etch layer count is enabled. This function is disabled if more than two etch layers are defined in the loaded data. Artwork files are defined for each etch layer and take the name of the defined layer, as specified in the data file or in the Etch Cross-section Details page.	
	Grid spacing	Specify the grid spacing for etch and non etch layers. The default value is `1 Mils`. The option is grayed if a data file is selected.
	Etch layer count	Specify the number of etch layers. Etch layers can also include power planes. The default count is `2`. The option is grayed, if more than two etch layers are defined in the loaded data file.

	Generate default artwork films	Select to generate artwork films for the layers based on default definitions. This option is selected by default.
	Don't generate artwork films	Select if you do not want to generate artwork films for the layers based on default definitions. This option is not selected by default.
	Next	Click to proceed to the Etch Cross-section details page. However, one of the following two pages might appear if no data exists in `BOARD GEOMETRY/DESIGN_OUTLINE` or `ROUTE KEEPIN` and `PACKAGE KEEPIN`. • Keepins page Appears if a loaded data file contains geometry on `BOARD GEOMETRY/DESIGN_OUTLINE`, but no data exists on `ROUTE KEEPIN` and `PACKAGE KEEPIN`. • Board Outline page Appears only if no data files are loaded or data does not exist on `BOARD GEOMETRY/DESIGN_OUTLINE`. • Etch Cross-section Details page
Board Wizard - Keepins	This page appears if a loaded data file contains geometry on `BOARD GEOMETRY/DESIGN_OUTLINE`, but no data on `ROUTE KEEPIN` and `PACKAGE KEEPIN`.	
	Route keepin	Enter the value for the route keepin distance from the board edge in design units.
	Package keepin	Enter the value for the package keepin distance from the board edge in design units.
Board Wizard - Board Outline	This page appears only if no data files are loaded or data does not exist on `BOARD GEOMETRY/DESIGN_OUTLINE`.	
	Circular Board	Click to define a circular board outline.

	Rectangular Board	Click to define a rectangular board outline.
Board Wizard - Circular Board Parameters		
	Diameter	*Specify* a value for the diameter of the circular board outline from the board edge in design units.
	Route keepin distance	*Specify* a value for the route keepin distance from the board edge in design units.
	Package keepin distance	*Specify* a value for the package keepin distance from the board edge in design units.
Board Wizard - Rectangular Board Parameters		
	Width	*Specify the width of the rectangular board.*
	Height	*Specify the height of the rectangular board.*
	Corner cutoffs	Click this option if your board contains corner cutoffs.
	Cut length	*Specify* a value for cut lengths.
	Package keepin distance	Specify the package keepin distance from the board edge.
	Route keepin distance	Specify the route keepin distance from the board edge.

	Next	Click to proceed to the summary page. However, the following two pages might appear depending on the absence or presence of etch layer and constraint definitions in loaded data files: • Etch Cross-section Details page Appears if imported data files defines two or fewer etch layers. • Spacing Constraints page Appears if constraints are not defined.
Board Wizard - Etch Cross-section Details	Define etch layer names and types from the layers that you added in the General Parameters page. You cannot change the name of the top and bottom layers, which are created with default names. You can rename any other layer that you created as a routing layer or power plane, with the option to define power planes as negative layers. This page does not appear if the loaded template or tech file defines more than two etch layers.	
	Layer name	Define a name for each etch layer.
	Layer type	Define a type for each etch layer.
	Generate negative layers for power planes	Check to define power planes as negative layers.
	Next	Click to proceed to the Summary page. However the Spacing Constraints page might appear if constraints are not defined.
Board Wizard - Spacing Constraints	Define basic spacing constraints and a default via padstack for your layout. ⚠ This page does not appear if the imported template or tech file defines constraints.	

	Minimum line width	Specify a value for the minimum line width. The same value is automatically assigned to the other spacing constraints. If necessary, modify the other spacing constraints.
	Minimum line to line spacing	Specify a value for the minimum line to line spacing.
	Minimum line to pad spacing	Specify a value for the minimum line to pad spacing.
	Minimum pad to pad spacing	Specify a value for the minimum pad to pad spacing.
	Default via padstack	Choose a default via padstack from the Board Wizard Padstack browser, or by specifying the padstack name in the via padstack field.
	Next	*Click to proceed to Summary page.* However, the Board Outline page might appear if `BOARD GEOMETRY/DESIGN_OUTLINE` does not contain any data.
Board Wizard - Summary	This is the final page of the board wizard, containing the name of your new layout file	
Finish	Click to create the new layout. The layout file is created in your current working directory. The layout file overwrites any existing files with the same name.	
Back	Click to return to the previous page.	
Cancel	Click to end the wizard process.	

Related Topics

- param in
- techfile

Creating a New Layout Using Board Wizard

1. Choose *File – New.*
   The *New Drawing* dialog box appears.
2. Specify a drawing name.
3. Select *Board (wizard)* and click *OK*.
   The Introduction page of the Board wizard appears.
   Follow the instructions for entering the required data on each of the wizard's dialog boxes, then click *Next* to move forward to the next dialog box. At any time before finishing the process, you can click:
   - *Back* to review or modify data
   - *Cancel* to end the wizard process.
     A new drawing containing no design data opens in the editor with the name you specified in the *New Drawing* dialog box.
4. When you have completed the last step in the wizard process, click *Finish*.
   The drawing is automatically opened in the layout editor.

Importing a Board Template File

1. Click *Yes* in the Template dialog box, and click the browse button.
   The Board Wizard Template Browser appears, listing all the template files in the path `WIZARD_TEMPLATE_PATH`. Reset this path to the location you want to keep your templates in, if different from the default location. (You must restart the layout editor to activate the change.)
2. Choose a template from the list, and click *OK*.
   The file name displays in the text field of the *Template* dialog box.
3. Click *Next*.

Importing a Technology File/Parameter File

1. Click *Yes* in the *Do you have a tech file that you would like to import in this board?* field to import a specified tech file. Otherwise, click *No*.
2. Click the ellipses (**...**) to display the file browser from which you can choose an existing filename. Or enter a tech file name.
   When you click the ellipses (**...**), the Board Wizard Tech File Browser appears, listing all the technology files in the path TECHPATH. Reset this path to the location you want to keep your tech files in, if different from the default location. You must restart the layout editor to activate the change.
3. Choose a tech file from the list, and click *OK*.
4. The file name appears in the text field.
5. Click *Yes* in the *Do you have a parameter file that you would like to import in this board?* field to import a specified database parameter file. Otherwise, click *No*.
6. Click the ellipses (**...**) to display the file browser from which you can choose an existing filename. Or enter a database parameter file name.
   When you click **...**, the Board Wizard Parameter File Browser appears, listing all the database parameter files in the path PARAMPATH. Reset this path to the location you want to keep your tech files in, if different from the default location. You must restart the layout editor to activate the change.
7. Choose a parameter file from the list, and click *OK*.
   The file name appears in the text field.
8. Click *Next*.

Importing a Board Symbol

1. Click *Yes* in the *Board Symbol* dialog box, and click the browse button.
   The Board Wizard Mechanical Symbol Browser appears, listing all the mechanical symbols in the path PSMPATH. Reset this path to the location you want to keep your board symbols in, if different from the default location. You must restart the layout editor to activate the change.
2. Choose a symbol from the list, and click *OK*.
   The file name displays in the text field of the Board Symbol dialog box.
3. Click *Next*.
   One of the following dialog boxes appears:
   - Import Default Data
   - General Parameters

Related Topics

- layout wizard
- techfile
- param in

lead editor

The `lead editor` command adds component lead contact area information in Allegro X PCB Editor or the Allegro Symbol Editor.

Use the command to do the following:

- Define the lead type
- Define lead physical details
- Position the lead in the symbol
- Graphically view the lead contact area
- Enable DFM lead checks

Supported Lead Types

You can assign leads to pins in a design.

In addition to ball, bump, and pillar, the following lead types are supported:

Ball-collapsing	Type of BGA ball where the pad is larger than the ball. Use this lead type for BGAs with pitch less than or equal to `.50 mm`. Provides an annular ring.
Ball non-collapsing	Type of BGA ball where the pad is smaller than the ball. Use this lead type for BGAs with pitch more than or equal to `.65 mm`. Define a non-solder mask larger than the pad for non-collapsing balls.
Butt lead	Also called an I lead, this is a Dual Input Pin (DIP) lead for Surface Mount Technology (SMT). It is positioned perpendicular to the pad.
Column	Column Grid Array (CGA) or Ceramic Column Grid Array (CCGA), non-collapsible lead used for Surface Mount Technology (SMT).
Corner Concave	Oscillators Corner Concave (OSCCC) lead.
Cylindrical end cap	Metal Electrode Leadless Face (MELF) cylindrical end cap leads.
Flat lead	Used in Small Outline Transistor Flat Lead (SOTFL) or Small Outline Diode Flat Lead (SODFL) packages. Leads come out of the package.
Flat lug	Flat thermal leads on Decawatt Package (DPAK).
Flat no lead bottom	Used in Small Outline No Lead (SONL) and with Pullback Quad Flat No-Lead (PQFN) packages. Lead not exposed on side of the package. Also called Pull-back. Can be in two shapes, rectangle or bullet (D-shape).
Flat no lead edge	Used in Small Outline No Lead (SONL) and Quad Flat No-Lead (QFN) packages. Leads start under the package and end at the edges of the package, not exposed on sides.
Flat thermal	Used as thermal pad in dual and quad flat thermal families: Quad Flat No-Lead (QFN), Quad Flat Packages (QFP), Small Outline No-lead (SON), and Small Outline Packages (SOP). Embedded in the package.

