cādence®

E Commands

Product Version 23.1
September 2023

Contents

1

E Commands

ecadmcad	echo	ecl_schedule
ecl param	ecl terminator	edit nets
editpad boundary	editpad restore	editpad restore all
edit parts	elong_by_pick	embed prop
emc audit	emc auditrep	emc autoprop
emc execrep	emc execute	emc init
emc manprop	emcontrol	emc results
emc rulesel	encore export	encore import
enved	esc	etchback
etchedit	etch length	excellon processing
exit	explot	export all structures
export creoview	extend segments	extract_ui
extracta		

ecadmcad

The `ecadmcad` command provides an interface to set up environment for exchanging physical design data between MCAD tools and layout editor whenever any update is available.

The command detects new or updated files, and notifies you by displaying an alarm. You can either initiate the import process immediately or set the alarm to remind you later. The design data when modified in layout editor can be passed on to MCAD using the same interface. This command also supports data exchange in an IDX format.

Related Topics

- Setting up Import File Notifications

- Setting up ECAD/MCAD Collaboration Environment

- Importing Physical Design Data from MCAD

- Exporting Physical Design Data to MCAD

ECAD/MACD Set Up Window

Provides options to setup MCAD collaborative files.You can dock or undock the window individually similar to the Find, Options, and Visibility panels.

Access Using

- Menu path: *Tools – MCAD Collaboration*

Setup	Opens *ECAD/MCAD Collaboration Setup* dialog box for setting up collaboration environment	
Collaboration		
	Push Updates	Exports the design data to MCAD. The status is displayed next to the color indicator.
	Pull Updates	Imports the baseline and incremental change files from MACD systems to layout editor. The color of the indicator changes according to the environment variable *ecadmcad_status_update_interval* and the status is displayed next to the button. Color indicator displays the import status of the latest imported file. • Green: Import is up to date • Yellow: File is available for import • Red: Import failed Click the color box to view the detailed information of the import files such as file names, date and time.
Info		
	Baselined	Displays the baseline status as YES or NO.
	Repository not defined	Displayed if baseline files directory location is not set in the *ECAD/MCAD Collaboration Setup* dialog box.

ECAD/MACD Collaboration Setup Dialog Box

Reminder time interval	Overwrites the file notification time interval specified by the environment variable *im port_file_alarm_interval.* The valid time interval range is from 1 to 720 minutes. The default value is 1 minute.
File repository location	Specifies the path of the directory of the baseline EDMD files.
Export filter	Setup object types for data exchanges.
Filter Options	Opens *IDX Out Filter Setup* dialog box.
Cancel	Exits the *ECAD/MCAD Collaboration Setup* without saving any modifications after the last *OK* or *Apply.*
Apply	Applies the change in settings in the `EcadMcadCFG.txt` and `importFileManagerConfiguration.txt` files.
OK	Saves the settings and closes the dialog box.
Help	Displays the documentation.

IDX Out Filter Setup Dialog Box	
OK	Applies the filter settings in the `idxFilterOut.config` file and exits the dialog box.
Cancel	Exits without applying any filter settings.
Reset	Restores default settings.
Help	Displays the documentation.

Related Topics

- Incremental Data eXchange (IDX) format

- Setting up ECAD/MCAD Collaboration Environment

- Importing Physical Design Data from MCAD

- Exporting Physical Design Data to MCAD

ECAD/MCAD Collaboration Setup Dialog Box

Access Using:

- Button: Click *Setup* in the ECAD/MCAD dialog box.

MCAD Connector	Specify the MCAD connector. @spb231In this release, only EDMD File Collaboration Flow is supported.spb231@
File Repository Location	Specify the repository location that will be used to collaborate. @spb231In this release, only network or Cloud locations are supported.spb231@
Update Time Interval	Specify the time interval in minutes after which the repository should be checked for updates. The default is 1 minute.
Export filter	Specify the list of objects to be excluded from the output to the repository using the IDX Out Filter Setup dialog box.

Setting up Import File Notifications

To set up notifications for import file, follow these steps:

1. Choose *Setup - User Preferences*.

2. Choose *File management – Miscellaneous* from the *Categories* list.

3. Select the checkbox in the Value column corresponding to the *import_file_alarm_enable* variable*.*

4. Specify the minutes in the *Value* column for the *import_file_alarm_interval* variable*.*

5. Specify the minutes in the *Value* column for the *ecadmcad_status_update_interval* variable*.*

6. Click *OK*.

Related Topics

- Incremental Data eXchange (IDX) format

- ecadmcad

- Importing Physical Design Data from MCAD

- Exporting Physical Design Data to MCAD

Setting up ECAD/MCAD Collaboration Environment

To set up the environment for exchanging physical design data between MCAD tools and layout editor, follow these steps:

1. Choose *Tools – MCAD Collaboration*.
   Alternatively, you can also type the `ecadmcad` command in the command window. The ECAD/MACD window is displayed.

2. Click *Setup* to the configure the collaboration process.
   The *ECAD/MCAD Collaboration Setup* dialog box displays.

3. Specify *Reminder time interval* to change the interval set by the environment variable.

4. Set the directory path to save the ECAD/MCAD files in the *File repository location* field.

5. Click *Filter Options* to exclude objects from IDX output.
   The filter settings are saved in the `idxFilterOut.config` file in the working design directory.

6. Click *OK* to apply the settings and close the dialog box.
   Settings are saved in the `EcadMcadCFG.txt` and `importFileManagerConfiguration.txt` files.

Related Topics

- Incremental Data eXchange (IDX) format

- ecadmcad

- ECAD/MACD Set Up Window

- Exporting Physical Design Data to MCAD

Importing Physical Design Data from MCAD

Once you set up the environment for ECAD/MCAD collaboration, the command checks the shared directories for new or updated IDX files and displays the status in the *Info* section of the ECAD/MACD window. Using this interface you can directly import the changed data without accessing committed import commands.

To import physical design data from MCAD, follow these steps:

1. Check the status of the *Baselined* field in the *Info* section of the ECAD/MACD window.
   The status appears as *NO*.

2. Verify the color indicator displayed corresponding to the *Pull Updates* button.
   The color turns yellow and the status changes to *New MCAD available*.

3. Click the color indicator corresponding to the *Pull Updates* button.
   The `report_importFileManager.txt` file is displayed that contains the name, time and date of the new files available from MCAD.

4. Click the *Pull Updates* button.
   The *IDX Flow Manager Import* dialog box appears.

5. Review the changes displayed in the grid.

6. Click *OK* to complete the import process.
   When completed, the *IDX Flow Manager Import* dialog box closes and the `idx_in` log file is created in the design directory. A transaction report is also generated in an HTML format.

