cādence®

I Commands

Product Version 23.1
September 2023

Contents

1

I Commands

iangle	iapick	iapick_to_grid
iapick_to_gridunit	icm_out	identify buses
identify nets	idf_in Batch Command	idf_out Batch Command
idf in	idf out	idx in
idx out	iff in	ifnvar
ifp	ifvar	image restore last
image restore userdefined	images	imouse_pos
import codesign die	import codesign pkg	import file manager
import logo	Import mentor	import timing
interface_planner	interface_vis	ipc356 out
ipc2581 in	ipc2581 out	ipc spec edit
ipick	ipick_to_grid	ipick_to_gridunit
irdrop	island_delete	

iangle

The `iangle` command lets you input an angle value, either an absolute angle or incremental from the current angle (`iangle`).

Use `iangle` for rotating elements in any command that allows rotation. For example, `move`, `add pin`, and `add symbol` commands have Rotate in pop–up menus. The `iangle` command can also be used for applications expecting angular input, where angular dynamics is active and position readout shows an angle value. As a substitute for Rotate , `iangle` provides the equivalent of choosing the Rotate pop–up, spinning the element to the appropriate angle, then clicking to choose that angle. When an application expects an angular input, `iangle` provides the equivalent to clicking to choose an angle. For example, `spin` rotates a chosen element and expects angular input. You can enter `iangle` instead of clicking.

You can enter angle coordinates from the console window prompt or bring up a dialog box into which you can enter the coordinates.

The `iangle` command is not valid for elements that do not have an angle as part of their instance data, for example, line segments.

```
iangle [+ –] <
degree value
>
```

[+] indicates clockwise (default).

[-] indicates counterclockwise.

Rotating Elements in a Running Command

1. Run a command that supports rotation of an element; for example, `move`.

2. Choose the element to affect.

3. Do one of the following:

   - Type `iangle` in the Console window, press `Enter` and type the angle coordinates in the dialog box that appears.

   - Type `iangle` in the Console window and the coordinates.

   The chosen element is rotated to that degree.

4. Click Done from the right-button pop-up menu.

Example

```
iangle + 225
```

Related Topics

- angle

iapick

The `iapick` command allows you to enter the incremental distances from the previous polar coordinates for objects you want to find and highlight, in addition to using the mouse to highlight objects in a drawing. You must be in a command mode — for example, `add connect` to activate the `iapick` command.

Highlighting Objects on the Canvas

1. Ensure that you are in command mode, for example, `add connect`.

2. Type `iapick` in the Command window.

3. Specify the distance and click OK.

4. Specify the angle and click OK.

> ⚠ You can also type the incremental coordinates on the command line after typing the command name.

Example

```
iapick 1000 45.
```

Related Topics

- add connect

iapick_to_grid

The `iapick_to_grid` command is used in scripts to record mouse clicks that must be mapped to the grid. The polar coordinate format is the same as that of the apick command. When the `iapick_to_grid` command is used in macro files, the coordinate system is relative to a pick sign.

Example

```
ipick_to_grid distance angle
```

Related Topics

- apick

iapick_to_gridunit

Internal command.

icm_out

The `icm_out` command lets you export an InterComm text file (*.bri*).

Access using

- Menu Path: *Export – InterComm*

Related Topics

- export creoview

identify buses

The `identify buses` command lets you create or edit a bus. You can group similar nets into a bus or you can manipulate a bus that has been imported from Allegro Design Entry HDL.

Access using

- Menu Path: *Logic – Identify Buses*

Identify Buses Dialog Box

Buses Area	
Buses list box	Displays a list of existing buses to which you can add or delete.
Add Bus	Displays a dialog box that enables you to specify the name of a bus to add to the list.
Delete Bus	Deletes the chosen bus from the list.

Nets not in the Bus Area	
Nets not in the Bus list box	Displays a list of nets not chosen in the bus. You can click a net to move it into the Nets in the Bus list or you can click All to move all nets to this list.
Filter	Limits the search for net selection. Examples of Filters

Nets in the Bus Area	
Nets in the Bus list box	Displays nets presently chosen in the bus list. You can click a single net to move it into the Nets not in the Bus list or you can click ß All to move all nets to this list.
Filter	Limits the search for net selection. Examples of Filters
Buttons	
All ->	Moves all the nets from left list box to right list box.
<- All	Moves all the nets from right list box to left list box.
OK	Applies the settings and dismisses the dialog box.
Apply	Applies the settings and retains the dialog box.
Cancel	Ignores the recently specified settings and dismisses the dialog box.

Related Topics

- Bus Rules

Creating a Bus

You can group existing nets into buses. You can also use buses created in Allegro Design Entry HDL, Allegro Design Editor GXL, or Composer. When creating buses, follow a consistent naming and numbering scheme for the nets that are grouped into a single bus.

1. Choose *Logic – Identify Buses.*
   Alternatively, type `identify buses` in the Command window. The Define Bus Nets dialog box appears and any existing buses display in the *Buses* list box.

2. Click *Add Bus.*
   A dialog box displays and prompts you to enter a name for the new bus.

3. Enter the name for the new bus and click *OK*.
   The new bus name appears in the Buses list box and the available nets display in the *Nets not in Bus* list box.

4. Leave the Filters fields set to * or use wildcards to list a subset of nets.

5. In the *Nets not in Bus* list box, click the nets that you want included in the bus or click All à to choose all available nets for inclusion in the bus.
   The chosen nets move to the Nets in Bus list box.

6. Click *OK* or *Apply* to create the bus.

Related Topics

- Bus Rules

Deleting a Bus

1. Choose *Logic – Identify Buses.*
   Alternatively, type `identify buses` in the Command window. The Define Bus Nets dialog box appears and any existing buses display in the *Buses* list box.

2. In the *Buses* list box, choose the bus to delete.

3. Click *Delete Bus*.
   A dialog box displays and prompts you to confirm the delete operation.

4. Click *Yes* to delete the bus.
   The bus name is removed from the *Buses* list box and the *Nets in Bus list* box is cleared.

5. Click *OK* or *Apply*.

Related Topics

- Bus Rules

identify nets

The `identify nets` command lets you assign a Voltage property on one or multiple nets so that the nets are identified as a DC Nets. This DC Net assignment is required for signal integrity analysis, a simplified ratsnest display, and performance improvements on large pin count nets. In addition, a DC Net provides a stop point when generating XNets topology preventing a large number of nets from being combined into one XNet.

For more details, see the *Routing the Design* user guide in your documentation set.

Access using

- Menu Path: *Logic – Identify DC Nets*

Identify DC Nets Dialog Box

Net filter	Filters the range of nets displayed in the DC nets list.
Net	Displays the name of the chosen net.
Voltage	Displays the voltage level of the net chosen.
Power Net Regex	Filters nets based on the value of *power_net_regex* user preference variable to expedite power voltage net assignment. You can specify a regular expression string that supports * and ? for wildcard matching and | between multiple wildcard strings. For example, `+1_?V, +3_*, VCC*, +1_?V|+3_*|VCC*` Setting the user preference variable enables this button.
Ground Net Regex	Filters nets based on the value of *ground_net_regex* user preference variable to expedite ground voltage net assignment. You can specify a regular expression string that supports * and ? for wildcard matching and | between multiple wildcard strings. For example, `GND|GND_V*` Setting the user preference variable enables this button.
Possible Voltage Net Rules	Opens Possible Voltage Net Rules dialog box to modify default rules for net selection.
Selected Net(s)	Displays information related to the selected net names.

	Name	The net name selected in the list box. `**` is displayed if multiple nets are selected in the list box.
	Voltage	Sets the DC voltage level of the chosen net (or pin). Enter NONE to remove a previously assigned voltage from a chosen net. `**` is displayed if multiple nets are selected in the list box.
	Delete	Removes voltage from the chosen net.

OK	Applies the settings and closes the dialog box.
Apply	Applies the setting and leaves the dialog box open.

Possible Voltage Net Rules

Voltage Net Name Formats	Displays a list that of formats for potential voltage net names. As you add more formats, the list expands.
Add New Voltage Net Format	Use this option to add new strings to define the format of possible net names.
Delete Selected Voltage Net Name Format	Use this option to delete existing strings that define possible net name formats.
Include nets that contain power or ground pins	Use this option to include nets with power or ground pins.
Include nets that contain more than n pins	This option lets you specify the minimum number of pins a possible voltage net must contain.

Choosing a Net to Carry a DC Voltage

1. Choose *Logic – Identify DC Nets*.
   Alternatively, type `identify nets` in the Command window. The Identify DC Nets dialog box appears.

2. Choose one or multiple nets either from the list or from the design.
   You can use filters to limit the search.
   The selected nets are highlighted in the design.

3. Assign a DC voltage in the *Net selected* section.
   If previously assigned, the voltage level for the net appears in the *Voltage* field, but can be changed if necessary. (To remove a previously assigned DC voltage from a selected net, click *Delete*.)

4. Click *Apply* to continue to define more DC nets, or click *OK*.

Related Topics

- Identify DC Nets Dialog Box

Defining an Extended Net

An extended net (XNet) traverses more than one net through a discrete device. To define an XNet, a device model must be assigned to the discrete device between the two (or more) nets.

1. Choose *Logic – Identify DC Nets*.
   Alternatively, type `identify nets` in the Command window. The Identify DC Nets dialog box appears.

2. Choose one or multiple nets either from the list or from the design.
   You can use filters to limit the search.
   The selected nets are highlighted in the design.

3. Enter a value in the *Voltage* field.

4. Run `signal model` to display the Signal Model Assignment dialog box.

5. Click *Auto Setup* and assign an ESpice model to discrete components.

   ⚠ If you have resistor packs, create the ESpice device model by selecting the device and then the *Create Model* button.

6. Run `show element` and choose one of the two nets to list the net name and the XNet group name.

   LISTING: 1 element(s)

   < NET >

   Net Name: NET2

   Member of XNet: NET1

   Member of Groups: NET1

7. Assign the PROPAGATION_DELAY (PD) or RELATIVE_PROPAGATION_DELAY (RPD) properties pin pairing to the XNet.

Related Topics

- show element
- signal model

idf_in Batch Command

The `idf_in` batch command translates design outline and component placement information from Intermediate Data Format (IDF) for use in an electrical design. You can import data into a new board/design or into an existing design.

Based on the mechanical system used, the editor looks for the file types described below. Any error or warning messages are stored in the `idf_in.log` file.

```
idf_in [-d
<name_type>
]<idf data file> [-o <output allegro database>] [-i <input allegro database> -[a[p|m|f]]
```

Optional Arguments

-d <name_type>	Switch specifying the name of the mechanical system. *<name_type>* can be one of the following: PTC, SDRC, or IDF. The default is IDF.
-o <output allegro database>	Specifies the name of the output board/substrate or drawing to be created/updated by `idf_in`. The default is a file with the `.brd` or `.mcm` extension.
-i <input allegro database>	Specifies the name of the input board/substrate or drawing to be created/updated by `idf_in`. The default is a file with the `.brd` or `.mcm` extension.
-p	Generates a .dra file. When loaded in the Symbol editor, the `.dra` file switches the mode of the editor to the `<sympackage>` type.
-m	Generates a .dra file. When loaded in the Symbol editor, the `.dra` file switches the mode of the editor to the `<symmech>` type.
-f	Generates a .dra file. When loaded in the Symbol editor, the `.dra` file switches the mode of the editor to the `<symformat>` type.
-a	Specifies the accuracy that is the number of decimal places. The valid range is 0-4. The default is 3.