Gull-Wing	Used in surface mount packages: Small Outline Integrated Circuit (SOIC), Quad Flat Packages (QFP), Ceramic Quad Flat Packages (CQFP), Small Outline Transistors (SOT), and Small Outline Diodes (SOD). Extends marginally out of the package before turning down slightly and then out again.
J-Lead	Used in surface mount packages, such as SOIC and Plastic Leaded Chip Carrier (PLCC). Goes straight down from the package edge before folding up. Not preferred for high-speed designs.
No Lead	Used in flat no-lead packages, such as Quad Flat No-lead (QFN) and Dual Flat No-lead (DFN). Also called leadless packages. Leads are at the bottom of the package instead of the periphery.
No connect	Any lead that is not connected to the PCB.
Other Surface	Used for Surface Mount Technology (SMT) if the lead type does not fit in any other defined types.
Press Fit	A press fit lead having either a solid or compliant press-in section is pressed into a plated through hole (PTH) on the PCB. The hole is smaller than the pin.
Rectangular end cap	Used in rectangular-end or square-end chip style packages, mainly resistors, capacitors, and inductors.
Ribbon L inward	Used in molded body components: Molded Inductors (INDM), Diodes (DIOM) and Polarized Capacitors (CAPMP). It goes down the edge of the package and then turns inwards under the package. Also referred to as Inward Flat Ribbon L.
Ribbon L outward	Used in SOT and SOD packages to reduce footprint size. Goes down the edge of the package and then turns outwards. Also referred to as Outward Flat Ribbon L.
Side concave	Used in Concave Chip Array (RESCAV, CAPCAV, INDCAV, OSCSC) packages. Embedded in an edge as a concave, usually running from the top to the bottom of the package.
Side convex	Used in Convex Chip Array (RESCAXE, RESCAXS) packages. Extends out of an edge, usually running from the top to the bottom of the package.
Side flat lead	Used in Flat Chip Array (RESCAF, CAPCAF, INDCAF) packages. Runs from the top to the bottom on the perimeter of the package.
Through other	Used in discrete packages and connectors. A PTH lead with a cross section that is not rectangular or round.

Through rectangular	Used in discrete packages and connectors. A PTH lead with a rectangular cross section.
Through round	Used in discrete packages and connectors. A PTH lead with a round cross-section.
Under body outward L	Used in Aluminum Electrolytic Capacitors and 2-pin SMT Crystals. Starts under the component and turns out towards an edge.

Related Topics

- Assigning Leads to Pins

Assign Pin Leads Dialog Box

Access Using

- Menu path: *Setup – Lead Editor*

Available Packages	Select a package to list its pins for lead assignment. Lists the packages available in the library and database, depending on the options you choose. You can filter the list using the *Filter package name* field at the bottom.
Filter package name	Use to filter the list of packages.
Show packages from database	Select to show packages from the database. This option is selected by default.
Show packages from library	Select to show packages from library. This option is not selected by default.
Pins	Select pins for lead assignment. Lists the pins of the selected symbols or packages to assign leads.
Quick view	Displays the location of the lead contact area(s) as positioned in the symbol. Use offsets to correctly position the lead geometry to its exact location. The graphics display of the symbol utilizes the color selection in the current Allegro drawing. Other window command such a turning on/off layer display, zoom and pan are available in the graphical display.
Assign leads	Choose a lead type to assign to the selected pins.
Parameters	Define the lead contact geometry parameters.
X-Offset from pad center	Specify the X-offset from the pad center.

Y-Offset from pad center	Specify the y-offset from the pad center.
Apply	Click to apply the settings.
Delete lead	Click to delete the applied lead.
Help	Shows a description of the lead type selected in *Assign leads*.
Dynamic View	Displays the contact area geometry before applying the geometry to the symbol.

Assigning Leads to Pins

To assign a lead to pins, do the following;

1. Choose *Setup – Lead Editor*.
2. Select a package.
3. Select pins to assign lead.
4. Choose a lead type from *Assign leads*.
5. Specify the settings in the *Parameters* box.
6. Click *Apply*.

Related Topics

- lead editor

lef lib

The `lef lib` command creates a library of macros using groups of LEF files.

Use this command to select LEF macro pins to create die pins. You can configure elements within chosen LEF files using various dialog boxes, such as LEF Library Manager, Filter options, and so on.

Related Topics

- Filter options Dialog Box
- Creating New Library Definition File Using LEF Library Manager
- Defining and Developing Libraries

LEF Library Manager Dialog Box

Access Using

- Menu path: *Setup – LEF Libraries*

Library definition file	
Current directory	Displays the current library directory path.
File name	Displays the name of the selected `.ldf` file. If you have not selected a library definition file, the default selection is `default.ldf`.
Library settings	
Current library from Library Definition File	A list of user-defined libraries from the library definition files. The selected library is used to define the associated LEF files.
Add	Opens a dialog box to add libraries. The new library name appears in the *Current library* field.
Remove	Removes selected libraries from the *Current library* field.
LEF files	The viewing window displays LEF file names and paths defined in the chosen current library. The Λ and **V** arrow buttons move a highlighted file up or down in the list. When you add LEF files, the first LEF file must be a technology file, which includes layer information. The contents of this file are used to populate the *Filters options* dialog box of the LEF Library Manager. If none of the LEF files is a technology file, a message is displayed. If one of the files you add is a technology file, the design tool automatically pushes it to the top of the list.
Add	Opens a dialog box to choose and add an LEF file to the current library.

Remove	Deletes the highlighted LEF file from the currently selected library.
Use LEF file path relative to LDF file	Check this box to specify a relative path rather than an absolute path to the LDF file. I*f checked, the absolute path is automatically converted to the relative path when you add LEF files using the Add button.*
CML Settings	
File name	Displays the name of the condensed macro library (`.cml`) file for the selected LEF file.
Status	Displays the status of the `.cml` file. Status conditions are: • *Up to date* • *Out of date* • *Does not exist*
Options	Displays the *Filter options* dialog box that displays the settings for automatic creation of the `.cml` file.
Auto create	Creates a new condensed macro library file for the selected LEF file, based on the settings in the *Filter options* dialog box.
OK	Saves the changes and closes the dialog box.

Related Topics

- Creating New Library Definition File Using LEF Library Manager

Filter options Dialog Box

The *Filter options* dialog box appears when you click the *Options* button in the *LEF Library Manager* dialog box.

It lets you create `.cml` files automatically. Changes that you make here are saved to the `.cml` of the LEF file specified in the LEF Library Manager dialog box.

The Filter options dialog box contains four tabs:

File Tab	This tab lets you load settings from the selected `.cml` file or automatically load the default settings from `default.cml`. You can then save the settings you choose as defaults.
Current directory	Displays the current directory path to the selected `.cml` file.
File name	Displays the current selected `.cml` file.
Browse	Displays a standard file browser to select a different `.cml` file.
Load defaults	Loads the options settings from `default.cml` by a. searching the directories within the $TECHFILE variable path b. searching the current working directory If the `default.cml` file is not found in either of the two locations, a warning message is flagged, and you are prompted to navigate to the file location using the *Browse* button.
Save as default	Saves the current options settings as the defaults.
General Tab	This tab displays information related to the selected LEF file. This is the default tab display.
Macros	
Available classes	Lists the available macros of class type PAD, ENDCAP, COVER and COVER BUMP found in the selected LEF file. Other macro types are not listed.
Number of macros	Specifies the number of macros in the LEF file of the type highlighted in the *Available classes* listing.

Total number	Specifies the total number of macros of class type PAD, ENDCAP, COVER and COVER BUMP found in the selected LEF file.
Macro Pins	
Pin names/Pin use	Lists the pin name and use pairs of all class type PAD, ENDCAP, COVER, and COVER BUMP macros in the current LEF file.
Total number of unique names	Specifies the total number of unique pin names used by macros of class type PAD, ENDCAP, COVER and COVER BUMP found in the selected LEF file.
Maximum size	Specifies the size of the largest pin of macro class type PAD, ENDCAP, COVER and COVER BUMP found in the selected LEF file.
Minimum size	Specifies the size of the smallest pin of macro class type PAD, ENDCAP, COVER and COVER BUMP found in the selected LEF file.
LEF Layers	Displays the LEF layer information for the current LEF library. If the current LEF file contains layer information, it is extracted from this LEF file. Otherwise, the first LEF file in the current LDF library that contains the layer information appears. *Name*: example, metal6 Default line width in microns: example, 1.00 *Mapping*; for example, *ignore*, *die pin*, or *IC routing*
Pins Tab	This tab shows the data that is used for the automated `.cml` file creation of solder bump or wirebond pads and connection points from macro pins.
Pin size filter	
Minimum die pin size	The read-only minimum width and height of the I/O pin, in user units defaulted to microns. The default values are `40.00 micrometer`. **Note:** Only pins in the size range of the default values are processed as solder bumps or wirebond I/O pads. Pins with minimum size smaller than `40.01 micrometer` are processed as connection point pins.
Pin names	
Die pins	Specifies the macro pins used to create die pins. The default list contains all pins that meet the minimum size requirements defined on the *Pins* tab.