7. Check the status again of the *Baselined* field in the *Info* section of the ECAD/MACD window.
   If the physical data exchange is initiated first time, the IDX baseline is imported and the status changes to *YES*.

8. Verify the color indicator assigned to *Pull Updates.*
   If incremental changes are available, the color remains yellow.

9. Repeat the steps from 4 to 6.

10. Verify the color indicator displayed corresponding to the *Pull Updates* button.
    The color turns green indicating no more files are available and the status changes to *No MCAD data available*.
    In the *File repository location*, an IDX response file is created showing that the changes are transferred to layout editor.

Related Topics

- Incremental Data eXchange (IDX) format

- ecadmcad

- ECAD/MACD Set Up Window

- Setting up Import File Notifications

Exporting Physical Design Data to MCAD

Design changes can be exported back to MCAD systems using the same interface. To transfer the IDX incremental data from layout editor to MCAD, follow these steps:

1. Verify the color indicator corresponding to the *Push Updates* button.
   The green color indicates that the design has changed after the baseline last time and incremental changes are ready to move to MCAD side.

2. Click the *Push Updates* button.
   The *IDX Flow Manager Export* dialog box appears.

3. Click *View Change Log* to review the export process.

4. Click *OK* to complete the process.
   When completed, the *IDX Flow Manager Export* dialog box closes and the `idx_out` log file is created in the design directory.

5. Verify the color indicator corresponding to the *Push Updates* button*.
   In the *File repository location* field, an IDX increment data file is created. The color turns red and the status changes to *Waiting for Response*.

Related Topics

- Incremental Data eXchange (IDX) format

- ecadmcad

- ECAD/MACD Set Up Window

- Setting up Import File Notifications

- Setting up ECAD/MCAD Collaboration Environment

echo

The `echo` command is used in conjunction with the `scriptmode` command while running scripts. During replay, script echoes the command to the appropriate window before executing the command. This command is disabled by default.

```
echo
```

Related Topics

- scriptmode

ecl_schedule

The `ecl_schedule` batch command generates a report of the nets with ECL properties.

Syntax

`ecl_schedule` *[-version]*

ecl param

The `ecl param` command displays the *ECL* dialog box for setting the ECL (Emitter Coupled Logic) parameters.

Related Topics

- ecl terminator
- Assigning Terminator to an ECL Net

ECL Dialog Box

Use this dialog box to choose the operating characteristics for the ECL program.

Access Using

- Menu path: *Logic – Terminator Assignment*

Terminator Assignment	
Max Terminator Distance	Indicates the maximum allowable distance between a pin and a terminator assigned to the pin. The default is 32767.
Assignment Mode	- All: Executes terminator assignment on all the ECL nets. - Partial: Executes terminator assignment on the ECL nets that have the ECL_TEMP property attached to them. The default is *All*.
Log File Name	Indicates the name of the log file to be generated. The default is `terminator.log`.

The *Load Report* section contains parameters that determine the values described in the Load Report. Ensure that the values you specify in the following fields are appropriate to your design process.

Max Loads Per Net	Indicates the maximum amount of load pins allowed per net.
Max Drivers Per Net	Indicates the maximum number of driver pins allowed per net.
Lump Loading Ratio	Indicates the longest distance between any two consecutive loads in a net, divided by the total length of the net.
Number of Lump Loads	Indicates the maximum number of lump loads allowed per net.
Max Vias Per Net	Indicates the maximum number of vias allowed per net.

Max Net Length	Indicates the maximum length allowed for a net.
Log File Name	Indicates the name of the load log file (not the Load Report). The default log name is eclrep.log.

Related Topics

- ecl terminator

Assigning Terminator to an ECL Net

To assign a terminator to an ECL Net, follow these steps:

1. Choose *Logic – Terminator Assignment*.
   Alternatively, you can also type `ecl param` in the command window. The *ECL* dialog box appears.

2. Edit the values in the dialog box as required.

3. Click *OK* to apply the parameters and close the dialog box.

Related Topics

- ecl terminator

- ecl param

ecl terminator

The `ecl terminator` command assigns one terminator on each end of a net, swaps terminators to minimize net length as defined by the NO_SWAP, NO_SWAP_EXT, and GROUP properties, and generates a terminator assignment log file.

```
ecl terminator
```

Related Topics

- ecl param
- GROUP

edit nets

edit nets

The `edit nets` command lets you create a netlist without having to draw a schematic, thereby allowing you to explore design types.

Related Topics

- Creating a New Net

- Adding Pins to a New Net

- Adding Pins to a New Net from the Design

- Deleting Nets

- Modifying Existing Nets from the Edit Nets Dialog Box

- Modifying Nets from the Design

- Removing Pins from Nets

- Renaming Nets

Edit Nets Dialog Box

Access Using

- Menu path: *Logic – Edit Nets*

Use this dialog box to view and edit the netlist.

Left and Right Main Selection Areas	The Left and Right Main Selection Areas are identical to each other. They choose either entire nets or devices on a net.
Select By	Sets the display of either nets or devices in the Net Selection List Box.

When Net is Selected:

Net Filter	Searches nets by net name.
Net Selection list box	Displays the chosen nets.

When Device is selected:

RefDes Filter	Searches on device data by refdes (reference designation).
Device Filter	Searches on net data by device (device name).
Sort	Displays device data sorted by refdes or device name in the list box.
Device Selection list box	Displays the chosen devices.
Left and Right Pin Selection Areas	The Left and Right Pin Selection Areas are identical to each other They move pins to a new net or between existing nets.
Highlight Pins This Side	Highlights device pins in the design area.
Net	Displays the name of the net whose pins are displayed in the Netlist box.
Netlist box	Displays the names of pins on a net.
Pin Typein	Use for entering a single pin name.

Clear Pins	Clears the list of pins in the Netlist box.
Delete Net	Deletes the net highlighted in the Net Selection list box.
Rename Net	Renames the net highlighted in the Net Selection list box.

Related Topics

- Adding Pins to a New Net

- Adding Pins to a New Net from the Design

- Deleting Nets

- Modifying Existing Nets from the Edit Nets Dialog Box

- Modifying Nets from the Design

- Removing Pins from Nets

- Renaming Nets

Creating a New Net

To create a new net, follow these steps:

1. Choose *Logic – Edit Nets*.
   Alternately, type `edit nets` into the command window.
   The *Edit Nets* dialog box appears.