Required Argument

<idf_data_file>	Base name of the IDF file. The -d <name_type> argument value detects the filename extension.

If You Import This File Name Type…	**Your Product looks for a Design File <drawing_name> with This Extension…**
PTC	`input_file.emn`
SDRC	`input_file.out`
IDF	`input_file.bdf`

Examples

The following command creates or updates the `in_test` board file by reading the IDF file `test.out`. The -d SDRC argument indicates that the IDF file (test) extension is `.out`.

```
idf_in –d SDRC –o in_test test
```

The following command creates or updates the `in_test.dra` file by reading the IDF file `test.emn`. The -d SDRC argument indicates that the IDF file extension is `.emn`. The -p argument indicates the generation of a `.dra` file.

```
idf_in –d PTC –p –o in_test test
```

The following command shows an example using only the required argument. The IDF file is test. Because there is no -d <name_type> argument, the default is IDF. The IDF file extension is `.bdf`. Because there is no outname argument, the current design name, test, is used.

```
idf_in test
```

Related Topics

- idf in

idf_out Batch Command

The `idf_out` batch command exports data from a design drawing for input to Intermediate Data Format (IDF).

The IDF Library file contains the package definitions used (their outlines and height). The layout editor's `idf_out` obtains height values in this order:

- Component definition's HEIGHT property

- Symbol definition's PACKAGE_HEIGHT_MAX property

- `idf_out` default height

The idf_out utility passes a component's HEIGHT value when both of the following conditions are met:

- No PACKAGE_HEIGHT_MIN and/or PACKAGE_HEIGHT_MAX properties are attached to the place bound shapes defined in the component instance's symbol

- The environment variable `idf_ignore_comp_height` is not set

Regarding the symbol outline, `idf_out` exports the union of all the place_bound shapes. IDF only supports one closed loop polygon per symbol definition.

Output files are produced with extensions described below. If you do not specify an output file name, the current design name is used. The `idf_out.log` file stores any error or warning messages.

```
idf_out –d <name_type> [–o <
filename
>] [–s <
source
>] [–h <
height
>] [–V <IDF Version>] [–b <
version
>][–c <configuration file>] <
brd>
```

If The Editor Exports to IDF in This File Name Type…	The Editor Creates a Board File with This Extension…	and a Library File with This Extension…

PTC	`myextract.emn`	`myextract.emp`
SDRC	`myextract.out`	`myextract.pro`
IDF	`myextract.bdf`	`myextract.ldf`

Optional Arguments   I

-d <name_type>	Switch specifying the name of the Mechanical System. <name_type> can be one of the following: PTC, SDRC, or IDF. The default is IDF.
-o <filename>	Specifies the name of the output file. Depending on the Mechanical System used, the output filenames are produced with different extensions. For example, if the specified file name is test , the resulting output file names are as follows: If no output file name is specified, the design name is used. For example, if no output name is specified and the Mechanical System is PTC, the output file names are as follows: `<input_file>.emn <input_file>.emp`
-s <source>	Specifies the string for source system identification that will appear in the HEADER section of the IDF file. The default value of <source> is null string.
-h <height>	Specifies the height of the components to be assumed, whose height is otherwise not stated in the input board/substrate. The default value of <height> is 0.
–V	Specifies the IDF version. Values can be 2.0 or 3.0. If no value is given, the default is 3.0.
-b <version>	Specifies the version number of the Board/Library file to be produced as output by the idf_out command. The default value of <version> is 1.
-c <configuration file>	Specifies the configuration file to be used for filtering out design information. No specific extension name is required.

Required Argument

<brd>	The name of the design file on which the `idf_out` command is run. If no input file is specified, `idf_out` prompts for the name of the design before executing. The `.brd` extension is not required.

Example

The following command runs `idf_out` on the board/substrate test file to generate the IDF file `out_test.bdf` and LIBRARY file `out_test.ldf`.

⚠ The `.ldf` file is created by default.

```
idf_out -d IDF -o out_test -s "ACME CAD 2.0" -b 2.0 -h 20 -V 3.0 test
```

idf in

The `idf in` command translates design outline and component placement information to Intermediate Data Format (IDF) for use in an electrical design. You can import data into a new board/design or into an existing design.

To represent the components when you run this command, you must have library symbols present. For additional information about IDF, see the *Transferring Logic Design Data* user guide in your documentation set.

Based on the mechanical system used, the editor looks for the file types described below. Any error or warning messages are stored in the `idf_in.log` file. Click *Viewlog* in the IDF dialog boxes to open this file.

If the editor Imports this file name Type…	It looks for a design file with this extension…
PTC	`input_brd.emn`
SDRC	`input_brd.out`
IDF	`input_brd.bdf`

Access using

- Menu Path: *File – Import – IDF*

IDF In Dialog Box

Use this dialog box to import design outline and component placement information in IDF.

IDF Board File	Specifies the name of the board file from which you are importing data.
...	Click this button to browse and locate the output file name.
Import	Runs the command and imports the data.
Viewlog	Displays the `idf_in.log` file, which contains errors and warnings from the translation.

Related Topics

- `idf in`

Creating a Design from an IDF File

1. Move or copy the IDF file to the platform on which you are running.

2. Type `idf_in` and arguments on a single line in the operating system Command prompt.

3. Press *Enter* to run the program.

idf out

The `idf_out` command exports data from a design drawing for input to Intermediate Data Format (IDF). For additional information about IDF, see the *Transferring Logic Design Data* user guide in your documentation set.

Output files are produced with extensions described below. If you do not specify an output file name, the current design name is used. Any error or warning messages are stored in the `idf_out.log` file. Click *Viewlog* in the IDF dialog box to open this file.

If The Editor Exports to IDF in This File Name Type…	The Editor Creates a Board File with This Extension…	and a Library File with This Extension…
PTC	`myextract.emn`	`myextract.emp`
SDRC	`myextract.out`	`myextract.pro`
IDF	`myextract.bdf`	`myextract.ldf`

Access using

- Menu Path: *File – Export – IDF*

IDF Out Dialog Box

Use the IDF Out dialog box to export design outline and component placement information from a design to IDF for use in a mechanical design group.

> ⓘ Warning: More than one closed loop defines the outline of *XXXXX*. An overall bounding box has been exported.

File Name Type	Specifies the IDF file extension: IDF, PTC, or SDRC. Note: File extensions are available for use in Structural Dynamic Research Corporation (SDRC) and Parametric Technology Corporation (PTC) systems. All files have the same content; only the extension differs.		
IDF Version	Lets you choose to export using either IDF Version 2.0 or IDF Version 3.0.		
		Design version	Specifies the version number of the board and library files you are creating. The default version number is 1.
Source identification	Specifies the string for source system identification. It defaults to the name of the current software.		
Default package height	Specifies the height of components if not otherwise stated in the current design. The default is 150.		
Use Filter	Click this box to filter design objects and then click the Filter button to choose the design objects.		
Filter	Displays the IDF Out Filter Setup Dialog Box for choosing design objects to exclude from the translator.		

Export	Click this button to begin the translation process.
Viewlog	Displays the `idf_out.log` file, which contains errors or warnings from the translation. Unrecognized sections of the IDF also generate error messages.

IDF Out Filter Setup Dialog Box

Use this dialog box to choose the design objects you want to exclude from the output files. These settings are in effect every time you run the `idf_out` command until you click *Reset*.

OK	Saves your changes and return to the IDF Out dialog box.
Cancel	Closes the filter dialog box without saving changes.
Reset	Deselects all prior selections, including those made in earlier sessions and saved to a configuration file.

Exporting Data from a Design Drawing

1. Type `idf_out` and arguments on a single line in the operating system Command prompt.

2. Press *Enter* to run the program.

idx in

The `idx in` command imports incremental physical design data from MCAD systems. The IDX interface translates Incremental Data Exchange (IDX) format data (including symbols, via structures, and design layers) into your design file.

Access using

- Menu Path: *File – Import – IDX*

Related Topics

- Allegro User Guide: Transferring Logic Design Data

IDX In Dialog Box

IDX file	Enter the name of a valid `.idx` input file.
Browse	Lets you choose an input file from the list of `.idx` files.
Use as baseline	Check to baseline the design for future design iterations.
Check For New IDX Files	Lets you view new IDX files. Set the *idxpath* variable in the *User Preferences Editor* dialog box to search for the IDX files.
MCAD Compare Report	Lets you compare the baseline IDX file with incremental data file.
Import	Click to begin importing the IDX file. Displays the *Select Items to Import* dialog box, if data changes are found.
Close	Click to close the IDX In dialog box.
Viewlog	Click to view the log file created during the import process.

IDX Flow Manager Import Dialog Box

Items	Displays the details of the items in the IDX file. You can edit the data in the *Reject Comment* field. Other fields are read-only.
Import	Click to select the item for import.
<−	Click to move the selected item up in the grid.
−>	Click to move the selected item down in the grid.
History	Displays the transaction history of the selected item.
IDX Accept/Reject file	Contains the updated transaction states (Accept/Reject/Cleared) and reject comments.
Select All	Check to select all items in the grid for import.
Roam and Zoom	Check to enable zoom into and highlight the selected item in the grid on the physical design.
Reset	Click to undo any changes you have made to the grid data.

Importing Incremental Physical Design Data from MCAD Systems

1. Choose *File – Import – IDX*.
   Alternatively, type `idx in` in the Command window. The *IDX In* dialog box appears.

2. Enter the name of an `.idx` file or browse to display the file browser and search for existing files.

3. Enable *Use as baseline* to specify the IDX data as baselined design for future iterations.

4. Click *Import*.
   The progress bar displays. On completion the *IDX Flow Manager Import* dialog box displays.

5. Review the changes displayed in the grid.

6. Check *Import* to select the objects to import.

7. Click *OK* to start the import process.
   The *IDX Flow Manager Import* dialog box closes and the `idx_in` log file appears.

8. Click *Close* to close the *IDX In* dialog box.

idx out

The `idx out` command or the `idx_out` batch command exports mechanical design data to the IDX data format for integration with MCAD systems.

Access using

- Menu Path: *File – Export – IDX*

`idx_out <brd> [-obhlpuv] [-c <config>] [-i <baseline>] [-f <delta_config>]`

Enter idx_out -help on the operating system prompt to see details on using this command in batch mode.

-o	The output file base name. Name of the resultant IDX files. The default is `<design_name>.idx`
-b	The board version. Valid Arguments are user-specified integer. The default value is `1`
-s	The system ID. Valid Arguments are user-specified string. The default value is ""
-h	The default height. This value applies to all package symbols without a specified package height. Valid Argument is a floating point value consistent with the original design units. The default value is the default height of drawing.
-c <config>	Filter configuration file. The name of the file is used to filter the specified parameters from the resultant IDX file.
-l <cnv>	Layer conversion file. It is an ASCII text file that specifies the mapping of a customized layer to Allegro class/subclass.
-u	Exports traces as outlines. The default setting is to export traces as lines.
-p	Exports compare file from current design.
-i <baseline>	Baseline file. The name of the file is used to create incremental IDX file.
-f <delta_config>	Incremental data configuration file. The name of the file is used to filter the specified objects from the resultant incremental IDX file.