Connection points	Specifies the macro pins used to create connection point pins. The default list contains all the pins that do not meet the minimum size requirements defined on the *Pins* tab.
Macros Tab	This tab displays information specific to macro settings.
Macros	
Available	Lists the macros of class types PAD, ENDCAP, COVER or COVER BUMP available in the selected LEF file that have at least one pin.
Selected	Lists the macros containing the specific rules for pins and connection points. The default list contains all macros of class types PAD, ENDCAP, COVER or COVER BUMP available in the selected LEF file that have at least one pin.
Pin names	
Die pins	Lists the macro pins that you choose as die pins when processing the macro. The default is that all pins meeting the minimum size requirements defined on the *Pins* tab are used to create die pins. When you click on the pin name in the list, you can see the values for *Size*, *Location*, *Pin Use*, and LEF layer name for the highlighted pin. The number of pins available for the highlighted macro is also displayed. By moving pins in or out of the *Die Pins* list, you can override the default behavior that processes specific pins of this macro as die pins or connection points. This is useful in situations, such as when a specific pin does not meet the minimum size requirement, but must be processed and presented as a die pin.
Connection points	Lists the macro pins that you select to use as connection point pins when processing the macro. The default is that all pins that do not meet the minimum size requirements defined on the *Pins* tab are used to create die pins. When you click a pin name in the list, you can see the values for *Size*, *Location*, *Pin Use*, and *Layer* for the highlighted pin. The number of pins available for the highlighted macro is also displayed. By moving pins in or out of the connection point pin list, you can override the default behavior that processes specific pins of this macro as die pins or connection points. This can be useful in the case when a specific pin does not meet the minimum size requirement, but must be processed and presented as a die pin or the other way round.
Size	Specifies the size of the highlighted pin.
Location	Specifies the location of the highlighted pin.

Pin use	Specifies the use of the highlighted pin.
Layer	Specifies the LEF layer name of the highlighted pin.
Number of pins	Specifies the number of pins available for the highlighted macro.
OK	Clicking *OK* automatically creates the `.cml` file.

Related Topics

- lef lib

Creating New Library Definition File Using LEF Library Manager

1. Choose *Setup – LEF Libraries*.
   The LEF Library Manager dialog box appears.
   The library definition file defaults to an empty `.ldf` file named `default.ldf` in your current working directory unless you have set up a different path.
   See Setting Up a Path for the Creation of a Default .ldf File.
2. Skip the following step if you are not creating a new `.ldf` file.
3. Click the *Browse* button to create a new library definition file. (See About the Library Definition File in the *Allegro User Guide: Defining and Developing Libraries* for details on the `.ldf` file.) If you set up a path for the default `.ldf` file, see Setting Up a Path for the Creation of a Default .ldf File.
4. From the *Selector* dialog box, navigate to the directory in which you want to place the file. To create a new library definition file, specify a file name and click *OK*.
   The new library definition file is created, and the selector box closes.
5. To create a new library, click the *Add* button in the *Library settings* section and specify a name. The definition file name appears in the *Current library from Library Definition* field. The purpose of the library is to group a common set of LEF files together that is required for DEF import.
   You can now add or remove LEF files from the Library Manager interface.
6. Select *Use LEF file path relative to LDF file* to specify a relative path rather than an absolute path to the LDF file. I*f checked, the absolute path is automatically converted to the relative path when you add LEF files using the* Add button.
7. To add LEF files to the library, click the *Add* button next to the *LEF files* field.
8. From the selector dialog box, navigate to the directory containing the LEF files and select a file. Repeat this step for each file you want to add to your library.
   The selected files appear in the *LEF files* field.
   Notes: You should add the technology file that contains layer information first. If several files contain technology information, information is used only from the first file on the list. Use the *UP* button to move another technology file to the top of the list after adding all the files.
   The list of LEF files that you add to a library define a set of IO macros that are designed to be used together in one IC. If you have different sets of macros that are not to be mixed, locate each set in a different library.
   The UP and DOWN arrows let you rearrange the order of LEF files that you add to your

library. If macros with the same name exist in multiple files, the first macro is used. Macros with the same name in subsequent LEF files are ignored.

9. Select each LEF file in turn and click *Auto create* to generate the `.cml` file for each LEF file. Default settings are used to create `.cml` files. To alter the settings in the Filter options dialog box, click *Options* in LEF Library Manager and display the *Filter options* dialog box.
   One `.cml` file is created for each LEF file in your library.

   ⚠ If you do not select any pins in a macro, the `.cml` for that macro is not created, so that macro is always ignored when you import DEF files.

10. Click *OK* to close LEF Library Manager.

Setting Up a Path for the Creation of a Default .ldf File

1. Choose *Setup – User Preferences* and click *Design Paths*.
2. Click the ellipsis (....) next to the `ldpath` preference.
3. Double-click in one of the lines in the *ldfpath Items* dialog box appears.
   A small icon with **....** appears to the right.
4. Click the icon.
   The Select Directory dialog box appears.
5. Choose the directory where you want to stop the `default.ldf` file to be located.
6. Click OK in the *Select Directory* dialog box.
7. Click OK in the *ldfpath Items* dialog box.

Using the LEF Library Manager with an Existing Library Definition File

1. Run `lef lib`.
   The LEF Library Manager dialog box appears.
2. Click the *Browse* button to locate an existing library definition file.
3. Choose the file and click *Open.*
   The library definition file name appears in the *File name* field. Libraries defined are listed in the *Current library from Library Definition file* field. From the drop-down menu, choose the library you want to use for importing the DEF file. By default, the first library in the list is highlighted.
   The LEF files defined in the current library definition file are listed in the *LEF files* field.
4. If the LEF file has changed since it was last defined in the session and you have the write permission to the LEF library, click *Auto create* to update the `.cml` file associated with the LEF file. Otherwise, contact your library administrator to update your read-only `.cml` file for you. Default settings are used to create `.cml` files. You can alter the settings in the Filter options dialog box by clicking *Options* in LEF Library Manager and displaying the Filter options dialog box.
5. Click *OK* to close LEF Library Manager.

Related Topics

- LEF Library Manager Dialog Box
- Filter options Dialog Box
- Defining and Developing Libraries
- lef lib

lef pin param

Associates die pins in your design with LEF macro cells of class COVER BUMP, as required by Cadence® First Encounter® technology (versions 3.1 and later). These associations are made based on padstack and pin use settings.

You must run this command before exporting LEF/DEF file for use by any IC tool, including First Encounter, that represents die pins as macro cells.

⚠ Available only in $BRANDX $PRODUCTAPD.

Related Topics

- Associating Die Pins with LEF Macro Cells
- LEF Pin Parameters Dialog Box
- Defining and Developing Libraries

LEF Pin Parameters Dialog Box

Access using:

- Menu Path: *Edit–LEF Pin Parameters*

.

Number of unassigned pins	Displays the number of die pins not associated with a macro cell.
Pins	
Padstack	Lists the padstacks associated with the die pins in the database file. The first padstack in the list is highlighted by default.
Pin Use	Lists the pin use associated with the die pins in the database file. The first pin use in the list is highlighted by default.
Pin Size	Displays the dimension of the pin selected in the Padstack list.
Number of matching pins	Displays the number of die pins that match the selected padstack/pin use combination.
Macro Data	
Macro List	Lists macro cells of class COVER BUMP present in the database file and in the current LEF library. The macro cell assigned to the selected padstack/pin use combination is highlighted by default. If the selected combination is unassigned, <NONE> is highlighted. **Note:** The double asterisk (**) indicates that the same padstack/pin use combination has been assigned different macro cells from the database file.
Pin Use	Displays the pin use for the selected macro cell present in the current LEF library.
Pin Size	Displays the dimension of the macro cell pad selected in the current LEF library.
Lef Library	Displays the name of the current LEF library.
LEF Library Manager	Opens the LEF Library Manager dialog box for selecting a different LEF library.

Allow Reassign	Select to be able to reassign a padstack/pin use combination to another macro cell. Not selected by default.
Unassign All Pins	Removes the macro cell associations from every padstack/pin use combination in the database file. You must confirm before the action is committed. Grayed if there are no assigned die pins in the database file.
Ok	Saves the changes you made during the session and terminates the command.
Cancel	Terminates the command without saving any changes.
Apply	Saves the macro assignments made to the chosen padstack/pin use combinations, but does not commit them to the database. ⚠ When you apply a macro assignment to a padstack/pin use combination, the count of unassigned pins at the top of the dialog box decreases. This allows you to keep track of your progress in reducing to zero the number of unassigned bumps.

Related Topics

- Associating Die Pins with LEF Macro Cells
- Reassigning Die Pins with LEF Macro Cells
- Removing Die Pins and LEF Macro Cell Association
- Defining and Developing Libraries
- lef pin param

Associating Die Pins with LEF Macro Cells

You must associate die pins with LEF macro cells before exporting LEF/DEF file to any IC tool, including First Encounter, that represents die pins as macro cells.

⚠ Available only in Allegro X Advanced Package Designer.

To associate die pins with LEF macro cells, do the following steps:

1. Choose *Edit–LEF Pin Parameters*.
   The LEF Pin Parameter dialog box appears with the number of unassigned pins in your database file shown at the top of the box.
2. If the status message at the bottom of the box reads "There are no LEF files in the current library," click *LEF Library Manager* to open the Library Manager dialog box, then follow the instruction in step a. Otherwise, proceed to step 3.
   a. Configure the LEF Library Manager to set the correct library and associated macros.
3. Assign a macro cell name to every unassigned padstack/pin use combination until the count of unassigned pins reaches 0.

   ⚠ If you click *Apply* after each new assignment, you can monitor the decrease of unassigned pins in the counter at the top of the dialog box.
4. When there are no more unassigned pins in your database, click *Apply.*
5. Click *OK* to close the dialog box.
   Changes are committed to the database.