2. In the *Left Pin Selection Area*, enter the name of a new net in the Net field.

3. Press Return.
   A prompt asks if you want to create a new net.

4. Click *Yes*.
   The name is added to the Net Selection Area window.

Related Topics

- edit nets

- Adding Pins to a New Net from the Design

- Deleting Nets

- Modifying Existing Nets from the Edit Nets Dialog Box

- Modifying Nets from the Design

- Removing Pins from Nets

- Renaming Nets

Adding Pins to a New Net

Follow these steps to add pins to a new net:

1. In the *Net Selection Area*, click the new net name.

2. In the *Right Main Selection Area*, set the radio button to show pins by Net or Device.
   Note: Under Net, you may show all unassigned pins. If you choose pins by Device, unassigned pins show "—" instead of a net name on their listings.

3. Use the *Right Pin Selection Area* to assign pins to the new net. When chosen, pins move to the left side, under the new net name.

4. When all pins are added, click *OK*.

Related Topics

- edit nets
- Edit Nets Dialog Box
- Deleting Nets
- Modifying Existing Nets from the Edit Nets Dialog Box
- Modifying Nets from the Design
- Removing Pins from Nets
- Renaming Nets

Adding Pins to a New Net from the Design

Follow these steps to add pins to a new net from the design:

1. Enter the name of a new net in one of the Net fields in the dialog box.

2. Above the field, click *Highlight Pins This Side*.

3. Click a device pin in the design.
   In the dialog box, the pin information is added to the Pin Selection list box beneath the new name.

4. Click another device pin in the design.
   A ratsnest line is drawn between the chosen pins. In the dialog box, the information for the second pin is added to the list box.
   Note: Each time you click a different pin in the design, a new ratsnest line is added and the pin information appears in the dialog box.

Related Topics

- edit nets

- Edit Nets Dialog Box

- Creating a New Net

- Modifying Existing Nets from the Edit Nets Dialog Box

- Modifying Nets from the Design

- Removing Pins from Nets

- Renaming Nets

Deleting Nets

To delete nets from your design, follow these steps:

1. Choose *Logic – Edit Nets*.
   Alternately, type `edit nets` into the command window.
   The *Edit Nets* dialog box appears.

2. Choose a side of the dialog box to work in and click *Select By: Net*.

3. Set the net filter to display the specified d nets.

4. Click *Highlight Pins This Side*.

5. Choose nets for deletion in any of the following ways:

   ○ Choose a net from the Net Selection list box

   ○ Enter a net name in the Net field

   ○ Click a connection line in the design.

6. Click *Delete Net*.

Related Topics

- edit nets

- Edit Nets Dialog Box

- Creating a New Net

- Adding Pins to a New Net

- Modifying Nets from the Design

- Removing Pins from Nets

- Renaming Nets

Modifying Existing Nets from the Edit Nets Dialog Box

You can use the *Edit Nets* dialog box to modify existing nets:

1. Choose *Logic – Edit Nets*.

   Alternately, type `edit nets` into the command window.

   The *Edit Nets* dialog box appears.

2. Click left on a pin in one Pin Selection list box to move it to the other Pin Selection Area list box.

   In the design, existing ratsnest lines are ripped up and new lines drawn.

   –or–

   Click left on All to move all pins from one list box to the other.

   Note: When you move all pins off of a net by clicking on All, a prompt appears to give you the option of keeping or deleting the empty net.

Related Topics

- edit nets

- Edit Nets Dialog Box

- Creating a New Net

- Adding Pins to a New Net

- Adding Pins to a New Net from the Design

- Removing Pins from Nets

- Renaming Nets

Modifying Nets from the Design

Follow these steps to modify nets from your design:

1. Choose *Logic – Edit Nets*.
   Alternately, type `edit nets` into the command window.
   The *Edit Nets* dialog box appears.

2. Make sure *Highlight Pins This Side* is active for the Pin Selection Area list box of your choice.

3. Choose a net from the dialog box or click the target net (line) in the design.

4. Click a pin in the design.
   A new ratsnest line is drawn to the highlighted pin (its previous ratsnest lines are ripped up).

5. Repeat the previous steps until you are finished.

Related Topics

- edit nets

- Edit Nets Dialog Box

- Creating a New Net

- Adding Pins to a New Net

- Adding Pins to a New Net from the Design

- Deleting Nets

- Renaming Nets

Removing Pins from Nets

To remove pins from nets, follow these steps:

1. Choose *Logic – Edit Nets.*
   Alternately, type `edit nets` into the command window.
   The *Edit Nets* dialog box appears.

2. In one of the Net Selection list boxes, click *<Unassigned Pins>.*
   The unassigned pins appear below in the Pin Selection Area list box.

3. In the other Net Selection Area list box, click the net containing the pin you want to remove.
   The pins for the chosen net appear below in the Pin Selection Area list box.

4. Click the pin you want to remove.
   The pin moves to the list of unassigned pins in the other Pin Selection Area list box. In the design, existing ratsnest lines are ripped up.

5. To remove all pins from the chosen net, leaving the net empty, click *All*, then click *No* in the pop-up window.

Related Topics

- edit nets

- Edit Nets Dialog Box

- Creating a New Net

- Adding Pins to a New Net

- Adding Pins to a New Net from the Design

- Deleting Nets

- Modifying Existing Nets from the Edit Nets Dialog Box

Renaming Nets

You can rename nets in the design as long as you do not attempt to use an existing net name.

1. Choose *Logic – Edit Nets*.
   Alternately, type `edit nets` into the command window.
   The *Edit Nets* dialog box appears.

2. Choose a side of the dialog box to work in, then click *Select By: Net*.

3. Set the net filter to display the specified nets.

4. Choose the net you want to rename.

5. Click Rename Net.
   A prompt appears.

6. Enter the new name for the net in the prompt field.

7. Click *OK*.
   The net name changes.

Related Topics

- edit nets

- Edit Nets Dialog Box

- Creating a New Net

- Adding Pins to a New Net

- Adding Pins to a New Net from the Design

- Deleting Nets

- Modifying Existing Nets from the Edit Nets Dialog Box

- Modifying Nets from the Design

editpad boundary

The `editpad boundary` command changes the geometry for a pad while maintaining a permanent association between the pad and the package or part symbol.

Access Using

- Menu path: Tools – Pad – Boundary

Related Topics

- editpad restore

Modifying a Pad Boundary

To modify the boundary of a pad, follow these steps:

1. Choose *Tools – Pad – Boundary*.
   Alternatively, you can also type `editpad boundary` in the command window.
   A message describes the method by which the grids are drawn and you are asked to choose the starting edit point on the pad boundary.