-v	IDX version.Valid Arguments are `1.2, 2.0, 3.0, and 4.0.` The default version is `3.0.`
-brd	Name of the design file.

Examples

1. Creating Baseline file:

   idx_out test.brd -o base -c idxFilterOut.config -h 150.00

2. Creating Incremental Data File:

   idx_out test.brd -o delta -c idxFilterOut.config -h 150.00 -i base

   The format of the filter file for baseline/incremental data is as follows:

   (filter Route_Keepout_sym Route_Keepout_board Via_Keepout_sym

   Via_Keepout_board Plated_Holes NonPlated_Holes

   Vias Board_Outline Route_Outline Package_Keepout

   Unplaced_Comp Placed_Comp)

3. Exporting IDX file in old format v1.2:

   idx_out test.brd -o base -c idxFilterOut.config -v 1.2

4. Exporting customized layer such as external copper layers:

   idx_out test.brd -o test_copper -l copper_l.cnv

5. Exporting compare file from current design:

   idx_out test.brd -o test_compare -c idxFilterOut.config -h 150.00

IDX Out Dialog Box

IDX feature mode	Lets you choose mode for exporting IDX data. By default, *Standard* mode is set. To enable the *Enhanced* mode, set *idx_enhanced_features* variable in the *User Preferences Editor* dialog box.
Output file name	Enter the name of the IDX file. If the current version of the design is not the baseline, this field displays the name of the incremental data IDX file.
Browse	Lets you choose an output file from the list of `.idx` files.
Design version	Specifies the version number of the board and library files you are creating. The default version number is 1.This field gets incremented after each export.
Source identification	Specifies the string for source system identification. It defaults to the name of the current software.
Export Filter	Click the button to display the IIDF Out Filter Setup Dialog Box. Select the items to exclude from the IDX output, and click *OK*.
Re-baseline	Click to baseline the design to the current version. If the current version of the design is already baselined, this option is disabled.
Export Compare File	Click to create a baseline file from the current design and filter configuration to compare it with MCAD tool. This option is available when *IDX feature mode* is set as *Enhanced.*
Export User Layers	Choose to include user-defined layer in IDX data. This option is available when *IDX feature mode* is set as *Enhanced.*
Clear IDX Data	Click to remove IDX properties, attachments, and mechanical bend areas from the design. This option is available when *IDX feature mode* is set as *Enhanced.*
Viewlog	Click to view the content of the log file.

IDX Out Filter Setup Dialog Box

Use this dialog box to choose the design objects you want to exclude from the output file. These settings are in effect every time you run the `idx_out` command until you click *Reset*.

OK	Saves your changes and return to the *IDX Out* dialog box.
Cancel	Closes the filter dialog box without saving changes.
Reset	De-selects all prior selections, including those made in earlier sessions and saved to a configuration file.

IDX Out for User Layers Dialog Box

Use this dialog box to map the layers with the IDX layers.

IDX file	Specifies the name of the `.idx` file.
Layer conversion file	Choose layer conversion file to which to map classes and subclasses to specific IDX layers.
Lib	Choose to select a layer conversion file form the list of files available in the Select IDX Layer Conversion File Dialog Box.
Edit	Choose to modify the selected layer conversion file form the IDX Out Edit Layer Conversion File Dialog Box.
Export external layer traces as outlines	Choose to export external copper layers such that pads, traces, and shapes as outlines.
Export	Click to start the export process.

Select IDX Layer Conversion File Dialog Box

Use this dialog box to choose the layer conversion file `.cnv`.

OK	Saves your changes and return to the IDX Out for User Layers Dialog Box.
Cancel	Closes the dialog box without saving changes.
Database	Select to display layer conversion file available in the database.
Library	Select to display layer conversion file available in the library.

IDX Out Edit Layer Conversion File Dialog Box

This dialog box displays specifications related to layers and the current mapping of classes and subclasses to IDX layers. Initially, layer mappings that currently exist in the layer-conversion file display. If the specified layer conversion file is empty, or if it does not exist, all classes or subclasses appear as unmapped.

Select all	Selects or deselects all classes or subclasses in the current design that currently display.
Class filter	Controls which classes appear. Initially, this field defaults to *All*, and all classes in the current design appear. Enter your own filters, which are added to the existing list for reuse in the current session.
Subclass filter	Controls which subclasses appear. Initially, this field defaults to *All*, and all subclasses in the current design display. Enter your own filters, which are added to the existing list for reuse in the current session.
Class	Displays the classes you chose using the *Class filter* field.
Subclass	Displays the subclasses you chose using the *Subclass filter* field.
IDX layer	Lets you change the IDX layer to which a class or subclass is mapped.
Map selected items	Use these fields to specify the IDX layers for mapping to classes and subclasses.
Use layer names generated from class and subclass name	Disables the *Layer* field and maps chosen subclasses to IDX layers with long names, the `<class name>` and `<subclass name>` are equally truncated.
Subclass name only	Enable to map only subclass name to the IDX layer. This option is enabled only if *Use layer names generated from class and subclass name* is enabled.
Layer	Selects a IDX layer for mapping. Initially this contains only layers read from the specified layer-conversion file. For an empty or new layer-conversion file, no entries appear here.
Map	Maps chosen classes and subclasses of the current design to IDX layers you choose. Only items that display and that are chosen are mapped.
Unmap	Clears the mapping for all currently chosen class/subclasses in the grid.

New IDX layer	Adds a new IDX layer name that is added to the *Layer* field and that is subsequently available for use in mapping.
Include external copper layers(pad, traces, shapes)	Include external copper layers in the exported data.
Show Selected Layers	Displays only the selected class/subclass in the design by turning off the visibility of the unselected layers.
Restore Layer Visibility	Click to turn on the visibility of all the classes/subclasses.
OK	Apply the mapping information and closes the dialog box.
Cancel	Closes the dialog box without saving changes.

Creating a IDX file from a Design

1. Choose *File – Export – IDX.*
   Alternatively, type `idx out` in the Command window. The *IDX Out* dialog box appears.

2. Enter the name the `.idx` file or click *Browse* to display the file browser and search for existing files.

3. In the *Source identification* field, specify the source system on which the board was originally created; for example, `allegro_17.4`. The default value is the name of the current software version.

4. Click the *Filter Options* to display the *IDX Out Filter Setup* dialog box. Select the items to *exclude* from the IDX Output, and click *OK.*

5. Click *Re-Baseline* to baseline the design.

   If the current version of the design is already baselined, this option is disabled.

6. Click *Export.* The *IDX Out* dialog box displays the progress of the export process.

7. Click *Close* to close the *IDX Out* dialog box.

Editing the IDX Out Layer Conversion File

1. You can edit the IDX Out Layer Conversion File or preview data in chosen classes or subclasses before exporting.

2. Click *Edit* on the *IDX Out for User Layers* dialog box to display the *IDX Out Edit Layer Conversion File* dialog box, which displays the current mapping of the classes and subclasses in the layer conversion file to IDX layers. If the specified layer conversion file is empty, or if it does not exist, all classes or subclasses appear as unmapped.

3. Enter the classes and subclasses you want to list in the *Class filter* and *Subclass filter* fields, respectively. The initial default is *All*. Filters you enter become part of the drop-down list, which you can reuse in the current session.

4. Use the IDX layer column to change mappings for subclasses on a one-by-one basis if necessary.

5. Use the *Select* check box to choose individual classes and subclasses to be mapped, or use *Select all* to choose all listed classes and subclasses.

6. In the *Map selected items* section, choose an IDX layer for the class and subclass from the *Layer* field, which contains only layers read from the specified layer conversion file.

   ○ To add a new IDX Layer name, click on the *New IDX layer* button. Enter the new layer name in the pop-up that appears.

   --or--

   ○ To map the chosen subclasses to IDX layers with names of `<class name>_<subclass name>`, choose the *Use layer names generated from class and subclass names* check box (the *Layer* field is disabled as a result). If the layer name thus constructed is excessively long, the `<class name> and <subclass name>` parts of the layer name are equally truncated.

7. Click the *Map* button to complete the mapping for all currently chosen classes and subclasses of the current design to IDX layers you choose. Or choose *Unmap* to clear the mapping for all currently chosen subclasses.

8. Click *OK* to write current mapping information for layers to the layer conversion file and return to the *IDX Out for User Layers* dialog box. Subclasses for which no mappings are specified are not written to the layer-conversion file and therefore are not exported into the editor.

9. Click *Export* in the *IDX Out for User Layers* dialog box to export the data or *Close* to close the

dialog box.

iff in

The `iff in` command translates Intermediate File Format (IFF) data (including symbols, via structures, and design layers) into your design file. See the *Transferring Logic Design Data* user guide in your documentation set.

Access using

- Menu Path: *File – Import – IFF*

HP IFF Interface Dialog Box

IFF filename	Enter the name of a valid `.iff` input file. The file must adhere to the Intermediate File Format (IFF) specification as described in the HP Intermediate File Format for CAE Framework Communication Reference Manual.
Browse	Lets you choose an input file from the list of `.iff` files.
Concept block Id	Specify the block ID when your Allegro Design Entry HDL or Allegro System Architect GXL schematic contains replicated hierarchy. Each reference designator is prefaced with the block ID that you specify here.
Keepout options	You can create a route keepout based on the following parameters.
Create keepout area	Choose to automatically generate a route keepout around the sub-circuit to prevent other connections from inflicting noise into the high frequency section. The keepout area will be created on all layers of the design.
Distance to keepout	Specifies a distance the keepout will be created outside the boundaries of the sub-circuit.
Width of breakouts	To connect to various points in the RF circuit and not create Darks, breaks must be created in the keepout area. Specifies the width of the breaks in the keepout area.
Width of keepout	Specifies the width of the lines used to draw the keepout.
Location of breakouts	Specifies breaks in the keepout in either the vertical (NORTH/SOUTH), horizontal direction (EAST/WEST), or at pin locations (pins only). Choose the vertical direction for breaks to appear mid-way through the circuit in the X direction and at the top and bottom of the keepout. Choose the horizontal direction for breaks to appear mid-way through the circuit in the Y direction and on the left and right sides of the keepout.Choose pin locations to create breaks in the keepout close to pins. A location is considered a pin if it is identified using the ARTPIN statement in the `.iff` file.

OK	Imports an IFF design. Once the .iff file is imported, an outline of the circuit attaches to the cursor so you may interactively place the circuit. When you place the circuit, the active parts list and the hfsymmap.txt file determine the circuit elements then added to the design. Your product tool creates a permanent group whose name is identical to the circuit name in the .iff file, and adds each item in the circuit to the group.
Cancel	Cancels input and closes the dialog box.
Edit layer map	Choose to display the IFF Layer Map dialog box and edit the hflayermap.txt layer mapping file prior to importing a .iff file. The button is disabled until you specify a valid .iff file in the IFF Filename field.

Translating IFF Data into Your Design File

1. Choose *File – Import – IFF.*
   Alternatively, type `iff in` in the Command window. The HP IFF Interface dialog box is displayed.