Related Topics

- LEF Pin Parameters Dialog Box
- Reassigning Die Pins with LEF Macro Cells
- Removing Die Pins and LEF Macro Cell Association
- Defining and Developing Libraries
- lef pin param

Reassigning Die Pins with LEF Macro Cells

To reassign a padstack/pin use combination to a different macro cell, do the following:

1. Choose *Edit–LEF Pin Parameters*.
2. Set *Allow Reassign* in the LEF Pin Parameter dialog box.
3. Reassign a new macro cell name to the padstack/pin use combinations.
4. Click *OK* to close the dialog box.
   Changes are committed to the database.

Related Topics

- LEF Pin Parameters Dialog Box
- Associating Die Pins with LEF Macro Cells
- Removing Die Pins and LEF Macro Cell Association

Removing Die Pins and LEF Macro Cell Association

To unassign a die pin and macro association, do the folloiwng:

1. Choose *Edit–LEF Pin Parameters*.
2. Check *Unassign All Pins* in the LEF Pin Parameter dialog box.
   Confirm if requested.
   All previously assigned padstack/pin use combinations are now unassigned. The count of unassigned pins at the top of the dialog box is updated to reflect the changes.
3. You can now begin to make new padstack/pin use assignments.
4. When finished, click *OK* to commit your changes to the database and terminate the command.

Related Topics

- LEF Pin Parameters Dialog Box
- Associating Die Pins with LEF Macro Cells
- Reassigning Die Pins with LEF Macro Cells

license_use

Displays the licenses currently in use.

Related Topics

- Displaying a Report of Product, Options, and Licenses Used
- Licenses in Use Window

Licenses in Use Window

Access using:

- Menu path: *Tools–Utilities–Licenses Used*

File – Save As	Saves the information in a text file. When you see this command, you are prompted for a file name and the program appends the.txt extension.
Close	Closes the window.

Related Topics

- Displaying a Report of Product, Options, and Licenses Used
- license_use

Displaying a Report of Product, Options, and Licenses Used

To display the product and licenses being used, do the following:

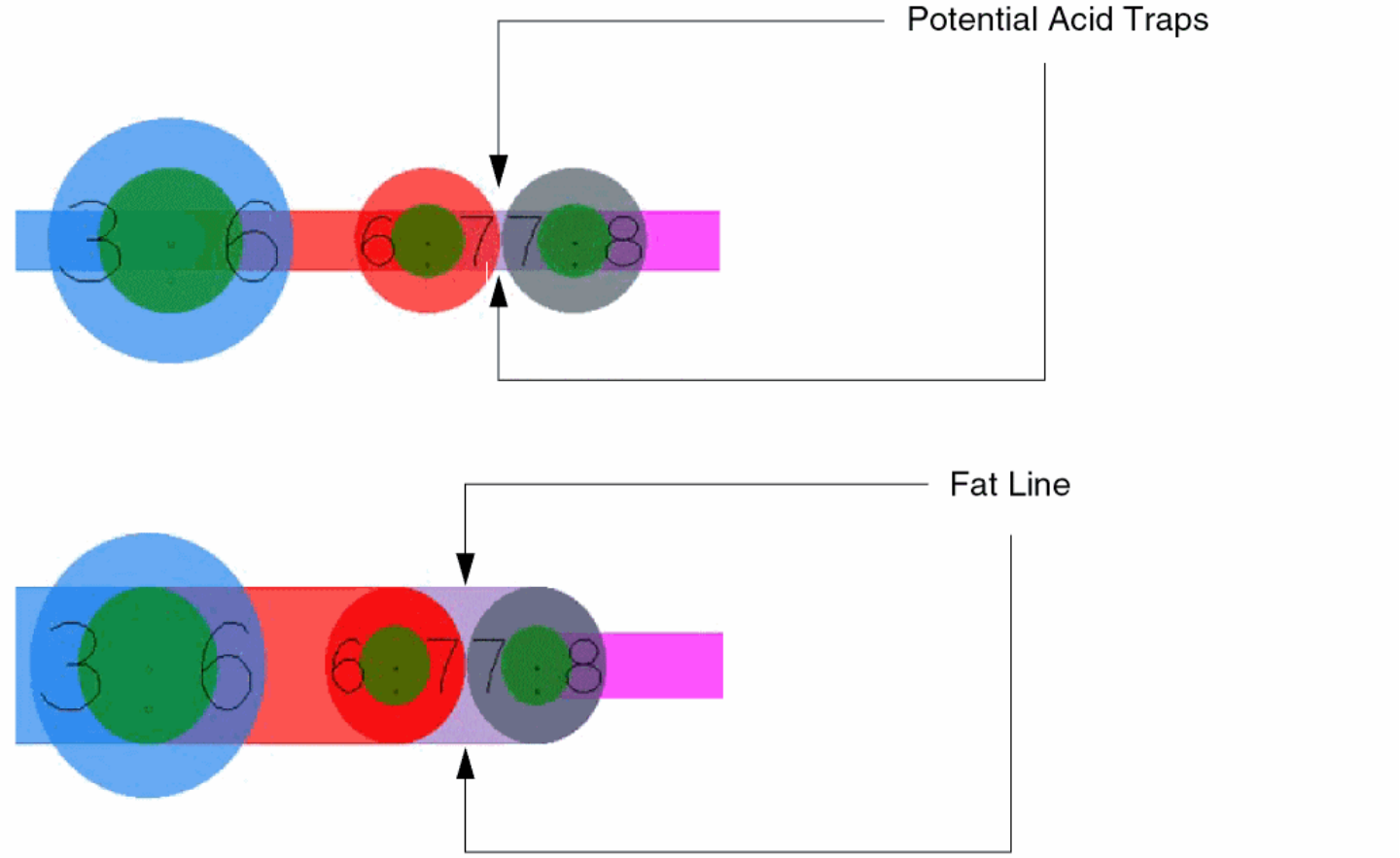
1. Choose *Tools–Utilities–Licenses Used.*
   The License in Use window appears.
   The window lists the product, options, and licenses being used currently.

Related Topics

- Licenses in Use Window
- license_use

line fattening

Eliminates potential acid traps by removing the acute angle formations at the junction of two tangent vias. The line width between the vias is increased to remove acute angles.



Run this post-route command near the end of the design process because you cannot reset line width.

Line Fattening rules:

- Line fattening occurs only between vias, typically HDI vias, with circular pads (not between pins and vias) on all conductor etch layers.
- Line fattening uses the width of the circle-pad diameter of the smaller via.

  ⚠ Line width is modified even if the etch belongs to a symbol (via structure).

- For line fattening to proceed, clines must already exist in your design.

  ⚠ Clines will not be created, and vertices will not be added or deleted.

Related Topics

- Eliminating Acid Traps
- Via-Via Line Fattening Dialog Box

Via-Via Line Fattening Dialog Box

Access using:

- Menu path: *Route – Resize/Respace – Via-Via Line Fattening*

Maximum Via-to-Via Spacing	Specify the maximum spacing allowed between vias. The default is zero.
Waive Impedance/max Line Width DRCs	Select to waive impedance or max line width design rule violations because of line fattening.
Entire Design	Select to apply to all clines in the design.Selected by default.
Selected Clines Only	Select to apply fattening to selected clines.
Run Line Fattener	Click to update all clines requiring fattening.

Related Topics

- Eliminating Acid Traps
- line fattening

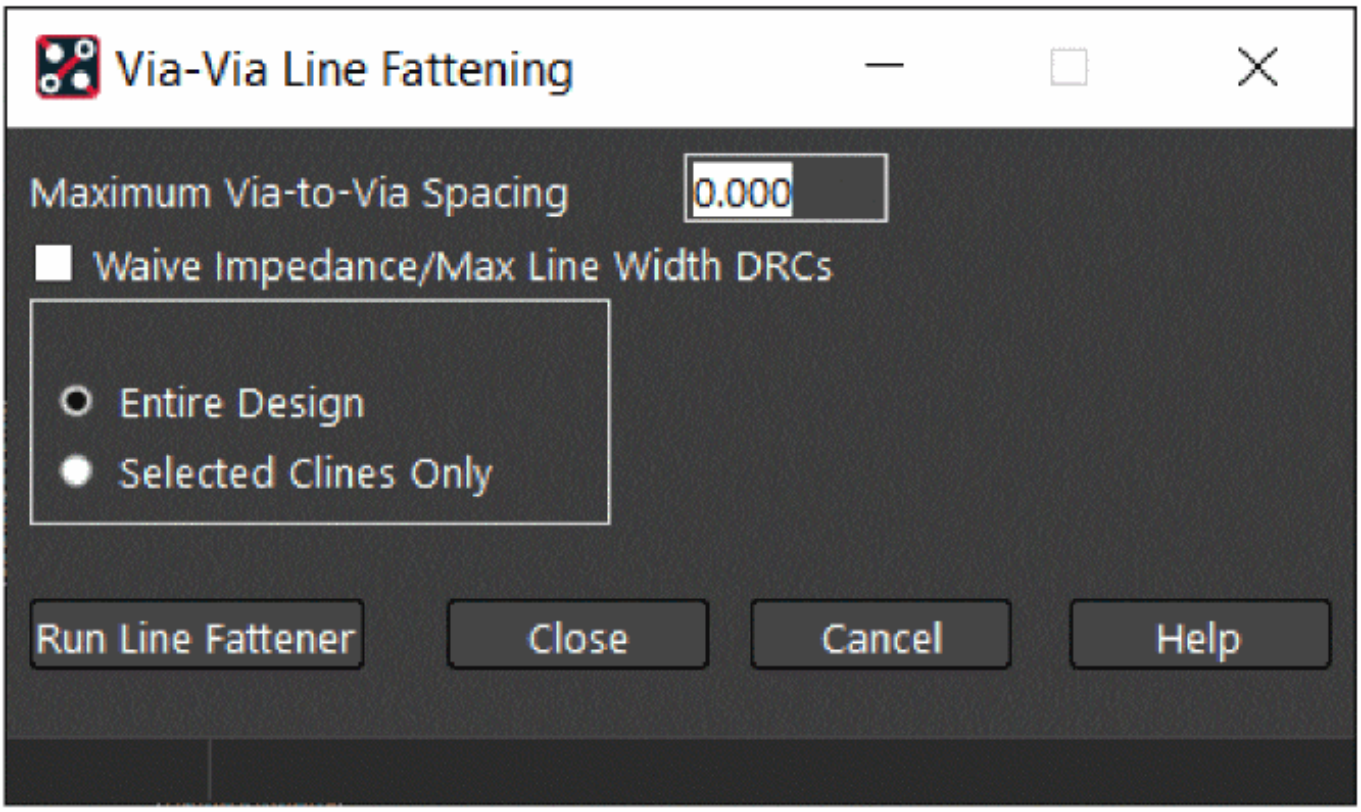
Eliminating Acid Traps

Acid traps occur because of acute angle formation between tangent vias, typically HDI vias. Lines are fattened to remove acid traps.