2. Do the following in the *Options* panel:

   a. Set the active class and subclass.

   b. Set the Line Lock box.
      You can choose from either lines or arcs at either `90`, `45`, or `no` (off) degrees.

   c. Set the grid value if it is different from the currently displayed value.

   d. If you want a separator character other than a dash (-), enter the character in the separator box.

3. Define the area to be trimmed or added to the shape by doing the following:

   a. ·Choose the starting edit point on a visible (active) pad subclass.
      The starting point must be on the pad boundary. You may find it helpful to zoom in or enlarge the view of the pad before you choose the starting edit point.

   b. Click the next point (vertex).
      If you want to finish editing the shape, proceed to step 3c. Otherwise, continue selecting points to further define the area that you want to trim or enlarge.

   c. Choose the closing point on the pad boundary by clicking on another point on the pad boundary.
      You can continue to edit another pad boundary or right-click to display the pop-up menu and choose Done.

The tool displays the trimmed or enlarged pad and identifies the resulting pad shape name and new padstack name. The tool uses the existing pad shape and padstack names and increments the names by a value of 1.

If you edit a pad shape more than nine times, the tool increments the pad shape name using an alphabetical character, starting with the letter A. For each subsequent edit to the pad, the tool increments the value by the next alphabetic character, up to the letter Z.

If the pad shape name becomes longer than 18 characters (including the separator character and the incremental value), the tool prompts you to enter a new pad shape or padstack name.

Related Topics

- editpad restore

editpad restore

The `editpad restore` command restores derived pads to their original padstacks.

Access Using

- Menu path: Tools – Pad – Restore

Related Topics

- editpad restore all

Restoring Individual Derived Pads to Their Original State

To restore individual derived pads to their original padstacks, follow these steps:

1. Choose *Tools – Pad – Restore*.
   Alternatively, you can also type `editpad restore` in the command window.
   You are prompted to choose the pin or via that you want to restore.

2. Click the derived pin or via that you want to be restored.
   The tool highlights the chosen pin or via pad, and does the following:

   - Displays the restored element.

   - Identifies the name of the original padstack for the pad.

   - Runs design rule checking on the restored pin or via pad.

3. Right-click and do one of the following:

   - To continue restoring other edited pins, choose *Next* and repeat step 2.

   - To end the restoration process, choose *Done*.

Related Topics

- editpad restore all

editpad restore all

The `editpad restore all` command restores all the derived pads to their original padstacks.

Access Using

- Menu path: Tools – Pad – Restore *All*

Related Topics

- editpad restore

Restoring All Derived Pads to Their Original State

To restore all the derived pads in your design to their original padstack, follow these steps:

1. Choose *Tools – Pad – Restore All*.
   Alternatively, you can also type `editpad restore all` in the command window.
   The tool highlights all of the derived pin and via pads in your design. The tool tells you the total number of pin and via pads that are chosen for restoration.
   The tool also temporarily restores the pin and via pads and displays the restored pin and via pads.

2. Right-click to display the pop-up menu and choose *Done*.
   The tool immediately restores the edited pads to their original padstacks and performs design rule checking on them.
   If you do not want to restore the edited pads, click *Cancel*. The pads return to their derived states.

edit parts

The `edit parts` command works similar to the `partlogic` command.

This command does not create a standalone symbol instance when copying or unplacing the component. It creates new component instances when making component copies by adding a reference designator for the component. When deleting a component, it removes the component instance.

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

Access Using

- Menu path: *Logic–Edit Parts List*

Related Topics

- power integrity

elong_by_pick

The `elong_by_pick` command increases the etch/conductor length, usually in inches or mils, to adhere to the timing rules.

Access Using

Menu path:

- For Allegro X Sigrity SI, Allegro X PCB Editor: *Route – PCB Router – Elongation By Pick*
- For APD: *Route – Router – Elongation By Pick*

Related Topics

- [Automatic Router Parameters Dialog Box](#)

Increasing the Etch/Conductor Length

To increase the etch or conductor length, follow these steps:

1. Choose *Route – PCB Router – Elongation By Pick*.
   Alternatively, you can also type `elong_by_pick` in the command window.

2. Right-click to display the pop-up menu and choose *Setup*.
   The *Automatic Router Parameters* dialog box appears with the *Elongate* tab selected.

3. Select the required options.

4. Click *OK* to save the changes and close the dialog box.

5. Choose a net or nets that you want to be elongated.

6. Right-click and choose *Done*.

Related Topics

- Automatic Router Parameters Dialog Box

embed prop

This is an obsolete command.

emc audit

This is an obsolete command. See emcontrol instead.

emc auditrep

This is an obsolete command. See emcontrol instead.

emc autoprop

This is an obsolete command. See emcontrol instead.

emc execrep

This is an obsolete command. See emcontrol instead.

emc execute

This is an obsolete command. See emcontrol instead.

emc init

This is an obsolete command. See emcontrol instead.

emc manprop

This is an obsolete command. See emcontrol instead.

emcontrol

The `emcontrol` command displays the *EMC Rule Checker* dialog box and enables you to repeatedly check your design for EMI violations against a pre-chosen sets of rules by running the EMControl system.

EMControl includes several default sets of EMI rules. You can also write your own rules to verify specific design, environment, and regulatory requirements. Running EMControl early in the design cycle often helps to detect potential EMI problems before they can significantly impact product development.

⚠ When you run EMControl in the current working directory for the first time, the *EMC System Configuration* dialog box displays to enable you to fill in your system configuration information first. You can also access this dialog box from the Setup tab of the *EMC Rule Checker* dialog box to change the settings.

ⓘ This command is available only with Allegro X PCB Editor.

Access Using

- Menu path: *Analyze – EMI Rule Checker*

emc results

This is an obsolete command. See emcontrol instead.

emc rulesel

This is an obsolete command. See emcontrol instead.

encore export

Available in a future release.

encore import

Accessible using an environment variable.

enved

The `enved` command displays the User Preferences Editor, which lets you set or unset environment variables (preferences) directly from a graphical user interface rather than in your local `env` file or from the console command window. A *My Favorites* category centralizes frequently accessed variables.