2. Enter the name of a valid `.iff` file or click *Browse* to display the file browser and search for existing files.
   If the hflayermap.txt layer mapping file does not exist, the program prompts you to create a mapping file interactively.
   Alternatively, you can choose Edit Layer Map to display the IFF Layer Map dialog box and modify an existing layer mapping file before you import the `.iff` file. The button is disabled until you specify a valid `.iff` file.

3. Specify each individual layer you want to import into the design by clicking the check box next to it; otherwise, any information on that layer in the `.iff` file is ignored. Initially all the layers are deselected.

4. Click *OK* to automatically generate the hflayermap.txt file based on the mappings you specify here and return to the IFF dialog box.

5. Enter an Allegro Design Entry HDL or Allegro System Architect GXL Block ID if your Allegro Design Entry HDL or Allegro System Architect GXLschematic contains a replicated hierarchy. Each reference designator will be prefaced with that ID.
   Example: Your Allegro Design Entry HDL or Allegro System Architect GXL design uses an RF block twice. The RF block contains two identical components, R1 and R2. Since they are identical, you design that block once in the EEsof environment, but put two instances in your schematic. Each component must have a unique reference designator in your design, so when the RF block reads into the tool, you cannot use R1 and R2 (because a conflict results for the second block).
   To avoid this situation, specify the block ID each time the RF block is loaded into the product tool. The first time you import an `.iff` file, for instance, you might specify A1; the second time, A2. The reference designators become A1_R1, A1_R2, A2_R1, and A2_R2.

6. Choose the *Create Keepout Area* option to automatically generate a route keepout around the sub-circuit and prevent other connections from inflicting noise into the high frequency section.

7. Specify keepout parameters, such as the width of the keep out and its distance from the sub-circuit.

8. Click *OK* from the IFF dialog box to import an IFF design.

Once the `.iff` file is imported, an outline of the circuit attaches to the cursor so you may interactively place the circuit. When you place the circuit, the active parts list and the hfsymmap.txt file determine the circuit elements then added to the design. the product tool creates a permanent group whose name is identical to the circuit name in the `.iff` file, and adds each item in the circuit to the group.

9. Save the design and proceed as necessary.

ifnvar

The `ifnvar` command lets you include variables in scripts and environment files to change from new to old names. You can modify the behavior of script recording and replaying through the use of environment commands entered at the user interface command line.

ifnvar *<variable>* "<then command>" "<else command>" ifnvar *<variable>* <then command> ifnvar *<variable>* ";" "<else command>"

Example

How `ifnvar` can be used to switch between the new and old menu sets:

ifnvar NEWCUI "set MENU = $GLOBAL/menus" "set MENU = $GLOBAL/cuimenus" set MENUPATH = .$MENU

Related Topics

- ifvar

ifp

The `ifp` command activates *Flow Planning* application mode. Interconnect Flow Planner (IFP) is the graphic user interface for the Global Route Environment (GRE). As an Allegro application mode, IFP customizes your environment to let you plan interconnect solutions for dense, highly constrained, high pin-count designs.

Access using

- Menu Path: *Setup – Application Mode– Flow Planning*

- Toolbar Icon:

Accessing Command Help for Right Mouse Button Options in IFP Application Mode:

1. Type `helpcmd` in the Allegro Command window.
   The Command Browser dialog box appears.

2. Enable the *Help* radio button at the top of the dialog box to place the browser in Help mode.

3. Scroll the command list and select (double-click) the command you want help on.
   The command documentation should display in the Cadence Help documentation browser momentarily.

> ⓘ The IFP command syntax listed in the browser may have the reverse spelling of its related command option on the menu. For example, the actual command syntax for the *Delete Bundle* menu option is `bundle delete`.

ifvar

You can modify the behavior of script recording and replaying through the use of environment commands entered at the user interface command line. The `ifvar` command lets you include variables in scripts and environment files to change from old to new names.

ifvar *<variable>* "*<then command>*" "*<else command>*" ifvar *<variable>* *<then command>* ifvar *<variable>* ";" "*<else command>*"

Example

How `ifvar` can be used to switch between the old and new menu sets:

ifvar OLDCUI "set MENU = $GLOBAL/menus" "set MENU = $GLOBAL/cuimenus" set MENUPATH = .$MENU

Related Topics

- ifnvar

image restore last

The `image restore last` command restores to your current design to the last graphical setting stored in a file (`.view`) created using the `images` command.

You can toggle back and forth between two graphical settings through this command.

Applying Custom Images

1. Type `image restore last` in the Command window to apply a graphical image that preceded the current graphical setting.

2. Re-run `image restore last` to toggle between two consecutive graphical images.

Related Topics

- images

image restore userdefined

The `image restore userdefined` command lets you choose and restore a previously-saved graphical setting of your design from the database or library created using the `images` command.

Restoring Previously Saved Custom Graphical Images

1. Run `image restore userdefined` to open the Select Image dialog box.

2. Choose an image from the list in the list window or enter an image name in the selection filter field. You can use wildcard characters ***** and **?** to filter the list of images.

3. Click *OK* to display the chosen graphical image in the design canvas.

Related Topics

- images

Select Image Dialog Box

Use this dialog box to restore customized images that you previously saved with
the `images` command.

Selection filter	The selection filter at the top of the dialog box controls the list of saved images in the list box. The wildcard characters * and ? allow you to filter the display of images.
List of saved images	Displays the image names of the design saved with View > Image Save.
OK	Displays in the user interface work area the image chosen in the selection filter and closes the dialog box.
Cancel	Closes the dialog box without changing the design image.
Database/Library	When checked, displays the saved images in the library or in the database of the design.

image restore v <1-4>

The `image restore <`*version* `>` command restores to your current design to the graphical settings stored in the predefined file versions 1, 2, 3, or 4 created using the `images` command.

The following settings are stored in a graphical image:

- Visibility of the classes and subclasses

- Colors of the visible classes and subclasses

- Magnification of the design window

- Location of the design in the window

Restoring Graphical Settings

1. Enter `image restore v <`*version number*> at the user interface command prompt.
   The graphical image you choose is applied to the layout.

Related Topics

- images

images

The `images` command lets you create, restore, change, or delete graphical images in your design. You can save four images to predefined file versions v1, v2, v3, or v4 using this command.

The following settings are stored in a graphical image:

- Visibility of the classes and subclasses

- Colors of the visible classes and subclasses

- Magnification of the design window

- Location of the design in the window

Custom Images Dialog Box

The Custom Images dialog box contains the following controls:

Image name	Indicates the name of the graphical image file you want to create, restore, change, or delete.
Restore	Restores the named graphical image to your layout design
Save	Saves the settings for the named graphical image in a file with a `.view` extension.
Delete	Deletes the named graphical image file
OK	Saves your changes and closes the dialog box
Cancel	Cancels your changes and closes the dialog box.

Modifying Graphical Images in Your Design

1. Run the `images` command.

   The Custom Images dialog box is displayed.

2. Choose one of the image versions from the *Image Name* field.

3. Based on what you want to do with the image, click the *Restore*, *Save*, or *Delete* buttons.

4. Click *OK*.

Related Topics

- image restore last

- image restore userdefined

imouse_pos

The `imouse_pos` command can be used to record the position of a design element in a macro. The command is normally used in conjunction with another command.

import codesign die

Internal command used by Allegro X Advanced Package Designer.

import codesign pkg

Internal command.

import file manager

The `import file manager` command provides an interface to set-up the tracking of different types of import files available for update. You can configure the set-up for design data files created by the MCAD vendors, such as IDX, IDF, DXF, and IPC2581 or the LOGIC file created by the schematic tools.

The command detects if there is any new or updated file is ready for import and notify by displaying an alarm. You can either initiate the import process immediately or set the alarm to remind you later.

The *Import File Manager* dialog box has options to add or delete different file types for tracking and import.

Access using

- Menu Path: *Tools – Import File Manager*

Related Topics

- import timing
- Import mentor

Import File Manager Dialog Box

Enable	Detects new or updated files for the selected File Type definition.
File Type	Specifies name for a specific kind of file. Right-click in the field to add or remove a *File Type* row.
File Extension	Specify file extensions that apply to the import function. The default options are: • LOGIC: `pstxnet.dat` and `cdsz` • IPC2581: `xml` and `cvg` • IDX: `idx` • IDF: `dbf` and `emn` • DXF: `dxf`
Shared Directory	Specifies the path of the directory where the files will be searched for importing.
Last Import Time	Displays the date and time from the log file when the import was run last time. This date and time was retrieved from the log file created by the import process in the following order: • Specified import log file available in the working directory • The l`ast_import_time.txt` log file available in the *Shared Directory*
Status	Display the import status of the latest file import by color. • Green: import is up to date • Yellow: file is available for import • Red: the last import was failed
Import Command	Specify the import command associated with the file type definition.
Import Log File	Specify the name of the log file name generated by the import process in the working directory.

Run Command	When selected, starts the import command. On adding the command name for a *File Type* a check is performed to validate the name. You can also custom command, such as a SKILL program that should be registered using the `axlCmdRegister` function.
OK	Applies the change in settings and closes the *Import File Manager* form.
Cancel	Exits the *Import File Manager* without saving any modifications after the last *OK* or *Apply*.
Apply	Applies the change in settings into the *Import File Manager*.
Report	Displays a report of the current status for all the selected *File Type* definitions.
?	Displays a quick tip form.

These parameters are saved in a text file (`importFileManagerConfiguratoin.txt`) in the working directory.

To enable the auto-detection of import files set the environment variable *import_file_alarm_enable* in the *User Preferences Editor*. When PCB Editor is re-opened, the *Import File Manager* checks the shared directories for new or updated files based on the time interval defined by the environment variable *import_file_alarm_interval*.

When a new or updated file is detected, the *Import File Alarm* is opened. The *Import File Alarm* displays the new/updated file name(s) with the time and date the file became available.

Import File Alarm Dialog Box

Reminder time interval	Specifies the number of minutes to delay the notification of the import files availability.
Remind Later	Closes the *Import File Alarm* dialog box and displays when the *Reminder time interval* elapses.
Disable Alarm	Deactivates the *Import File Alarm* for the current session. The alarm is re-activated when a new session is started.
Launch Manager	Opens the *Import File Manager* dialog box to initialize the file import commands.

Setting up Import File Manager

1. Choose *Tools – Import File Manager*.
   Alternatively, type `import file manager` in the Command window. The *Import File Manager* dialog box is displayed.

2. Click *Enable* to the select file types for reporting.

3. Specify *File Type* to define the category for the import files.

4. To add a new file type definition, right-click to add a new *Add File Type*.

5. Specify a valid *File Extension* for the import function.

6. Specify the I*mport Command.*
   A check is performed to validate the command.

7. Specify the name of the log file generated by the import function.

8. Click *OK* to apply the settings and closes the *Import File Manager*.

Setting up Import File Notifications

1. Type `enved` in the Command window.
   The *User Preferences Editor* dialog box is displayed.

2. Open *User Preferences – File Management – Miscellaneous* .

3. Enable the variable *import_file_alarm_enable.*

4. Specify the number of minutes in the value for the variable *import_file_alarm_interval.*

5. Click *OK* in the User Preferences Editor.

6. Re-start PCB Editor to apply the user preferences settings.

Related Topics

- enved

Updating Design from Import File Manager

When you open PCB Editor, the *Import File Manager* checks the shared directories for new or updated files and opens the *Import File Alarm*. The alarm displays the file name(s) with the time and date the file became available.