As this command will change line width, it is recommended that you run the command near the end of the design process.

To eliminate acid traps created due to acute angles between tangent vias, do the following:

1. Choose *Route – Resize/Respace – Via-Via Line Fattening*.
   The *Line Fattening* dialog box appears.



2. Enter the *Maximum Via-to-Via Spacing* value (defaults to zero).
3. Optionally, enable the *Waive Impedance/Max Line Width DRCs* check box to waive resulting impedance or max line width design rule violations.
4. Choose either *Entire Design*, or *Selected Clines Only*. On selecting *Selected Clines Only mode,* you may select single clines, or select multiple clines by drawing a window or polygon.
5. Click *Run Line Fattener*.
   All clines that require fattening are redrawn on the canvas. A message appears with the results of the operation.

6. Optionally, repeat Steps 2 through 4 as needed.
   For each iteration, line widths reset to the values present when you invoked this command.
7. Click *Close* to save any line width changes, and dismiss the *Line Fattening* dialog box
   Click *Cancel* to discard any changes, and dismiss the *Line Fattening* dialog box.

   ⚠ Starting a new command before closing the *Line Fattening* dialog box results in a *Close* operation.

Related Topics

- Via-Via Line Fattening Dialog Box
- line fattening

linefont

Specifies a font for a line.

Related Topics

- Options Panel for the Linefont Command

Options Panel for the Linefont Command

Access using:

Toolbar Icon: 

Line Font	Select a font for lines. Available fonts: Solid, Hidden, Phantom, Dotted, and Center.

Related Topics

- linefont

list

Lets you choose items from a file list of element names.

Syntax

```
list <element type> <file name>
```

Element type	One of the find-by-name types, such as net and refdes, that indicates the type of name in the file.
File name	Name of a file that contains a list of element name of the same type. Default extension is `.lst`.

Example

```
list net net_list
```

The net_list.lst output file has a list of net names (one per line):

```
CLOCK

DATA1

ADDRESS1
```

Load Cadence Artwork Dialog Box

Access using:

- Menu Path: *File – Import – Artwork*

> ⓘ Only Cadence artwork is supported.

File Name	Specifies the name of the Gerber file to load.
Browse	Displays an Open browser window for indicating the Gerber artwork file name. By default, it is set to working directory. ⚠ To change the default file location, set the path of the directory to the environment variable *ads_sdart.*
Format	Determines the file format automatically from the header block in the file name.
Manual	For files that were not generated, choose *Manual* and choose the proper format: *Gerber 4X00, Gerber 6X00, Gerber RS-274X, Barco DPF, MDA,* or *Automatic*.
Class	Specifies the class. The default class is ETCH/CONDUCTOR.
Subclass	Specifies the subclass into which you want the Gerber file loaded. The default is TOP/SURFACE.
The Display Pads option is available only with vector-based Gerber data.	
No	Disregards pad flashes in the artwork file as the Gerber data is read. All of the necessary pad information is provided during symbol placement. This is the default setting.
Yes	Displays the flash geometry if the symbol definition corresponding to the flash is present in the PSMPATH environment variable. Otherwise, a triangular pad target appears at the flash location.

Re-Use Last Mirror/Rotation/Location	Indicates you want the Gerber data loaded using the same location and orientation as the previously loaded data.
The Origin option is available only with raster-based Gerber data.	
*Data Origi*n	Indicates you want to use the lower left point of all the elements in the artwork as the input to position the artwork.
Absolute Origin	Indicates you want to use the 0,0 point as the input to position the artwork.
The *Add Offset* option is available only with raster-based Gerber data.	
No	Indicates that film offsets are to be ignored.
Yes	Indicates that film offsets, if any, are added to the film while loading the artwork.

Related Topics

- Loading Vector-Based Data
- load gerber

load gerber

Loads Gerber artwork files and creates the appropriate line and pad figure elements in the design database using FPOLYs rather than POLYs. The `load gerber` command is identical to `load photoplot`.

Related Topics

- Loading Vector-Based Data
- Load Cadence Artwork Dialog Box
- Preparing Manufacturing Data

Loading Vector-Based Data

Before loading the data, do the following:

1. Make sure that the appropriate artwork aperture (`art_aper.txt`) and parameter (`art_para.txt`) files are present.
2. If your Gerber files are on tape, run the `gb_from_tape` script.
3. Open a drawing that is at least as big as the original drawing from which the Gerber file was created. The units and accuracy of the board should be equal to the units and accuracy of the artwork file.
4. Define the appropriate ETCH/CONDUCTOR, BOARD/SUBSTRATE GEOMETRY, DRAWING FORMAT, and MANUFACTURING subclasses.

Do the following steps to load Gerber 6x00 and Gerber 4x00 photoplotter format types:

1. Choose *File – Import – Artwork.*
2. Enter, or browse for, the name of the artwork file that you want to add.
3. From the drop-down menu, choose the class to load.
4. Choose a subclass.
5. In the *Option* section, choose whether you want the pads to be displayed as targets. Depending on your choice of class/subclass, this dialog box may or may not appear.
6. Click *Load File*.
   A dynamic rectangle that represents the extents of the Gerber data appears in the UI work area. You can rotate, move, or mirror the rectangle using the pop-up menu, but once you place it, the position is fixed.
7. Once you have moved and rotated the rectangle, position it.
   The cursor appears at coordinates 0,0 of the Gerber data.

To load raster-based data, such as RS274X, DPF, MDA, or Automatic data, do the following steps:

1. Choose *File – Import – Artwork* .
   The Load Cadence Artwork dialog box appears.
2. Enter, or browse for, the name of the artwork file to add.
3. From the drop-down menu, choose the class to load.
4. Choose a subclass.

5. In the *Option* section, choose *Absolute Origin* or *Data Origin* as the artwork origin.
   Absolute Origin: The points to which the coordinates in the artwork file have been specified. Use this option to align all the artwork generated from the same board/substrate. When you choose this option, the dialog box displays the *Add Offset* check box which allows you to add existing film offsets to the film while loading the artwork.
   Data Origin: The lower-left point of all the elements in the artwork. If you choose this option, the dialog box displays the *Re-Use Last Mirror/Rotation/Location* check box after you have loaded the first artwork file. This allows you to choose again the last pick-point along with the mirror/rotation operations. This check box remains visible when you choose *Absolute Origin* again.
6. Click *Load File*.
7. A dynamic rectangle that represents the extents of the data appears in the UI work area. You can rotate, move, or mirror the rectangle using the pop-up menu, but once you place it, the position is fixed. Once you have moved and rotated the rectangle, position it.
   The cursor appears at coordinates 0,0 of the artwork data.
8. Continue placing any additional artwork files.

Related Topics

- Load Cadence Artwork Dialog Box
- load gerber

load photoplot

Loads Gerber artwork files and creates the appropriate line and pad figure elements in the design database using FPOLYs rather than POLYs. The `load gerber` command is identical to `load photoplot`.

Related Topics

- Loading Vector-Based Data
- Load Cadence Artwork Dialog Box
- load gerber
- Preparing Manufacturing Data

load plot

Gives a preview of the contents of an intermediate plot file before you plot the data.

Related Topics

- Previewing IPF Files
- Load Plot Dialog Box

Load Plot Dialog Box

Access using:

- Menu Path: *File – Import – IPF*

Look in	Allows you to navigate higher in the directory hierarchy.
File Name	Type the name of the file you want in the field, or choose the file from the list. Selecting a directory in the list by double-clicking on it pushes into the directory. Typing a directory causes the browser to display the contents of that directory. This is useful on UNIX if you want to access another user's directory via the automount home (/home or /hm) or system (/net). Navigating these directories may be time-consuming because the browser mounts them all. To navigate to the home directory of `user1`, for instance, typing "`/hm/user1`" is faster than navigating with the mouse through "`/hm`."
Files of type	By default, provides the typical extension of the file required. The drop-down provides access to additional extensions. You can override the extensions by typing a name with one or more wildcard characters in the File name: field. For example; to display all symbol dra files starting with "dip" you would type "dip*.*".
Change directory	Displays the directory of the chosen file. By default this is set when browsing databases and not set when browsing other files. With respect to the initial directory displayed: ◦ When opening or saving a database the browser always opens in the current working directory this appears on the title bar of the main window. In this instance the *Change directory* check box is set. ◦ All other browsers use the "stickyness" mode. In these instances the Change directory check box is initially deselected. Browsing to a new directory and selecting it does not change the working directory of the main window but the browser remembers this directory to use as a starting point for the next browser session using this mode.