Related Topics

- Setting Environment Variables
- Adding Variables to the My Favorites Category

User Preferences Editor

Access Using

- Menu path: *Setup – User Preferences*

Categories	A functionally classified tree view of the environment variables that you can enable or disable. When you choose a category, the individual variables (preferences) associated with the category display.
Search for preference	Enter a variable name or other string and click the *Tab* key or *Search* button to search variable names in all categories for a match. Use the * wildcard to enter a partial string; for example, drc*.
Include Summary in Search	Click to search the summary description in addition to the categories for the variable name or string entered in the *Search for preference* field.
Category: <type>	
Preference	A list of the environment variables (preferences) associated with the chosen category.
Value	Clicking a check box sets variables with on/off states. Edit fields appear if string or number data must be entered. Some preferences have values that can be chosen from a drop-down menu; in these instances, selecting the empty entry deselects the value. Path values must be entered in the Physical Paths window that displays for preferences in the *Paths* category.
Effective	Indicates when the changed state of the variable takes effect. *Command*: When you run the next command related to the preference *Immediate*: As soon as you click *OK* in this dialog box *Repaint*: When you reset your view of the work area *Restart*: After restarting the tool
Favorite	Click to include the variable in the *My Favorites* category if it is unchecked, or remove it if it is already checked.
Summary description	Describes the function of a variable or edit box when the cursor hovers over it.

OK	Updates the current session for any user preference changed by updating the local `env` file, then closes the dialog box.
Cancel	Cancels any changes and closes the editor without updating the `env` file.
List All	Displays a viewer listing all current environment variables.
Info	Summarizes help descriptions for all environment variables.

Related Topics

- Adding Variables to the My Favorites Category

Setting Environment Variables

To setup environment variables, follow these steps:

1. Choose *Setup – User Preferences*.
   Alternatively, you can also type `enved` in the command window.

2. Choose a category from the *Categories* list, or enter a name in the *Search for preference* field. To search the summary description, enable *Include Summary in Search.*

3. Click the *Tab* key or *Search* button. The list of preferences (environment variables) associated with the category displays. If the list extends beyond one page, the *Previous/Next* button appears to allow scrolling.

4. Do any of the following to change the values for preferences:

   - Select or deselect the check box

   - Entering or deleting data in the edit field (by typing in a value or selecting one from a drop-down menu, where available)

   - Resetting paths in the physical paths windows in the *Path* category.

   Information related to the variable appears in the *Summary description* section.

5. To view a text file listing all current settings, click *List All*.

6. Click *OK* to save your changes and close the editor.

Related Topics

- enved

Adding Variables to the My Favorites Category

The *My Favorites* category in the *User Preferences Editor* dialog box displays the frequently accessed variables. Selecting the *Favorites* check box next to a variable includes it in this category. To add a variable to *My Favorites*, follow these steps:

1. Choose *Setup – User Preferences* to display the User Preferences Editor.

2. Choose a category from the *Categories* list, or enter a name in the *Search for preference* field.

3. Click the *Favorites* check box next to the variable to include it in the *My Favorites* category (if it is unchecked) or to remove it, (if it is already checked).

Related Topics

- enved
- User Preferences Editor

esc

This is an obsolete command.

etchback

The `etchback` command lets you create etch-back shorting elements that connect nets together (generally in a daisy-chain) for the purpose of plating bar connectivity. It also lets you create etch-back masks that are used during manufacturing to remove the etch-back shorting elements after the plating process is complete.

The log file for this command contains information about all that happened during execution time as well as information about the detailed processing that did not appear in the Command window prompt. The log file is saved as `etchback.log`.

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

Related Topics

- pbar check

- xsection

- Creating Etch-Back Traces

Etch-Back Dialog Box

Access Using

- Menu path: *Manufacture – Etch-Back*

- Toolbar Icon:

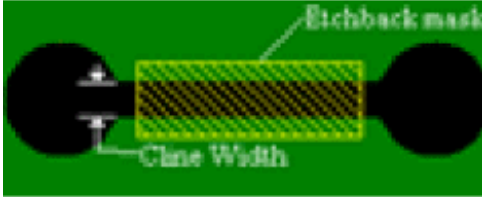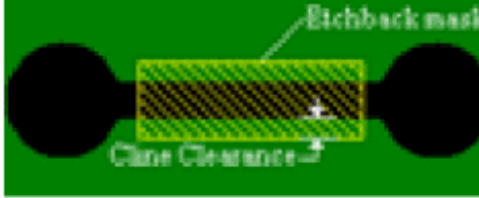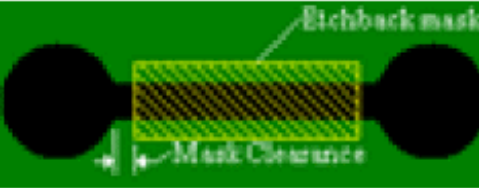|	
Conductor	Specifies the name of the conductor layer where the etch-back shorting occurs. You can use the etch-back tool only on the top and bottom layers. The default setting is the top conductor layer of the current design. Bonding wire and plane layers are excluded from the top and bottom layer considerations.
Masks	Specifies the manufacturing subclass name for the *Conductor* class (specified above) on which to draw the etch-back masks. This value defaults to a layer containing etch-back masks, or when there are no masks, it defaults to an arbitrary manufacturing layer that you can change. Once the tool finds a layer with masks or you create masks on the layer, you can rename the layer but cannot create more than one mask layer for each substrate layer. For easier reference in the tool, it is recommended that you use a similar naming scheme for your trace and mask layer pairs, for example: TOP_COND (existing conductor layer) EB_MASK_TOP_COND
Etch-back Mode	
Add trace	Specifies trace mode. Click this button to create etch-back traces. You can edit the parameters for mask mode during trace mode when you enable the automask option.
Add mask	Specifies mask mode. Click this button to create etch-back masks. You cannot edit the parameters for trace mode when you are in mask mode.
Delete etch-back objects	Allows you to delete etch-back objects. If you select masks, the tool deletes the mask and flags the trace as unmasked. If you delete the trace, the tool removes the trace and related DRC markers.
Trace Parameters	

Line lock	Specifies the corner-style to use when adding etch-back shorting traces. Choices are *Off*, *45*, and *90*. The default value matches the value currently set for the `add connect` command.
Line width	Specifies the width of the etch-back shorting traces as shown below. The default value is the minimum line width constraint value on the conductor layer of the design (see the command or check your physical constraints).
Allow any-angle pin escape	Check this box if you do not want the default setting in which the tool snaps the current rubber band segment to the via or pin origin. If you check this box, the tool snaps to pins by creating a segment to the exact area on the pin or via that you request. Then it creates an any-angle segment from that point to the center of the pin. This is critical for dense designs as there may not be enough room to snap to the center of the pin from the normal entry angle.
Do not void shapes around traces	Check this box if you do not want dynamic shapes to avoid the etch-back trace.
Auto create mask using settings below	Check this box to automatically create a mask as you create etch-back traces using the parameters you set in the *Etch-back Mask Parameters* section of the *Etch-Back* dialog box. This option eliminates adding the mask as a second step after you create the etch-back traces. If automatic mask creation fails, you will get an unmasked trace DRC error. You can adjust the mask settings and create a new mask or delete the etch-back trace.
Mask Parameters	
Style	Select the style of the etchback mask to be created. Available options are: • *Maximum Removal* • *Square Ends*