1. Click *Launch Manager* in the *Import File Alarm*.
   The *Import File Manager* dialog box is displayed.

2. Verify the color in the *Status* column for the file types available for import.
   The color of the colmn must be yellow.

3. Click *Import* in the *Run Command* column.
   The user-interface associated with the import command is launched.

4. Run import process.

5. Verify the color in the *Status* column.
   The color of the column becomes green.

6. Click *Report* to review the status of all the selected file types.

7. Click *OK* to close the *Import File Manager*.

import logo

The `import logo` command lets you import a bit map(.bmp) file. By default the bitmap file is written to the Board Geometry class and Silkscreen_Top subclass. You can specify the scaling factor, rotation and location before importing the bitmap file. After import, the command also provide an option to modify the image settings.
This command is only available in the Symbol Editor.

Access using

- Menu Path: *File – Import – Logo Import*

Logo Import Dialog Box

Use this dialog box to specify the logo parameters.

Import Logo file	Select the bit map file.		
Class and Subclass Selection			
Class	Choose to specify the Class for placing logo on the symbol.		
Subclass	Choose to specify the Subclass for placing logo on the symbol.		
Import	Click to import the bit map file		
Logo Parameters			
	Scaling Factor	Specify the scale factor to resize the bit map file. Default vale is '1'.	
	Rotation	Specify the angle for rotation. Valid angles are: 90, 180, and 270 degrees. Default value is '0' degree.	
	Location X	Specify the length of the grid in the X (horizontal) direction.	
	Location Y	Specify the length of the grid in the Y (horizontal) direction.	
Modify	Click to modify the imported bit map file		
OK	Click to place the bit map file		
Cancel	Reverses the current settings and returns to the default state.		
Viewlog	Click to view the log file.		

Placing the Imported Symbol into PCB Editor

1. Copy the .osm file in the database directory.

2. Open the layout design in the PCB Editor.

3. Click *Place – Manually.*
   The the Placement dialog box is displayed.

4. In the *Advanced Settings* tab, check the *Library* box in the *List construction* field.

5. In the *Placement List* tab, choose *Format Symbols* from the drop-down menu.
   The imported format symbol is displayed in the list.

6. Choose the symbol for placement. Right-click and choose *Done* to complete the command.

Importing a Bitmap File into Symbol Editor

1. Choose *File – Import – Logo Import.*
   Type `import logo` in the Command window. The Logo Import dialog box is displayed.

2. Choose *Class* and *Subclass.*

3. Specify Logo Parameters.

4. Click *Import.*
   On successful import ,following messages are displayed in the command window and *Modify* button becomes active.

   Performing a partial design check before saving.

   Writing design to disk.

   'imported_logo_module.mdd' saved to disk.

   In case an error occurred, open the file in MS Paint, save it as 16 Color Bitmap file (`.bmp`), and re-import.

5. Click *Viewlog* to view the log file created during the import process.

6. Optionally, change the logo parameters and click *Modify* to re-import the logo.

7. Click *File – Save As* to save the symbol as format symbol.

8. Click *OK* to close the dialog box.

Import mentor

The `import mentor` command lets you import a Mentor CAD database into Allegro PCB SI L, XL, or GXL. The first time you import a design from Mentor to Allegro PCB SI L, XL, or GXL, you will need to define the electrical data for the design and save it to the Allegro PCB SI L, XL, or GXL database. The Allegro PCB SI L, XL, or GXL Database Advisor will guide you through parts of this task. You can perform the remaining tasks with Allegro PCB SI L, XL, or GXL commands.

Access using

- Menu Path: *File – Import – Mentor*

The following information is created in the Allegro PCB SI L, XL, or GXL database when a Mentor design is imported.

- Design units

- Layer information

- Shapes (board outline, keepins, keepouts, rooms, filled shapes, etc.

- Padstacks

- Package symbols

- Component definitions / gates and swap codes

- Component instances

- Logical component refdes / placed package symbol mapping

- Allegro function / component instance mapping

- SIGNAL_MODEL, TERMINATOR_PACK, VALUE properties

- Nets

- Via grid

- Connect Lines and vias

- Cline and via pin escapes

- Physical and spacing constraint set information (used for automatic routing)

You need to add the following electrical data to the Allegro PCB SI L, XL, or GXL database for the imported Mentor design before you can work with the design in Allegro PCB SI L, XL, or GXL.

- Stack-Up definition

- Constraints

- Voltage properties

- Signal model assignments

File Import Mentor Dialog Box

Use the File Import Mentor dialog box to import a Mentor design to a Allegro PCB SI L, XL, or GXL database.

Files Tab

Defines the names and locations of the various Mentor files to be imported and any archived Allegro PCB SI L, XL, or GXL database to be accessed for electrical information during the import.

Mentor Directory Area

Identifies the directory where the translator will look by default for input files.

Mentor Directory	Displays the path to the directory containing the Mentor files. This path is propagated to each of the file paths in the Mentor Files area. You can override this path for individual files.
Browse	Displays the Import Mentor dialog box that lists directories containing Mentor files. The Import Mentor directories browser is displayed by clicking Browse in the Mentor Directory area of the Import Mentor dialog box. 1. In the list box, scroll the available directories to locate the directory containing the default Mentor files to import to Allegro PCB SI L, XL, or GXL. 2. In the list box, click to choose the directory containing the Mentor files. The chosen directory path displays in the Directories field above the list box. 3. Click OK to dismiss the browser and display the directory path in the Mentor Directory field of the Import Mentor dialog box.The directory path chosen here specifies the directory where the translator will look for default Mentor files.

Mentor Files Area

The fields in this area display paths to the individual Mentor files. For each individual file, you can override the default directory path copied from the Mentor Directory: field.

Geometries ASCII File	Displays the path to the Mentor ASCII geometries file.
Tech File	Displays the path to the Mentor tech file.

Nets File	Displays the path to the Mentor net list file.
Components File	Displays the path to the Mentor component placement file.
Gates File	Displays the path to the Mentor gate and pin swapping data file.
Pins File	Displays the path to the Mentor pin properties file.
Traces File	Displays the path to the Mentor traces file.
Testpoints File	Displays the path to the Mentor testpoint generation file.
Browse	Displays a file browser set to display the appropriate Mentor file type.

Allegro PCB SI L, XL, or GXL Files Area

Displays the path to an archived Allegro PCB SI L, XL, or GXL database from which to retrieve electrical constraints and parameters. The Allegro PCB SI L, XL, or GXL database would typically have been created and archived during a previous import.

Template Board	Displays the path to an archived Allegro PCB SI L, XL, or GXL database from which electrical constraints and parameters should be retrieved.
Browse	Displays a file browser set to display `.brd` files.

Options Tab

Defines how various details of the translation will be performed.

Output Units Area

Indicates which units the Allegro output board should be created in. The default is mils.

Device Class Mapping Area

Defines how Allegro PCB SI L, XL, or GXL device classes are assigned to components.

Discrete, IC RefDes, and IO RefDes Names areas

For each of the three Allegro PCB SI L, XL, or GXL device classes: Discrete devices, Integrated Circuits, and IO (connector) devices, create a list of filter strings against which component reference designators are matched during translation. Choose one of the three as the default device class.

During translation, a component whose reference designator matches a filter is assigned that device class. A component whose reference designator does not match any of the filters is assigned the default device class.

Default Device Class	Designates one of the three device classes as the default.
Add Filter	Specifies a filter string to add to the list box.
List Box	Lists the filter strings for the device class. Discrete RefDes defaults: C*, L*, and R*. IC RefDes defaults: IC*, and U*. IO RefDes defaults: None.
Remove	Removes a chosen filter string from the list box.
Clear	Removes all filter strings from the list box.

Post Processing Area

Specifies any post-processing operations to perform after the Mentor database is imported.

Dump Libraries	Performs a library dump operation.
Db/Check/Db/Fix	Automatically runs dbcheck and dbfix on the translated design.

Common Buttons

Import	Imports the Mentor data from the locations specified on the Files tab using the defaults established on the Options tab. The Mentor data is added to the currently active Allegro PCB SI L, XL, or GXL design. The importMentor.log file is created and displayed in a text window once the import is complete.

Importing a Mentor CAD Database into Allegro PCB SI L, XL, or GXL

1. Choose *File – Import – Mentor.*
    Alternatively, type `import mentor` in the Command window. The Import Mentor dialog box is displayed.

2. Configure the dialog box as described above.

3. Click *Import*.

import timing

The `import timing` command displays the Import Timing dialog box that lets you include device delay and cycle time data in the Timing Spreadsheet by importing a MOTIVE timing file.

Importing the MOTIVE file puts integrated circuit delays and setup/hold requirements from the extracted timing model file into the design database, applying the imported data to all component instances listed in the ICs for Import list box.

You can then view and reset these numbers in the Switch Settle area of the Timing Spreadsheet.

Import Timing Dialog Box

Use the File – Import Timing dialog box to import timing information from a MOTIVE file and apply the imported data to all component instances listed in the ICs for Import list box.

Input Timing File Area	Identifies the file from which Allegro Package SI reads the timing information for a device.
Refdes Filter	Specifies a reference designator that Allegro Package SI uses to narrow the list of ICs in the Candidate ICs list box.Click Sort: Refdes to display all ICs with the specified reference designator.
Device Filter	Specifies a device name that Allegro Package SI uses to narrow the list of ICs in the Candidate ICs list box. Click *Sort: Device* to display all ICs with the specified device name.
Sort: Refdes/Device	Filters the list of Candidate ICs by the specified reference designator or device name.
Candidate ICs	Lists the integrated circuits with the specified refdes or device name.
Move All =>Button	Copies all ICs from the Candidate ICs list to the ICs for Import list.
Move <= All Button	Removes all ICs from ICs for Import list.
ICs for Import	Lists integrated circuits whose timing data will be imported when you click Import.
Import Button	Imports timing data from the ICs in ICs for Import list.
Cancel	Cancels the import operation.

Importng a MOTIVE Timing File

1. Type `import timing` in the Command window.

2. Choose the MOTIVE file to import.

3. Click *Browse* to navigate to the directory and file you want.

4. Click *OK*.
   The File Import Timing dialog box is displayed. The Candidate ICs area lists the ICs whose timing data you can import.

   - To narrow the list to ICs with a certain reference designator, type the reference designator in the Refdes Filter field and click Sort: Refdes.

   - To narrow the list to all instances of a particular device, type the device name in Device Filter field and click Sort: Device.

5. Click *Move All*.

6. Click *Import*.

interface_planner

Interface Planning application mode customizes your environment to plan and create the port assignments and optimize existing assignments after creating and mapping interfaces in a design for co-design flow. An application mode provides an intuitive environment in which commands used frequently in a particular task domain, such as editing or moving, are readily accessible from right-mouse- button popup menus, based on a selection set of design elements you have chosen.

Use *Setup – Application Mode – None* to exit from the current application mode and return to a menu-driven editing mode, or verb-noun use model, in which you choose a command, then the design element.