Related Topics

- Previewing IPF Files

- load plot

Previewing IPF Files

1. Open a drawing or create a new, blank board that is as big as the size of the paper you are using to assure proper alignment and offsetting.
2. Choose *File – Import – IPF* to display the file browser.
3. Choose the name of the IPF file to load and click OK .
   When the file is loaded, a dynamic rectangle appears in the design window.
4. To alter the way the design is placed on the work area, right-click to display a pop-up menu. These are the options:

Done	Ends the process without placing the design in the work area.
Cancel	Ends the process without placing the design in the work area.
Mirror	Flips the elements about the Y-axis.
Rotate	Turns the elements by 90 degrees counterclockwise.
Scale	Scales the elements. Input is floating-point.

5. Position the rectangle to place the IPF file on the work area.
   The lower left corner of the rectangle is the actual cursor position.
6. Click on the work area to place the IPF file elements in the design.

Related Topics

- Load Plot Dialog Box
- Creating Penplot Files for Negative Plane Layers
- Viewing a Plot File
- load plot

Creating Penplot Files for Negative Plane Layers

Use this procedure to create penplot files from a design for negative plane layers.

1. Adjust visibility and color priorities as required.
2. Choose *Setup – Design Parameters* to display the Design Parameter Editor window.
3. In the Display tab, enable the following:
   - Filled pads (in Windows) or Filled pads and cline endcaps (in UNIX)
   - Thermal pads
4. Choose *File – Plot Setup* to display the Plot Setup dialog box.
5. In the IPF setup section, set the parameters of the IPF file. Choose *Vectorize text* and specify a line width in the width field.
6. Display the part of the drawing you want to output to the IPF file by using the View commands as needed.
7. Choose *File – Export – IPF* to display the Create Plot browser.
8. Enter the name of the plot file you want to create.
   The tool automatically appends the `.plt` extension.
9. Click *OK*.
   The tool creates the plot and control files, for example, F001.plt and F001.ctl

Related Topics

- Load Plot Dialog Box
- Previewing IPF Files
- Viewing a Plot File
- load plot

Viewing a Plot File

You can view the contents of plot files before plotting by opening a plot file in a design window. Using this process you can also combine plot files into one design.

1. Open a drawing or create a new, blank board that is as big as the size of the paper you are using to assure proper alignment and offsetting.
2. Choose *File – Import – IPF* to display the load plot browser.
3. Choose the name of the IPF file to load and click *OK*.
   A dynamic rectangle appears in the design window.
4. To alter the way the design is placed on the work area, right-click and choose a command.

Done	Ends the process without placing the design in the work area.
Cancel	Ends the process without placing the design in the work area.
Mirror	Flips the elements about the Y-axis.
Rotate	Turns the elements by 90 degrees counterclockwise.
Scale	Scales the elements. Input is floating-point.

5. Position the rectangle to place the IPF file on the work area.
   The lower left corner of the rectangle is the actual cursor position.
6. Click on the work area to place the IPF file elements in the design.
7. When you have loaded the files you need, you can either create a new plot file or plot your design.

Related Topics

- Load Plot Dialog Box
- Previewing IPF Files
- Creating Penplot Files for Negative Plane Layers
- load plot

load stream

Creates a design file using geometric data from a GDSII Stream file (`.sf` or `.gds`). The geometric data includes the stream elements PATH, BOUNDARY, and TEXT.

Related Topics

- load stream
- Stream In Dialog Box
- Stream In View Data Dialog Box
- Stream In Edit Layer Mapping Dialog Box
- Creating a Design from a GDSII Stream File
- Viewing and Importing Data on GDSII Stream Layers
- Viewing Data on GDSII Stream Structures
- Mapping or Unmapping GDSII Stream Layers
- Adding a User-Defined Subclass for Mapping Purposes
- Creating a Stream Layer Conversion File Using a Text Editor

Stream In Dialog Box

Access using:

- Menu Path: *File – Import– Stream*

Stream data	
Stream file	Indicates the name of the stream file to import. The stream filename entered here automatically generates a layer-conversion filename based on the entered stream filename that defaults into the Layer Conversion File field. For example, if you enter Stream_File1 here, Stream_File1_l.cnv displays in the Layer Conversion field. To search for existing files, click … to display the file browser.
View Data	Displays the Stream In View Data dialog box where you can selectively view stream data for a group of layers or a group of structures.
Conversion profile	
Scale Factor	Indicates how the entries are to be scaled vertically and horizontally. For example, a value of 0.5 reduces each entry by 50 percent; a value of 2.0 increases each entry by 100 percent.
Cursor origin	Sets the origin to use for the cursor. The values are *Stream Origin* (default), *Lower-Left Corner*, and *Center*.
Placement rotation	Specify the rotation of the imported stream around the origin specified in *Cursor origin*. The value is `0.00` by default.
Mirror around origin	Select to apply a mirror geometry around the origin specified in *Cursor origin*.

Layer conversion file	Specifies the name of the layer conversion file to map design classes and subclasses to stream data layers. To search for existing files, click … to display the file browser. ⚠ The BONDING_WIRE class is supported in conversion files. The subclasses are the wire profile names from the database. You cannot import this class and subclasses to the layers because you cannot have standalone bond wire objects. They must be connected at both ends. If you try to import to this class, an error appears when the tool reads the conversion file.
Layer Mapping	Opens the *Stream In Edit Layer Mapping* dialog box from which you can map stream layers to design classes and subclasses.
Import	Imports stream data into the tool using the specified layer conversion file. If you do not supply or choose an existing stream layer-conversion assignment file, which maps GDSII stream layer numbers to design classes/subclasses, the tool automatically creates one.
Close	Closes the *Stream In* dialog box without saving any changes made during the session or running the Stream In program.

Related Topics

- Stream In Edit Layer Mapping Dialog Box
- Creating a Design from a GDSII Stream File
- Viewing and Importing Data on GDSII Stream Layers
- Viewing Data on GDSII Stream Structures
- Mapping or Unmapping GDSII Stream Layers
- Adding a User-Defined Subclass for Mapping Purposes
- Creating a Stream Layer Conversion File Using a Text Editor

Stream In View Data Dialog Box

Layers	Lets you selectively view stream data based on stream layers. Click *Layer Mapping* to open the *Stream In Edit Layer Mapping* dialog box, where you can map stream layers in the layer conversion file for final import of stream data. You can selectively view incoming GDSII stream data files either on a layer-by- layer or a structure-by-structure basis before you import the files. You can choose from a list of layers or a list of top-level structures and view the corresponding data. Choose only layers or structures you want to import from a particular GDSII stream file, and exclude unwanted layers or structures that the file may contain. In parallel, you can choose layers or structures you ultimately want to import, generate/edit mappings for those layers or structures on the Stream In Edit Layer Mapping dialog box, and then import.
Stream layer filter	Controls which stream layers display. Initially, this field defaults to All, and all layers in the stream file display. Enter your own filters, which are added to the existing list for reuse in the current session.
Select all to view	Click or deselect all layers in the stream file that currently appear.
Select all to import	Click or deselect to import all stream layers that currently appear.
View	Click to graphically view the data on a stream layer.
Import	Click to import data on that layer after viewing it. For each specified layer, all data resident on the layer is imported.
Stream layer name	Displays the names of the layers in the GDSII stream file.
View selected layers	Displays data on all currently chosen layers in the design window. After selecting additional new layers, click this button again to preview the current selection of layers. The data for the current selection of layers is imported into a temporary, secondary design. For viewing these layers' data, you cannot specify the subclasses to which data is imported. Selected layers that do not display are unaffected.

Layer mapping	Click to display the *Stream In Edit Layer Mapping* dialog box, which shows layers for which you chose the Import check box. If the layer conversion file exists on disk, then the tool uses mappings for the layers from the layer conversion file. If you chose a layer to import, but it is not mapped in the layer conversion file, a default mapping is initially used for the layer.
Close	Closes the dialog box and returns to the *Stream In* dialog box
Structures	Lets you view stream data for top-level structures selectively, which are not referred to by any other structure. Data in a stream file is organized in the form of structures. In addition to containing its own data, a structure can refer to multiple structures and thereby include their data as part of itself. Data on each structure can reside on multiple Stream layers. Given a layer, there can be multiple structures in the stream file that have part of their data on the layer. You cannot import data on a structure-by-structure basis.
Stream structure filter	Controls which stream structures display. Initially, this field defaults to All, and all structures in the stream file display. Enter your own filters, which are added to the existing list for reuse in the current session.
Select all to view	Selects/deselects all structures in the stream file that currently display.
Select	Click to graphically view the structure data.
Stream structure name	Displays the names of the structures in the GDSII stream file.
View selected structures	Displays data on all currently chosen structures in the design window. After selecting additional new structures, click this button again to preview the current selection of structures. The data for the current selection of structures is imported into a temporary, secondary design. For viewing these structures' data, you cannot specify the subclasses to which data is imported. Selected structures that do not display are unaffected.
Close	Closes the dialog box and returns to the *Stream In* dialog box.

Related Topics

- load stream
- Creating a Design from a GDSII Stream File
- Viewing and Importing Data on GDSII Stream Layers
- Viewing Data on GDSII Stream Structures
- Mapping or Unmapping GDSII Stream Layers
- Adding a User-Defined Subclass for Mapping Purposes
- Creating a Stream Layer Conversion File Using a Text Editor

Stream In Edit Layer Mapping Dialog Box

Use this dialog box to edit an existing a layer conversion profile or create a new one, specifying the classes and subclasses to which GDSII stream layers are to be mapped.