Line clearance	Specifies the exact distance required between the shorting element line and the etch-back mask as shown below. The default value is the value of the design's *Minimum aperture for artwork fill* constraint found in the *Global Dynamic Shapes* dialog box (see the `shape global param` command).
Antenna length	Specifies the mask pullback from the objects connected by the etch-back trace. The minimum mask spacing is used to check for spacing violations between the newly generated etch-back mask and other surrounding objects.
Min mask clearance	Specifies the exact distance required between the etch-back mask and conductors other than the shorting element as shown below. This value does not constrain how close etch-back masks can be to each other. The masks can overlap. The default value is the value of the design's *Minimum aperture for artwork fill* constraint found in the *Global Dynamic Shapes* dialog box (see the `shape global param` command). The mask clearance may also include conductors surrounding the mask that are not directly involved with the source and destination conductors of the etch-back trace.
Max mask width	Specifies the maximum width that an etch-back mask can have. The default value is ten times the minimum mask width. However, the tool rejects mask generation requests if they result in masks being larger than the other values you specified.

Min mask width	Specifies the minimum width that an etch-back mask can have. The default value is the value of the *Suppress shapes less than* constraint found in the *Global Dynamic Shapes* dialog box.
Allow DRCs to surrounding metal	Check this option to create the etch-back mask with the defined line clearance even if the etch-back mask shape does not have enough clearance to conductor shapes along its path. As a result, spacing DRC violations appear in the design.
Allow masking of plating traces	If you check this box, the tool lets you select a plating trace and generates the appropriate mask for it. The default state for the check box is disabled. This option improves the electrical characteristics of the package.
Check for mask violations	If you click this button, the tool checks the entire design, and highlights and attaches DRC markers to any etch-back traces that do not have valid etch-back masks.

Related Topics

- xsection

Creating Etch-Back Traces

You can create etchback traces on shapes, clines, pins, and vias on the exposed (top and bottom) substrate layers only. All etchback traces not covered by etchback mask are treated as clines by the signal integrity and electrical analysis tools.

1. Run the `pbar check` command to check the plating bar violations.

2. Choose Manufacture - Etch Back
   Alternatively, you can also type `etchback` in the command window.
   The *Etch-Back* dialog box appears. Ensure that you select the correct *Conductor* layer.

3. Select the *Auto create mask using settings below to automatically check box to auto-create the mask as you create the traces.*

4. Select the starting point in the design (pin, cline, or pad) and move the cursor towards the ending point.
   Rubber banding follows the cursor. Any click following the start click that is not an end point (pin, net, or pad) acts as an anchor point for the line. Mask appears as soon as you complete the etchback traces.

5. Repeat step 3 until you have created all the etch-back traces.

6. Click *Check for Mask Violations* in the *Etch-Back* dialog box to check for any unmasked traces.

7. Right-click and choose *Done*.

8. Run the `pbar check` command to check that all the plating bar violations are gone.

Upon exiting the etch-back command, the tool generates a report that lists all the etch-back violations.

Related Topics

- pbar check

- xsection

- etchback

etchedit

The `etchedit` command enables the Etch-edit application mode that allows you to customize your environment to perform etch-editing tasks such as adding and sliding connections, delay tuning, and smoothing cline or cline segment angles. An application mode provides an intuitive environment in which commands used frequently in a particular task domain, such as etch editing, are readily accessible from right-mouse- button popup menus, based on a selection set of design elements you have chosen.

This customized environment maximizes productivity when you use multiple commands on the same design elements or those in close proximity in the design. Application mode configures your layout editor for a specific task by populating the right-mouse-button popup menu only with commands that operate on the current selection set.

To exit from the current application mode and return to a menu-driven editing mode, choose *Setup – Application Mode – None*.

Access Using

- Menu path: *Setup – Application Mode– Etch Edit*

- Toolbar icon:

Related Topics

- noappmode

- Etch-edit Application Mode Automatic Command Execution

etch length

The `etch length` command displays the current pin-to-pin and total net etch lengths on the second status line of the design window. You can use this command during *Route – Connect* processing.

The display format is

```
<refdes.pin#> – <refdes.pin#> = <connection length> Net = <net_length>
```

This command shows the current length changing as the rubber band cursor moves. This lets you create connections of a required length. Once you start the etch length display, it stays in effect until you end the Connect process by clicking on Done .

excellon processing

The `excellon processing` command changes A codes to I and J codes in your Excellon file, adds and delete sequence numbers and changes the end-of-block character used in the Excellon file.

Excellon File Processing Dialog Box

Input File	Identifies the file to be processed.
Output File	Identifies the name of the file that is created during processing.
Change A to I/J codes	Indicates whether the A codes in the input file are changed to I and J codes.
Process sequence numbers	Indicates whether sequence numbers are processed. You must specify whether sequence numbers are added or deleted.
Change EOB character	Indicates whether the end-of-block character is changed during processing. You must identify the current EOB character and specify the new EOB character. If you leave the *Current eob char* field blank, the editor interprets the blank as a null string and places the new end-of-block character at the end of each line. After entering the values for Excellon file processing, choose Execute to process the file. The editor stores the results in a temporary file. Press Done to save the results in the file you specified as the Output file.

Related Topics

- gerber processing

exit

The `exit` command saves the active layout, exits the editor, and returns to the host operating system. The command displays a browser window asking for a name under which to save the active layout. The default is the name of the active layout. If you do not enter a name but click OK , the command displays a dialog box asking whether you want to overwrite the existing layout and exits. If you enter a new name, the command writes the layout to that filename and exits.

In a co-design environment, the `exit` command checks for unsaved co-design dies and asks whether to save or discard the changes.

Access Using

- Menu path: *File – Exit*

explot

The `explot` batch command creates Intermediate Plot Files (`.plt`) and control files (`.ctl`) from Excellon Drill files derived from a design, which are used as input for `hp_plot`. Use this latter file to generate plots of your drill files.

Before executing the `explot` command, the NC Drill parameter file should be accessible through the NCDPATH environment variable. If it is not found, a default set of parameter values is used. Each drill is plotted as a circle of the specified diameter. The command outputs a summary of numbers of each drill size and estimated tape length at the terminal.