For more information on the Interface Planning application mode, see the Getting Started with Physical Design user guide in your documentation set.

Access using

- Menu Path: *Setup – Application Mode – Interface Planning*

Related Topics

- noappmode

Interface Planner Tree View

The Interface Planner tree view displays the co-design components in a design as nodes with the top-level interfaces and all ports that are not member of any interfaces listed under each node.

You can click a node to list the number of assignments required, number of partial assignments, and the detailed assignments for the interface or PortGroup. It also displays the current color for an Interface.
A random color is assigned, by default. To change the color click on the swatch and choose a different color.

OK	Updates the database with changes.
Oops	Lets you undo last change while still in the command.
Cancel	Closes the dialog box and terminates the command without changing database.
Hide	Hides the Interface planner tree view form. Choose *Show Interface Form* from the pop-up menu to show the form again.
Help	Displays help for this form.

Assigning Pins to PortGroups and Ports

You can assign pins to PortGroup or assign a single physical pin to a logical Port.

To assign pins to a PortGroup from the tree view:

1. Select the component to assign in the tree.

2. Choose *Assign* from the pop-up menu.
   Pins that can be assigned are highlighted in the design.

3. Select from the highlighted pins.

   If you select a Port in the tree view and assign a pin to it, the next Port is selected.

4. Click *OK*.

To assign pins to a PortGroup from the canvas:

1. Select any or all of Groups, Pins, Clines or Ratsnests in the Find filter.

2. Select the pins.
   If the tree view has a PortGroup selected, the pins assigned to the PortGroup are selected by default.

3. Choose *Assign PortGroup* from the pop-up or enter `assign portgroup`

To assign a pin to a port:

1. Select the pin you want to assign to a port

2. Choose *Assign Port* from the pop-up menu and then choose the port from the sub-menu.
   The *Assign Port* option is available only if the selected pin is not already assigned to any port.
   If the selected pin is not assign ed to any PortGroup, the ports that are not member of any PortGroup are listed.

Related Topics

- Optimizing Pin Assignments

- Swapping Pins Between PortGroups

- Coloring of PortGroup Pins

Removing Pin Assignments from PortGroups and Ports

You can remove pin assignments from PortGroups or ports.
When you remove a pin assignment from a PortGroup, it becomes a member of the parent PortGroup. A pin assignment is removed from a top-level PortGroup, the pin is removed from the membership of all PortGroups.

A pin assignment removed from a port remains a member of the PortGroup.

To unassign pins from the tree view:

1. Choose *Unassign* from the pop-up menu.

2. Click the pins you want to unassign.

3. Click *OK*.

To unassign pins from the canvas:

1. Select the pins

2. Choose *Unassign* from the pop-up menu or enter `unassign`.

Removing Pins from All PortGroups

You can unassign pins from all PortGroups, including the top-level groups.

To remove pins from all PortGroups from the tree view:

1. Choose *Unassign All* from the pop-up menu

2. Select the pins you want to unassign from all PortGroups.
   The pins are removed from all PortGroups.

To remove pins from all PortGroups from the canvas:

1. Select the pins you want to remove from all PortGroups

2. Choose *Unassign All* from the pop-up menu or enter `unassign all`.

Related Topics

- Optimizing Pin Assignments

- Swapping Pins Between PortGroups

- Coloring of PortGroup Pins

Removing Pins from All PortGroups

You can unassign pins from all PortGroups, including the top-level groups.

To remove pins from all PortGroups from the tree view:

1. Choose *Unassign All* from the pop-up menu

2. Select the pins you want to unassign from all PortGroups.
   The pins are removed from all PortGroups.

To remove pins from all PortGroups from the canvas:

1. Select the pins you want to remove from all PortGroups

2. Choose *Unassign All* from the pop-up menu or enter `unassign all`.

Related Topics

- Optimizing Pin Assignments

- Swapping Pins Between PortGroups

- Coloring of PortGroup Pins

Optimizing Pin Assignments

You can refine assignments by using the iterative optimize utility that minimizes rat crossings. Note that physical pins that are assigned to ports will be unassigned by this utility.

- Do one of the following:

    - To optimize assignments from the tree view, choose *Optimize* from the pop-up menu.

    - To optimize from the canvas, select the pins and choose *Optimize* from the pop-up menu or enter `optimize`.

Related Topics

- Assigning Pins to PortGroups and Ports

- Removing Pin Assignments from PortGroups and Ports

- Coloring of PortGroup Pins

Swapping Pins Between PortGroups

You can swap the pins or ports in a PortGroup with the pins and ports of another PortGroup, if both the PortGroups:

- Are sub-PortGroups of the same top-level PortGroup

- Contain same number of pins and ports

To swap PortGroups:

1. Either in the canvas or tree view choose *Swap* from the pop-up menu.
   The pins of PortGroups that can be swapped with the current PortGroup are highlighted.

2. Select a highlighted pin of the PortGroup you want to swap the current PortGroup.
   Selecting a pin selects the entire PortGroup and the pins and ports are swapped.

Related Topics

- Assigning Pins to PortGroups and Ports

- Removing Pin Assignments from PortGroups and Ports

Coloring of PortGroup Pins

The pins of a co-design component are colored with the Interface color in the canvas. The color of a pin is based on the current selection in the tree view, as described:

Component	Pins belonging to the top-level PortGroup are colored in the corresponding NetGroup color.
PortGroup	Pins of the PortGroup and its peer are colored with their respective NetGroup color.

- To change color click on the *Color* swatch and select a color. Alternatively, you can choose *Display – Color/Visibility* and select *Nets* to change pin color.

Related Topics

- Assigning Pins to PortGroups and Ports

- Removing Pin Assignments from PortGroups and Ports

- Optimizing Pin Assignments

interface_vis

Internal command.

ipc356 out

The `ipc356 out` command or `ipc356_out` batch command lets you export information from the current SPB design to an output file that maps directly to either the IPC-D-356 format and supports standard electrical TEST record structure, or the IPC-D-356A format, which additionally supports buried and blind via extended records. An output file with a `.ipc` file extension is created in your current working directory.

This command:

- Uses unique names for dummy nets, which is useful for identifying unused probe pins.

- Includes a comment in the output file when you use the NET_SHORT property and indicates which nets are shorted together, where they are shorted, and by which pin or via.

- Supports non-orthogonal padstacks rotation, which allows you to place components at non-orthogonal rotation and obtain the correct output.

During translation, the program creates the *ipc356_out.log* file in your current working directory. If signal pins are unplaced, they are skipped, and the following message displays on the console window prompt and in the log file:

<refdes>-<pin num> on net <net name> is unplaced - SKIPPING.

Access using

Menu Path: *File – Export – IPC 356*

`ipc356_out` can be run in batch mode if you do not want to run the program as an interactive process. The command line switches correspond to the fields that appear on the dialog box when you run `ipc356 out`.

To export IPC data in batch mode type the following command at a system prompt:

% ipc356_out [-t] [-i] [-r] [-f] [-A] [-c] [-b] [-e]<Input board name> <ipc_filename>

-t	The title of the IPC output file you are creating (optional).
-i	Board identification number (optional).
-r	Revision level (optional).
-f	Header File (optional).

-A	Generates complex records for blind/buried vias (to distinguish them from through-hole), and suppresses Y dimensions for circular figures. This supports the IPC-D-356A format. Use uppercase for this switch. When you include the optional command line switches, this information is listed in the output file
-c	Ignores wirebond layers.
-b	Ignores backdrill data.
-e	Exports embedded components.
Input board name	The name of your design board (required). The `.brd` extension is not required.
ipc_filename	The name of the IPC output file (required). The `.ipc` extension is not required.

Example

ipc356_out -i 120 -r D test.brd test.ipc

IPC-D- 356 Dialog Box

Output file	The name of the output file you want to create. You can use the Browse button to find a previously created file.
IPC Version	Choices are: *IPC-D-356 –Supports the base substrate electrical test data format.* *IPC-D-356A – Supports the standard format as well as buried and blind via extended records.*
Board title	A title for the board data you are extracting. If you do not enter a title here, the current board filename is used.
Identification number	A number to identify this data. This number indicates the part number for data you want to define in a job set or in a parameter record, such as the artwork number, or schematic number. The program assigns 001 as the identification number if you do not enter a number.
Revision	A revision number to help track your data. The program assigns A as the revision level if you do not enter a revision identifier.
Header file	The header file you want to use. Click the button to the right to browse for the correct header file. The header file is a text file that contains any comments, specifications, or descriptive data you want to include in the output file. You create this with a text editor in your current working directory. The editor inserts the information from the header file as comments into the beginning of the output file.
Ignore backdrill	If checked, the editor does not export backdrill data.
Ignore die layers	If checked, the editor does not export bond finger via pads, which exist on die stack layers. Die pins on these layers continue to be exported.

Exporting Information to an Output File Supporting IPC-D-356 or IPC-D-356A Formats

1. Choose *File – Export – IPC 356.*
   Alternatively, type `ipc356 out` in the Command window. The IPC - 356 - D dialog box appears.

2. Configure the dialog box as described above.

3. Click *Export* to run the program.

During translation, the editor creates the output IPC file (*filename.ipc*) and a log file, *ipc356_out.log* in your current working directory.

ipc2581 in

The ipc2581 in command or `ipc2581_in` batch command process imports graphical data into physical design data and creates objects under the class Manufacturing. This import is used only for reference or comparison.

Access using

- Menu Path: *File – Import – IPC2581*

```
ipc2581_in <ipc2581 data file> [-xg][-o <output allegro database>][-i <input allegro
database>
```

-X	Imports layer stackup. Default is Off.
-g	Imports layer features. Default is Off.
-o	Output board file. Default is `<ipc2581_data_name>.brd`
-i	Input board file name. By default the output allegro database is used.
<ipc2581 data file>	Name of the IPC2581 data file.

Related Topics

- Allegro User Guide: Transferring Logic Design Data

IPC2581 In Dialog Box

IPC2581 file	Enter the name of a valid `.cvg`, `.zip`,`.xml` input file or click the browse button to choose an input file.
Import Options	
Layer stackup	Enable to import layer stackup
Layer features	Enable to import layer features
Purge Import	
Compare	Click to compare the IPC layers against the design layers.
Import	Click to begin importing the IPC2581 file.
Close	Click to close the IPC2581In dialog box.
Viewlog	Click to view the log file created during the import process.

IPC2581 Compare Dialog Box

Global visibility	Turn on/off all layers
IPC2581 layer	These are IPC layers for comparison.
IPC	Click the checkbox to turn on/off an IPC layer.
Film	Click the checkbox to turn on/off a film record.
Browse	Displays a dialog box to edit the currently mapped film that corresponds to an IPC layer.
Close	Click to close the *IPC2581Compare* dialog box.

Translating IPC2581 Data into Your Design File

1. *Choose File – Import – IPC2581.*
   Alternatively, type `ipc2581 in` in the Command window. The *IPC2581 In* dialog box is displayed.

2. Enter the name of a valid `.cvg or .xml or .zip` file or click *Browse* to display the file browser and search for existing files.

3. Click *Compare* after an import to open the *IPC2581 Compare* dialog box to turn on/off the IPC2581 layers. You can also change layer color for comparison.