This dialog box appears if you click the *Layer Mapping* button from one of the following:

- The *Stream In* dialog box
- The *Stream In View Data* dialog box

⚠ Once you decide on appropriate mapping for layers, close this dialog box and perform a final import from the Stream In dialog box.

Depending on whether the specified layer conversion file exists and from where you invoke this dialog box, grid contents in terms of layers and the initial mappings displayed for them differ as shown in the following table.

Invoked from…	Conversion file?	Grid displays…
Stream In View Data	yes	Layer mappings that currently exist in the layer-conversion file for layers you chose to import in the Stream In View Data dialog box. Default mappings are provided for remaining layers in the grid.
Stream In View Data	no	Default mappings for layers you chose to import in the Stream In View Data dialog box.
Stream In	yes	Layer mappings that currently exist in the layer-conversion file for all layers in the stream file. Default mappings are provided for remaining layers in the grid.
Stream In	no	Default mappings for all layers in the Stream file.

Stream layer filter	Controls the display of stream layers. Initially, this field defaults to All, and all layers in the stream file are displayed. Enter your own filters, which the tool adds to the existing list for reuse in the current session.

Show all data types	Displays all data types to assign or modify the design class/subclass mapping to stream layers. Enable this option to apply mapping for blank stream layers.
Select all	Selects all layers in the stream file that currently display.
Select	Select to display a particular stream layer for viewing or mapping.
Stream layer	Displays the names of the layers in a stream file.
Datatype	A value (-1 to 255) that identifies a data type of element and maps it for a particular layer to different class/subclass combinations. The value -1 means all datatypes. A row displays for each datatype associated with a layer in the Stream file. For example, if layer 5 has datatypes of 2, 7, or 9 in the Stream file, then three rows appear in the grid for layer 5. In each of the three rows, you can toggle between -1 and either 2, 7, or 9 (depending on which value is valid for that row). For the first row, you can toggle between 2 and -1; the second, between 7 and -1; and the third, between 9 and -1. To map the different datatypes to different class/subclass combinations, change the datatype value from -1 to enable previously disabled rows. Initially, layer mappings that currently exist in the layer-conversion file display. If the different datatypes map to different subclasses, each individual mapping displays. If some datatypes for the layer still remain unmapped, they display without any mappings. When datatype -1 is mapped in the layer conversion file, all data types for the layer map to the same class/subclass, and the grid displays this mapping and disables the remaining rows. For default mappings the datatype -1 for the layer is mapped to some default class/subclass combination. All the remaining rows for the layer are disabled in the grid.
Mapped Class	Displays the design classes available for mapping to stream layers and allows you to change mappings one-by- one.
Mapped Subclass	Displays the design subclasses available for mapping to stream layers and allows you to change mappings one-by-one.

Map selected items

Use the following fields to specify the classes and subclasses to which you want to map stream layers. All classes contained in the design display in the *Class* field. The *Subclass* field displays an initial list of corresponding standard design subclasses, as well as user-defined subclasses in the layer-conversion file.

Use stream layer as subclass name	Disables the *Subclass* field and maps the chosen stream layers to subclasses having the same name as the stream layer. If a stream layer name is illegal as a subclass name, the layer name maps to a valid subclass name.
Include data type	Includes the data type as part of the subclass name. The format of naming is <subclass>_<data type>.
Class	Displays the class to which you want to map chosen stream layers.
Subclass	Displays an initial list of standard design subclasses corresponding to the class currently chosen in the *Class* field, as well as user-defined subclasses in the layer-conversion file. When you add new subclasses using *New Subclass*, they display here as well.
Map	Maps chosen stream layers that display to the design classes and subclasses you chose. Selected layers that do not display are not mapped. Specifying a layer mapped to a subclass that cannot legally accommodate a layer's entities generates a warning requiring confirmation. If impending data loss is acceptable, you can choose to proceed.
Unmap	Clears the mapping for all currently chosen layers that display. Selected layers that do not display are unaffected.
New subclass	Adds a new subclass name for a class when you choose the specified class from the *Class* field. If the maximum number of subclasses permitted for a class is exceeded, clicking on this button generates an error. If the class you chose in the *Class* field is one that does not allow user-defined subclasses, this button is disabled, along with the *Use stream Layer as Subclass Name* field.
View selected layers	Displays data on all currently chosen layers in the design window. After selecting additional new layers, click this button again to preview the current selection of layers. The data for the current selection of layers is imported into a temporary, secondary design. For viewing these layers' data, you cannot specify the subclasses to which data is imported. Selected layers that do not display are unaffected.
OK	Writes the current mapping information for layers to the layer conversion. Layers for which no mappings are specified are not written to the conversion files and are consequently not imported.
Cancel	Exits the dialog box and reloads the original design.

Related Topics

- load stream
- Stream In Dialog Box
- Viewing and Importing Data on GDSII Stream Layers
- Viewing Data on GDSII Stream Structures
- Mapping or Unmapping GDSII Stream Layers
- Adding a User-Defined Subclass for Mapping Purposes
- Creating a Stream Layer Conversion File Using a Text Editor

Creating a Design from a GDSII Stream File

Before you can import GDSII stream data, you must have a GDSII `.sf` file containing geometric data.

A stream layer conversion file is required to map the stream layer numbers to the desired class/subclass in your design. The Stream In dialog box helps you create a layer conversion file for the chosen stream file. You can also create a stream layer conversion file using a text editor.

You can use the stream-layer-conversion file that you use to import GDSII stream data to also export GDSII stream data.

To import stream data and create a design file that contains geometric data from the GDSII file, do the following:

1. Open the database and add any user-defined classes/subclasses listed in the stream layer conversion file before you invoke `load stream`.
2. Run `load stream` to display the *Stream In* dialog box, from which you can:
    a. Click on the View Data button to graphically preview GDSII stream format data prior to importing it into a design.
    b. Click on the Layer Mapping button to display the Stream In Edit Layer Mapping dialog box to create or edit existing layer mappings and then return to this dialog box and import the data.
3. Specify the name of the stream file to import in the Stream File field. The filename you specify automatically generates a layer-conversion filename based on it that defaults into the Layer Conversion File field. For example, if you enter Stream_File1 here, Stream_File1_l.cnv displays in the Layer Conversion field. To search for existing files, click the ... (ellipses) button to display the file browser.
4. Click View Data to selectively view data on GDSII stream layers or the Structures tab to view data on GDSII stream structures. The *Stream In View Data* dialog box appears.
5. Enter a Scale Factor, which indicates how the entries are to be scaled vertically and horizontally. For example, a value of 0.5 reduces each entry by 50 percent; a value of 2.0 increases each entry by 100 percent. The default is 1.0. Enter the layer conversion file name in the Layer conversion file field or accept the default name based on the GDSII stream file name. To search for existing files, click the ... (ellipses) button to display the file browser. You can also create a layer conversion file by using a text editor as outlined in Creating a Stream Layer Conversion File Using a Text Editor.
6. To create or edit the current layer conversion profile and/or view data in chosen stream layers

before you import, Click Layer Mapping to map stream layers to design classes and subclasses. The Stream In Edit Layer Mapping dialog box appears.

7. Click **Import** to import the stream data and create a design file that contains geometric data from the GDSII file or **Close** to close the Stream In dialog box without importing GDSII data.

Related Topics

- load stream
- Stream In Dialog Box
- Stream In View Data Dialog Box
- Viewing Data on GDSII Stream Structures
- Mapping or Unmapping GDSII Stream Layers
- Adding a User-Defined Subclass for Mapping Purposes
- Creating a Stream Layer Conversion File Using a Text Editor

Viewing and Importing Data on GDSII Stream Layers

You can selectively view stream data based on stream layers. In parallel, you can also choose layers you ultimately want to import. Click the Layer Mapping button to map stream layers to design classes/subclasses in the layer conversion file for final import of stream data. The Stream In Edit Layer Mapping dialog box appears.

1. Click the View Data button on the Stream In dialog box to display the Layers tab of the Stream In View Data dialog box. Enter the GDSII stream layers you want to list in the Stream layer filter field. Initially, this field defaults to All, and all layers in the stream file display. Enter your own filters, which are added to the existing list for reuse in the current session.
2. Choose Select All to View to preview all layers in the GDSII stream file that currently display and/or Select All to Import to import all stream layers that currently display.
3. Choose View to graphically view the data on a particular stream layer or choose Import to import data on that layer after viewing it. For each specified layer, all data resident on the layer is imported.
4. Click the View selected layers button to display data on all currently chosen layers in the design window. After selecting additional new layers, click this button again to preview the current selection of layers. The data for the current selection of layers is imported into a temporary, secondary design. For viewing these layers' data, you cannot specify the subclasses to which data is imported. Chosen layers that do not display are unaffected. The following message appears in the dialog box:

   Importing Stream data for viewing...

   Click Yes or No in the popup dialog box that appears.

   --or--

   Click the Layer Mapping button to map stream layers in the layer conversion file for final import of GDSII stream data. The Stream In Edit Layer Mapping dialog box appears, which shows layers for which you chose the Import check box. If the layer conversion file exists on disk, then the tool uses mappings for the layers from the layer conversion file. If you chose a layer to import, but it is not mapped in the layer conversion file, a default mapping is initially used for the layer.
5. Click Close to return to the Stream In dialog box.
6. Click Import in the Stream In dialog box to import the GDSII Stream data you chose on the *Stream In View Data Layers* tab or Close to close the dialog box.
7. Review the *stream_in.log* file, which details the processing status (for example, when

processing begins and ends), the library name, and number of entities converted, once you have imported the data.