Syntax

`explot` *[-r] [-p] drill_file_name [penplot_file_name]*

-r	Indicates that the Excellon drill file was created using the `ncroute` command. The drill file has an extension of `.rou`. This option must be used if running `explot` on a drill file created in this way.
-p	Displays the path that the drill head takes in the resulting IPF file. This field is optional. When this option is used, a line connects the drill points in the order in which they are processed. This option does not affect the results when `explot` is run on an Excellon drill file created using the `ncroute` program.
drill_file_name	Name of the existing Excellon drill file. This field is required. The `.drl` or `.rou` extension is not required. If you do not enter a drill file name, or the name you enter cannot be found, `explot` asks for a drill file name.
penplot_file_name	Name of the output file. This field is optional. If you do not enter an output file name, `explot` uses the drill file name for the `.plt` and `.ctl` files. When an input file name is not supplied, and `explot` asks for the `drill_file_name`, it also asks for the `penplot_file_name`. You can enter a name or press Enter only for this second file name. If you press Enter , `explot` uses the drill file name for the output.

Examples

- The following example reads the Excellon drill file `thru_drills.drl` and creates the IPF file `thru_drills.plt` and the control file `thru_drills.ctl`.

  explot thru_drills

- The following example reads the Excellon drill file `thru_drills.drl` and creates the IPF file `plot1.plt` and the control file `plot1.ctl`. The IPF file contains lines indicating the path of the drill head.

  explot -p thru_drills plot1

- The following example reads the Excellon drill file `route.rou` and creates the IPF file `plot1.plt` and the control file `plot1.ctl`.

  explot -r route plot1

Related Topics

- Using Drill Template Files

export all structures

The `export all structures` command extracts all structures from a design and save in a user-defined directory. The command internally checks for all the structures in a design and opens a file bowser to choose a directory to saves the structure definitions in an XML format.

Access Using

- Menu path: *Route – Structures – Export All*

Related Topics

- export creoview

export creoview

The `export creoview` command extracts the design data `(.brd)` and converts it into a PTC's Creo View compatible database `(.bri)`. The command internally checks for the Creo View Interface for ECAD in the install hierarchy. If the interface does not exist, you are directed to the PTC website.

Access Using

- Menu path: *File – Export – Creo View*

extend segments

The `extend segments` command extends two non-parallel lines or arc segments to a projected intersection point.

Access Using

- Menu path: *Manufacture – Drafting – Extend Segments*

Extending Line Segments

To extend two non-parallel lines or arc segments to a projected intersection point, follow these steps:

1. Choose *Manufacture – Drafting – Extend Segments.*
   Alternatively, you can also type `extend segments` in the command window.

2. Select a line or an arc segment.
   The selected segment is temporarily extended and highlighted.

3. Select another line or an arc segment.
   Both the extended segments are temporarily extended and highlighted and a possible intersection is displayed.

4. Click to choose an intersection point.
   The segments are extended and joined at the selected point.

5. Right-click and choose *Next* to continue or *Done* to complete the operation.

extract_ui

The `extract_ui` command defines new report configurations that you customize using `extracta` command files or modifies existing custom reports. You can configure reports using all existing `extracta` data fields, properties, and groups. Standard `extracta` object type qualifiers may be applied to the properties included as data fields in the report.

- You cannot create reports based on multiple views

- Any record that contains an equal sign (=), known as a selection record, is not supported.

- Any record with only the word OR is not supported

Related Topics

- extracta

- Creating a Customized Report Based on a Predefined View

Extract UI Dialog Box

This section describes the field available in the different tabs of the Extract UI dialog box.

Data Fields Tab

Use this tab to choose the view and the type of elements that you want as data fields in the custom report you are defining.

Select Database view	Choose a predefined view, which contains data fields defining the type of elements to extract from the design database and include in the custom report. Changing this field to another view prior to saving or loading your report clears the right pane of all choices you made.
Available Fields	Choose the data fields to include in the report. The data fields that display are those associated with the view chosen in the *Select Database* field. Click on a data field to include it in the custom report. Your choices appear in the right pane. Click the arrow buttons next to the right pane to re-arrange the order in which the data fields appear in the generated report if necessary
Current Configuration	
Database View	Displays the view chosen in the *Select Database* field.
Cancel	Click to close the dialog box without creating a custom report.
Load	Click to open a file browser from which you can choose to modify an existing custom report with the changes you have made.
Save	Click to save a new custom report configuration to the output file you specified. The report is saved with a .txt extension.

Properties Tab

Use this tab to extract a property as a data field record and include it in a custom report by doing either of the following:

- Specifying the property name

- Combining the property with an object type qualifier

For example, to create a pick and place report, select the `PART_NUMBER` property as a data field record.

Property Filter	Choose to display all properties or only those that can be attached to *Geometry, Logic,* or *Symbol*-related elements in the *Available Properties* list. User-defined properties also appear in the list if you choose *All*.
Object Qualifier	Choose an object type qualifier to combine with a property. Specifying *<object_t ype>* and *<property_name>* causes the report to include the property only if it is attached to that object type, where *<object_type>* is *BOARD, COMP, FUNC, GEO, GRP, NET, PIN, SYM, or VIA*.
Available Properties	
double click to select	Choose the property to include as a data field in the custom report. Click on a property to include it as a data field in the custom report. Your choices appear in the right pane. Click the arrow buttons next to the right pane to re-arrange the order in which the data fields appear in the generated report if necessary
Current Configuration	
Database View	Displays the view chosen in the *Select Database view* field.
Cancel	Click to close the dialog box without creating a custom report.
Load	Click to open a file browser from which you can choose to edit an existing custom report.
Save	Click to save a new custom report configuration to an output file with a `.txt` extension.

Miscellaneous Tab

Group Names	Choose the group names to include the membership of that group in the report. Click the arrow buttons next to the right pane to re-arrange the order in which the data fields appear in the generated report if necessary For instance, use `XNET_GRP_NAME` to report the nets within each extended net (xnet). For additional information, see *Working with Groups and Modules* in the *Placing the Elements* user guide in your documentation set, or Working with Objects in the *Constraint Manager User Guide* for information on defining group membership.
Sort Commands	Choose the data fields by which to sort extracted data fields in the report. Each data field derives from its corresponding original data field (for example, `FUNC_DES_ SORT` from `FUNC_DES`) by separating the field into an alphabetic and numeric subfield, and expanding the numeric subfield to a wide, right-justified field. This assures that a standard sort of the file results in a reasonable sort order. For example, the function designators `FUNC1, FUNC2, FUNC3`, and `FUNC10` sort to `FUNC1, FUNC10, FUNC2`, and `FUNC3`.
Current Configuration	
Database View	Displays the view chosen in the *Select Database view* field.
Cancel	Click to close the dialog box without creating a custom report.
Load	Click to open a file browser from which you can choose to modify an existing custom report with the changes you have made.
Save	Click to save a new custom report configuration to the output file you specified. The report is saved with a `.txt` extension.