4. Click *Import*.

5. Click *Viewlog* to see the log file.

6. Click *Close* to exit the *IPC2581 In* dialog box.

ipc2581 out

The ipc2581 out command or `ipc2581_out` batch command exports physical design data to the IPC2581 data format.

Access using

- Menu Path: *File – Export – IPC2581*

```
ipc2581_out [-ufdblRnpstcgkvexyzPDOIMS] [-g <attr_file>] [-o <output_file>] <brd>
```

-u	Output units: Inch, Millimeter, Micron. Default Is Inch (Inch/mils), Millimeter (Meter/millim/centim), Micron (Micron)
-o	Output file name. Default name is `<drawing>_ipc2581`
-s	Source tool. The default is CadenceTool.
-f	The IPC2581 format version to write. Valid values are: 1.00, 1.01(Amendment 1), 1.02(IPC2581-A), 1.03(IPC2581-B), and 1.04(IPC2581-C) The default version is 1.03.
-q	Export the following default properties: • Component/LOGICAL_PATH • Component/PRIM_FILE • Net/LOGICAL_PATH Use [-g] argument for exporting more properties.

-g	Property configuration file. It is a ASCII text file that specifies the property name for components and nets in the following format: • Component/DFA_DEV_CLASS_UD • Component/DFA_DEV_TYPE • Net/DIFFP_PHASE_TOL_DYNAMIC • Net/MAX_VIA_COUNT
-P	Specifies global package pin1 orientation. Valid arguments are: LOWER_LEFT, LEFT, LEFT_CENTER , UPPER_LEFT, UPPER_CENTER, UPPER_RIGHT, RIGHT, RIGHT_CENTER, LOWER_RIGHT, LOWER_CENTER, CENTER, OTHER The default value is OTHER
-b	Bill of material (BOM). Default is Off.
-l	Layer stackup. Default is Off.
-K	Backdrill layers. Default is Off.
-R	Regular drill layers.Default is Off.
-n	Logical and physical netlist. Default is Off.
-G	Logical netlist
-Y	Physical netlist
-p	Component package. Default is Off.
-t	Device land pattern. Default is Off.
-d	Device Descriptions. Default is Off.

-c	Component assembly. Default is Off.
-D	Documentation layers. Default is Off.
-O	Outer copper layers. Default is Off.
-I	Inner copper layers.Default is Off.
-M	Miscellaneous fabrication layers. Default is Off.
-S	SolderMask/SolderPaste Legend layers. Default is Off.
-A	Solder Mask Layers
-B	Solder Paste Layers
-C	Silkscreen(Legend) Layers
-U	Profile(Outline) for Rev-C and above
-k	Padstack Definitions. Default is Off.
-v	Cavities for embedded component board. Default is Off.
-y	Export cross-section data only. Default is Off.
-e	Export vector text. Default is Off.
-z	Zip file. Default is Off.
-x	Layer mapping configuration file. Default is Off. The data entry format is `<Artwork Film Name>, <Layer mapping category>` A sample file `IPC2581_LayerMappingCfg.txt` is located at `<insatll_hierarchy>\share\pcb\text\`

Required Argument

| **<brd>** | The name of the design file on which the command is run. |

Example

1. Generate IPC2581 file `test.xml` which contains physical design objects on outer copper layers, inner layers, documentation layers, miscellaneous layers, and soldermask/solderpaste layers.

   ipc2581_out test.brd -o test -O -I -D -M -S

   You can define these layers in the IPC2581 Layer Mapping Editor dialog box and save the definition into the design.

2. Generate IPC2581 file `test.xml` which contains net list, package, and component description with output units as millimeter.

   ipc2581_out test.brd -o -test -n -p -c -u -MILLIMETER

Related Topics

- Allegro User Guide: Transferring Logic Design Data

IPC2581 Export Dialog Box

IPC2581 Export Tab	
Output file name	Enter the name of the IPC2581 file.
Browse	Lets you choose an output file from the list of .xml files.
IPC2581 version	Specifies the version number IPC2581 data format. Supported versions are: IPC2581-1, IPC2581-A, IPC2581-B, and IPC2581-C
Output units	Specifies the output unit to export. Supported values are: Millimeter, Micron, and Inch
Global package pin one orientation	Specifies pin1 orientation of global package
File Segmentations and Function Apportionment	Displays default selections of schema selection based on the function mode and function level.

		Function Level	Specifies the data complexity of the IPC2581 file needed for each functional mode. There are three valid values 1, 2, and, 3. With version IPC2581-C, this option is not required.
Layer Mapping Edit	Opens IPC2581 Layer Mapping Editor to specify class/subclass for outer copper layers, inner layers, documentation layers, solder mask, solder paste, silkscreen legend layers, and miscellaneous fab layers.		
Film Creation	Opens Artwork Control Form to add/update film records for layer mapping.		
Vector text	Select to export the text characters as line segments.		
Compress output file (.zip)	Select to generate the compressed IPC2581 file.		
Export	Click to generate IPC2581 file.		

Close	Click to close the *IPC2581 Export* dialog box.
Viewlog	Click to view the log file.
Export Property Tab	
Available properties	Click to add properties to export for component and net.

IPC2581 Layer Mapping Editor Dialog Box

You can define the Class/Subclass for outer copper layers, inner layers, documentation layers, solder mask, solder paste, silkscreen legend, and miscellaneous fab layers. This definition is saved into the database and you can use it for future exports or batch exports.

OK	Save your changes and return to the *IPC2581 Export* dialog box.
Cancel	Closes the *Layer Mapping Editor dialog* box without saving changes.

Artwork Control Form

You can create a new film record for IPC2581 export. The new film is created for all the four domains (including Artwork, PDF, IPC2581, and, Visibility). You can however create the new film for any of the domains. For this purpose *Domain Selection* can be used to change the domain for each film record.

Unused pad can be suppressed by enabling the *Dynamic unused pads suppression* using *Setup – Cross-section – Unused Pads Suppression.* This option suppresses unconnected pads for the selected object types (pin/via) on the selected inner layers. When exporting IPC2581, it is recommended to use the *Dynamic unused pads suppression* option, even if the *Suppress Unconnected Pads* option on the Artwork form is enabled.

The version IPC2581-C provides a capability to define IPC2581 layer name in the output using artwork film names by setting an environment variable i*pc25881_enable_artwork_filename_affixes.* You can also omit the square drill data and export it as a slot by setting another environment variable *ipc2581_output_square_hole_as_slot* in the User Preferences Editor.

Translating Design File to IPC2581 format

1. Choose *File – Export – IPC2581.*
   Alternatively, type `ipc2581 out` in the Command window. The *IPC2581 Export* dialog box is displayed.

2. Enter the name of the `.xml` file or *Browse* to display the file browser and search for existing files.

3. Specify the IPC2581 version.

4. Specify the output units.

5. Choose pin1 orientation for global package.

6. Select the functional mode.

7. Select Function level.

8. Click *Layer Mapping Edit* to display the *IPC2581 Layer Mapping Editor* dialog box. Select the layers for export, and click *OK*.

9. Click *Film Creation* to display the *Artwork Control Form* dialog box. Select the layers and domain for recording film.

10. Click *Vector text* checkbox to export the text characters as line segments.

11. Click *Compress output file(*`.zip`*)* checkbox to create the compressed output file.

12. Click *Export* to generate the IPC2581 file.

13. Click *Viewlog* to see the `ipc2581_out.log` file.

14. Click *Close* to exit the *IPC2581 Export* dialog box.

ipc spec edit

The `ipc spec edit` command assigns IPC2581 spec to design elements, or changes, or deletes existing IPC2581 specs.

Access using

- Menu Path: *Edit – IPC2581 Specs*

Edit IPC2581 Specs Dialog Box

Use this dialog box to choose the IPC2581 specs you want to add, edit, or delete. When you choose a IPC2581 spec from the *Available IPC2581 Specs* list, or type in the spec name, it appears in the panel on the right-hand side of the dialog box.

Available IPC2581 Specs	Lists all IPC2581 spec that are attached to the chosen elements.
Name	Lets you type the name of the IPC2581 spec you want to delete or change instead of choosing it from the *Available IPC2581 Specs* list.
Delete	Marks the chosen IPC2581 spec for deletion from the selected objects.
Info	Identifies the IPC2581 spec that you are adding, deleting, or modifying.
Spec Name	Displays the IPC2581 spec name that is selected in the *Available IPC2581 Specs* list.
Reset	Resets the dialog box, which clears the right-hand panel.
Apply	Adds or deletes marked IPC2581 specs from the selected objects.
Show	Displays instances of the selected IPC2581 specs property.

Related Topics

- ipc2581 out

- ipc2581 in

Assigning IPC2581 Specifications to Design Elements

1. Choose *Edit – IPC2581 Specs.*
   Alternatively, type `ipc spec edit` in the Command window.

2. Enable object types in *Find* filter.

3. Hover your cursor over an object.
   The tool highlights the element and a data tip identifies its name.

4. Click to select the object or window select to choose a group of objects.
   The Edit IPC2581 Specs dialog box appears.

5. In the Edit IPC2581 Specs dialog box, choose a IPC2581 spec from the list of *Available IPC2581 Specs* or enter a spec name into the *Name text* field.
   The selected spec appears in the right side of the dialog box. You can choose any number of specs.

6. To assign a spec to a design element, click *Apply* .
   The Show Specs window updates to display the current spec applied to the objects.

7. To delete a spec, click *Delete to the left of the spec name and then click Apply.*

8. Click *Close to close the Edit IPC2581 Specs dialog box.*

9. Repeat steps 2 to 8 to complete IPC2581 spec assignments.

10. Right-click and choose *Done* from the pop-up menu to end the command.

Related Topics

- ipc2581 out
- ipc2581 in

ipick

The `ipick` command allows you to enter the incremental distances from the previous coordinates for objects you want to find and highlight. You must be in a command mode—for example, `add connect` to activate the `ipick` command.

You can specify Cartesian coordinates in terms of X and Y or polar coordinates in terms of distance and angle.

Highlighting Objects

1. Ensure that you are in command mode, for example, `add connect`.

2. At the console window prompt, type `ipick`.

   The Pick dialog box appears.

3. Select *XY Coordinate* to specify cartesian coordinates or select *Distance + Angle* to specify polar coordinates.

4. Type the incremental coordinates in the *Value* field. Be sure to leave a space between the numbers.

5. To position the point on the nearest grid from the coordinates, check the *Snap to current grid* box.

   The *Relative (from last pick) box* is already enabled because you are using the `ipick` command.

6. Click *OK* to dismiss the dialog box and establish the point.

You can also type the incremental coordinates on the command line after typing the command name. For example:

```
ipick 300 -400
```


Related Topics

- Pick dialog box

- iapick

- add connect

ipick_to_grid

The `ipick_to_grid` command is used in scripts to record mouse clicks that must be mapped to the grid. The coordinate format is the same as that of the `pick` command. When the *command is used in macro files, the coordinate system is relative to a pick sign.*

Example

ipick_to_grid x y

Related Topics

- Pick dialog box

- iapick_to_grid

- pick

ipick_to_gridunit

The `ipick_to_grid` command moves selected database elements in 1-grid increments according to the design's database units.

ipick_to_gridunit <direction> <grid_units>

In conjunction with this command, the *Ctrl* or *Shift* keys plus the *Up, Down, Left*, and *Right* arrow keys, which are defined as default aliases in the system `env` file, let you move selected elements in 1-grid increments in the desired direction. (The system `env` file is located at `share\pcb\text\env`.)