Related Topics

- load stream
- Stream In Dialog Box
- Stream In View Data Dialog Box
- Stream In Edit Layer Mapping Dialog Box
- Mapping or Unmapping GDSII Stream Layers
- Adding a User-Defined Subclass for Mapping Purposes
- Creating a Stream Layer Conversion File Using a Text Editor

Viewing Data on GDSII Stream Structures

Data in a stream file is organized in the form of structures. In addition to containing its own data, a structure can refer to multiple structures and thereby include their data as part of itself. Data on each structure can reside on multiple Stream layers. Given a layer, there can be multiple structures in the stream file that have part of their data on the layer. You can selectively view GDSII stream data for top-level structures, which are not referred or by any other structure.

You cannot import data on a structure-by-structure basis doing the following steps:

1. Click View Data on the Stream In dialog box to display the Structures tab of the Stream In View Data dialog box.
2. Enter the GDSII stream structures you want to list in the Stream Structure Filter field. Initially, this field defaults to All, and all structures in the GDSII stream file display.
3. Enter your own filters, which are added to the existing list for reuse in the current session.
4. Choose Select All to View to preview all structures in the GDSII stream file that currently display or choose Select to graphically view the data on a particular stream structure.
5. Click the View Selected Structs button to graphically view data on all currently chosen structures in the design window. After selecting additional new structures, click this button again to preview the current selection of structures. The data for the current selection of structures is imported into a temporary, secondary design. For viewing these structures' data, you cannot specify the subclasses to which data is imported. Chosen structures that do not display are unaffected.
6. Click Close after viewing the structures to return to the Stream In dialog box.
7. Click Import in the Stream In dialog box to import the GDSII Stream data you chose on the Stream In View Data Layers tab or Close to close the dialog box.
8. Review the *stream_in.log* file, which details the processing status (for example, when processing begins and ends), the library name, and number of entities converted, once you have imported the data.

Related Topics

- load stream
- Stream In Dialog Box
- Stream In View Data Dialog Box
- Stream In Edit Layer Mapping Dialog Box
- Creating a Design from a GDSII Stream File
- Adding a User-Defined Subclass for Mapping Purposes
- Creating a Stream Layer Conversion File Using a Text Editor

Mapping or Unmapping GDSII Stream Layers

Before importing, you can edit the Layer Conversion Profile to change the mappings of design classes/subclasses to chosen GDSII stream layers on a one by one basis or change mappings for a group of layers. Or you can simply view data in the chosen GDSII stream layers prior to importing it.

To map or unmap stream layers, do the following steps:

1. Click Layer Mapping on the *Stream In* or *Stream In View Data* dialog box to display GDSII stream layers mapped to design class/subclasses. The *Stream In Edit Layer Mapping* dialog box appears.Initial mappings and layers that display differ depending on whether the specified layer conversion file exists and from where you invoke this dialog box.
2. Enter the layers you want to view or edit in the GDSII Stream Layer Filter field. The initial default is All. Filters you enter become part of the drop-down list, which can be reused in the current session.
3. Use Select All to display all listed layers or Select to choose individual GDSII stream layers to be mapped. The names of the layers in a GDSII stream file display in the Stream Layer column.
4. Map the datatypes for each GDSII stream layer to design class/subclasses in the Datatype column. Enter a value (-1 to 63) that identifies a data type of element and maps it for a particular layer to different class/subclass combinations in the Datatype column. The value -1 means all datatypes. A row displays for each datatype associated with a layer in the Stream file.
   For example, if layer 5 has datatypes of 2, 7, or 9 in the Stream file, then three rows appear for layer 5. In each of the three rows, you can toggle between -1 and either 2, 7, or 9 (depending on which value is valid for that row). For the first row, you can toggle between 2 and -1; the second, between 7 and -1; and the third, between 9 and -1.
   - Map all datatypes for a layer to the same class/subclass in the tool by selecting -1 as the datatype, which disables the remaining rows for each datatype associated with that layer.
   - Map the different datatypes to different class/subclass combinations by changing the datatype value from -1 to enable previously disabled rows.

   Initially, layer mappings that currently exist in the layer-conversion file display. If the different datatypes map to different subclasses, each individual mapping displays.
5. If some datatypes for the layer still remain unmapped, they display without any mappings.

When datatype -1 is mapped in the layer conversion file, all datatypes for the layer map to the same class/subclass. This mapping displays and disables the remaining rows. For default mappings, the datatype -1 for the layer maps to a default class/subclass combination. All the remaining rows for the layer are disabled.Use the Class and Subclass columns to change mappings for layers on a one-by-one basis if necessary. The Class column displays the class to which chosen stream layers are to be mapped. The Subclass column displays an initial list of standard design subclasses corresponding to the class currently chosen in the Class field, as well as user-defined subclasses in the layer-conversion file. When you add new subclasses using New Subclass, they display here as well.

6. Use the fields in the Map Selected Items section to specify the classes and subclasses to which to map chosen stream layers. You can use these fields to map several layers simultaneously. All classes contained in the design display in the Class field. The Subclass field displays an initial list of corresponding standard design subclasses, as well as those user-defined subclasses in the layer-conversion file.
7. Choose a class for the GDSII stream layer from the Class field, which contains all classes present in the design, for mapping the GDSII stream layers.
8. Choose a subclass for the GDSII stream layer from the Subclass field, which contains the design subclasses for the class currently chosen in the Class field, and user-defined classes in the layer-conversion file.
    - To add a new user-defined subclass to a class, see Adding a User-Defined Subclass for Mapping Purposes.
    - To map the chosen GDSII stream layers to subclasses with the same name as the GDSII stream layer, choose Use Stream layer as subclass name. (The Subclass field is disabled as a result.) If a GDSII stream layer name is illegal as a subclass name, the tool maps the layer name to a valid subclass name.
9. Click Map to complete the mapping for all chosen layers that currently display to the design classes and subclasses you chose. Specifying a layer mapped to a subclass that cannot legally accommodate a layer's entities generates a warning requiring confirmation. If impending data loss is acceptable, you can choose to proceed. Click Unmap to clear the mapping for all chosen layers that currently display. Chosen layers that do not display are unaffected. Click OK to write current mapping information for layers to a new Layer Conversion File or overwrite an existing layer-conversion file and return to the Stream In dialog box. Layers without specified mappings are not written to the layer-conversion file, and therefore are not imported.

10. Click Import in the *Stream In* dialog box to import the data or Close to close the dialog box.
11. Review the *stream_in.log* file, which details the processing status (for example, when processing begins and ends), the library name, and number of entities converted, once you have imported the data.

Related Topics

- load stream
- Stream In Dialog Box
- Stream In View Data Dialog Box
- Stream In Edit Layer Mapping Dialog Box
- Creating a Design from a GDSII Stream File
- Viewing and Importing Data on GDSII Stream Layers
- Creating a Stream Layer Conversion File Using a Text Editor

Adding a User-Defined Subclass for Mapping Purposes

You can define a subclass for mapping purposes by doing the following steps:

1. Click Layer Mapping on the *Stream In* or *Stream In View Data* dialog box to display the GDSII stream layers mapped to A design class/subclasses. The Stream In Edit Layer Mapping dialog box appears.
2. Enter the layers you want to view or edit in the GDSII Stream Layer Filter field. The initial default is All. Filters you enter become part of the drop-down list, which can be reused in the current session.
3. Choose the class from the Class field in the Map Selected Items section to which you want to define the subclass. The New Subclass button is initially disabled. Selecting a class to which it is permissible to add a user-defined subclass enables New Subclass. Otherwise, you cannot add a user-defined subclass to the chosen class.
4. Click on the New subclass button (if enabled) and enter the new subclass name in the popup dialog box that appears. If the maximum number of subclasses permitted for a class is exceeded, clicking on New Subclass triggers an error.
5. Click OK in the popup dialog box to make the new subclass available in the Subclass field for the class chosen in the Class field map the new subclass to the specified stream layers.

Related Topics

- load stream
- Stream In Dialog Box
- Stream In View Data Dialog Box
- Stream In Edit Layer Mapping Dialog Box
- Creating a Design from a GDSII Stream File
- Viewing and Importing Data on GDSII Stream Layers
- Viewing Data on GDSII Stream Structures

Creating a Stream Layer Conversion File Using a Text Editor

Use the following file-record format when using a text editor to create a stream layer-conversion file for importing stream data:

- Separate each field in the line by one or more blank spaces.
- Do not use non printable characters. (You may use tab characters.)

To create the conversion file, do the following steps:

1. Assign a filename to your stream layer conversion file.
   The filename must follow UNIX file-naming conventions, which dictate that a filename can be up to 14 characters long and consist of any characters, including the letters a-z or A-Z, 0-9, and underscore (_), period, and minus sign. Uppercase and lowercase letters are considered different in filenames. The import process assumes that the file extension is `.cnv` if you do not specify one.
2. Create a comment line by starting a line with the pound sign (#). For example:

   ```
   # layer_number data_type class_name subclass_name
   ```

3. Enter each layer conversion record on a separate line using the following record format.

Related Topics

- load stream
- Stream In Dialog Box
- Stream In View Data Dialog Box
- Stream In Edit Layer Mapping Dialog Box
- Creating a Design from a GDSII Stream File
- Viewing and Importing Data on GDSII Stream Layers
- Viewing Data on GDSII Stream Structures

lock symbol

The `lock symbol` command locks the selected instances of a symbol by attaching the `LOCKED` property to the instances.

Select a symbol instance and right-click to access the command.

Related Topics

- unlock symbol