Related Topics

- extracta

Creating a Customized Report Based on a Predefined View

You can customize a report based on a predefined view.

1. Run `extract_ui` or click *New/Edit* on the *Reports* dialog box that appears when you run `reports`.
   The *Extract UI* dialog box appears.

2. In the *Select Database view* field on the *Data Fields* tab, choose the view from which you want to extract data fields and include in the custom report. All data fields associated with that view appear in the *Available Fields* list.

3. In the *Available Fields* list, click the fields you want to include in the report. The data fields you choose appear in the right pane.

4. In the *Property Filter* field on the *Properties* tab, choose to display all properties or only those you can attach to *Geometry, Logic,* or *Symbol* elements.

5. Choose the properties to include in the report in the *Available Properties* list by clicking on each one.The data fields you choose appear in the right pane.

6. Choose an object type qualifier as required.

7. In the *Group Names* field on the *Miscellaneous* tab, choose group names to include the membership of these groups in the report as required.

8. In the *Sort Commands* field, choose a command by which to sort the report.

9. Click the arrow buttons next to the right pane to re-arrange the order in which the data fields appear in the generated report, if necessary. Double click a field to delete it if necessary.

10. To save a new custom report based on your choices, click *Save*. In the *Save As* dialog box that appears, enter the name of the `.txt` file to which to save the report, then click *Save*.

11. To load an existing custom report and overwrite it with the modifications you have made, click *Load*.

12. Choose the named `.txt` file you created and click *Open* to save the file.

13. Click *Exit* on the *Extract UI* dialog box.

14. On the *Reports* dialog box, select the report name you just created or modified from the *Available Reports* list by double clicking on it. The chosen report appears in the *Selected Reports* list.

15. To display the report on screen in an HTML-enabled window, enable the *Display Report* field.

16. To write the report to a text file in Comma Separated Value format, enable the *Write Report* field.

Related Topics

- extract_ui

extracta

The `extracta` batch command obtains flattened information from a design from information contained in the `cmdfile`.

To control which data is extracted from a design database, `extracta` references a command file as input during the extract process. This command file must exist before you can run `extracta`.

Syntax

`extracta` *[args] [<drawing>] [<cmdfile>] [<outfile>...]*

-c	Dumps interior of cross hatch shapes as individual lines.
-d	Dumps all field names. The file names are unused, and the output goes to the log (extract.log) file.
-k	Precludes generation of errors if the `cmdfile` has illegal field names.
-m	Overrides the default behavior of the command of renaming the outermost etch layers to TOP/BOTTOM. This only works for APD designs.
-q	Prevents `extracta` from displaying status messages during processing (quiet mode).
-r	Reuses the log file (delete before execution).
-l\<logfile\>	Name of the logfile. Overrides the default logfile name (extracta.log) and location; defined by ads_sdlog environment variable.
-s	Provides a short output format (only A record).
-A	Lists all database attachments in the design. ⚠ This option does not require any other options.

-a\<name>	Extracts the attachment with a given name to a file \<name>.dat. To extract more than one attachments, use this option multiple times on the command line. ⚠ This option does not require any other options.
-w	Excludes the date in output file.
-z	Generates a unique net name for items on dummy net in non-net views.
\<drawing>	Name of design database.
\<cmdfile>	Name of extract command file. Uses TEXTPATH to locate file if relative path is given
\<outfile>...	Name of output file. If multiple views are given in the cmdfile then one output file is needed for each file. If no outfile files are given then output is dumped to *stdout*.

Related Topics

- extract_ui
- Running Extracta on a Unix Machine
- Running Extracta in a Windows Machine

Creating an Extract Command File

To create an extracta command file, follow these steps:

1. Create a command file using any text editor.
   The file can be located anywhere in the $EXTPATH setting but is typically kept in the current working directory of the database.

2. Enter the view name and data field names that defines the types of elements to be extracted when you run `extracta`, or use a baseview and modify it to meet your requirements.
   Example:

   #

   # SAMPLE SYM_COMP COMMAND FILE NAMED SYMCOMP.TXT

   #EXTRACT THE NAMES OF ALL MECHANICAL SYMBOLS

   SYMBOL

     SYM_TYPE = "MECHANICAL"

     SYM_NAME

   END

   #NOW EXTRACT REDDES AND DEVICE TYPE FOR ALL IC AND IO COMPOMENTS

   COMPONENT

   COMP_CLASS = "IC"

   OR

   COMP_CLASS = "IO"

   REFDES

   COMP_DEVICE_TYPE

   END

   This simple example contains various record types (lines in a text file). The data generates the output files `symcomp.txt`, `symbol_output.txt`, and `component_output.txt`, containing one line of text for each mechanical symbol in the database, the name of every symbol in the database, and all the component reference designators in the database that are either an IC or an IO.

> ⚠ To facilitate the writing of command files, the editor provides predefined views called `baseviews`, text files that you can copy and edit for use in your command files. For additional information, see Baseview Files in the Completing the Design user guide in your documentation set.

3. Save and close the file after you have completed entering data.

Related Topics

- Sample Output Files
- Running Extracta in a Windows Machine

Running Extracta on a Unix Machine

1. At your system prompt, enter the `extracta` command on a single line.
   If you enter `extracta` at an operating-system prompt and do not enter any file names, the editor prompts you for the name of the design, command file name, and extraction file name, as shown in the following example (`italics` indicate file names you provide).

   $ extracta

   Layout name (*.brd): *abc*

   Extract command file (*.txt): *board_baseview*

   Extract output file name (*.txt): *ext1*

   File 'ext1.txt' already exists, overwrite it (yes/no)? y

   Additional output file name (<return> if none) (*.txt)

   Extract output file name [.txt] <Return>

   Extract started: command file is 'board_baseview.txt'.

   Extract ended .

You can view a copy of the command file and any error messages generated by the `extracta` command in the `extract.log` file.

Related Topics

- extract_ui
- extracta

Running Extracta in a Windows Machine

Follow these steps to run the command `extracta` in a Windows environment:

1. Open a Run dialog box.

2. Enter the extracta command with the appropriate switches and arguments. Or, specify the file names when prompted.

3. Click *OK* to run the extract process.

You can view a copy of the command file and any error messages generated by the `extracta` command in the `extract.log` file.

Related Topics

- extract_ui

- extracta

- Creating an Extract Command File