- The *Ctrl* key aliases function in the menu-driven editing mode, in which you choose a command (verb), then the design element (noun).

- The *Shift* key aliases function in placement-edit application mode, in which the tool defaults to a pre-selection use model, letting you choose a design element (noun), and then a command (verb) from the right mouse button pop-up menu.

To use these aliases, first choose an element, and ensure that it remains highlighted, at which point the *Ctrl* or *Shift* plus arrow keys can be used to move it incrementally. In placement-edit application mode, *Shift* click to select and move an element incrementally if no interactive command is active. The *Rotation Point* should not be set to *User Pick* when you are using either the *Ctrl* or *Shift* key aliases. (Right-click to display the pop-up menu and choose *Move,* then *Rotation Point* to access this option). If *Rotation Point* is set to *User Pick*, add another `ipick 0` to the default aliases to set the pick point.

Examples

ipick_to_gridunit 0 +1	increments by 1 grid unit in the y direction
ipick_to_gridunit 0 -1	decrements by 1 grid unit in the y direction
ipick_to_gridunit +1	increments by 1 grid unit in the x direction
ipick_to_gridunit -1	decrements by 1 grid unit in the x direction

Default Aliases in the System env File

alias CUp ipick 0; ipick_to_gridunit 0 +1 alias SUp move; ipick_to_gridunit 0 +1	increments by 1 grid unit in the y direction
alias CDown ipick 0; ipick_to_gridunit 0 -1 alias SDown move; ipick_to_gridunit 0 -1	decrements by 1 grid unit in the y direction
alias CRight ipick 0; ipick_to_gridunit +1 alias SRight move; ipick_to_gridunit +1	increments by 1 grid unit in the x direction
alias CLeft ipick 0; ipick_to_gridunit -1 alias SLeft move; ipick_to_gridunit -1	decrements by 1 grid unit in the x direction

irdrop

The `irdrop` command lets you perform static IR-Drop analyses for nets. This functionality is detailed in Analyzing for Static (IR) Drop in the *PCB SI User Guide*.

Access using

- Menu Path: *Analyze – IR-Drop*

Related Topics

- Analyzing for Static (IR) Drop

IR-Drop Analysis Dialog Box

The IR-Drop Analysis dialog box is a tabbed form. Use this dialog to set up net information and analysis preferences, check the integrity of your set-up, then run your IR-Drop simulation analysis.

Net Information Tab

Net List Section

Filter	Lets you tailor the list of net names using alphabetic and wildcard combinations; for example, `A*`.
DCNet Only	When checked, displays only the voltage nets in the design instead of the voltage and signal nets.
Identify DC Nets	Launches the Identify DC Nets dialog box, from where you can change the voltage of the DC nets in the design.
Net List Columns	*Select:* Adds the selected net for IR-Drop analysis.
	Name: The name of the net.
	Voltage (V): The voltage of the DC net, set in the Identify DC Nets dialog box. (Signal nets should not have a voltage.)
	Threshold (V): The threshold value of the selected net. The default value for all nets is 0.1V.
	Note: Values below the threshold are flagged with an asterisk (*) in the generated analysis reports.

Components on Selected Net Section

Filter	Lets you tailor the list of components in the selected nets using alphabetic and wildcard combinations; for example, `A*`.
Select All	Highlights the entire list of components.
Deselect All	De-highlights the entire list of components.

Pins on selected Components Section

Pin List Columns	*Pin Name: The name of the pins in the selected components.*
	Port Type: The port type of the associated pin, either *Open, Source,* or *Sink.* Pins assigned the *VOLTAGE_SOURCE_PIN* property are automatically set *to Source* port type.
	Current (A): The current value of the selected pin if designated as a *Sink* type.
	Note: You can assign negative current values only to sink pins in a ground (GND) net.

Preferences Tab

Drill Plating Information Section

Selected Net Only	When checked, this box displays only the padstacks associated with the nets selected in the Net List section of the Net Information tab. Unchecked, all padstacks in the design are listed.
Padstack Name	Displays the padstacks used, either in the selected net only, or in the entire design.
Drill Size	Displays the shape and size of the padstack.
Plating Thickness	Displays the plating thickness of the padstack. This field is inactive unless one or more padstacks are highlighted.
Material	Displays the materials available to plate the padstack.
	A drop-down menu lets you change the material selection for individual padstacks, or you can right click on the *Material* column header to change the material for all the padstacks in the list.

Shape Mesh Information Section

Radio Buttons	*Fine:* Selecting this button sets the size of the mesh cells in the design to 0.1 mm. *Regular:* Selecting this button sets the size of the mesh cells in the design to 0.5 mm. *Coarse:* Selecting this button sets the size of the mesh cells in the design to 1.0 mm. *Custom:* This button lets you define the size of the design's shape mesh cells.
X Size	Enabled when you select the *Custom* button, this field lets you enter an *X* value for the design's shape mesh cell size.
Y Size	Enabled when you select the *Custom* button, this field lets you enter an *Y* value for the design's shape mesh cell size.
Temperature rise threshold	Lets you define the maximum allowed temperature rise for the net you are analyzing. If high current or current density on any signal traces or shapes cause a violation of the threshold setting, a warning is generated in the result report.

Common Functional Buttons

These buttons are common to both tabs:

OK	Saves the changes you have made to the nets in the design and closes the dialog box, returning SI to an idle state.
Cancel	Closes the dialog box without saving any changes you have made in it, returning SI to an idle state.
Analyze All	Runs an IR-Drop analysis simulation on the nets you have configured in the dialog box.
Hide	Close the dialog box without terminating the `irdrop` command. Redisplay the dialog box by right-clicking in the SI canvas and selecting *Show Main* from the pop-up menu.

Material	Displays the material selection form that lets you view the materials present in the `materials.dat` file. When you access the material selection form from here, the materials attributes are read-only. See define materials for detailed information on this form.

Analyzing Static IR-Drop for Nets

The use model for performing IR-Drop analysis relies extensively on a dynamic graphical user interface. The following steps outline the high-level procedure for performing voltage drop analyses.

1. Choose *Analyze – IR-Drop* in the PCB SI user interface to open the IR-Drop Analysis dialog box.
   Alternatively, type `irdrop` in the Command window.

2. In the Net Information tab of the form, select the net or nets you want to simulate.

3. Select the components in the selected nets for which you will set pin information.

4. Set the port types and current values for the pins in the selected components.

5. Check the *Select* box next to the selected net or nets to run a preliminary check of your setup.

6. Review the information in the pop-up and make any required changes to the setup of the pin settings.

7. Click the Preferences tab in the form and set the drill plating information and mesh size.

8. Set a temperature rise threshold or accept the default value of 5 degrees Celsius.

9. To perform an analysis for all selected nets, click *Analyze All*. –or– To analyze for a single net (of multiple selected nets) in the Options tab:

   a. Highlight the net and right-click.

   b. Select *Analyze* from the pop-up menu.

10. Review the results in the following ways:

    a. View the color display and associated voltages, currents, or temperature rises in the Options tab of the SI user interface.

    b. Create a reference point on the voltage-drop color display in the SI canvas and view the relative and absolute values displayed in the Options tab.

    c. Review the text report listing the voltage result at any sink pins.

11. If required, make any necessary changes to the setup of the nets and run the analysis again, repeating the above steps.

island_delete

The `island_delete` command lets you remove islands, which are non-conductive isolated areas of copper. Islands can be deleted via the design window using the standard selection mechanism or using the Options tab to navigate through the islands on the current layer, as described in the procedure below. You can also select the highlighted islands in the design window to delete them. The command can only be used on dynamic shapes when the *Dynamic Fill* field on the Global Shape Parameters dialog box is set to *Smooth*.

Once removed, a manual void that represents the exact shape of the island is inserted to prevent the island area from refilling during future edits to the dynamic copper fill shape. Use *Shape – Manual Void – Delete* to remove these voids.

If an area appears as an island, and the editor has not identified it as such, it is probably conductive through a via into another layer of the PCB. To delete a minimum size island for an entire board, specify the desired size in the *Suppress Shapes Less Than* field on the Void Controls tab of the Global Shape Parameters dialog box for all dynamic shapes, the Shape Instance Parameters dialog box for a particular dynamic shape, or the Static Shape Parameters dialog box for static shapes.

Access using

- Menu Path: *Shape – Delete Islands*

- Toolbar Icon

Related Topics

- shape void delete

- Global Shape Parameters

- Static Shape Parameters

Options Tab for the island_delete Command

Process Layer	Indicates the layer from which to delete unconnected etch/conductor shapes. Only layers that contain islands display here.
Total design	Displays the number of unconnected shapes in the design. This number decreases by one each time you delete or skip an unconnected shape
Total on layer	Displays the number of unconnected shapes on the layer shown in the *Process Layer* field. This number decreases by one each time you delete or skip an unconnected shape.
Delete all on layer	Select to delete every island on the layer shown in the *Process Layer* field.
Current Island	
Net	Displays the net name of the currently highlighted shape. This field is blank if the shape is not on any net.
First or Next	The *First* button initially displays. Click it to zoom to the first island in the design. The *Next* button replaces the *First* button on the Options tab once you click the *First* button. Click *Next* to skip the currently highlighted shape and zoom to the next island on that layer.
Delete	Deletes the current highlighted unconnected shape.
Report	Click to generate the Shape Island Report, which lists current shape islands on the design by layer, extents, and net, and displays in a long message window.

Deleting Unconnected Shapes

1. Choose *Shape – Delete Islands.*
   Alternatively, type `island_delete` in the Command window.
   The names of the etch/conductor layers that contain islands display in the *Process Layer* field, all islands on the first layer listed highlight in the design window, and the *First* button displays in the Options tab.
   The *Total design* field displays the number of islands in the entire design, and the *Total on Layer* field displays the number of islands on the layer that displays in the *Process Layer* field.

   If the design contains dynamic shapes whose *Dynamic Fill* on the Global Shape Parameters dialog box is set to is set to *Rough* or *Disabled*, the following displays:

   Dynamic Shapes present are not Smooth. Should I update the shapes to Smooth (yes) or exit command (no)?
   Click *Yes* to update out-of-date dynamic shapes to *Smooth*.

2. Do one of the following:

   - Click *First* to zoom to the first island in the design.
     The *Next* button replaces the *First* button on the Options tab, and as each shape centers in the design window, its net name displays in the *Net* field.

   - Click *Delete all on Layer* to delete every island on the layer that displays in the *Process Layer* field*.

3. Do one of the following:

   - Click *Delete* on the Options tab to remove the currently highlighted shape, or right mouse click, select by window, or select by group in the design window. Only highlighted shapes (islands) can be deleted.

   - Click *Next* to skip the currently highlighted shape and zoom to the next island on that layer. If you scroll through all the islands on that layer without deleting any, the following displays:
     Reached end of current island list for this layer, would you like to see earlier islands that you did not delete

4. Click *Yes* to redisplay the first island on that layer; the *First* button replaces the *Next* button; otherwise click *No.*